國立中央大學土木工程學系

Finite Element Method

Final Project：

Reinforced Concrete Column Blind Prediction Contest

組員：109302512 陳泰安

109302516 劉晏彤

109302519 目崎勇太

109302526 李宗霖

111322089 黃亭耀

112322022 郭瑞嫺

中華民國　113 年 1 月 8 號

I. ABSTRACT

So, this report digs into checking if those reinforced concrete columns can handle loads that keep coming back. We're using structural analysis software to test if their design is good. We will discuss about how the reinforced concrete columns break and the ways to make them better.

Basically, the report splits into six parts including this part : Abstract、Introduction、Theorem and algorithm、Description、Results and discussion, and the final part is Conclusions

We used SAP2000 and ABAQUSs to analyze this report, so that can give us different angles to show what we found.

In this part of the outline, we're going to talk about what our report's all about and the cool software we're using for structural analysis. We'll give you a quick preview of each section.

The second part of the introduction, we'll delve into the background behind Taiwan's frequent earthquakes and explore the concept of pushover analysis while understanding the arrangement of reinforcement.

Then, in the third part, Theorem and Algorithm, we're going to dig into the condition of those reinforcements and try to derive the hysteresis loop.

Moving on to part four, Description, we're going to use those structural analysis software tools to build models. We'll show you exactly how we do it and explain all the juicy data involved.

In part five, Results and Discussion, we'll chat about what we found, and finally, we will make conclusions in the end.

II. INTRODUCTION

Taiwan lies within the Pacific Ring of Fire and at the junction of the Eurasian Plate and the Philippine Sea Plate. This geographical location results in frequent seismic activity, with an average of tens of thousands of earthquakes occurring annually, about hundreds of which are perceptible. In the recent year of 2022, a total of 184 perceptible earthquakes were recorded.[1][2]

Taiwan gets hit by earthquakes a lot, so it's super important to keep an eye on seismic activity and research here. There are tons of old buildings around. When they get wrecked by earthquakes or other stuff, their strength goes down. That's why we gotta figure out how much shaking they can handle at lower levels and check out columns when they're put through specific repetitive loads.

With the rise of performance-based design methods, we gotta come up with analysis tricks that match. We did a lot of pushover analysis studies to understand how thrust force, shear force diagrams, and bending moment diagrams all relate under displacement control.

Pushover analysis is a static procedure that uses a simplified nonlinear technique to estimate seismic structural deformations.[3] Structures redesign themselves during earthquakes. As individual components of a structure yield or fail, the dynamic forces on the building are shifted to other components.

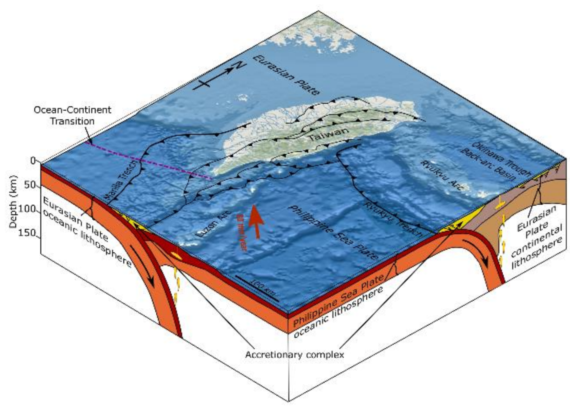


Figure 1 Taiwan seismic zone[4]

Specimen information

Table 1 Specimen information

Specimen number	Failure Mode	L(mm)	Cross Section (cm2)	Cover Concrete (mm)	f'c (MPa)	Fy (MPa)	Fyt (MPa)	S(mm)	ρ (%)	Axial Force
FF-15S-0.1	bending failure	1800	40*40	40	36.31	484.7121	351.48	150	0.61	0.1Agf'c

