

PEOPLE'S DEMOCRATIC REPUBLIC OF ALGERIA  
MINISTRY OF HIGHER EDUCATION AND SCIENTIFIC RESEARCH  
MOHAMED BOUDIAF UNIVERSITY - M'SILA



FACULTY OF TECHNOLOGY  
DEPARTMENT: CIVIL ENGINEERING

DOMAIN: Science and Technology  
STREAM: CIVIL ENGINEERING  
OPTION: STRUCTURES

*The influence of nonlinear finite element  
modeling assumptions on the seismic response  
demands of steel buildings*

Dissertation submitted to the department of Civil Engineering in partial  
fulfilment of the requirements for the Degree of Master

Submitted by:

**HAFIDI SAFIA**

Board of Examiners

Dr. Abderachid BOULAOUAD	University of M'sila	Chairperson
Dr. Djamel OUZENDJA	University of M'sila	Examiner
Dr. Abdallah Yacine RAHMANI	University of M'sila	Supervisor

**JUIN 2023**

## **Dedication**

Dedication to all those who have supported and encouraged me throughout this work.

*SAFIA HAFIDI*

## ***Acknowledgements***

I would like to express my gratitude to God for my success in this work.

This thesis would not have been possible without the unconditional love and support of my parents, Ameer Hafidi and Khaira, my sisters, Amina and Zahia with the encouragement of my friends Cherifa and Yacine, to them, I want to express all my love and my gratitude since all I am is because of them.

I would like to express acknowledge the valuable guidance and advice provided by my supervisor, Professor Abdullah Rahmani Yacine.

I would like to extend my sincere gratitude to the committee overseeing the evaluation of this work, I appreciate their time and effort in thoroughly reviewing and evaluating the content, and I am grateful for their valuable insights and recommendations. and I am truly thankful for their guidance and support.

I would like to express my thanks to my friends, with a special mention to Maryam, and Imane. Their support has been invaluable throughout this endeavor.

***SAFIA HAFIDI***

# Summary

RESUME.....	i
الملخص.....	i
ABSTRACT .....	i
List of Figures: .....	ii
Table List.....	iii
GENEAL INTRODUCTION .....	1
I.1. INTRODUCTION:.....	3
I.2. EARTHQUAKES: .....	3
I.3. Earthquakes and their influence on buildings .....	4
I.4. STEEL BUILDING IN SEISMIC AREAS: .....	6
I.5. LINEAR AND NONLINEAR ANALYSES .....	6
I.5.1. Linear analysis .....	7
I.5.2. Nonlinear analysis .....	8
I.6. Structural nonlinear behavior:.....	10
I.7. Plastic hinge .....	10
I.8. Seismic analysis methods in seismic codes: .....	11
I.8.1. Linear Static Method (Equivalent static method):.....	12
I.9. Non-Linear Static Method (Pushover Method).....	13
I.9.1. Definition of pushover analysis:.....	13
I.9.2. Pushover curve .....	14
I.10. Response spectrum method.....	14
I.10.1. Definition of a Response Spectrum .....	14
I.11. Nonlinear time history analysis .....	15
I.12. Conclusion:.....	16
Chapter II. Nonlinear finite element .....	17
II.1. INTRODUCTION:.....	17
II.2. Element modeling .....	17

II.2.1. Elastic element: .....	18
II.3. FIBER ELEMENT.....	24
II.4. Inelastic Element.....	25
II.4.1. The inelastic behavior of materials and structures .....	26
II.5. Curves in nonlinear elements.....	28
II.6. Non-Linear Element Classes: .....	30
II.6.1. Inelastic force-based frame element type – infrmFB .....	31
II.6.2. Inelastic force-based plastic hinge frame element type- infrmFBPH .....	31
II.6.3. Inelastic displacement-based plastic hinge frame element type – infrmDBPH.....	33
II.6.4. Inelastic displacement-based frame element type - infrmDB .....	34
II.7. CONCLUSION.....	35
Chapter III. Pushover analysis .....	36
III.1. Introduction .....	36
III.2. Pushover analysis .....	36
III.3. Theoretical background of pushover analysis .....	37
III.4. Conventional pushover analysis methods.....	38
III.5. The N2 Method.....	39
III.5.1. . Definition of the equivalent inelastic system SDOF.....	39
III.5.2. Seismic demand .....	40
III.5.3. Global seismic demand for the MDOF model .....	41
III.6. Pushover analysis advantages.....	42
III.7. Pushover analysis Limitations .....	42
III.8. Conclusion .....	42
Chapter IV. Case study.....	43
IV.1. Introduction: .....	43
IV.2. Description of the Frame Structures:.....	43
IV.2.1. 3.1.2 Material Properties of the frame: .....	45
IV.3. Presentation of the used Software .....	45

IV.4. ELEMENT MODELLING.....	46
IV.5. RESULTS OF THE STEEL STRUCTURE .....	47
IV.5.1. 5.1. Priliminare results .....	47
IV.5.2. 5.2 Pushover curve : .....	48
IV.5.3. 5.3 Storey displacement profile .....	48
IV.5.4. 5.4 Inter-storey drift ratio: .....	49
IV.6. RESULTS OF THE RINFORCED CONCRETE (RC) STRUCTURE.....	50
IV.6.1. 6.1. Priliminare results .....	50
IV.6.2. 6.2 Pushover Curve.....	50
IV.6.3. 6.3 Storey displacement profile: .....	51
IV.6.4. 6.4 Inter-storey drift ratio: .....	52
IV.7. CONCLUSION: .....	52
GENERALE CONCLUSION .....	53
References .....	54

## RESUME

Cette étude examine l'impact des hypothèses du modèle élémentaire sur le comportement sismique global des bâtiments en acier auto stable. Deux types de structures, un bâtiment en acier et une structure en béton armé, ont été étudiés afin d'évaluer leurs caractéristiques dynamiques non linéaires. Les résultats montrent que les hypothèses du modèle élémentaire ont une influence significative sur le comportement sismique global des bâtiments en acier et en béton. Par conséquent, il est essentiel de choisir ces modèles avec attention. Ces résultats soulignent l'importance d'une représentation précise et d'un étalonnage approprié des modèles élémentaires pour garantir des prédictions fiables des performances sismiques dans la conception et l'analyse structurale.

## الملخص

تبحث هذه الدراسة في تأثير افتراضات النموذج الأساسية على السلوك الزلزالي العام للمباني الفولاذية ذاتية الاستقرار. تمت دراسة نوعين من الهياكل، مبنى فولاذي وهيكلي خرساني مسلح، من أجل تقييم خصائصها الديناميكية غير الخطية. تظهر النتائج أن افتراضات النموذج الأولي لها تأثير كبير على السلوك الزلزالي العام للمباني الفولاذية والخرسانية. لذلك، من الضروري اختيار هذه النماذج بعناية. تسلط هذه النتائج الضوء على أهمية التمثيل الدقيق والمعايرة المناسبة للنماذج الأساسية لضمان تنبؤات موثوقة للأداء الزلزالي في التصميم والتحليل الهيكلي.

## ABSTRACT

This study investigates the impact of element model assumptions on the global seismic behavior of moment-resisting steel buildings. Two types of structures, a steel building and a reinforced concrete structure, were examined to evaluate their respective compression characteristics. The findings reveal that the element model assumptions significantly influence the overall seismic behavior of both steel and concrete buildings. Therefore, careful consideration must be given to the selection of these models. The results emphasize the importance of accurate representation and proper calibration of element models to ensure reliable predictions of seismic performance in structural design and analysis.

## List of Figures:

<b>Figure I:1.</b> Seismic Hazard map.....	4
<b>Figure I:2.</b> The behaviour of structural elements under the influence of various earthquakes . (b): Before and after Turkey's earthquake: a modern residential building in Alexandrina .....	4
<b>Figure I:3.</b> Some earthquake effects on buildings. ....	5
<b>Figure I:4.</b> Photos from Turkey and Syria's recent earthquake .....	6
<b>Figure I:5.</b> Linear relationship between force and displacement.....	8
<b>Figure I:6.</b> Nonlinear load-displacement curve .....	8
<b>Figure I:7.</b> Geometric nonlinearity .....	9
<b>Figure I:8.</b> Material non linearity .....	9
<b>Figure I:9.</b> Plastic hinge .....	11
<b>Figure I:10.</b> Plastic hinge length .....	11
<b>Figure I:11.</b> Time cost scale for seismic analysis techniques. ....	12
<b>Figure I:12.</b> Pushover-curve-of-structure .....	14
<b>Figure I:13.</b> Response spectrum curve. ....	15
<b>Figure I:14.</b> Time history analysis .....	15
<b>Figure II:1</b> elastic behaviour of materials ;(b) Stress-strain relationship of a linear material. ..	20
<b>Figure II:2.</b> Elastic behaviour of section. ....	20
<b>Figure II:3.</b> Longitudinal (tension, compression). ....	21
<b>Figure II:4.</b> Three examples of springs: spring with non-attachment coils used according to its axis (fig. on the left), spring with a blade applied in bending (in the centre), spring with a blade applied in torsion (on the right). ....	22
<b>Figure II:5.</b> Equivalent diagonal strut model. ....	23
<b>Figure II:6.</b> Beam. ....	23
<b>Figure II:7.</b> Wall on plate.....	24
<b>Figure II:8.</b> Fiber Element Scheme-definitions. ....	25
<b>Figure II:9.</b> Fibre section model. ....	25
<b>Figure II:10.</b> Schematic of nonlinear beam-column elements. ....	26
<b>Figure II:11.</b> Inelastic behavior of materials and structures. ....	26
<b>Figure II:12.</b> stress-strain curve. ....	27
<b>Figure II:13.</b> Inelastic Behavior from Material. ....	28
<b>Figure II:14.</b> Inelastic Behavior to Section. ....	28
<b>Figure II:15.</b> Hysteresis curve. ....	29
<b>Figure II:16.</b> Load-displacement curves. ....	29

<b>Figure II:17.</b> Moment-curvature curve. ....	30
<b>Figure II:18.</b> Force-displacement curve. ....	30
<b>Figure II:19.</b> Local axes and output notation type-infrmFB. ....	31
<b>Figure II:20.</b> Beam-Column a plastic hinge. ....	32
<b>Figure II:21.</b> Local axes and output notation type-infrmFBPH. ....	33
<b>Figure II:22.</b> Local axes and output notation type-infrmDBPH. ....	34
<b>Figure II:23.</b> Local axes and output notation type-infrmDB. ....	35
<b>Figure III:1.</b> Pushover Analysis curve. ....	36
<b>Figure III:2.</b> Transformation from MDOF system to SDOF one. ....	37
<b>Figure III:3.</b> Capacity curve (a) of MDOF system (b) of SDOF system. ....	38
<b>Figure III:4.</b> Relationship $R_{\mu} - \mu - T$ ....	40
<b>Figure III:5.</b> Estimation of the target displacement by the N2 method when $T < T_c$ . ....	41
<b>Figure III:6.</b> Estimation of the target displacement by the N2 method when $T \geq T_c$ . ....	41
<b>Figure IV:1.</b> (a): Reinforced concrete frame; (b): Steel frame. ....	44
<b>Figure IV:2.</b> (a): HEB Section; (b): IPE Section ....	44
<b>Figure IV:3.</b> SeismoStruct software interface. ....	46
<b>Figure IV:4.</b> Element classes in SeismoStruct. ....	47
<b>Figure IV:5.</b> pushover curves of the steel structure using different element model. ....	48
<b>Figure IV:6.</b> Comparison of displacement between floors. ....	49
<b>Figure IV:7.</b> Comparison of Inter-story drift ratio. ....	50
<b>Figure IV:8.</b> Comparison of pushover curves for the RC frame. ....	51
<b>Figure IV:9.</b> Storey displacement profile of the RC structure. ....	51
<b>Figure IV:10.</b> Comparison of Inter-story drift ratio. ....	52

## Table List

<b>Tableau I:1.</b> analysis type. ....	7
<b>Tableau IV:1.</b> Geometrical details of columns and beams (steel structure). ....	44
<b>Tableau IV:2.</b> Geometrical details of columns and beams (reinforced concrete). ....	45
<b>Tableau IV:3.</b> Periods and mass participation of 1st mode for each steel model. ....	47
<b>Tableau IV:4.</b> Periods and mass participation ratios for the RC structure in first mode: ....	50

## List abbreviation

$[M]$	mass matrix of MDOF system
$[C]$	damping matrix of MDOF system
$\{F\}$	storey force vector of MDOF system
$\{\ddot{u}_g\}$	ground acceleration history
$\{\ddot{U}\}$	relative acceleration vector of MDOF system
$\{\dot{U}\}$	relative velocity vector of MDOF system
$\{1\}$	influence vector
$\{\Phi\}$	mode shape vector
$\varphi$	mode shape at the roof
$\varphi_i$	mode shape at the $i$ th -storey
$U$	relative displacement vector of MDOF system
$u_t$	roof/top displacement of MDOF
$\ddot{u}_t$	relative acceleration of roof
$\dot{u}_t$	relative velocity of roof
$u^*$	reference displacement of ESDOF system
$\ddot{u}^*$	reference acceleration of ESDOF system
$\dot{u}^*$	reference velocity of ESDOF system
$M^*$	mass of ESDOF system
$C^*$	damping of ESDOF system
$F^*$	force relationship of ESDOF system
$V_b$	base shear of MDOF system
$V_y$	yield strength of MDOF
$K_e$	effective elastic stiffness of MDOF
$U_y$	yield displacement of MDOF system
$K_s$	hardening/softening stiffness
$\alpha$	strain-hardening ratio
$T_{eq}$	elastic period of ESDOF system
$K^*$	elastic stiffness of ESDOF system
$POA$	pushover analysis
$MPA$	modal pushover analysis
$CSM$	capacity spectrum method
$DCM$	displacement coefficient method

<i>ADRS</i>	acceleration-displacement spectra
<i>AMP</i>	adaptive modal pushover analysis
<i>AMC</i>	adaptive modal combination procedure
<i>i</i>	storey number
<i>j</i>	mode number
$F_y^*$	yield force of ESDOF system
$u_y^*$	yield displacement of ESDOF system
$F_i$	applied force at $i^{\text{th}}$ storey
$W_i$	weight of $i^{\text{th}}$ storey
$\varphi_{ij}$	$i^{\text{th}}$ storey element of the mode shape vector for mode
<i>j</i>	
$h_j$	height of the $i^{\text{th}}$ storey
<i>n</i>	total number of modes, total number of storey
$S_d(T_n)$	acceleration ordinate of the design spectrum at the
fundamental period $T_n$	
$T_n$	fundamental period of vibration
<i>W</i>	weight of structure
<i>K</i>	coefficient dependent on the fundamental period $T_n$
$a_{mr}$	modification factor in Kunnath load pattern
$\Gamma_j$	participation factor for mode <i>j</i>
$M_i$	mass of the $i^{\text{th}}$ -storey
$S_a$	spectral acceleration of inelastic ESDOF
$S_d$	spectral displacement of inelastic ESDOF
$R_\mu$	strength reduction factor
$S_{ae}$	pseudo-acceleration ordinate from the response
spectrum	
$S_v$	pseudo-velocity ordinate from the response spectrum
$S_{de}$	elastic displacement ordinate from the response
spectrum	
$S_d$	target displacement of ESDOF from N2 method
$S_{ay}$	yield acceleration from the capacity spectrum
$T_c$	characteristic period of the ground motion
<i>T</i>	period of vibration

$C_0$	modification factor to relate the SDOF spectral displacement to MDOF roof displacement
$C_1$	modification factor to relate the expected maximum inelastic SDOF displacement divided by the elastic SDOF displacement
$C_2$	modification factor to represent the effect of hysteresis shape on the maximum displacement response
$C_3$	modification factor to represent increased displacements due to second-order effects
$T_e$	effective fundamental period
$R$	ratio of inelastic strength demand to calculated yield strength coefficient
$\mu$	ductility factor
<i>infrmFB</i>	Inelastic force-based frame element.
<i>infrmFBPH</i>	Inelastic force-based plastic hinge frame element type.
<i>infrmDBPH</i>	Inelastic displacement-based plastic hinge frame element type.
<i>infrmDB</i>	Inelastic displacement-based frame element type.

# *General Introduction*

## **GENERAL INTRODUCTION**

Throughout history, there have been many earthquakes. Caused damage to buildings, serious human injuries and deaths. Recently, potential activity in seismic areas has raised concerns about the security and vulnerability of buildings and their residents. Earthquakes are spectacular and devastating manifestations, and it is difficult to estimate seismic risks because predictions are uncertain and random events. Seismic phenomena are not fully known and only major earthquakes lead the population to question. Among the major earthquakes was the 2003 Boumerdas earthquake, which raised many questions about the adequacy of framed structures to resist strong movements, with many buildings badly damaged and collapsed.

Seismic design is a vital process of structural analysis while designing a building, which is subjected to earthquake ground motions. For the seismic analysis linear or nonlinear procedures can be used to assess the response of buildings and their elements. Nonlinear finite element analysis is a widely used technique to simulate the response of structures subjected to earthquake loads. However, the accuracy of the results obtained from the analysis depends on the assumptions made during the modeling process. Among these hypotheses, we can find the element modeling assumption.

This work comprises four chapters. These four chapters provide a comprehensive exploration of the subject matter, covering various aspects such as bibliographic study, nonlinear finite element analysis, pushover analysis, and a case study analysis. By delving into these topics, we aimed to enhance our understanding of structural behavior.

In the first chapter, we conducted a bibliographic study, which involved an extensive review of existing literature and research related to our topic. This study allowed us to identify the key concepts, theories, and methodologies that formed the basis of our research.

In the second chapter, we focused on the study of nonlinear finite elements. This involved understanding and implementing techniques to account for nonlinearities in the behavior of structures. We explored various nonlinear effects, such as material nonlinearity, geometric nonlinearity, and boundary conditions, to accurately model and analyze the structural response.

In the third chapter, we delved into pushover analysis, a valuable method for assessing the seismic performance of structures. By subjecting the structures to incremental lateral loads, we examined their response and identified critical failure modes and weaknesses. This analysis provided insights into the capacity and behavior of the structures under seismic conditions.

Lastly, in the fourth and final chapter, we conducted a case study that involved analyzing two structures using nonlinear static pushover analysis. To enhance the accuracy of our results, we

utilized five different types of finite elements, employing the software program Siesmostruct. By comparing and contrasting the obtained results, we gained a comprehensive understanding of the structural behavior, highlighting the effectiveness and limitations of the selected finite element types.

Through this comprehensive study, we aimed to contribute to the knowledge and understanding of structural analysis and design, particularly in the context of nonlinear behavior and seismic performance.

# ***Chapter I:***

## ***Bibliographic study***

## **I.1. INTRODUCTION:**

Earthquakes can significantly affect buildings, and the use of steel construction can help mitigate damage. However, the use of steel construction can also increase the risk of damage in certain situations. For example, if a steel building is not properly designed or constructed, it may be more susceptible to collapse or failure during an earthquake. Steel buildings also tend to be lighter and more flexible than other types of structures, which can lead to greater displacement and damage during seismic events.

Overall, the influence of earthquakes on the seismic behavior of buildings depends on many factors, including the design and construction of the building, the intensity and duration of the earthquake, and the soil and site conditions where the building is located. Hence, proper design and construction, in combination with appropriate seismic characteristics, can help ensure that buildings can withstand seismic forces and minimize damage during earthquakes.

## **I.2. EARTHQUAKES:**

An earthquake is a natural phenomenon characterized by a sudden and rapid shaking of the ground, usually caused by the movement of tectonic plates beneath the Earth's surface. It is a result of the release of energy that has been built up over time due to the slow movement of these plates, which suddenly gives way, causing seismic waves to radiate out from the point of origin, known as the earthquake epicenter. Earthquakes can range in magnitude and intensity, and can cause significant damage to buildings and infrastructure, as well as loss of life.