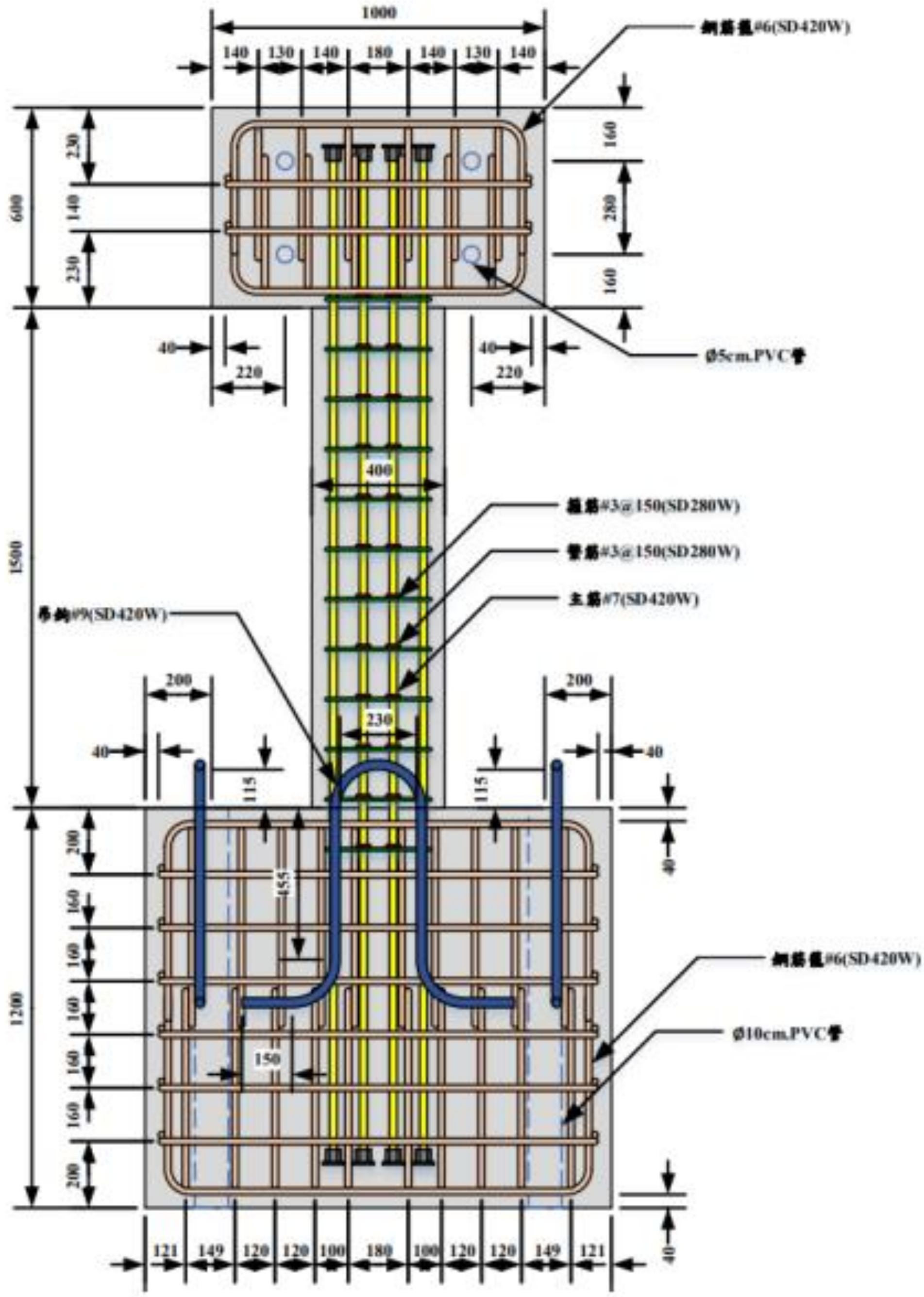


Figure 2 Specimen side view configuration

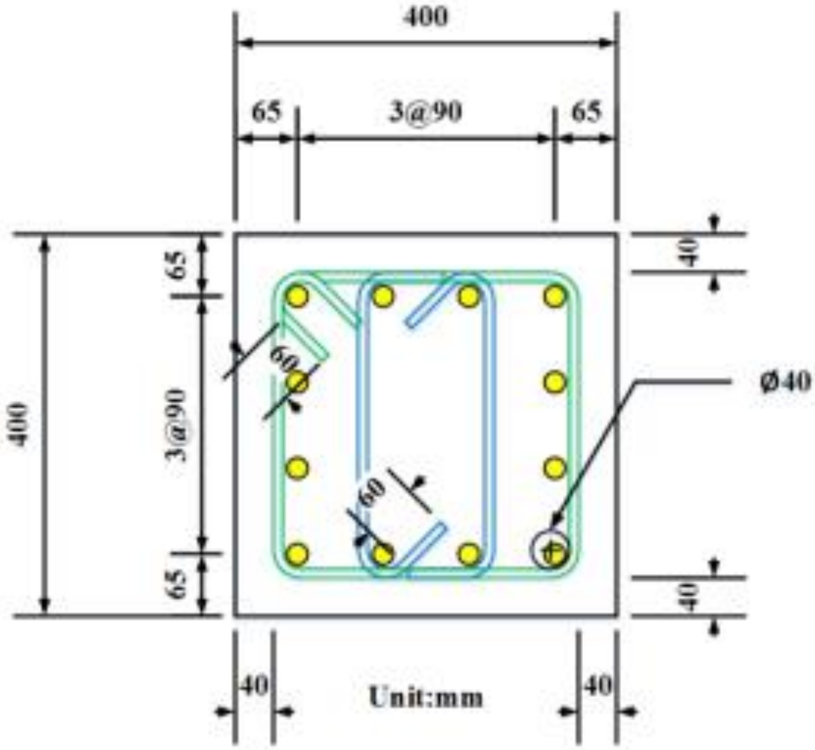


Figure 3 Specimen section configuration

Concrete Material Properties

Table 2 Concrete Properties

Specimen number	Sampling Number	Specimen size(cm)		Age (days)	Compressive Strength(MPa)	Average Compressive Strength(MPa)	Axial pressure(0.1AgF'c)(KN)
		Diameter	Height				
FF-15S-0.1	1	10	20	428	33.93	36.31	581
	2				35.74		
	3				39.25		