The major earthquakes on Earth tend to occur in belts that align with the margins of tectonic plates. This has been apparent for a long time, as evidenced by early catalogs of felt earthquakes, and is even more evident in modern seismicity maps that show instrumentally determined epicenters. The most significant earthquake belt is the Circum-Pacific Belt, which affects many populous coastal regions around the Pacific Ocean, including those of Japan, New Zealand, New Guinea, the Aleutian Islands, Alaska, and the western coasts of North and South America. Approximately 80 percent of the energy released in earthquakes is believed to come from those with epicenters in this belt. The seismic activity is not uniform throughout the belt, and there are several branches at various points. Due to the fact that volcanic activity is associated with the Circum-Pacific Belt at many locations, it has been popularly called the "Pacific Ring of Fire." [1]

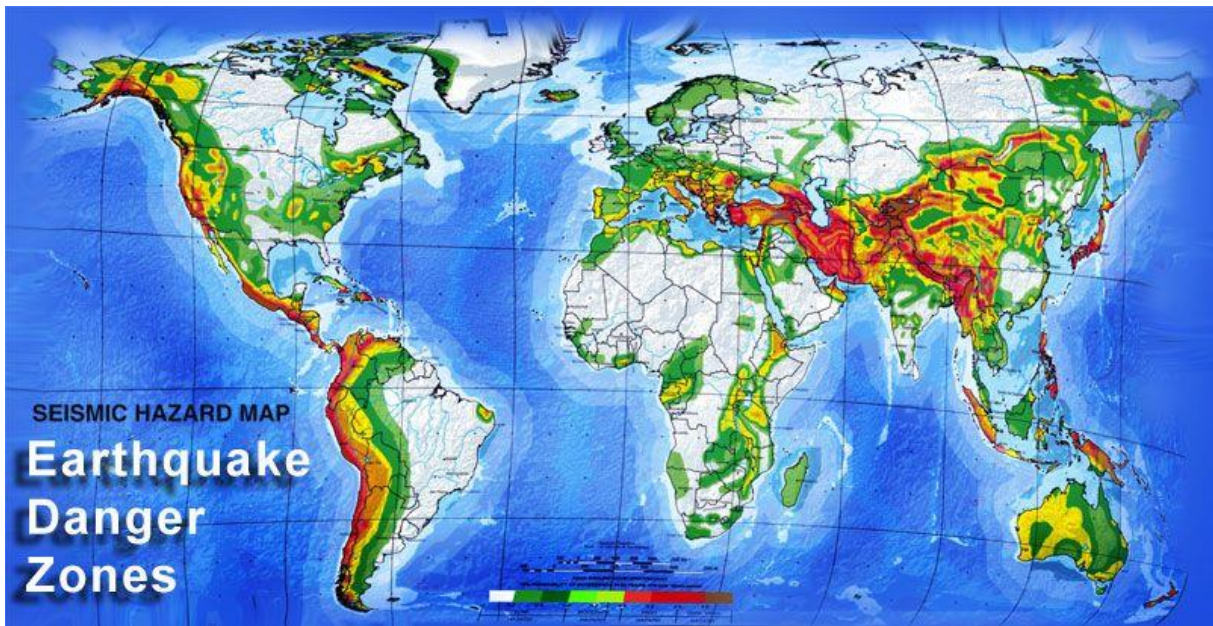


Figure 0:1. Seismic Hazard map.[2]

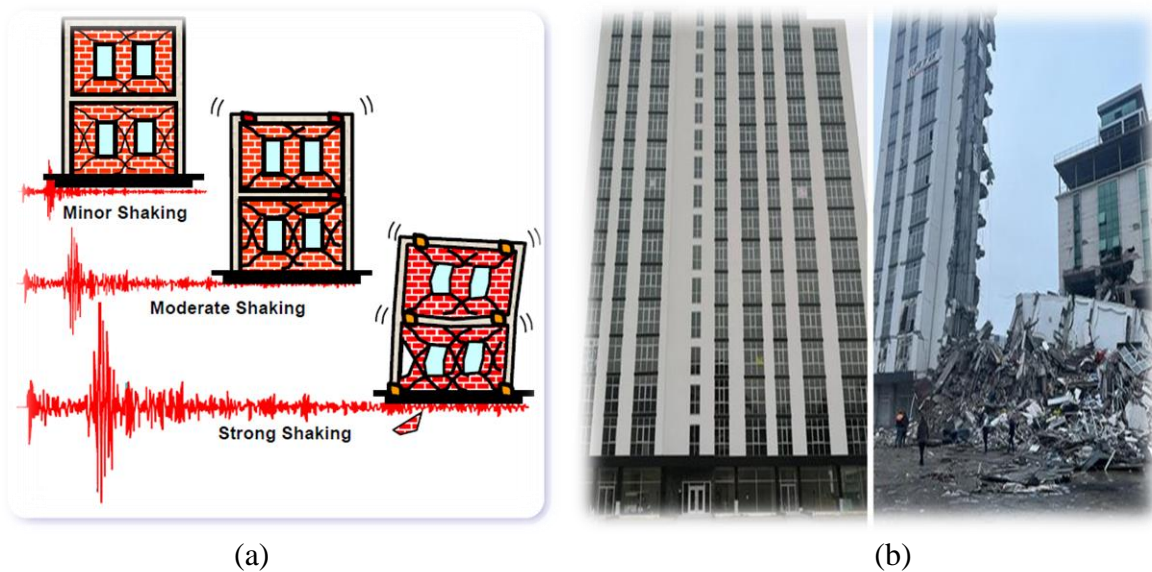


Figure 0:2. The behaviour of structural elements under the influence of various earthquakes [3].

(b): Before and after Turkey's earthquake: a modern residential building in Alexandrina [4].

### I.3. Earthquakes and their influence on buildings

Earthquakes can have a significant influence on construction, particularly in areas that are prone to seismic activity. The shaking and ground motions caused by earthquakes can cause damage to buildings and infrastructure, leading to structural failure, collapse, and loss of life. As a result, it is essential for engineers and architects to consider seismic design and construction practices to

minimize the impact of earthquakes on buildings and other structures. Here (Figure 1.3) are some of the ways earthquakes can influence construction.



[5]



[6]



[7]



[8]

**Figure 0:3.**Some earthquake effects on buildings.

As we witnessed in the recent earthquake in Turkey and Syria, it left a total devastation and human loss of approximately more than 50 thousand people.



**Figure 0:4.**Photos from Turkey and Syria's recent earthquake [9] [10].

#### **I.4. STEEL BUILDING IN SEISMIC AREAS:**

Steel buildings are a popular choice for seismic areas due to their ductility and ability to deform under seismic forces. This makes them more resilient to earthquakes compared to other construction materials. However, the behavior of steel buildings in seismic areas can depend on several factors, including the design, construction, and location of the building.

One study conducted by researchers at the University of California, San Diego, analyzed the seismic behavior of steel moment-resisting frame buildings. The study found that these buildings were generally resistant to seismic forces but could exhibit different behaviors depending on the location and magnitude of the earthquake [11].

Overall, it is important to ensure that steel buildings in seismic areas are designed and constructed according to appropriate codes and standards to ensure their stability and safety during earthquakes. Proper design and detailing are crucial to ensure that the steel building can withstand seismic forces and remain resilient in the event of an earthquake.

#### **I.5. LINEAR AND NONLINEAR ANALYSES**

Seismic analysis is a process used in engineering to evaluate the behavior of structures when subjected to seismic forces, such as those caused by earthquakes. The primary objective of seismic analysis is to ensure that structures are designed and constructed to withstand such forces, minimizing the risk of damage, injury, and loss of life.

There are several methods used in seismic analysis, including linear and nonlinear static analysis, linear and nonlinear dynamic analysis, and time history analysis. Linear static analysis is

the simplest and most commonly used method, in which the seismic forces are estimated using simplified procedures and assumptions. Nonlinear static analysis, on the other hand, considers the nonlinear behavior of the structure, accounting for the inelastic response of the structure.

Linear dynamic analysis involves the application of time-varying forces to the structure, simulating the effects of an earthquake (may use a response spectrum). The dynamic response of the structure is then calculated using mathematical models and numerical techniques. Nonlinear dynamic analysis considers the effects of both nonlinear structural behavior and the nonlinear nature of the earthquake ground motion. [12]

**Tableau 0:1.** Analyses types.

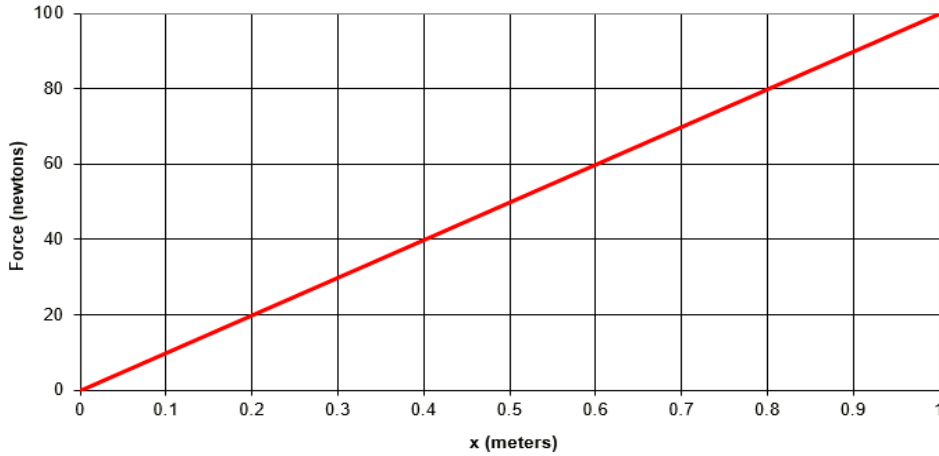
Analysis type	Linear	Nonlinear
Static	Strength-based	Static-pushover
Dynamic	Response-spectrum	Time-history

#### I.5.1. Linear analysis

Linear analysis is a mathematical method used to study the behavior of a system or structure. It involves the use of linear equations to describe the relationship between input and output variables of the system, assuming that the system behaves linearly.

In the context of structural analysis, linear analysis is used to calculate the behavior of a structure under different loading conditions. It assumes that the behavior of the structure is linearly proportional to the applied loads and can be represented by linear equations. This method is used to determine the structural response to loads such as gravity, wind, and seismic forces. In linear analysis, the stiffness matrix of the structure is assumed to be constant, regardless of the level of deformation. This allows for the use of simplified equations and methods of analysis. Linear analysis is often used in the preliminary design stage of a structure, where approximate values for loads and geometry are used to determine the feasibility of the design. It is also commonly used in the design of smaller, simpler structures such as beams and columns.

Force vs. Displacement



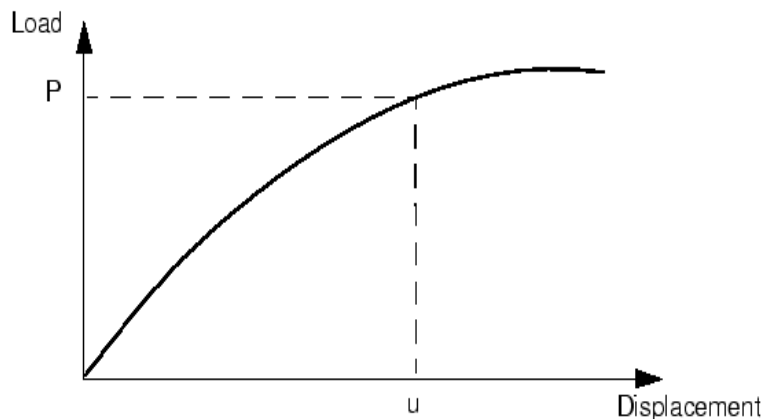
**Figure 0:5.**Linear relationship between force and displacement [13].

However, linear analysis has limitations when applied to complex structures and systems. In reality, the behavior of structures is often nonlinear due to material properties, geometric configurations, and other factors. As a result, linear analysis may underestimate the actual response of the structure, leading to unsafe or inefficient designs.

To overcome these limitations, nonlinear analysis techniques are used, which consider the nonlinear behavior of the structure. These techniques can provide more accurate results but are more complex and computationally demanding.

I.5.2. Nonlinear analysis

Nonlinear analysis takes into account the nonlinear behavior of materials and structures under loading. It involves the use of nonlinear equations to describe the relationship between input and output variables of the system, which better reflects the actual behavior of structures under extreme loading conditions such as earthquakes, explosions, and impacts.



**Figure 0:6.**Nonlinear load-displacement curve [14].

Non-linear analysis can be divided into major categories: non-linear engineering analysis, non-linear material analysis and non-linear communication.

Geometric nonlinear analysis considers the nonlinear behavior of a structure due to large deformations. As the load increases, the deformation of the structure may become so large that the stiffness and geometry of the structure change. In this case, linear analysis assumptions are no longer valid, and the response of the structure becomes highly nonlinear. Examples of geometric nonlinear behavior include large deflections, buckling, and collapse.

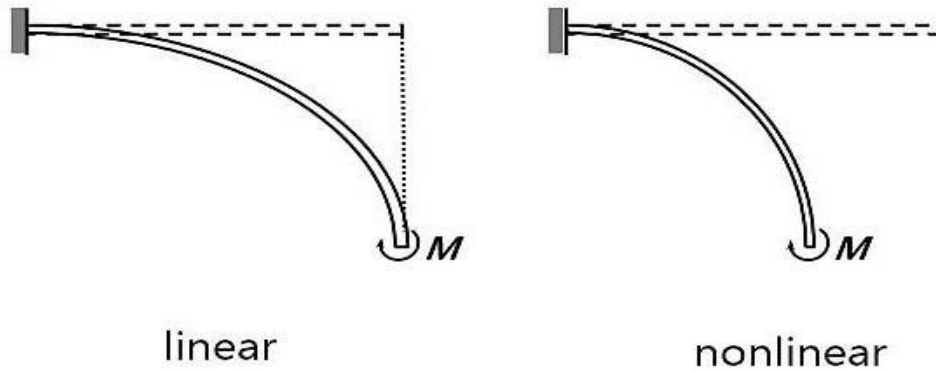


Figure 0:7. Geometric nonlinearity [13].

Material nonlinear analysis considers the nonlinear behavior of a structure due to the behavior of the materials used in the structure. Materials such as concrete, steel, and composite materials exhibit nonlinear behavior under certain conditions, such as yielding, strain hardening, and strain softening. Material nonlinear analysis takes into account these material behaviors and their effect on the overall response of the structure.

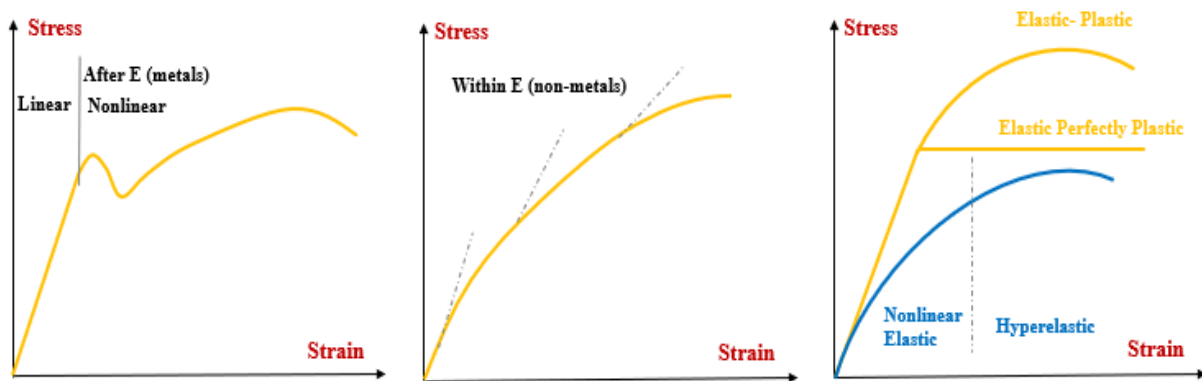


Figure 0:8. Material non linearity [13].

Contact Nonlinearity «Contact" is a type of event that involves a change in the status of a system. This can occur when bodies come into or out of contact with each other, causing an abrupt change in stiffness due to the changing nature of contact between the components during motion [15].

Nonlinear analysis techniques are more complex and computationally demanding than linear analysis techniques. However, they can provide more accurate results and can better represent the actual behavior of structures under extreme loading conditions. Nonlinear analysis techniques are often used in the design of critical structures, such as high-rise buildings, bridges, and nuclear power plants.

### **I.6. Structural nonlinear behavior:**

Nonlinear behavior refers to the behavior of a system or material that does not follow a linear relationship between cause and effect. In other words, the response of the system is not proportional to the applied input, and may exhibit complex and unpredictable behavior. In engineering and physics, nonlinear behavior is often encountered in systems that are subject to extreme conditions, such as high stresses, temperatures, or loads. Examples of nonlinear behavior include plastic deformation of metals, the response of structures to seismic or wind loads, and the behavior of fluids under extreme conditions.

Nonlinear behavior can be analyzed using various mathematical and computational techniques, such as numerical simulations, finite element analysis. These techniques can help to predict the behavior of complex systems and to identify potential failure points or areas of instability.

Overall, nonlinear behavior is an important consideration in many areas of engineering and science, and understanding and predicting nonlinear behavior is essential for designing safe and efficient systems and structures.

### **I.7. Plastic hinge**

The plastic hinge, in structural engineering, refers to the deformation of part of the beam wherever plastic bending occurs. The hinge means it does not have the ability to resist the moment. Therefore, the plastic hinge acts like a standard hinge - allowing free rotation. The concept of plastic hinge is important in understanding structural failure [16].

The formation of a plastic hinge is a critical design consideration in the performance and integrity of structures, especially in seismic design. In steel structures, plastic hinges occur in beams and columns when they are exposed to excessive loads or seismic events. Plastic hinges allow structures to absorb and dissipate energy, thereby reducing the risk of sudden and catastrophic failure.

A plastic hinge is a specific region (in a structure) designed to “fail” first to dissipate devastating energy in a stable manner to protect the rest of the building during a severe earthquake. [17]

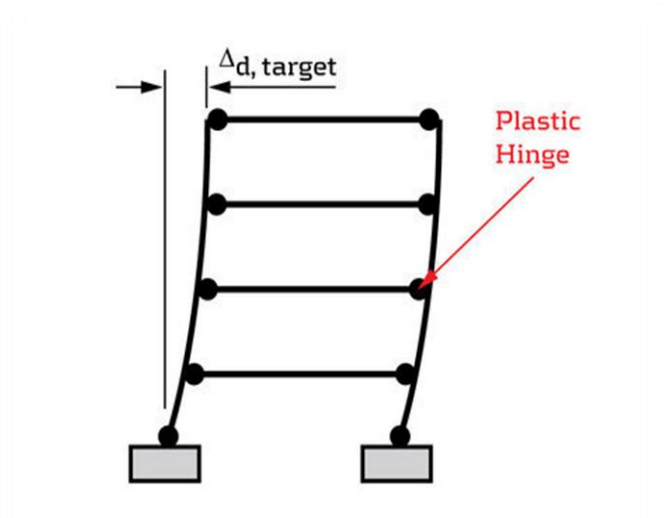


Figure 0:9. Plastic hinge [17].

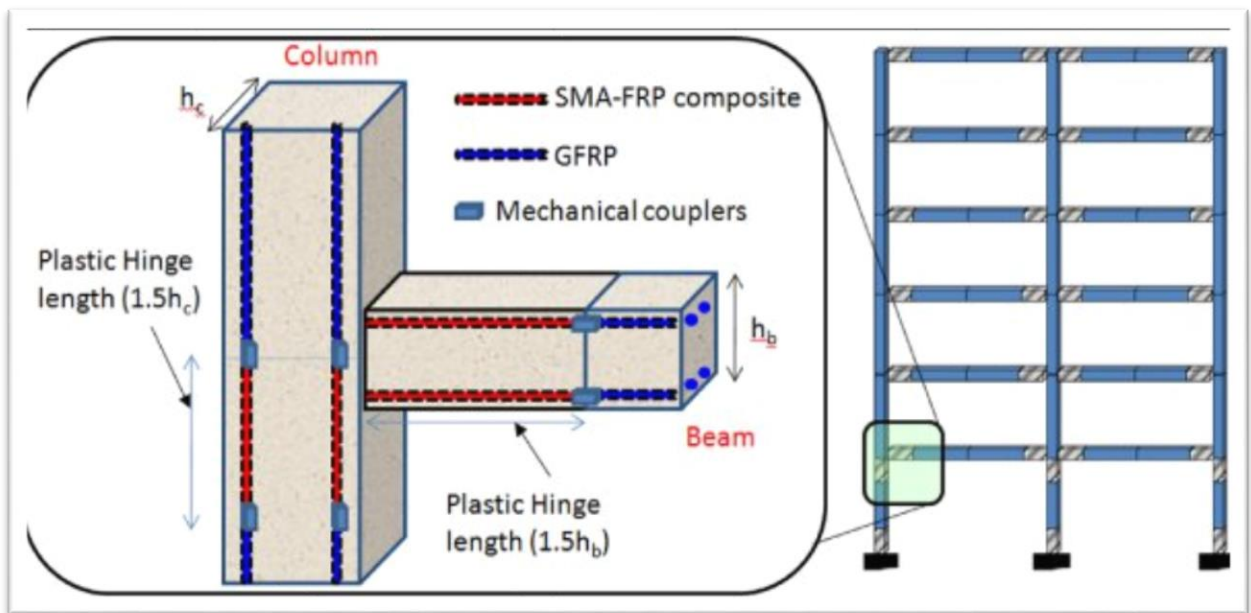
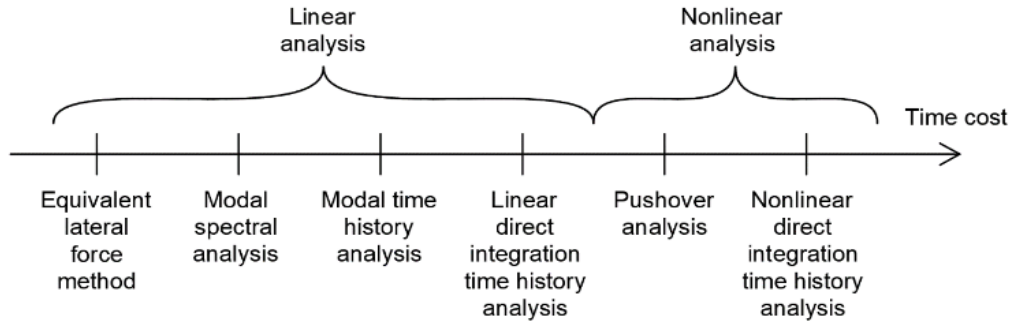


Figure 0:10. Plastic hinge length [18].

### I.8. Seismic analysis methods in seismic codes:

Seismic codes are regulations and standards put in place to ensure the safety of buildings and structures in areas prone to earthquakes. These codes are based on the understanding of the potential impact of seismic activity on buildings and the surrounding environment. The methods used in seismic codes include the identification of seismic hazards in a specific region, the establishment of design criteria for structures, and the creation of guidelines for construction and

inspection of buildings. Seismic codes also take into account the soil conditions and topography of a given area, as well as the seismic resistance of materials used in construction. The goal of seismic codes is to reduce the potential for damage and loss of life in the event of an earthquake. [19]



**Figure 0:11.**Time cost scale for seismic analysis techniques [20].

I.8.1. Linear Static Method (Equivalent static method):

The equivalent fixed method is a simplified approach to earthquake-proof design used in structural engineering. It is a simplified analysis method that estimates seismic forces on the structure by applying a constant equivalent load based on the earthquake's maximum projected ground movement at the site.

The equivalent fixed method is commonly used to design low-height structures or structures with normal shapes and simple configurations. It is a direct and conservative approach that can provide reasonable estimates of seismic forces on the structure. However, the equivalent fixed method has limitations and is not suitable for designing more complex or irregular structures. For these types of structures, dynamic analysis methods are used, such as response spectrum analysis or time history analysis.

I.8.1.1. **Base shear calculation**

The Eurocode 8 [21] requires regularity in elevation as a condition for the application of this method, the fundamental principle of this method is to undertake a static analysis using static forces equivalent to those dynamic of seismic action and is based on the fundamental mode.

According to this method, as set out in the Algerian seismic regulation RPA99v2003 [21], the maximum cutting force at the base is given by the relation

$$V = \frac{A \cdot D \cdot Q}{R} \cdot W$$

V: The total seismic (shear) force

D: mean dynamic amplification coefficient

Q: quality coefficient

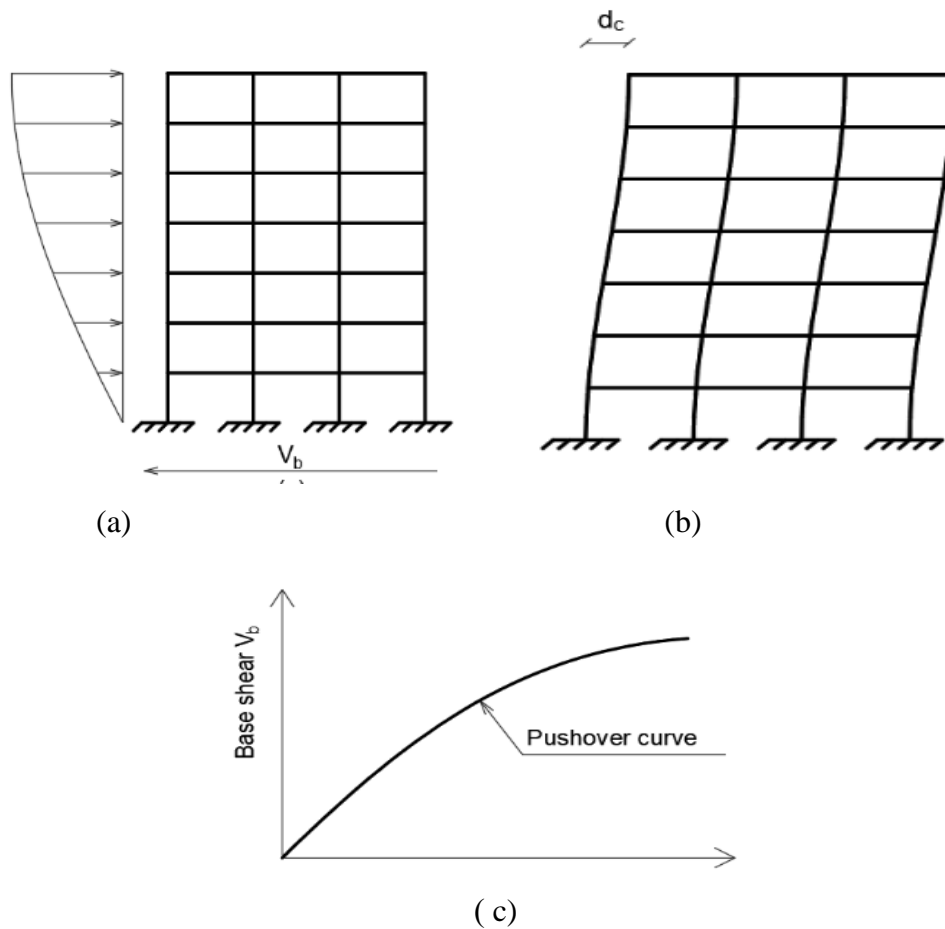
R: coefficient of behaviour

A: zone acceleration coefficient it depends on the use group of the building and the seismic zone.

### I.9. Non-Linear Static Method (Pushover Method)

#### I.9.1. Definition of pushover analysis:

Pushover analysis is a static analysis technique utilized to evaluate the performance of a building under seismic loads. A building model is created in a computer software, incorporating all load-resisting elements and their corresponding force-deformation relationships. This includes pre- and post-yield behavior, along with dead loads and average live loads



**Figure I: 12.** Pushover analysis.

. The analysis involves applying a series of horizontal forces, which mimic ground motions, and calculating the resulting deformations. The applied forces are gradually increased in steps, producing a graph of base motion versus deformation. By studying this graph, the maximum base motion that the building can withstand without experiencing total or partial collapse can be determined [22].

I.9.2. Pushover curve

The pushover curve presents the base shear versus the roof displacement of the building, usually presented in natural form where the base shear is divided by the weight of the building. The curve is created by gradually increasing the lateral force applied to the building until the structure reaches its limit, or the point at which it begins to suffer significant damage or collapse. The pushover curve is used to identify weaknesses in the building's structural system and to assess the structure's performance under different load conditions. It can be used to compare different design options and to improve building design to improve performance and safety. In general, the exercise curve is a useful tool for engineers and designers to assess the performance of buildings under harsh loading conditions, designing flexible and safe structures in the face of natural disasters and other hazards.

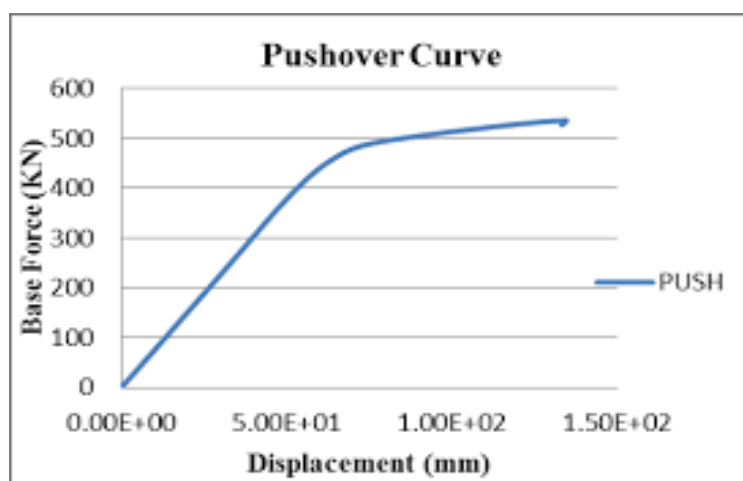


Figure 0:13.Pushover-curve-of-structure [23].

**I.10. Response spectrum method**

Response spectrum analysis is a method to estimate the structural response to short, nondeterministic, transient dynamic events. Examples of such events are earthquakes and shocks. Since the exact time history of the load is not known, it is difficult to perform a time-dependent analysis. Due to the short length of the event, it cannot be considered as an ergodic ("stationary") process, so a random response approach is not applicable either.

The response spectrum method is based on a special type of mode superposition. The idea is to provide an input that gives a limit to how much an eigenmode having a certain natural frequency and damping can be excited by an event of this type.

I.10.1. Definition of a Response Spectrum

A response spectrum is a function of frequency or period, showing the peak response of a simple harmonic oscillator that is subjected to a transient event. The response spectrum is a

function of the natural frequency of the oscillator and of its damping. Thus, it is not a direct representation of the frequency content of the excitation (as in a Fourier transform), but rather of the effect that the signal has on a postulated system with a single degree of freedom (SDOF).

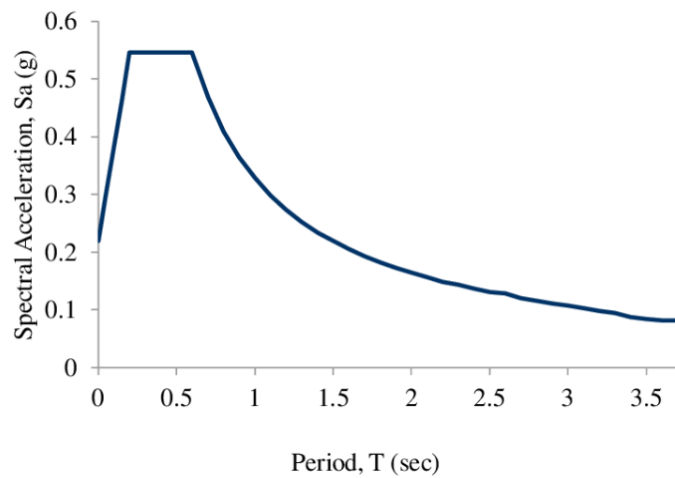


Figure 0:14. Response spectrum curve [24].

### I.11. Nonlinear time history analysis

A time history analysis is a step by step analysis of the dynamic response of a structure to real earthquake ground accelerations that vary with time. [17]



Figure 0:15. Time history analysis [17].

Nonlinear time history analysis is a more complex approach compared to linear analysis, as the stiffness matrix of the system changes continuously over time due to material and geometrical nonlinearities. To address this, the concept of a tangent stiffness matrix  $[KT(\{U(t)\})]$  is introduced to represent the stiffness matrix at a specific displacement  $\{U(t)\}$ . With the changing stiffness matrix and nonlinear calculations, time history analysis becomes computationally intensive, especially when dealing with multiple base excitations in different directions [22].

**I.12. Conclusion:**

This chapter provided some basic concepts and techniques for seismic analysis that engineers can use to calculate structures in earthquake-prone areas. Depending on the complexity and importance of the building, as well as the available computational resources, an engineer who understands the basic assumptions and limitations of each method can choose with confidence the most technical fit for its specific situation. In addition, the chapter deals with the different behavior types such as linear and nonlinear element behavior. It is shown that nonlinear analyses reflect very well the real structure behavior.

# ***Chapter II:***

*Nonlinear finite*

*Element Modelling*

## **Chapter II. Nonlinear finite element**

### **II.1. INTRODUCTION:**

The response of buildings to seismic events relies on the complex interaction of various structural elements, each contributing to stiffness, resistance, and ductility. When these components are appropriately balanced, structures can effectively control displacement, withstand seismic forces, and dissipate energy. However, traditional seismic design and assessment methods have often relied on linear approaches that fail to accurately capture the behavior of asymmetric structures.

Fortunately, advancements in material understanding, element modeling, and computational methods have made non-linear analysis more accessible. Yet, there is still a need for user-friendly software and simplified, accurate non-linear methods that can be easily implemented within the time constraints of design offices. Additionally, engineers may lack sufficient knowledge and experience in dealing with non-linear behavior.

The primary objective of this chapter is to provide a comprehensive overview of current non-linear models and present a specific models in SeismoStruct (2016), a software with both linear and non-linear analysis capabilities. By utilizing this software and evaluating its performance in the case study, we aim to assess the reliability and feasibility of non-linear modeling approaches for seismic design and assessment. [25]

### **II.2. Element modeling**

In modern times, element modeling in the finite element method (FEM) continues to be of utmost importance in various engineering and scientific disciplines. With advancements in computational power and simulation techniques, element modeling has evolved to address more complex and sophisticated problems.

In recent years, researchers and practitioners have developed specialized elements to model specific phenomena accurately. For example, in structural engineering, elements capable of simulating nonlinear material behavior, large deformations, and contact interactions have gained prominence. These advancements have allowed engineers to analyze structures with greater accuracy, including those subjected to extreme loading conditions or exhibiting intricate behavior.

Additionally, advancements in computational algorithms and numerical methods have led to the development of high-order elements. These elements incorporate higher-degree polynomial functions for more precise representation of the solution within an element. High-order elements

have the potential to enhance accuracy while reducing the computational cost associated with a fine mesh, making them particularly useful in large-scale simulations.

Another area of focus in element modeling is the incorporation of multiphysics phenomena. With the growing demand for coupled analyses involving fluid-structure interactions, thermal-electrical interactions, and other multiphysics problems, researchers have developed elements that can effectively capture the interaction between different physical domains. These multiphysics elements enable engineers to study complex systems where the behavior of multiple interconnected phenomena needs to be accurately represented.

Furthermore, advancements in material science, such as the development of advanced composites or smart materials, have prompted the creation of specialized elements to model their unique behavior. These elements allow engineers to study the performance and characteristics of these advanced materials, facilitating the design of innovative structures and components.

Overall, in today's computational era, element modeling in FEM continues to evolve to address the increasing complexity of engineering and scientific problems. Through advancements in element types, interpolation functions, numerical algorithms, and the incorporation of multiphysics phenomena, element modeling enables accurate and efficient simulations, leading to improved designs and enhanced understanding of various physical systems.

In finite element method (FEM), a widely used numerical technique for solving complex engineering problems, the accurate representation of physical systems is crucial for obtaining reliable and meaningful results. At the core of FEM lies the concept of discretizing the domain into smaller regions called elements, where the behavior of the system is approximated using a set of mathematical functions. The selection and modeling of these elements play a pivotal role in the overall accuracy and efficiency of the analysis. By carefully designing and defining the element properties, such as shape, size, and interpolation functions, engineers and scientists can tailor the FEM model to accurately capture the physical phenomena of interest. In this context, the element modeling becomes a vital step in the FEM workflow, influencing the fidelity of the simulations and providing valuable insights into the system's behavior.

#### II.2.1. Elastic element:

An elastic element refers to a component or material that exhibits elastic behavior, meaning it can deform under the application of external forces or loads and return to its original shape when the forces are removed. In structural analysis and engineering, elastic elements are often used to represent the behavior of structural members, such as beams, columns, and connections, in which the deformation is assumed to occur within the elastic range of the material.

Elastic elements are characterized by their stiffness, which quantifies their resistance to deformation. The stiffness of an elastic element is typically described by its elastic modulus or Young's modulus, which is a measure of the material's stiffness and ability to withstand stress without permanent deformation.

In structural analysis, elastic elements are commonly modeled using mathematical equations or finite element methods, allowing engineers to predict and analyze the structural response under different loading conditions. By considering the elastic behavior of elements, engineers can assess factors such as stress distribution, deflection, and overall structural integrity, helping to ensure the safety and performance of the structure.

It is important to note that the assumption of elastic behavior is suitable for many engineering applications, especially when the loads and deformations are within the linear range of the material. However, in situations where the loads or deformations exceed the elastic limit, nonlinear behavior and plastic deformation may occur, requiring the consideration of additional factors in the analysis.

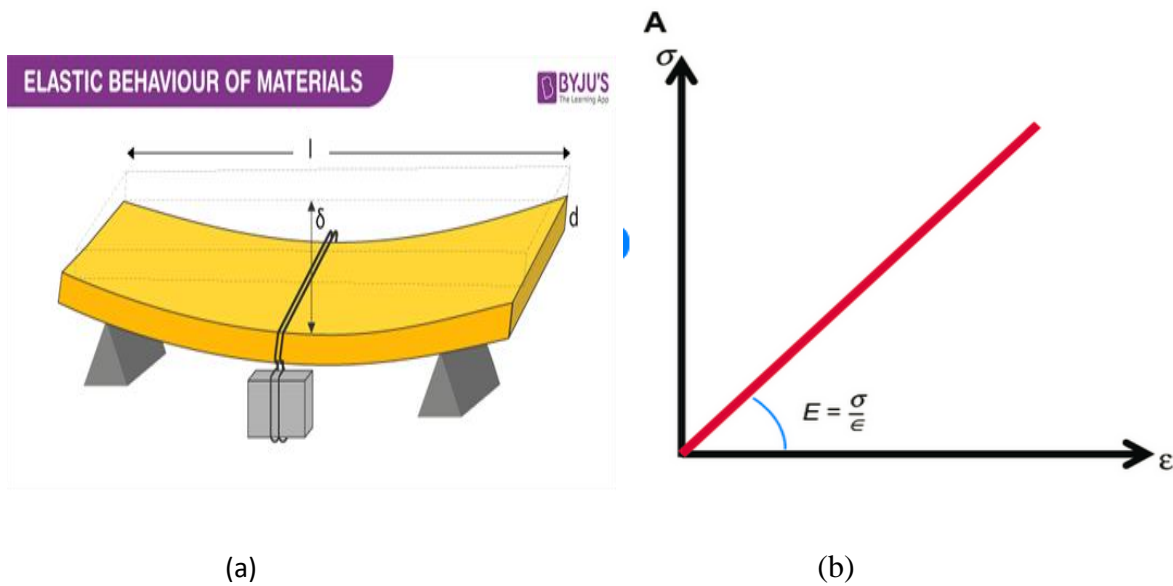
The ability of elastic materials to absorb shock, lessen vibration, and resist deformation is essential to the safe and effective operation of many equipment and devices. Elastic materials are a basic part of many engineering systems and construction

The modulus or Young's modulus, often referred to as the elastic modulus, is a fundamental property of a material that measures its stiffness or resistance to deformation when subjected to an applied force or stress. It is denoted by the symbol  $E$  and is defined as the ratio of stress to strain within the material's elastic range.

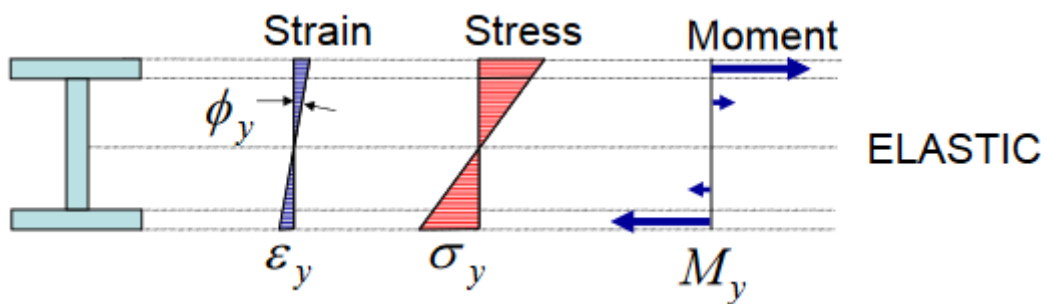
#### **II.2.1.1. *The elastic behavior:***

The elastic behavior of materials and structures refers to their response when subjected to external loads within their elastic limit. Elastic behavior is characterized by the material's ability to deform reversibly under the applied stress and return to its original shape once the load is removed.

In elastic behavior, the relationship between stress and strain follows Hooke's Law, stating that the stress ( $\sigma$ ) is directly proportional to the strain ( $\epsilon$ ) within the linear elastic range. This relationship can be expressed as:  $\sigma = E * \epsilon$



**Figure II:1.** elastic behaviour of materials [26] ;(b) Stress-strain relationship of a linear material. [27]



**Figure II:2.**Elastic behaviour of section. [28]

II.2.1.2. *The elastic law:*

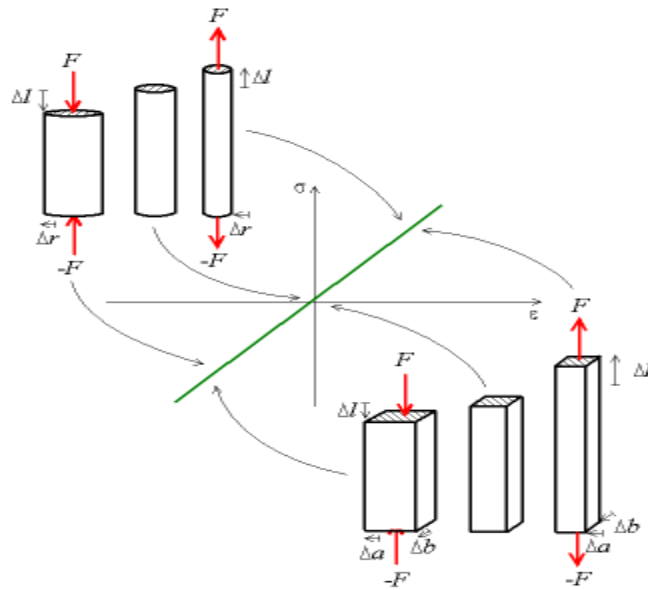
The elastic law states that the stress in a material is directly proportional to the strain it experiences within the material's elastic limit. This relationship is described by Hooke's Law, which states that stress ( $\sigma$ ) is equal to the product of the material's elastic modulus ( $E$ ) and the strain ( $\epsilon$ ):

$$\sigma = E * \epsilon$$

Here,  $\sigma$  represents the stress,  $E$  represents the elastic modulus (also known as Young's modulus), and  $\epsilon$  represents the strain. The elastic modulus is a material property that defines its stiffness and ability to deform under stress.

It is important to note that this relationship holds true only within the elastic limit of a material. Once the material surpasses its elastic limit, it may exhibit plastic deformation, where the stress-strain relationship becomes nonlinear.

The elastic law is fundamental in analyzing and designing structures, as it allows engineers to predict the response of materials and ensure that they remain within their elastic limits for optimal performance and safety.



**Figure II:3.** Longitudinal (tension, compression). [29]

**II.2.1.3. Young's modulus**

Young's modulus represents the slope of the linear portion of the stress-strain curve for a material. It quantifies how much a material elongates or contracts in response to an applied force. The higher the value of Young's modulus, the stiffer the material is considered to be. Mathematically, Young's modulus is expressed as:

$$E = \sigma / \epsilon$$

where:

E: is Young's modulus,

$\sigma$ : is the applied stress,

$\epsilon$ : is the resulting strain.

Young's modulus is typically expressed in units of pressure or stress, such as Pascals (Pa) or pounds per square inch (psi). It is an important parameter in structural engineering and materials science as it helps determine the behavior of materials under different loading conditions. By knowing the value of Young's modulus, engineers can assess the stiffness, deformation, and overall mechanical response of a material or structural element. It is used in various calculations and analyses, such as determining deflections, predicting structural stability, and designing structures to withstand expected loads and deformations.

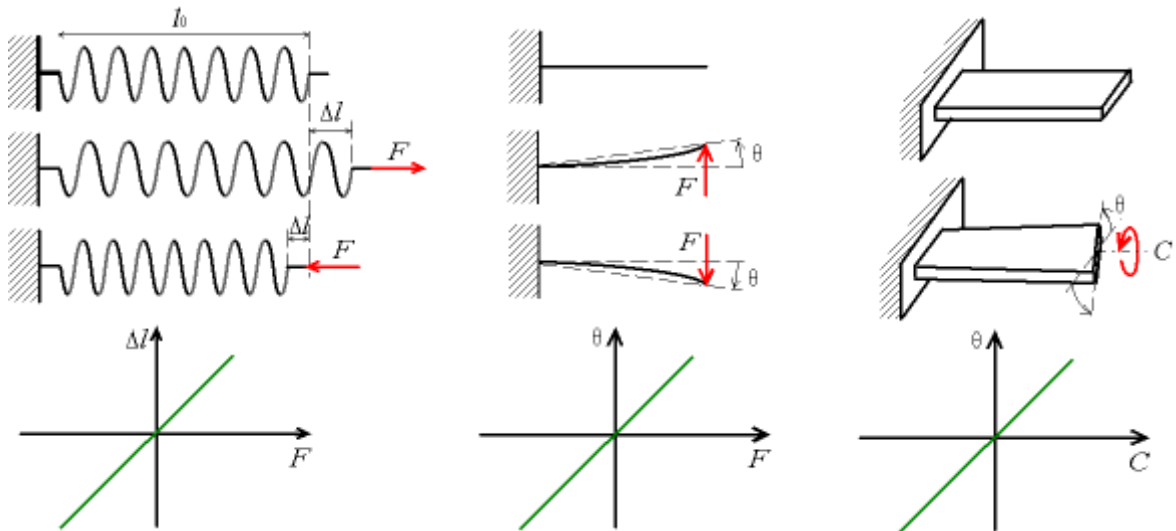
It's worth noting that Young's modulus is specific to each material and can vary depending on factors such as composition, temperature, and manufacturing processes. Therefore, it is essential to use accurate and appropriate values for Young's modulus when performing structural analyses or material selection for engineering applications.

**II.2.1.4. Types of elastic element**

Elastic elements are components that exhibit linear elastic behavior, meaning that their deformation is directly proportional to the applied load or stress within their elastic limit. There are several types of elastic elements, including [30]:

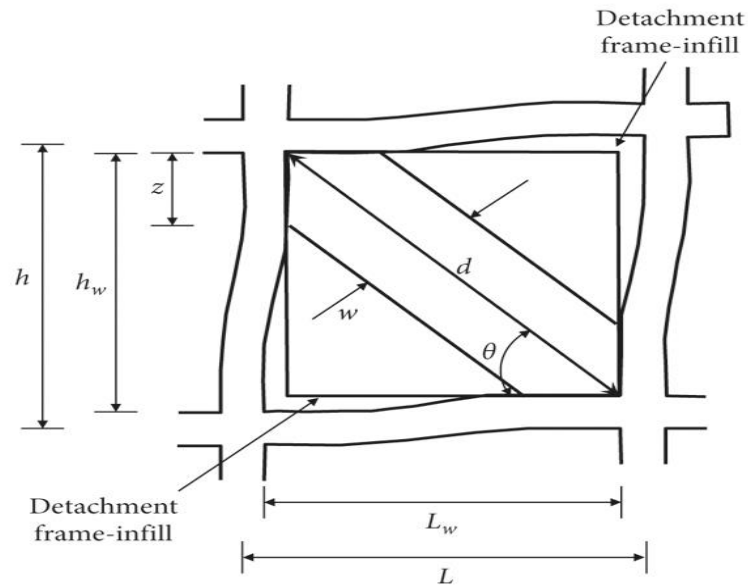
1. **Springs:** Springs are one of the most common types of elastic elements and are used to store and release mechanical energy. They can be made from various materials, including metal, plastic, and rubber.

A mechanical device known as a linear elastic spring can only handle axial loads, and the lengthening or shortening of the spring is inversely proportional to the applied axial load. The term "spring constant," "spring rate," or "spring stiffness  $k$ " refers to the proportionality constant, which has units of force per unit length, between deformation and load. We use an element coordinate system (sometimes referred to as a local coordinate system) with an  $x$  axis aligned along the length of the spring since an elastic spring can only handle axial loading.



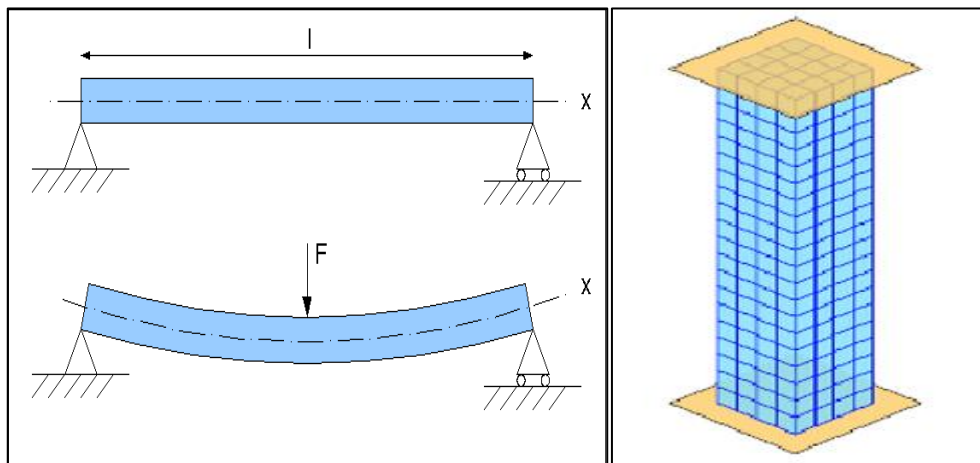
**Figure II:4.** Three examples of springs: spring with non-attachment coils used according to its axis (fig. on the left), spring with a blade applied in bending (in the centre), spring with a blade applied in torsion (on the right). [29]

2. **Struts:** Struts are elastic elements that resist compressive loads and are commonly used in structures such as buildings and bridges.



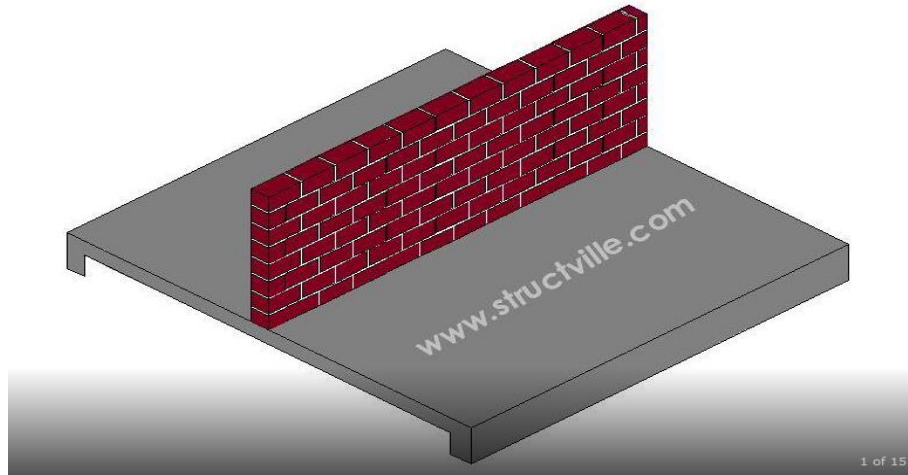
**Figure II:5.** Equivalent diagonal strut model [31].

3. **Beams:** Beams can be elastic elements that resist bending and torsion and are commonly used in structures such as buildings, bridges, and aerospace applications.



**Figure II:6.** Beam [32].

4. **Plates:** Plates are elastic elements that resist bending and are commonly used in structures such as aircraft wings and ship hulls.



**Figure II:7.** Wall on plate.

5. **Cables:** Cables are elastic elements that resist tension and are commonly used in structures such as suspension bridges and cable-stayed bridges.
6. **Columns:** Columns are passive elastic elements that provide resistance to compressive loads or forces. They are commonly used in structural systems to support vertical loads.

Due to its ability to store and release energy as well as withstand mechanical loads, flexible materials are important parts of many engineering applications. The use of a certain type of flexible element depends on the application and the necessary performance standards.

#### II.2.1.5. *Elastic Frame Element:*

The Elastic Frame Element (elfrm) is a finite element used in structural analysis to model the behavior of elastic frames. It is a three-dimensional element that can simulate the behavior of a wide range of frame structures, including buildings, bridges, and other types of structures.

The frame structure is represented by a set of interconnected nodes and elements that make up the elastic frame element. Six degrees of freedom are available for each node, including three translations (in the x, y, and z directions) and three rotations (around the x, y, and z axes). Typically, the components are represented as beam or truss elements.

To use the elfrm in structural analysis, the structure is first discretized into finite elements using a finite element mesh. The properties of each element, including its material properties and geometry, are then defined. The loads and boundary conditions are applied to the structure, and the behavior of the structure under these conditions is analyzed using the elastic frame element.

### II.3. FIBER ELEMENT

A fiber element is a type of finite element used in structural analysis to model the behavior of structures made of fiber-reinforced materials, such as composites. It is a one-dimensional element that simulates the behavior of a fiber within a composite material.

In a fiber element, the material properties of the fiber, such as its modulus of elasticity and Poisson's ratio, are defined. The element is typically modeled as a beam or truss element, with a cross-sectional area representing the fiber. The element's length is defined to represent the length of the fiber within the composite material.

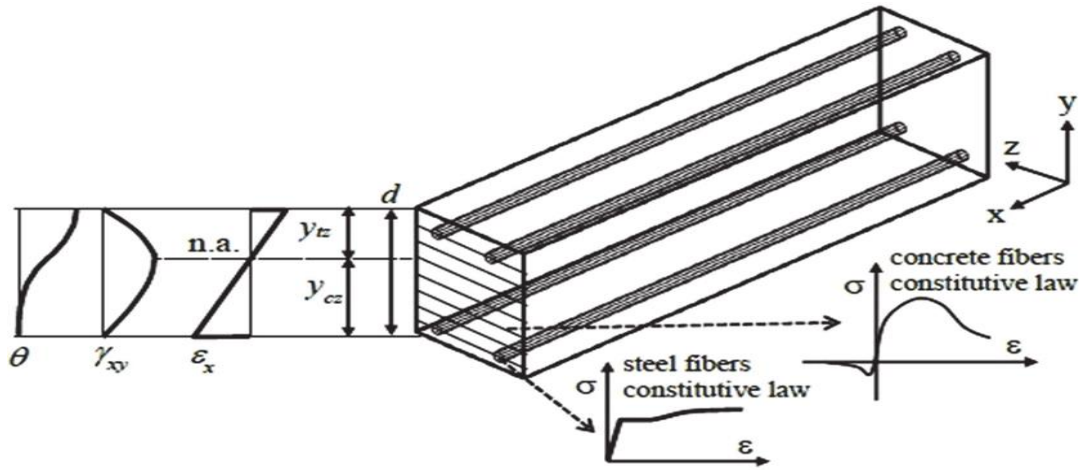


Figure II:8. Fiber Element Scheme-definitions [33].

Fiber elements are commonly used in the analysis of composite materials, which have unique material properties that cannot be modeled using traditional isotropic materials. By modeling the behavior of individual fibers within the composite material, engineers can gain insight into how the material will behave under various loading conditions and design structures that are optimized for performance. [34]

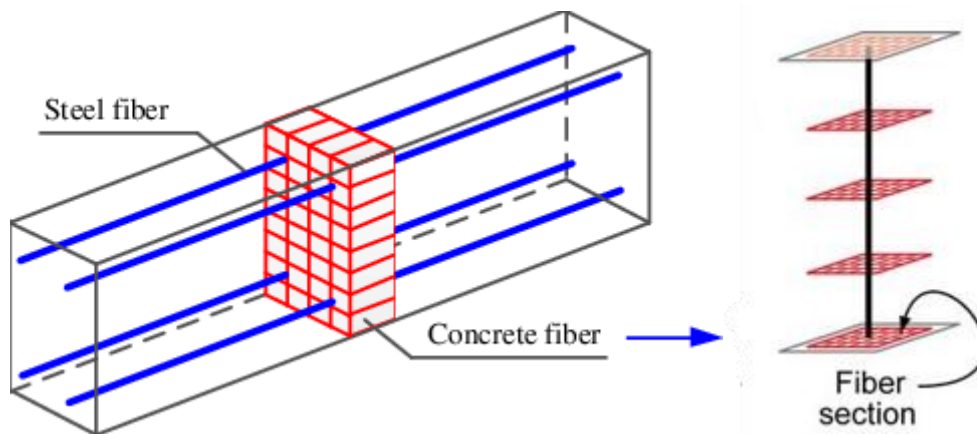


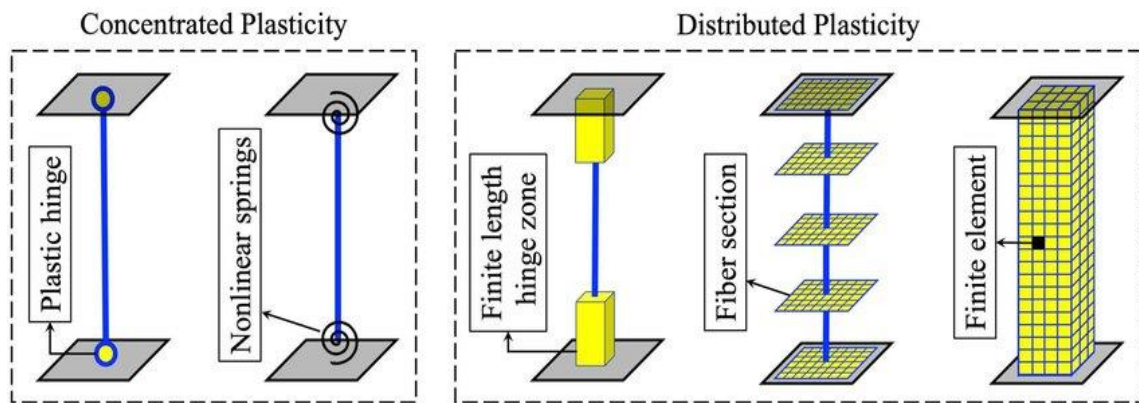
Figure II:9. Fibre section model [35].

#### II.4. Inelastic Element

Inelastic element is a type of element used in finite element analysis, which is used for the analysis of structures under cyclic loading. This type of element can capture the main characteristics of materials under cyclic loading such as hysteresis, strain-hardening, and constitutive equations. These elements are used to consider the time-dependent behavior of the

structure, such as plasticity, viscoelasticity, etc., and for the design in accordance with seismic codes. The inelastic element can be used to find the displacement, strain or stress of the material under cyclic loading. This type of element is generally more complex than traditional elastic elements, since it requires more precise integration of equations in order to capture the different elements of the constitutive equations.

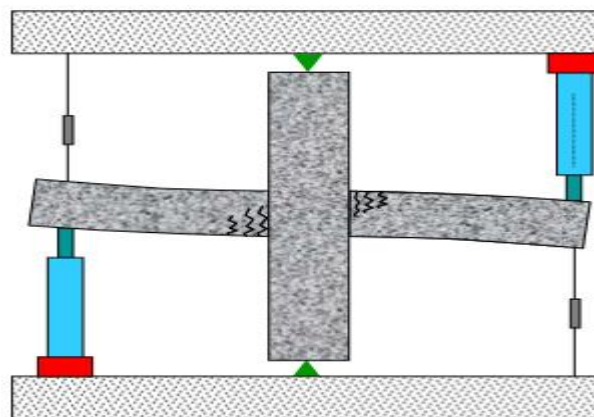
An inelastic element is a component in a mechanical system that experiences permanent deformation or damage when subjected to an external load or force. Unlike elastic elements that can recover their original shape after deformation, inelastic elements exhibit plastic deformation or failure under certain loading conditions.



**Figure II:10.** Schematic of nonlinear beam-column elements [36].

#### II.4.1. The inelastic behavior of materials and structures

The inelastic behavior of materials and structures refers to their response when subjected to loads that exceed their elastic limit or cause permanent deformation. Inelastic behavior occurs due to various factors, including yielding, plastic deformation, creep, fatigue, and fracture. Understanding and analyzing inelastic behavior is crucial in engineering to ensure the safety, reliability, and performance of materials and structures.



**Figure II:11.** Inelastic behavior of materials and structures [28].

Yielding is the onset of plastic deformation in a material when the applied stress exceeds its yield strength. Plastic deformation occurs when the material undergoes permanent changes in shape or size, even after the applied stress is removed. This behavior is common in ductile materials, such as metals, which can undergo significant plastic deformation before failure.

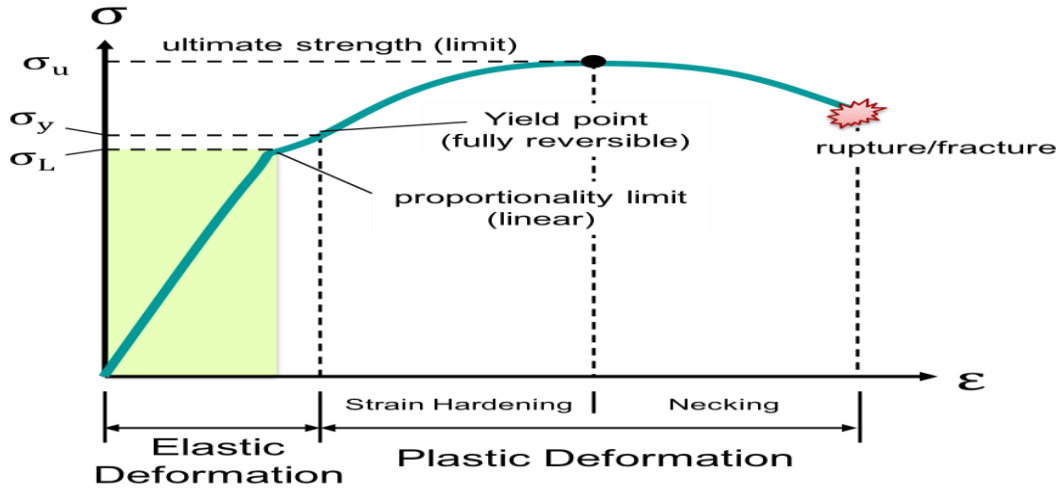


Figure II:12. stress-strain curve [37].

Creep is the time-dependent deformation that occurs when a material is subjected to a constant load or stress over an extended period. Creep is particularly relevant in materials exposed to high temperatures or long-term loading conditions, such as in power plants or structural components.

Fatigue refers to the weakening and failure of a material or structure when subjected to repeated loading and unloading cycles. Fatigue failure can occur at stress levels significantly below the material's ultimate strength, making it critical to consider in the design of structures that experience cyclic loading, such as bridges or aircraft components.

Fracture is the complete separation or rupture of a material when it cannot sustain the applied stress. Fracture can occur suddenly and catastrophically, leading to structural failure. It is essential to understand the fracture properties and behavior of materials to design structures with sufficient strength and resistance to fracture.

To accurately analyze and predict the inelastic behavior of materials and structures, advanced techniques such as nonlinear finite element analysis and material testing are employed. These methods consider the nonlinear stress-strain relationships, plasticity models, and failure criteria to capture the complex behavior of materials and ensure structural safety and performance.

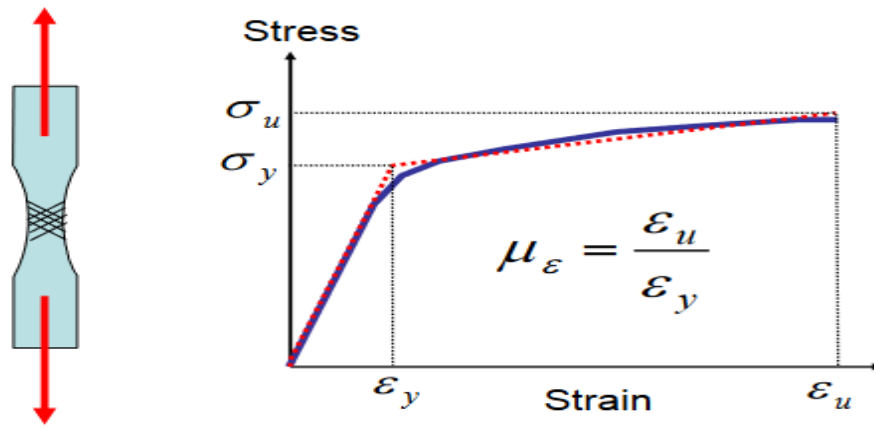


Figure II:13. Inelastic Behavior from Material [28].

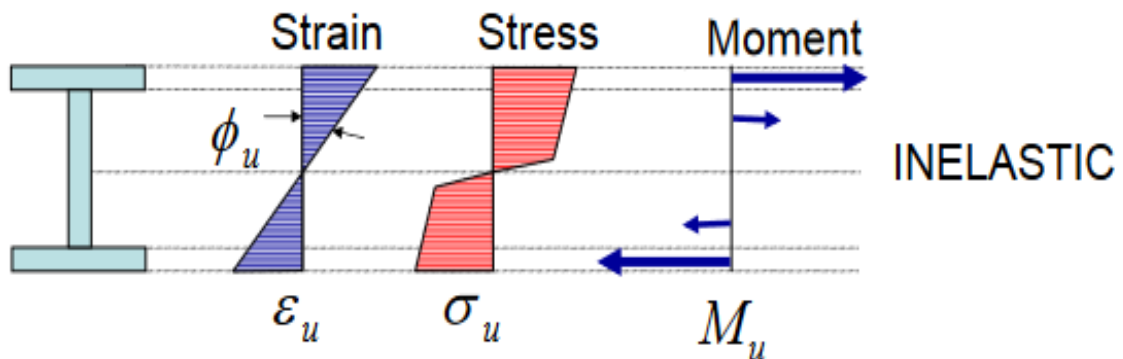


Figure II:14. Inelastic Behavior to Section [28].

### II.5. Curves in nonlinear elements

In nonlinear elements, the stress-strain relationship is not linear, meaning that the deformation of the element is not proportional to the applied stress. This results in curves that describe the relationship between stress and strain, which can take various shapes depending on the specific behavior of the material and the loading conditions.

The possibilities of curves in nonlinear elements refer to the behavior of these elements under different loading conditions. Nonlinear elements can exhibit various types of curves, including:

1. **Hysteresis curves:** These curves show the relationship between stress and strain during a loading and unloading cycle. They exhibit a loop shape due to the inelastic behavior of the element.

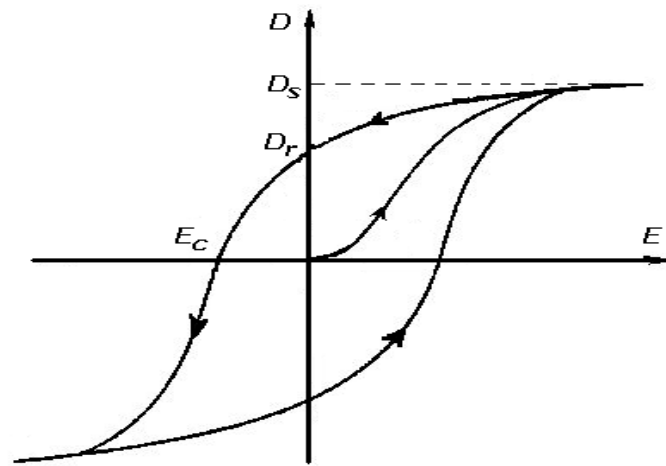


Figure II:15. Hysteresis curve [38].

2. **Load-displacement curves:** These curves show the relationship between the applied load and the resulting displacement in the element. They can be used to determine the stiffness and strength of the element.

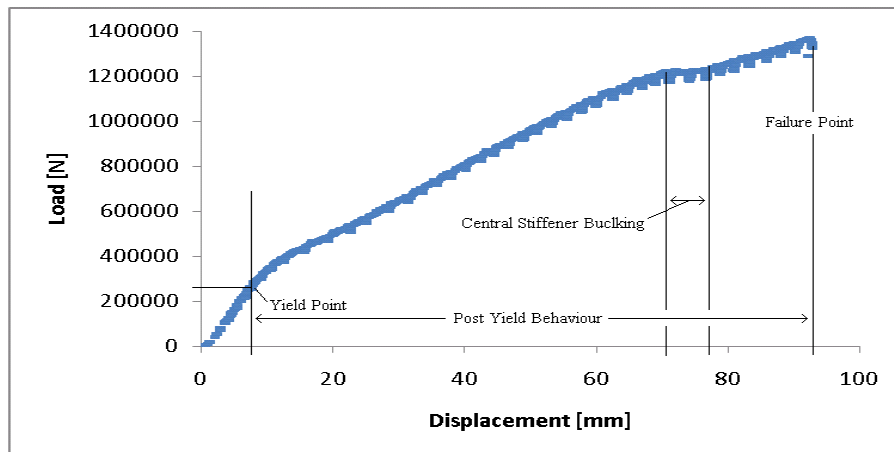


Figure II:16. Load-displacement curves [39] .

3. **Moment-curvature curves:** These curves show the relationship between the applied bending moment and the resulting curvature in the element. They are commonly used in the design of reinforced concrete structures.

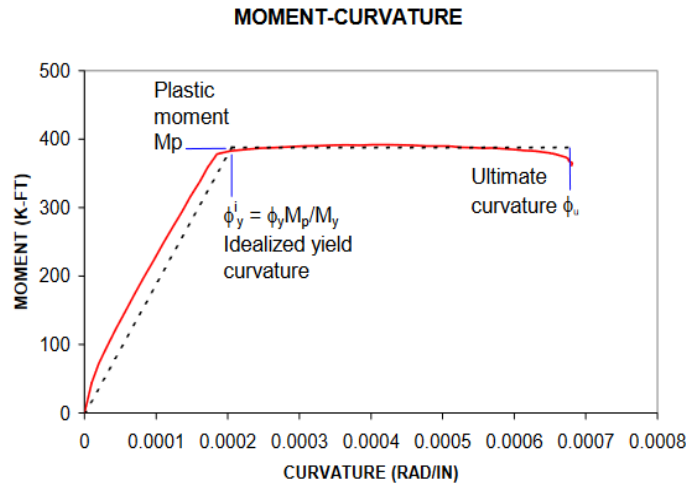


Figure II:17. Moment-curvature curve [40].

4. **Force-displacement curves:** These curves show the relationship between the applied force and the resulting displacement in the element. They are commonly used in the design of mechanical systems.

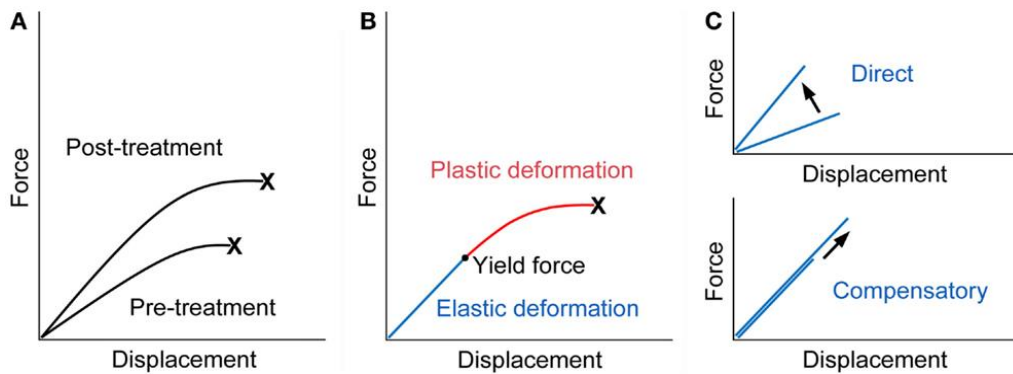


Figure II:18. Force-displacement curve [41].

The type of curve exhibited by a nonlinear element depends on its material properties, geometry, and loading conditions. Engineers use these curves to analyze and design mechanical systems that incorporate nonlinear elements.

**II.6. Non-Linear Element Classes:**

In our study, we used four classes of non-linear elements available in Siesmostruct. These elements allowed us to model the behavior of structures that show accurately non-linear responses, such as plastic deformation, and large displacement. Using these non-linear elements enables us to more accurately predict system behavior under different load conditions, ensuring a safer and more reliable design.

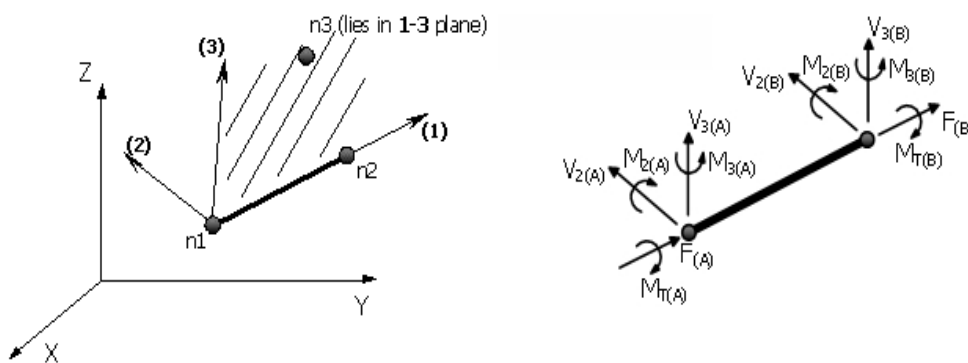
II.6.1. Inelastic force-based frame element type – infrmFB

An inelastic force-based frame element is a finite element used in structural analysis to model the behavior of inelastic materials, such as steel or reinforced concrete, under cyclic loading. This type of element takes into account the nonlinear behavior of the material, including yielding and post-yielding behavior.

The element is formulated based on the principles of force equilibrium and compatibility. The forces and deformations are calculated at discrete points along the length of the element, and the element is then assembled with other elements to form the complete structural system. The element typically consists of two nodes, each with three degrees of freedom (displacements in the x, y, and z directions).

The inelastic behavior of the element is typically modeled using a plastic hinge approach. Plastic hinges are assumed to form at discrete locations along the element where the moment exceeds the yield moment capacity of the material. The element is then divided into segments, with each segment representing a plastic hinge. The forces and deformations are calculated separately for each segment.

One important feature of the inelastic force-based frame element is the use of a displacement-based plasticity model. This means that the element is capable of tracking the displacement history of the material as it undergoes cyclic loading, which is important for accurately capturing the material behavior.

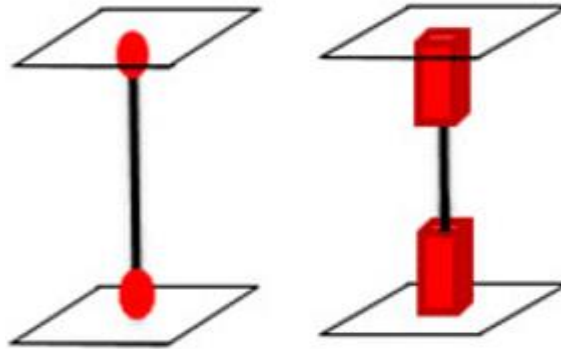


**Figure II:19.** Local axes and output notation type-infrmFB [42].

II.6.2. Inelastic force-based plastic hinge frame element type- infrmFBPH

An inelastic plastic-hinge force-based frame element is a finite element used in structural analysis to model the behavior of inelastic materials under seismic or cyclic loading conditions. This type of element takes into account the nonlinear behavior of the material, including yielding and post-yielding behavior, and is commonly used in the design of steel or reinforced concrete

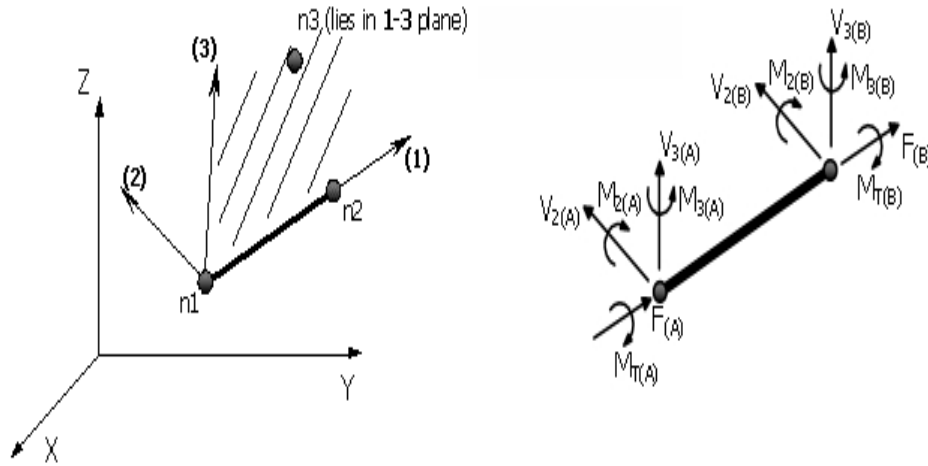
structures. This element is a plastic-hinge version of the inelastic force-based frame element (infrmFB). It uses a similar distributed inelasticity forced-based approach, but limits the inelasticity to a fixed length of the element. This approach was proposed by Scott and Fenves in 2006. The benefits of this formulation include reduced analysis time, as fiber integration is only required for the two-member end sections. Additionally, it provides full control and calibration of the plastic hinge length, which helps to overcome issues related to localization, as discussed by Calabrese et al. in 2010.



**Figure II:20.** Beam-Column a plastic hinge [43].

To carry out equilibrium computations at the element's end sections, a defined number of section fibers is required. The number of section fibers needed varies depending on the shape and material characteristics of the element, as well as the level of inelasticity the element will be subjected to. Generally, single-material sections are adequately represented by 100 fibers, while more complex sections subjected to high levels of inelasticity require 200 or more fibers. However, determining the optimum number of section fibers requires a sensitivity study on a case-by-case basis.

The plastic hinge length also needs to be defined, and guidance on this can be found in the literature, such as Scott and Fenves (2006), Papadrakakis (2008), and Calabrese et al. (2010). [42]



**Figure II:21.** Local axes and output notation type-infrmFBPH [42].

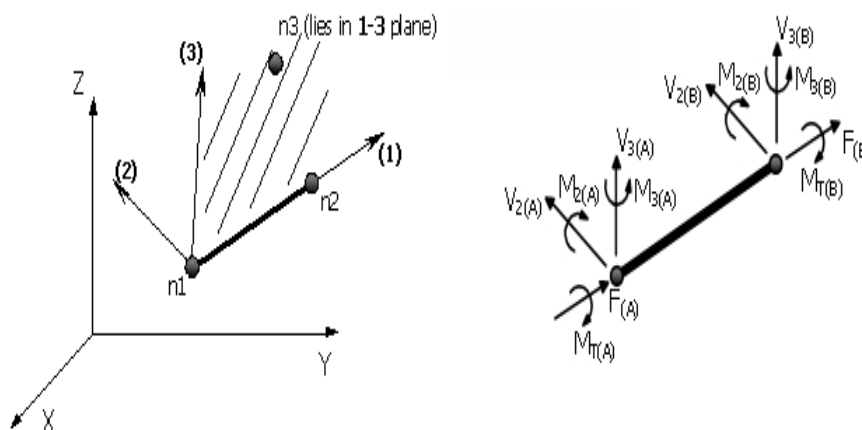
### II.6.3. Inelastic displacement-based plastic hinge frame element type – infrmDBPH

This 3D beam-column element with concentrated plasticity at the two ends is a displacement-based plastic-hinge element. It follows the one-component Giberson model [Giberson, 1967], which consists of one elastic girder and four nonlinear rotational springs attached at the two ends of the element in both the 2nd and 3rd local axes.

Nonlinear deformations of the element are confined to these rotational springs, while the rest of the member remains elastic. The moment-rotation curves in the two local axes at each end are independent. Although this element simplifies the force-based plastic hinge element by confining inelastic deformations to a finite region at the ends of the girder and uncoupling the behavior in the two local axes, it provides increased stability and significantly shorter analysis times.

The DBPH formulation accommodates geometric nonlinearities, while material inelasticity at the plastic hinges is modeled with an asymmetric Takeda type of curve featuring different yield values in tension and compression. The monotonic curve is described by a trilinear skeleton curve, which considers cracking of concrete and yielding of reinforcing steel.

The program automatically calculates the hysteretic curve parameters, so users need only specify the member's section. Like the infrmFB and infrmFBPH elements, changes in reinforcement details within the same member can be achieved using multiple sections per element. [42]



**Figure II:22.** Local axes and output notation type-infrmDBPH [42].

#### II.6.4. Inelastic displacement-based frame element type - infrmDB

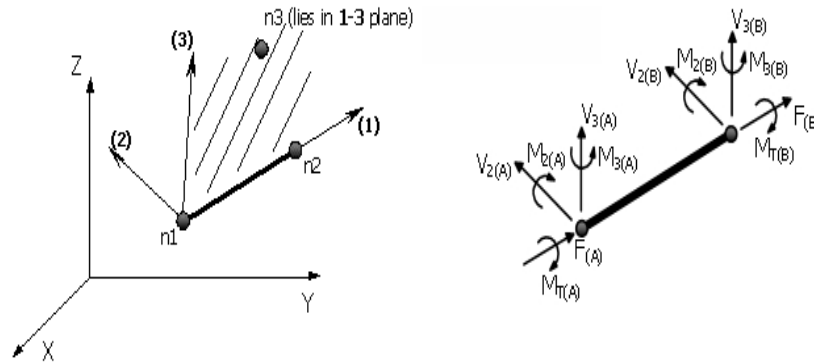
The inelastic displacement-based frame element type, called infrmDB, is a structural element used in the analysis of seismic response in buildings and structures. This element is designed to simulate the inelastic behavior of the structure under large earthquake motions, where traditional linear methods are no longer appropriate. This is the displacement-based 3D beam-column element type capable of modelling members of space frames with geometric and material nonlinearities. As described in the Material inelasticity paragraph, the sectional stress-strain state is obtained through the integration of the nonlinear uniaxial material response of the individual fibres in which the section has been subdivided, fully accounting for the spread of inelasticity along the member length and across the section depth. Unlike traditional linear frame elements, the infrmDB element allows for inelastic behavior to be modeled within a finite section of the element, rather than concentrating it at specific points like plastic hinges. This allows for a more accurate representation of the actual behavior of the structure, as it undergoes inelastic deformations throughout the length of the element.

The infrmDB element is based on a displacement-based formulation, which means that the element deformation is described in terms of nodal displacements, rather than element forces. This allows for a more intuitive understanding of the element behavior, as the deformations are directly related to the nodal displacements.

The inelastic behavior of the element is modeled using a series of nonlinear springs, which are designed to represent the nonlinear response of the structure under large earthquake motions. These springs are distributed along the length of the element, allowing for a more accurate representation of the inelastic behavior of the structure.

The infrmDB element is a powerful tool in the analysis of seismic response, as it allows for a more accurate representation of the inelastic behavior of the structure. However, its use requires a

good understanding of nonlinear structural behavior and the appropriate modeling of the material properties, as well as the selection of appropriate analysis parameters.



**Figure II:23.** Local axes and output notation type-infrmDB [42].

## II.7. CONCLUSION

In conclusion, finite element analysis has revolutionized the field of engineering and has become an essential tool for understanding and predicting the behavior of complex structures and systems. Through the study of finite element methods and the different types of elements used in analysis, we have gained insight into the fundamental principles and assumptions underlying these methods. By applying this knowledge to real-world problems, we can accurately model and analyze structures under a variety of loading conditions, leading to safer and more efficient designs.

## ***Chapter III :***

### ***Pushover Analysis***

## Chapter III. Pushover analysis

### III.1. Introduction

Modern seismic design codes allow engineers to use either linear or nonlinear analyses to compute design forces and design displacements. For example, Eurocode 8 contains four methods of analysis: linear simplified static analysis, linear modal analysis, nonlinear pushover analysis and nonlinear time-history analysis. These methods refer to the design and analysis of framed structures, mainly buildings and bridges. The two nonlinear methods require advanced models and advanced nonlinear procedures in order to be fully applicable by design engineers. This Chapter introduces the use of pushover analysis for assessment purposes.

### III.2. Pushover analysis

Pushover analysis, also known as nonlinear static analysis, is a seismic analysis used in structural engineering to assess the seismic performance of buildings and other structures. It involves subjecting a structure to a series of incrementally increasing lateral loads or displacements to evaluate its nonlinear behavior and identify potential failure modes. The analysis is typically performed by modeling the structure as a series of interconnected elements with predefined plastic hinge locations.

In pushover analysis, the structure's response is determined by calculating the internal forces and deformations at each step as the applied load or displacement increases. Nonlinear behavior is accounted for by incorporating plastic hinges at critical locations, which represent regions of localized yielding and damage. These plastic hinges are assumed to deform in a nonlinear manner, absorbing and redistributing the applied energy.

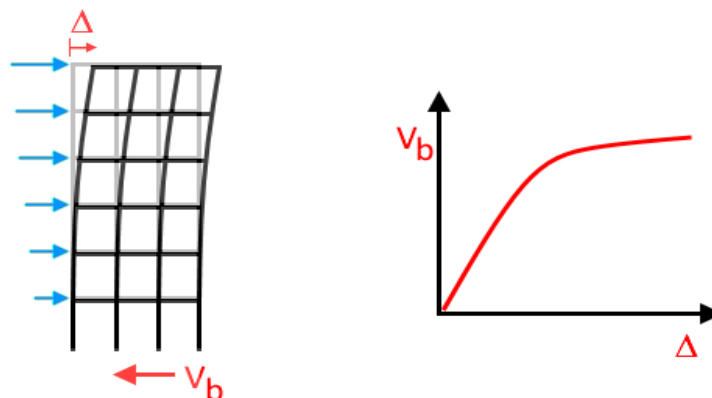
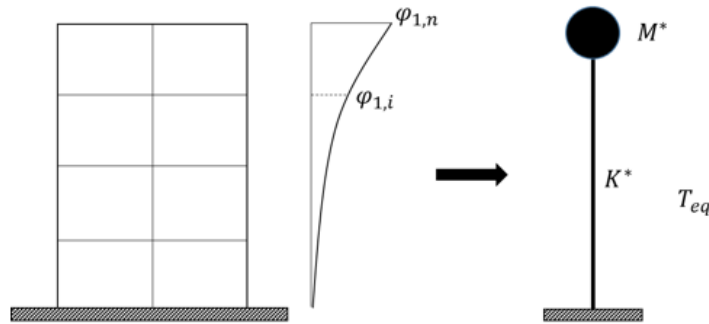


Figure III:1.Pushover Analysis curve.

The analysis aims to estimate the structure's global capacity curve, which represents the relationship between the applied lateral load or displacement and the corresponding resisting force or deformation capacity. By comparing the capacity curve with predetermined performance objectives or acceptance criteria, engineers can assess whether the structure can withstand a given seismic event without experiencing excessive damage or collapse.

### III.3. Theoretical background of pushover analysis

The conventional pushover method has no strict theoretical basis. It is mainly based on the assumption that the response of the structure is controlled by the first vibration mode(s), and that this shape remains constant during the elastic and inelastic response of the structure. This provides the basis for transforming a dynamic problem into a static problem, which is theoretically wrong. Moreover, the response of a MDOF structure is related to the response of an equivalent SDOF system, as shown in Figure III.2.



**Figure III:2.** Transformation from MDOF system to SDOF one.

The differential equation of the SDOF system will be:

$$[M]\{\varphi\}\ddot{u}_t + [C]\{\varphi\}\dot{u}_t + [K]\{\varphi\} u_t = -[M]\{1\}\ddot{u}_g \quad (3.1)$$

where  $[M]$  is the mass matrix,  $[C]$  is the damping matrix,  $[K]$  is the stiffness matrix of the structure,  $\{1\}$  is a unit vector,  $\ddot{u}_g$  is the acceleration of the floor and  $\{\varphi\}$  is a unique mode shape vector. The differential equation described the response of the SDOF equivalent system is written as follows:

$$M^*\ddot{u}^* + C^*\dot{u}^* + K^*u^* = -M^*\ddot{u}_g \quad (3.2)$$

In which:

$$M^* = \{\varphi\}^T [M] \{1\} \quad (3.3)$$

$$C^* = \{\varphi\}^T [C] \{\varphi\} \frac{\{\varphi\}^T [M] \{1\}}{\{\varphi\}^T [M] \{\varphi\}} \quad (3.4)$$

$$K^* = \{\varphi\}^T [K] \{\varphi\} \frac{\{\varphi\}^T [M] \{1\}}{\{\varphi\}^T [M] \{\varphi\}} \quad (3.5)$$

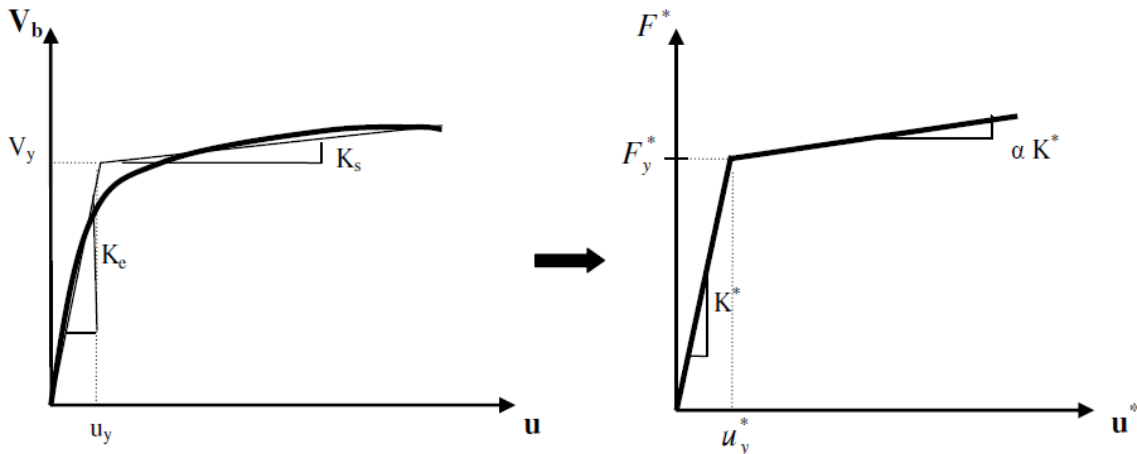
Thus, the period  $T_{eq}$  of the equivalent SDOF system will be:

$$T_{eq} = 2\pi \sqrt{\frac{M^*}{K^*}} \quad (3.6)$$

where,  $K^*$  defines the elastic stiffness of the SDOF system which is given by:

$$K^* = \frac{F_y^*}{u_y^*} \quad (3.7)$$

The idealized pushover curve (see Figure 3.3) can be used to define the properties of the SDOF system.



**Figure III:3.** Capacity curve (a) of MDOF system (b) of SDOF system.

The maximum displacement of the SDOF system subjected to a given earthquake can be found from the elastic or inelastic spectra or from a temporal dynamic analysis. Then, the corresponding displacement of the MDOF system can be estimated by:

$$u_t = \frac{\{\varphi\}^T [M] \{1\}}{\{\varphi\}^T [M] \{\varphi\}} u^* \quad (3.8)$$

#### III.4. Conventional pushover analysis methods

There are several conventional pushover analysis methods, all with the same general steps but different details. Among these methods we find:

- ✓ Capacity Spectrum Method (CSM) - ATC-40 [44]

- ✓ N2 Method [45]
- ✓ FEMA356 coefficient method (DCM) [46]

In this section, the N2 method is detailed.

### **III.5. The N2 Method**

The method is presented by Fajfar et al. [47] as an alternative to the CSM method. The basic idea of the N2 method comes from the **Q** model developed by Saiidi and Sozen [48]. The main difference between the N2 method and the CSM method is the type of demand spectrum used to estimate the target displacement. The steps of the method are presented in the following sections.

#### **3.5.1. Nonlinear static analysis of the MDOF structure**

A lateral load distribution is applied to the assumed structure based on the fundamental mode of vibration. Other lateral load distributions can also be used. A nonlinear static analysis is then performed to obtain a shear force at the base-displacement curve at the top of the MDOF structure (capacity curve).

##### III.5.1. . Definition of the equivalent inelastic system SDOF

The pushover curve is converted to the equivalent SDOF system capacity spectrum using the following equations:

$$S_a = \frac{V_b}{\Gamma_j \cdot M^*} \quad (3.9)$$

$$S_d = \frac{u}{\Gamma_j \varphi_n} \quad (3.10)$$

where  $M^* = \sum M_i \varphi_{ij}$  is the effective mass of the building,  $\varphi_n$  is the corresponding roof mode component of the structure and  $\Gamma_j$  the modal participation factor of mode j. An approximate bilinear idealization of the capacitance spectrum is performed to determine the yield strength  $F_y^*$ , the corresponding displacement  $u_y^*$  and the effective period  $T_{eq}$  of the SDOF system using Equation 3.6.

III.5.2. Seismic demand

The damped elastic acceleration spectrum to be used is defined in the ADRS format. The inelastic spectra are then calculated using the ductility reduction factor  $R_\mu$  and the ductility factor  $\mu$ . The reduction factor  $R_\mu$  is defined as follows:

$$R_\mu = \frac{S_{ae}}{S_{ay}} \quad (3.11)$$

Where  $S_{ae}$  is the pseudo-acceleration ordinate of the response spectrum and  $S_{ay}$  is the limit elastic acceleration of the capacity spectrum. The acceleration ordinate  $S_a$  and the spectral displacement  $S_d$  of an inelastic SDOF system can be calculated as follows:

$$S_a = \frac{S_{ae}}{R_\mu} \quad (3.12)$$

$$S_d = \frac{\mu}{R_\mu} S_{de} \quad (3.13)$$

Where  $S_{de}$  is the elastic spectral displacement. The factor  $R_\mu$  is usually expressed in terms of ductility  $\mu$  and period  $T$ , through the so-called relations  $R_\mu - \mu - T$ . An example of bilinear relation  $R_\mu - \mu - T$  presented in [85], is shown graphically below (Figure 3.4).

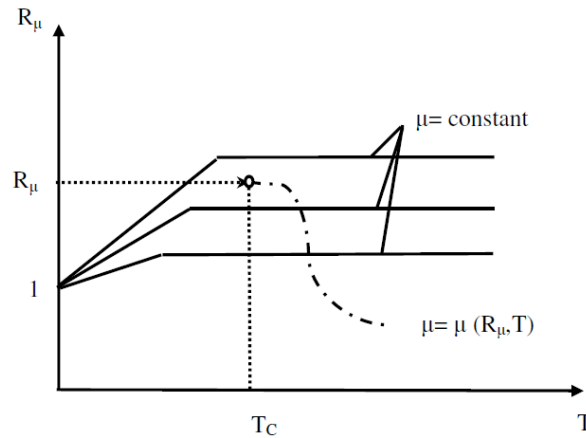


Figure III:4. Relationship  $R_\mu - \mu - T$  [49].

$$R_\mu = \begin{cases} (\mu - 1) \frac{T}{T_c} + 1 & T < T_c \\ \mu & T \geq T_c \end{cases} \quad (3.14)$$

Where,  $T_c$  is the characteristic period of the earthquake, defined as the transition period between the constant acceleration domain and the constant velocity domain of the spectrum.

3.5.4. Seismic demand for equivalent SDOF system

The displacement demand  $S_d$  of the ESDOF system can be determined by:

$$S_d = \begin{cases} \frac{S_{de}}{R_\mu} \left( (R_\mu - 1) \frac{T_c}{T} + 1 \right) & T < T_c \\ S_{de} & T \geq T_c \end{cases} \quad (3.15)$$

Equation 3.15 implies that the displacement estimate will always be larger than the initial elastic displacement for short-period structures ( $T < T_c$ ). This is illustrated in Figure 2.9. Figure 2.10 shows the procedure applied in the case of medium and long period structures.

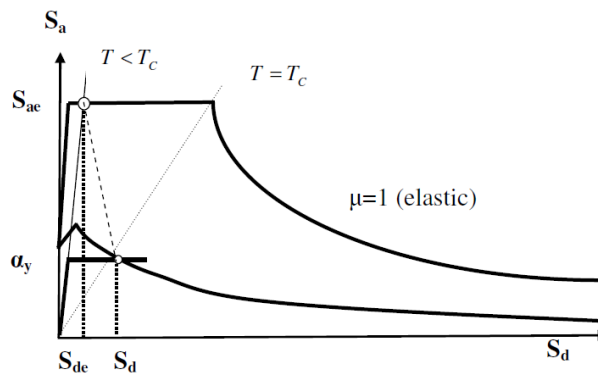


Figure III:5. Estimation of the target displacement by the N2 method when  $T < T_c$  [49].

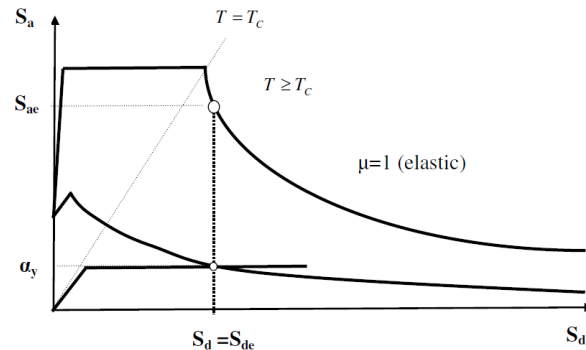


Figure III:6. Estimation of the target displacement by the N2 method when  $T \geq T_c$  [49].

III.5.3. Global seismic demand for the MDOF model

The displacement  $S_d$  of the SDOF equivalent system can be transformed into the roof displacement  $u_t$  of the MDOF model using the following equation:

$$u_t = \Gamma_j S_d \quad (3.16)$$

Local quantities of interest such as rotations, inter-storey displacements... corresponding to  $u_t$ , can then be determined.

### **III.6. Pushover analysis advantages**

Pushover analysis can provide the following advantages:

- ✓ It allows us to evaluate overall structural behaviors and performance characteristics.
- ✓ It enables us to investigate the sequential formation of plastic hinges in the individual structural elements constituting the entire structure.
- ✓ When a structure is to be strengthened through a rehabilitation process, it allows us to selectively reinforce only the required members, thereby maximizing the cost efficiency.

### **III.7. Pushover analysis Limitations**

Pushover analysis procedures in general involve a separation between structural capacity and seismic demand. However, many research works have attempted to establish an interconnection between structural capacity and seismic demand. These procedures also assume that structural damage is a function only of lateral deformation of the structure, neglecting duration effects and the cumulative demand for energy dissipation.

Also, the conventional pushover procedures do not take into account progressive changes of the modal properties during the non-linear deformation of the structure, which also leads to the lengthening of the period and thus to different spectral amplifications.

### **III.8. Conclusion**

Despite the shortcomings of these types of analyses, they provide the engineer with an effective alternative to expensive inelastic dynamic analysis and also overcome some constraints in the practice of seismic codes.

# ***Chapter IV:***

## *Case study*

## **Chapter IV. Case study**

### **IV.1. Introduction:**

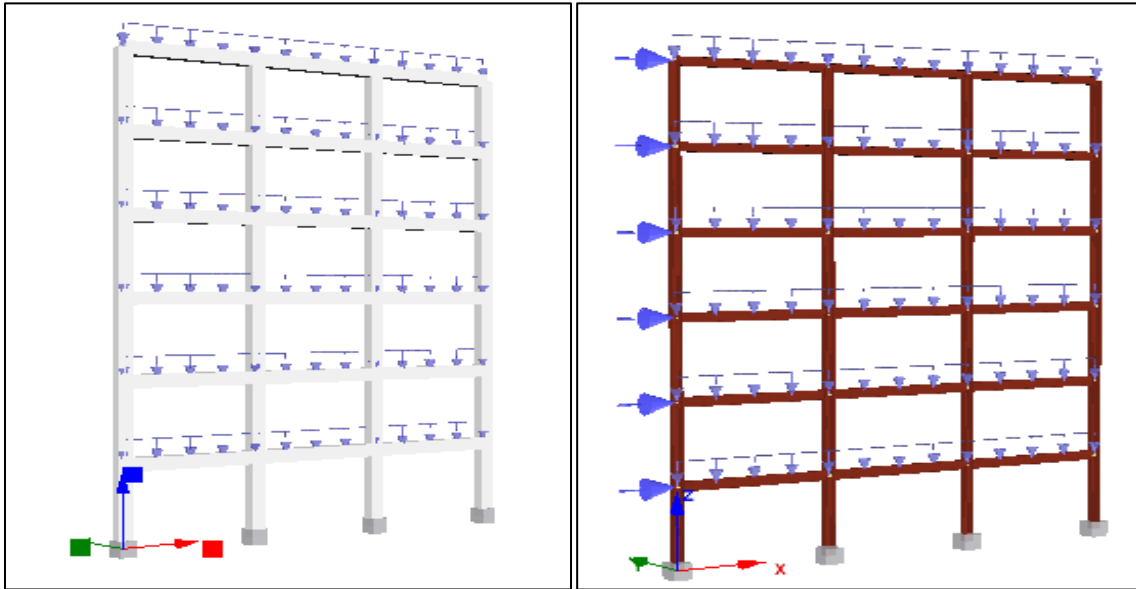
This chapter focuses on examining the impact of element modeling assumptions on the overall performance of steel structures. Specifically, a comprehensive study is conducted on 6-storey buildings sourced from existing literature. These buildings are modeled using SeismoStruct software. The modal information of each building is extracted, followed by conducting pushover analysis for different types of element models. The obtained results include pushover curves, storey displacements, and the drift storey ratio, which holds significant importance in evaluating structures situated in seismic zones.

This chapter also includes a comparative analysis by studying a reinforced concrete structure of the same number of storeys. The objective is to investigate the influence of different element model types. This comparison aims to provide a clearer understanding of the complexities involved in selecting an appropriate element model for analyzing building structures. By exploring the variations and implications of different model types, a comprehensive picture can be obtained regarding the importance of this choice in accurately assessing the behavior of structures.

### **IV.2. Description of the Frame Structures:**

In the current work, two dimension (2D) concrete and steel structures containing 6-storeys with 3 and 6 m bay each, and 3 m height of each floor, are designed to assess the impact of the element model assumption on the global response of buildings. The buildings are designed according to the provisions of the European structural code Eurocodes [4,12] based on peak ground acceleration (PGA) equal to 0.35 g, damping coefficient  $\xi = 5\%$ , soil class B and behavior factor  $q = 4.0$ . It is that gravity loads on beams is equal to 27.5 kN/m (dead and live loads of floors)

, for steel structure, steel beams are made of S235 grade. The first four stories have columns with HEB280 sections and beams of IPE360 sections, while the next two floors have columns with HEB260 sections and beams with sections IPE330. For the Concrete structure: the compressive strength is about 25 MPa and reinforcement bars yield strength is 400 MPa. The section sizes of the beams and columns are of (40\*50) cm<sup>2</sup>. More details about these buildings are presented below in figures 4.1 and 4.2, and also tables .1 and 4.2 [50]



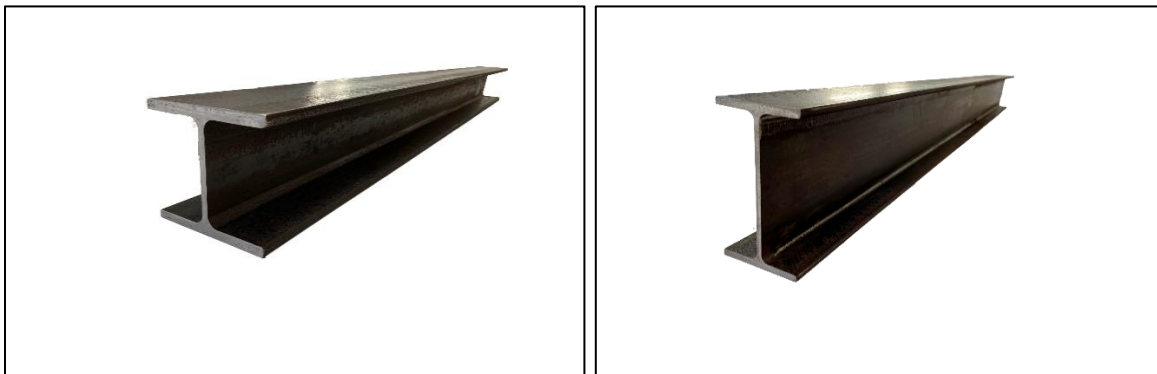
(a)

(b).

**Figure IV:1.** (a): Reinforced concrete frame; (b): Steel frame.

Tableau IV:1: Geometrical details of columns and beams (steel structure).

Storey	Columns	Beams
1-4	HEB280	IPE360
5-6	HEB260	IPE330

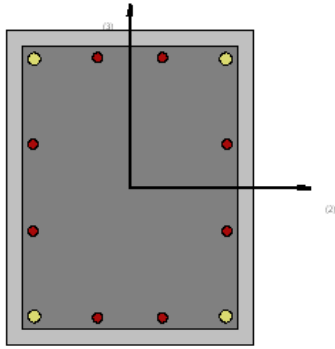


(a) 2

(b)3

**Figure IV:2.** (a): HEB Section[51]; (b): IPE Section[52].

**Tableau IV:2.** Geometrical details of columns and beams (reinforced concrete).

storey	Columns	Beams
1-6	Section height: 50 cm Section width: 40 cm Cover thickness: 2.5 cm	Section height:50 cm Section width: 40 cm Cover thickness: 2.5 cm
<b>Beams reinforcement bars:</b> Left right_ sides: (4 $\phi$ 16 mm) Top bottom_sides: (4 $\phi$ 16 mm) Corners: (4 $\phi$ 20 mm)		

IV.2.1. 3.1.2 Material Properties of the frame:

Reinforced concrete structure model is employed for defining the concrete material with the following parameters:

- $F_c = 25 \text{ MPa}$  ;  $f_t = 0.001 \text{ MPa}$  ;  $\epsilon_c = 0.002 \text{ mm/mm}$

Then, the bilinear model with kinematic strain-hardening is employed for defining the steel material with the following parameters:

- $E_s = 200000 \text{ MPa}$ ;  $f_y = 459 \text{ MPa}$ ;  $\mu = 0.004$

Steel model is employed for defining the steel materials: for steel structure steel beams and columns are made of S235 grade

- Modulus of elasticity:  $E = 210000 \text{ MPa}$
- Yield strength:  $f_y = 235 \text{ MPa}$
- Ultimate strength  $f_u = 360 \text{ MPa}$

**IV.3. Presentation of the used Software**

**SeismoStruct** is a software program widely used for structural analysis and design, specifically for seismic applications. It offers several important features and capabilities that make it a preferred choice for seismic assessment.

One of the key features of SeismoStruct is its ability to perform advanced seismic analyses, including pushover analysis, response spectrum analysis, and time history analysis. These

analysis methods allow engineers to assess the structural response under different seismic loading conditions and evaluate the performance of the structure.

The software provides a user-friendly interface and intuitive workflow, making it easy to model complex structures and define various structural elements such as beams, columns, slabs, and walls. It also offers a wide range of material models and element types, allowing for accurate representation of different structural components.

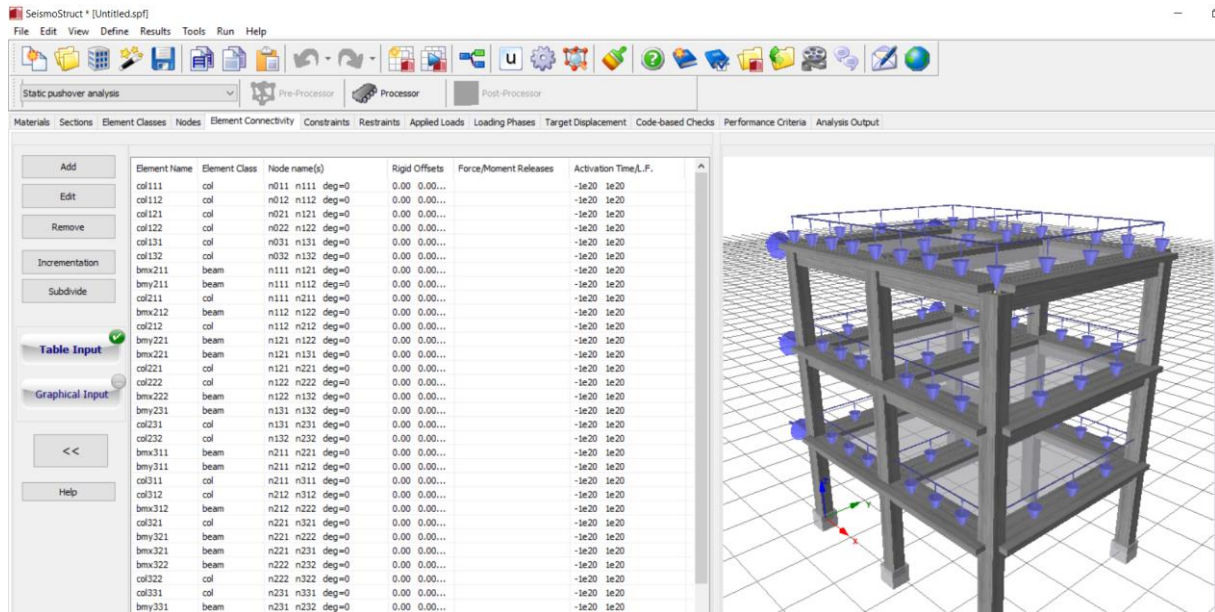


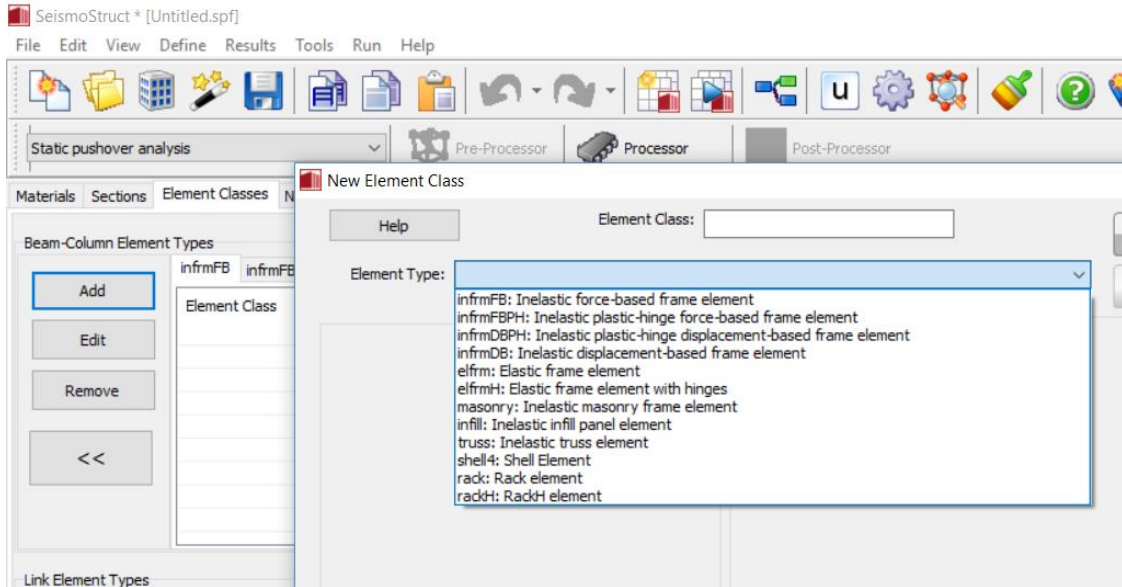
Figure IV:3. SeismoStruct software interface.

#### IV.4. ELEMENT MODELING

We have created five different models for each structure using SeismoStruct software. The difference between them is only the element model assumption. The five element model are:

- ✓ **Elfrm**: elastic frame
- ✓ **InfrmFB**: inelastic frame force-based
- ✓ **InfrmFBPH**: inelastic frame force-based plastic hinge
- ✓ **InfrmDBPH**: inelastic frame Displacement-based plastic hinge
- ✓ **InfrmDB**: inelastic frame Displacement-based

For the steel structure, the analysis using the **InfrmDBPH** is not performed because the software does not provide the model parameters.



**Figure IV:4.**Element classes in SeismoStruct.

These models provided by SeismoStruct take into consideration several factors such as flexibility, linear/nonlinear behavior, fiber count, force/displacement control, and frame type. The utilization of different design approaches within the SIESMOSTRUCT software allows us to cater to various design scenarios and project requirements, ensuring the integrity and safety of the structures under consideration.

## IV.5. RESULTS OF THE STEEL STRUCTURE

### IV.5.1. 5.1. Preliminary results

Table IV.3. Presents the results in terms of periods of vibration and mass participating mass ratio of the 1<sup>st</sup> mode of vibration for the steel structures using different element models. As seen in the table, all the element models give the same results which means that all the models have the same characteristics in the elastic phase in particular the element stiffness.

**Tableau IV:3.** Periods and mass participation of 1st mode for each steel model.

Element Modelling	Mode	1 <sup>st</sup> Period	Mass participation
<b>elfrm</b>	1	1.19	81.09%
<b>infrmFB</b>	1	1,19	81,09%
<b>infrmFBPH</b>	1	1,20	81,16%
<b>InfrmDBPH</b>	1	1,20	81,16%
<b>infrmDB</b>	1	1,19	81,09%

IV.5.2. 5.2 Pushover curve :

Figure IV.5 depicts pushover curves of the steel structure and for the four element models considered in this study. For the **elmf** model, the whole response is linear and the base shear is proportional to the displacement as seen in the Figure. For the three rest of models the behavior is devised in two phases; Elastic and plastic phases. It is clear that all the models have the same initial stiffness. However, they do not have the same elastic limit and they do not have the same maximum strength and ductility. **InfrmDB** have the greatest strength can exceed 1000 kN, while **InfrmFBPH** and **InfrmFB** give equal strength and the largest ductility (here we consider  $\mu = 0.6/0.1 = 6$ ).

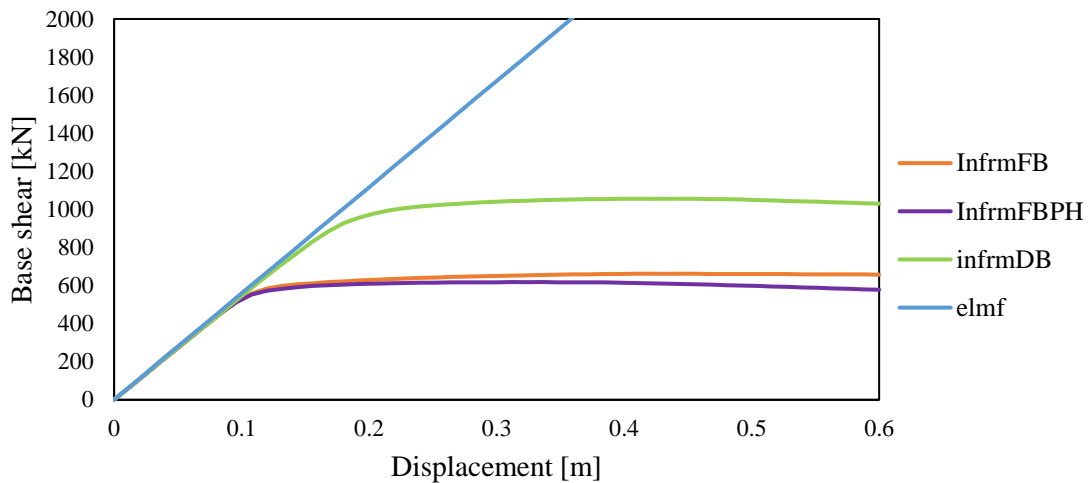


Figure IV:5. pushover curves of the steel structure using different element model.

IV.5.3. 5.3 Storey displacement profile

For comparison purposes a displacement equal to **0.2 m** is selected to stop the pushover analysis and evaluate the displacements profiles. Storey displacements are shown in Figure IV.6 for the steel structures and considering different element models. **infrmFB** and **infrmFBPH** give the same and the largest displacements for all the storeys. While, **elmf** model give the smallest displacement.

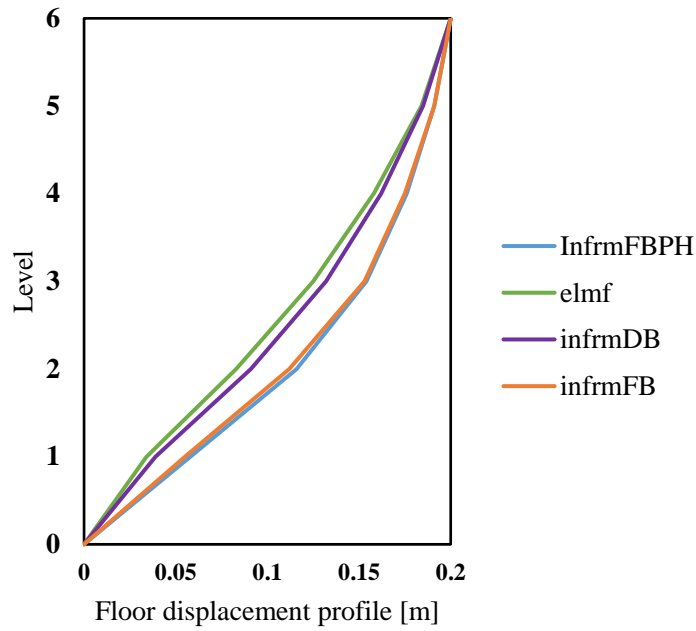
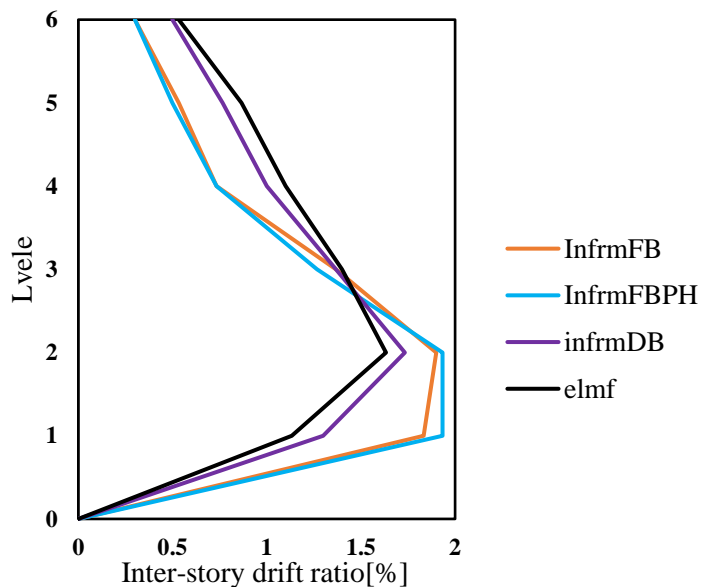


Figure IV:6. Comparison of displacement between floors.

IV.5.4. 5.4 Inter-storey drift ratio:

Inter-Storey drift ratio is calculated by dividing the inter-story by the height of the storey and multiplying the results by 100. Figure 4.7 present these results for the steel structure. For the 3 first storeys it can be seen that the force-based models (infrmFB and infrmFBPH) give the largest values in terms of storey drift (can exceed 1.9%). However, these two models give small values in mid and upper storeys. Elmf gives the smallest values for the lower storeys witch make the design Insafe. infrmDB also provides the smallest values but this time in upper-storeys.



**Figure IV:7.** Comparison of Inter-story drift ratio.

**IV.6. RESULTS OF THE RINFORCED CONCRETE (RC) STRUCTURE**

IV.6.1. 6.1. Preliminary results

Table IV.4 presents the results in terms of periods of vibration and mass participating mass ratio of the 1<sup>st</sup> mode of vibration for the RC structures using different element models. Table IV.4 shows that all the element models give comparable results, which means that all the models have the same characteristics in the elastic phase (the same global/local stiffness).

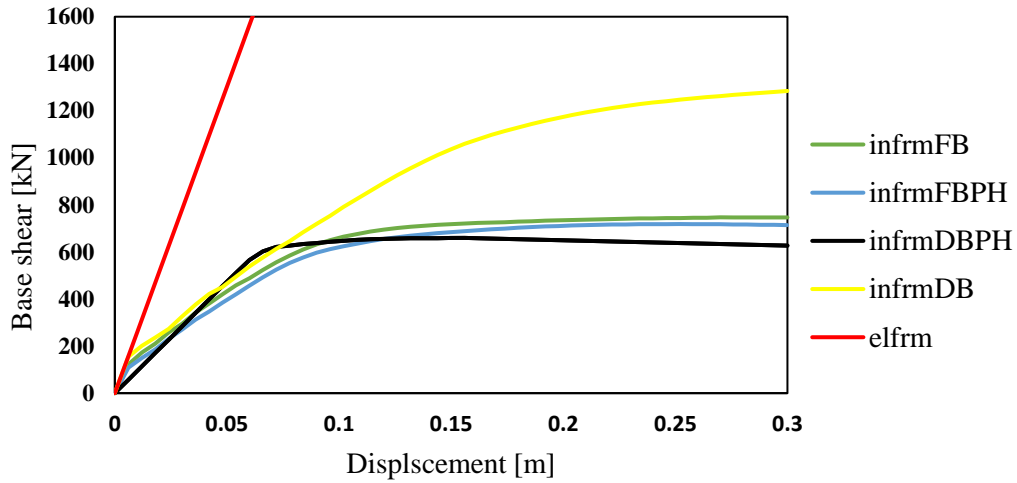
**Tableau IV:4.** Periods and mass participation ratios for the RC structure in first mode:

Element Modelling	Mode	1 <sup>st</sup> Period	Mass participation
elmf	1	0.67	82.29%
infrmFB	1	0,63	82,28%
infrmFBPH	1	0,64	82,31%
infrmDBPH	1	0,67	82,29%
infrmDB	1	0,63	82,28%

IV.6.2. 6.2 Pushover Curve

Figure IV.8 depicts pushover curves of RC structure and this time for the five element models considered in this study. Ass seen in the case of steel structure, the response, obtained from the using of the **elmf** model, is linear and the base shear is proportional to the displacement as shown in the Figure IV.8.

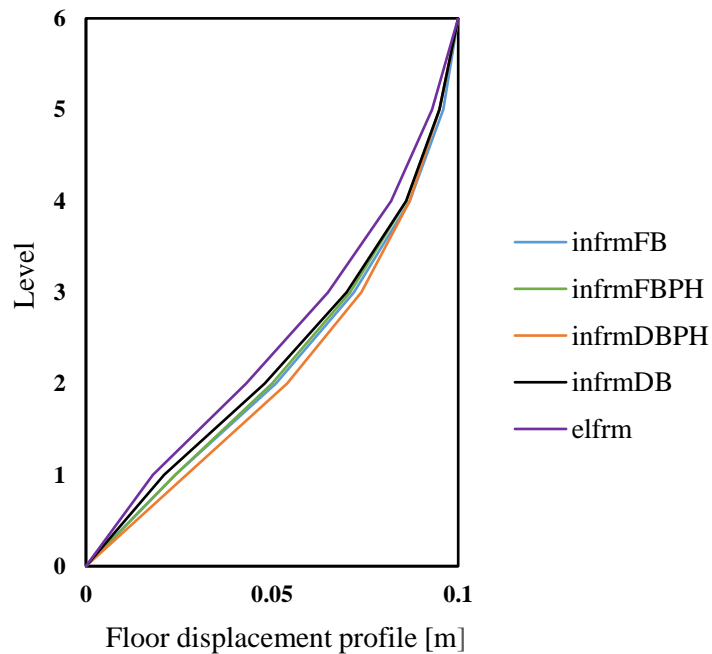
The four inelastic models show a response devised in two phases; Elastic and plastic phases. It is clear that all the models have a comparable initial stiffness. However, they do not have the same elastic limit and they do not have the same maximum strength and ductility. As observed in the steel structure InfrmDB have the greatest strength can reach 1200 kN, while InfrmFBPH and InfrmFB give equal strength and ductility (here we consider  $\mu = 0.3/0.1 = 3$ ). InfrmDBPH give the smallest value of the strength (600 kN) among the other models. However, this element model record the largest value of the ductility ( $\mu = 0.3/0.06 = 5$ ).



**Figure IV:8.** Comparison of pushover curves for the RC frame.

IV.6.3. 6.3 Storey displacement profile:

For comparison purposes a displacement equal to **0.1 m** is selected to stop the pushover analysis and evaluate the displacements profiles. Storey displacements are shown in Figure IV.9 for the RC structures and considering different element models. This time, *infrmDBPH* gives the largest displacements for all the storeys. While, *elfrm* model give the smallest displacement.



**Figure IV:9.** Storey displacement profile of the RC structure.

IV.6.4. 6 .4 Inter-storey drift ratio:

Figure IV.10 present the results in terms of storey drift ratio for the RC structure. For the 2 first storeys it can be seen that the infrmDBPH gives the largest values in terms of storey drift (can reach 0.8% in the first storey). However, this model gives the smallest value in mid and upper storeys. Elmf gives the smallest values for the lower storeys witch make the design Insafe. InfrmDB, infrmFB and infrmFBPH values still between the boundary values.

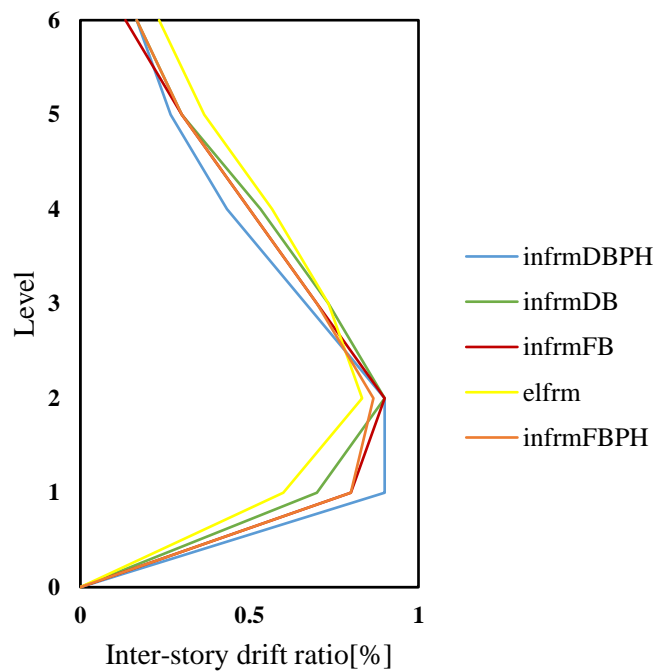


Figure IV:10. Comparison of Inter-story drift ratio.

IV.7. CONCLUSION:

In conclusion, this chapter focused on the linear and nonlinear analysis of steel and concrete frames using the Pushover analysis. We examined the differences between the analysis of a concrete structure and a steel structure. Additionally, we emphasized the importance of the finite element model used in the analysis and how they impact the results, particularly in terms of base shear and inter-story drift. Through our analysis, we have confirmed the fact that different element types may yield different results in terms of base shear and inter-story drift, highlighting the importance of selecting appropriate element types that best represent the structural behavior.

# ***GENERALE CONCLUSION***

## **GENERAL CONCLUSION**

In conclusion, this study has examined the influence of nonlinear finite element modeling assumptions on the seismic response demands of steel buildings. By considering the nonlinear behavior of structural elements, as well as the geometric and material properties of the buildings.

Five element models are used in this research which are; elastic frame, inelastic frame force-based, inelastic frame force-based plastic hinge, inelastic frame Displacement-based plastic hinge, inelastic frame Displacement-based plastic hinge and inelastic frame Displacement-based. We have find that:

- Periods and participation mass ratios show that all the element models give comparable results, which means that all the models have the same characteristics in the elastic phase (the same global/local stiffness). For that, there is no difference between all the used models in the elastic phase.
- The four inelastic models show a response devised in two phases; Elastic and plastic phases. It is clear that all the models have a comparable initial stiffness. However, they do not have the same elastic limit and they do not have the same maximum strength and ductility.
- In terms of displacement, inelastic frame Displacement-based plastic hinge gives the largest displacements for all the storeys. While, elastic model give the smalest displacement.
- The diffrence between the results of the nonlinear models is large for the steel structure in comparision to the RC structure specialy in terms of storey drift ratio.

Finaly, we demonstrated in this study that different element types can provide varied outcomes in terms of base shear and inter-story drift, emphasizing the necessity of choosing appropriate element types that appropriately depict structural behavior.



# References

## References

### Chapter I:

- [1]: Bolt, B. A. (Invalid Date). earthquake. Encyclopedia Britannica.  
<https://www.britannica.com/science/earthquake-geology>
- [2]: <https://www.pinterest.com/pin/earthquake-danger-zone-seismic-hazard-zones-world-map->
- [3]: <https://www.syr-res.com/article/21001.html>
- [4]: <https://www.bbc.com/arabic/middleeast-64609500>
- [5]: <https://www.insureon.com>
- [6]: <https://www.pbs.org/newshour/world>
- [7]: <https://www.nationalnursesunited.org>
- [8]: <http://learningfromearthquakes.org>
- [9]: <https://www.alhurra.com/zlzal-trkya-wswrya>
- [10]: <https://www.aljazeera.net/news/>
- [11] - Ricles, J. M., Deierlein, G. G., Sause, R., Uang, C. M., & Whittaker, A. S. (2011). Seismic performance of steel moment-resisting frame buildings. *Journal of Structural Engineering*, 137(10), 1083-1092.
- [12]- Uang, C. M., & Goel, S. C. (2002). Seismic behavior and design of steel buildings with eccentrically braced frames. *Journal of Structural Engineering*, 128(10), 1304-1311.
- [12]: Chopra, A.K. (2012). *Dynamics of Structures: Theory and Applications to Earthquake Engineering* (4th Edition). Prentice Hall.
- [13]: <https://www.mechead.com/what-is-linear-and-nonlinear-analysis/>
- [14] : <http://130.149.89.49:2080/v2016/books/gsa/default.htm?startat=ch08s02.html>
- [15]: <https://yasincapar.com/nonlinearities-in-fem/> / Nonlinearities in FEM
- [16]: <https://www.corrosionpedia.com/definition/1610/plastic-hinge-structural-engineering>
- [17]: <https://www.tinoseismic.co.nz/articles>
- [18]: <https://www.quora.com/What-is-plastic-hinge-formation>
- [29]: Naeim, F., & Kelly, J. M. (2013). *Design of seismic isolated structures: from theory to practice*. John Wiley & Sons.
- [20]: Ayoub, N. (2021). *Reduced order finite element models for the nonlinear dynamic analysis of reinforced concrete structures under seismic excitations* (Doctoral dissertation, HESAM Université)
- [21] : REGLES PARASISMIQUES ALGERIENNES RPA 99 / VERSION 2003

- [22]: Robert V. Whitman, in Encyclopedia of Physical Science and Technology (Third Edition), 2003
- [23]: Virkhare, D., Variegate L., & Nair, V. (July-2015). Pushover Analysis of Water Tank Staging. international Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056  
Volume: 02 Issue: www.irjet.net p-ISSN: 2395-0072
- [24]: Sajed, Tofiq & Mukhlis, Mohammad. (2021). EFFECTS OF PLAN ASPECT RATIO ON SEISMIC RESPONSES OF RC BUILDINGS BY TIME HISTORY ANALYSIS.
- [25]: Carvalho, G., Bento, R., & Bhatt, C. (2013). Nonlinear static and dynamic analyses of reinforced concrete buildings-comparison of different modelling approaches. Earthq. Struct, 4(5), 451-470.
- [26]: <https://byjus.com/physics/elastic-behaviour-of-materials/>
- [27]: <https://www.researchgate.net/publication/301796039/figure/fig6/AS:6689315765780771536497124625/Linear-versus-non-linear-elastic-behavior-Notes-A-Stress-strainrelationship-of-a.png>
- [28]: Bathe, K. J. (2016). Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM) Finite Element Analysis (FEA) or Finite Element Method (FEM). 1065.
- [29]: [https://fr.wikipedia.org/wiki/D%C3%A9formation\\_%C3%A9lastique](https://fr.wikipedia.org/wiki/D%C3%A9formation_%C3%A9lastique)
- [30]: Hibbeler, R. C. (2015). Mechanics of materials (10th ed.). Pearson.
- [31]: <https://static.hindawi.com/articles/ace/volume-2019/3725373/figures/3725373.fig.0012.jpg>
- [32]: [https://2.bp.blogspot.com/-nhXuLaF975U/U3rqDYdKZI/AAAAAAAAABYA/QJadGeeqzaU/s1600/Beam\\_bending.png](https://2.bp.blogspot.com/-nhXuLaF975U/U3rqDYdKZI/AAAAAAAAABYA/QJadGeeqzaU/s1600/Beam_bending.png)
- [33]: <https://www.researchgate.net/profile/Konstantinos>
- [34]: Reddy, J. N. (2004). Mechanics of laminated composite plates and shells: theory and analysis. CRC press.
- [35]: <https://www.researchgate.net/profile/De-Cheng-Feng/publication/325278896/figure/fig4/AS:697765579485184@1543371686508/Fiber-section-model-with-shear-deformation.png>
- [36]: <https://www.researchgate.net/profile/Milad-Roohi/publication/344727512/figure/fig1/AS:947991531769856@1603030202008/Schematic-of-nonlinear-beam-column-elements-Data-from-Deierlein-et-al-2010.jpg>
- [37]: <https://ocw.tudelft.nl/course-readings/1-1-6-main-takeaways-on-stress-and-strain/>

- [38]: <https://upload.wikimedia.org/wikipedia/commons/4/4d/Ehysteresis.PNG>
- [39]: <https://www.researchgate.net/publication/283703497/figure/fig2/AS:669959986364428>
- [40]: [http://www.structsource.com/pdf/Momcurv\\_web.pdf](http://www.structsource.com/pdf/Momcurv_web.pdf)
- [41]: <https://www.frontiersin.org/articles/10.3389/fendo.2015.00006/full>
- [42]: SeismoStruct help system
- [43]: <https://www.researchgate.net/publication/308668978/figure/fig9/AS:410944686575616@1474988256058/Idealized-Models-of-Beam-Column-Elements-a-Plastic-hinge-concentrated-at-the-two.png>
- [44]: Freeman, S. A. (2004). Review of the development of the capacity spectrum method. ISET Journal of Earthquake Technology, 41(1), 1-13.
- [45]: Fajfar, P., Kilar, V., Marusic, D., Perus, I., & Magliulo, G. (2005, August). The extension of the N2 method to asymmetric buildings. In Proceedings of the 4th European workshop on the seismic behaviour of irregular and complex structures (Vol. 41).
- [46]: Abass, H. A., & Jarallah, H. K. (2021). Comparative Study of the Seismic Assessment According to ATC-40, FEMA-356 and FEMA-440 for Existing Hospital Building Located at Baghdad City. no. June.
- [47]: Fajfar, P., Krawinkler, H., & Fischinger, M. (1997). Nonlinear seismic analysis of RC buildings with the capacity-spectrum method. Earthquake Engineering & Structural Dynamics, 26(9), 979-994.
- [48]: Saiidi, M., & Sozen, M. A. (1983). Seismic response of reinforced concrete structural walls. Journal of the Structural Division, 109(4), 827-843.
- [49]: Themelis, S. (2008). Pushover analysis for seismic assessment and design of structures (Doctoral dissertation, Heriot-Watt University).
- [50]: Yahmi, D., Branci, T., Bouchaïr, A., & Fournely, E. (2018). Evaluating the behaviour factor of medium ductile SMRF structures. Periodica Polytechnica Civil Engineering, 62(2), 373-385
- [51]: [https://www.acvm-metal.fr/img/img\\_upload/acvm-poutrelle-heb.png](https://www.acvm-metal.fr/img/img_upload/acvm-poutrelle-heb.png)
- [52]: [https://www.acvm-metal.fr/img/img\\_upload/acvm-poutrelle-ipe-cop\\_12.png](https://www.acvm-metal.fr/img/img_upload/acvm-poutrelle-ipe-cop_12.png)



# **Annexes**

## **Annex 1: PRESENTATION OF THE ANALYSIS PROGRAM**

SeismoStruct is a software package based on Finite Element Analysis that can accurately predict the behavior of space frames subjected to static or dynamic loads. It takes into account both geometric nonlinearities and material inelasticity, making it capable of simulating large displacements. The software offers various material models such as concrete, steel, frp, and sma, and provides a wide range of pre-defined steel, concrete, and composite section configurations through a comprehensive library of 3D elements.

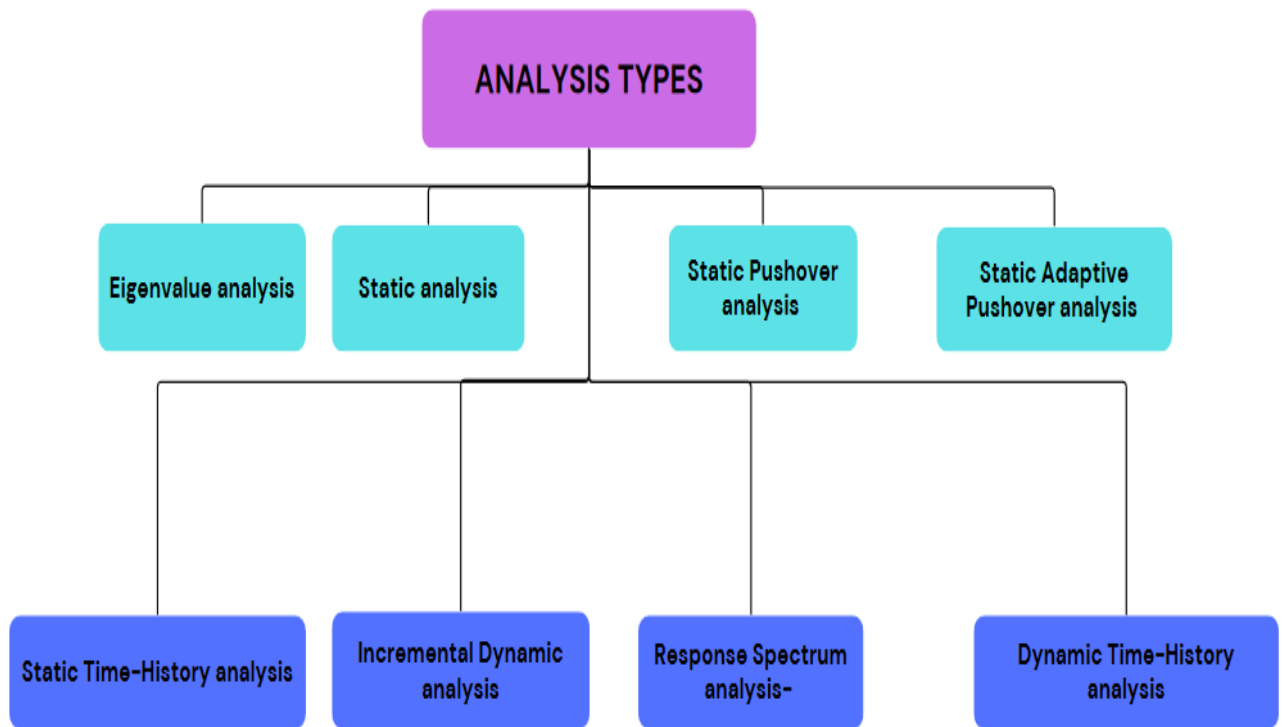
One of the key features of SeismoStruct is its explicit modeling of inelastic behavior along the length of structural members and across the depth of their sections. This enables a precise estimation of damage distribution and allows for accurate assessment of structural integrity. The program also ensures numerical stability and accuracy even at high strain levels, which is essential for determining the inelastic response and collapse load of any frame-type structural configuration.

SeismoStruct supports both static actions, including forces and displacements, and dynamic actions, including accelerations. It offers various analysis capabilities such as eigenvalue analysis, nonlinear static pushover analysis (conventional and adaptive), nonlinear static time-history analysis, nonlinear dynamic analysis, incremental dynamic analysis, and response spectrum analysis.

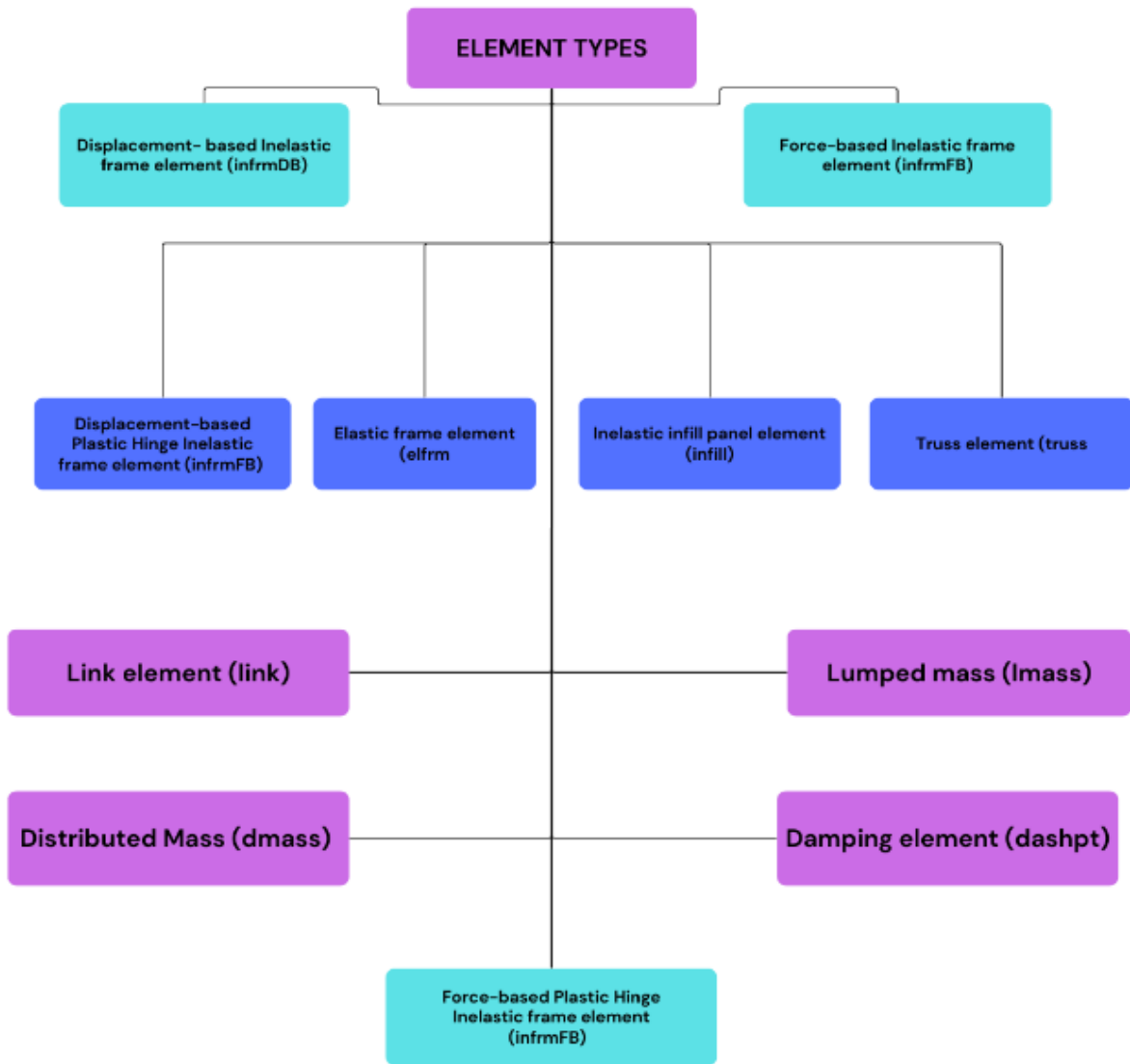
Overall, SeismoStruct is a powerful software tool for analyzing and evaluating the performance of space frames under different loading conditions, considering nonlinear behavior and accurately predicting their response to seismic and other dynamic events.

## **Annex 2: PROGRAM FEATURES COVERED BY THE PROGRAM**

This section aims to show which program aspects (i.e., types of analyses, elements, restrictions, limitations, and applied loads) are handled in each example of the current study through the charts presented below.



**Figure .2.** Analysis types in SeismoStruct.



**Figure .3.** Element types in SeismoStruct.

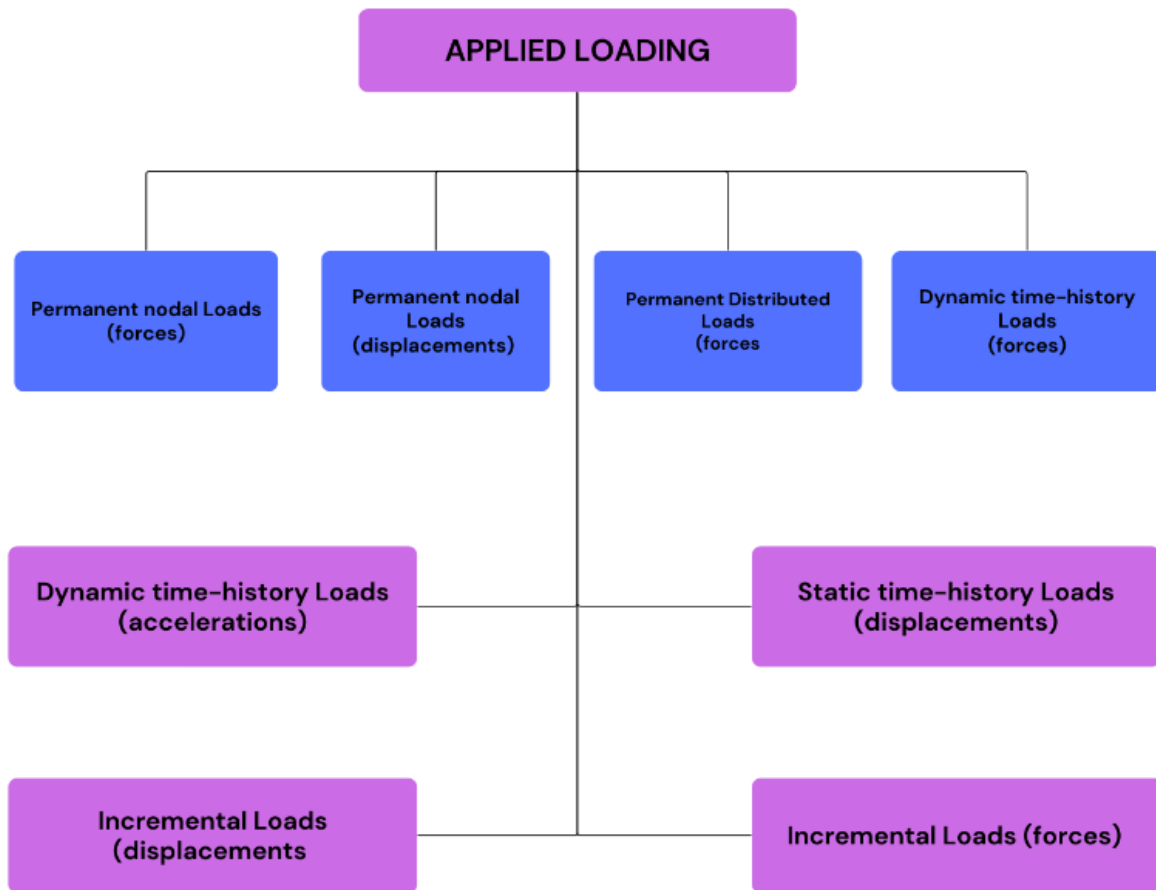


Figure .4. Applied Loading in SeismoStruct.

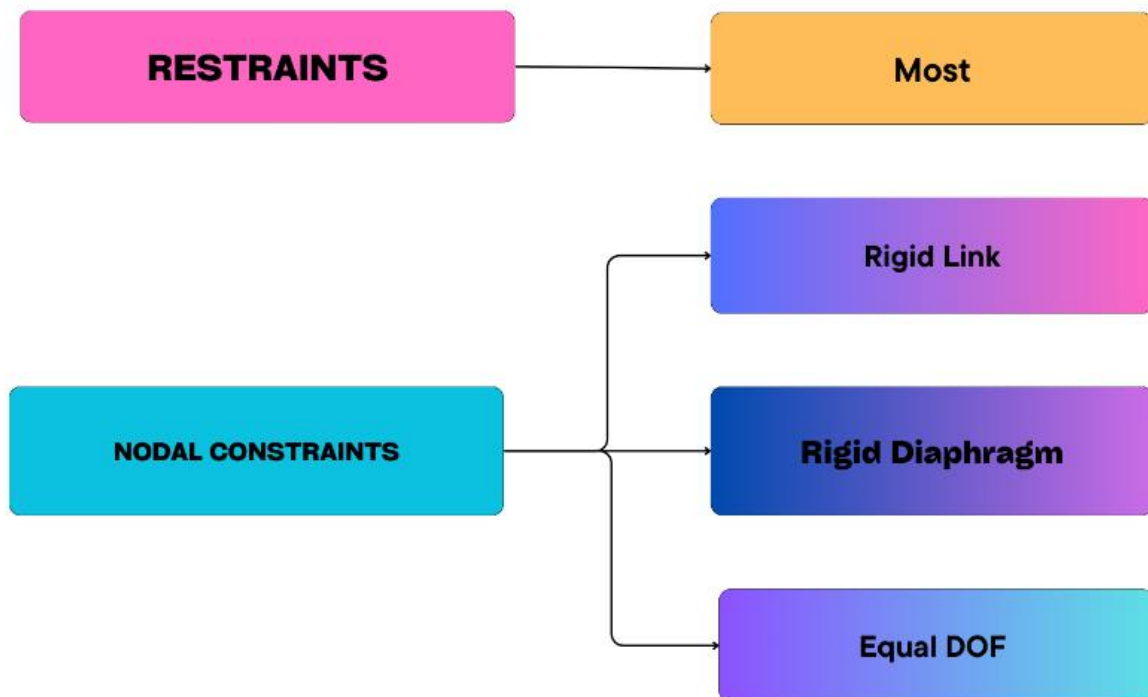


Figure. 5. Restraints and Nodal constraints.

*Hafidi Safia*

**The influence of nonlinear finite element  
modelling assumptions on the seismic response  
demands of steel buildings**

**الملخص**

تبحث هذه الدراسة في تأثير افتراضات النموذج الأساسية على السلوك الزلزالي العام للمباني الفولاذية ذاتية الاستقرار. تمت دراسة نوعين من الهياكل، مبنى فولاذي وهيكلي خرساني مسلح، من أجل تقييم خصائصها الديناميكية غير الخطية. تظهر النتائج أن افتراضات النموذج الأولي لها تأثير كبير على السلوك الزلزالي العام للمباني الفولاذية والخرسانية. لذلك، من الضروري اختيار هذه النماذج بعناية. تسلط هذه النتائج الضوء على أهمية التمثيل الدقيق والمعايرة المناسبة للنماذج الأساسية لضمان تنبؤات موثوقة للأداء الزلزالي في التصميم والتحليل الهيكلي.

**Abstract**

This study investigates the impact of element model assumptions on the global seismic behavior of moment-resisting steel buildings. Two types of structures, a steel building and a reinforced concrete structure, were examined to evaluate their respective compression characteristics. The findings reveal that the element model assumptions significantly influence the overall seismic behavior of both steel and concrete buildings. Therefore, careful consideration must be given to the selection of these models. The results emphasize the importance of accurate representation and proper calibration of element models to ensure reliable predictions of seismic performance in structural design and analysis.