Steel Material Properties

Table 3 Rebar Properties

Material	Application	Yield strength (fy)	Tensile strength (fu)
		MPa	MPa
#3(SD280W)	Hoops	351.48	462.356
#7(SD420W)	Vertical (longitudinal) reinf.	484.7121	695.02

Loading Protocol

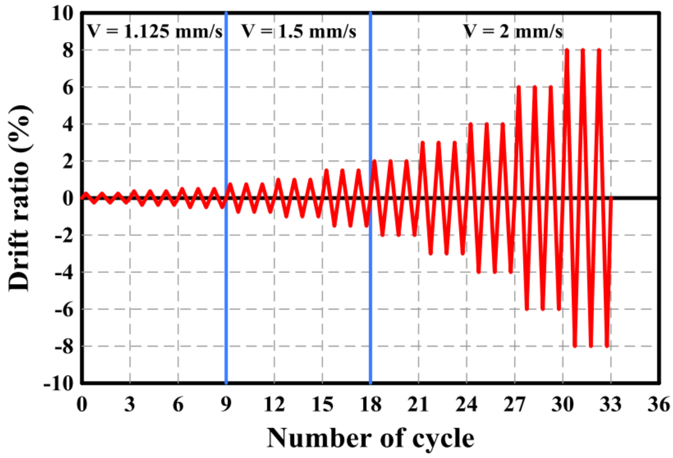


Figure 4 Cyclic loading

Table 4 Loading information

Drift ratio (%)	Cycles	L (mm)	Disp. (mm)	Velocity (mm/s)	Time (s)	Stage	Data Number (筆/步)	資料筆數	Data Time (s)
0.25	3	1800	4.5	1.125	48	4	1	4	57.6
0.375	3	1800	6.75	1.125	72	6	1	6	86.4
0.5	3	1800	9	1.125	96	8	1	8	115.2
0.75	3	1800	13.5	1.5	108	9	1	9	129.6
1	3	1800	18	1.5	144	12	1	12	172.8
1.5	3	1800	27	1.5	216	18	1	18	259.2
2	3	1800	36	2	216	18	1	18	259.2
3	3	1800	54	2	324	27	1	27	388.8
4	3	1800	72	2	432	36	1	36	518.4
6	3	1800	108	2	648	54	1	54	777.6
8	3	1800	144	2	864	72	1	72	1036.8
Total Time (minutes)					52.8	Total Time (minutes)			63.36

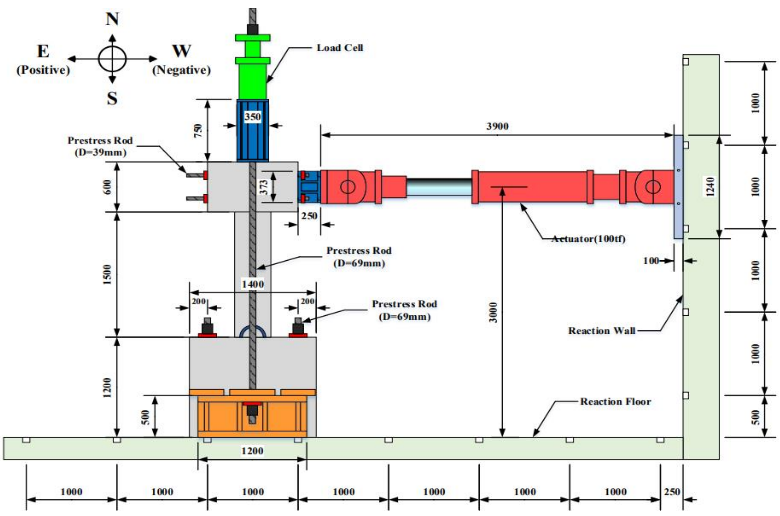


Figure 5 Drift calculation Schematic

III. Theorem and algorithm

Introduction

In architectural design (Figure 1), it is common to consider the seismic resistance of structures to assess the risks associated with earthquakes.

The Performance-Based Earthquake Engineering (PBEE) is a common performance indicator used to evaluate the behavior of structures during earthquakes. It is employed in the design of earthquake-resistant buildings to achieve seismic effectiveness, thereby reducing the potential for casualties and significant property damage.[5]

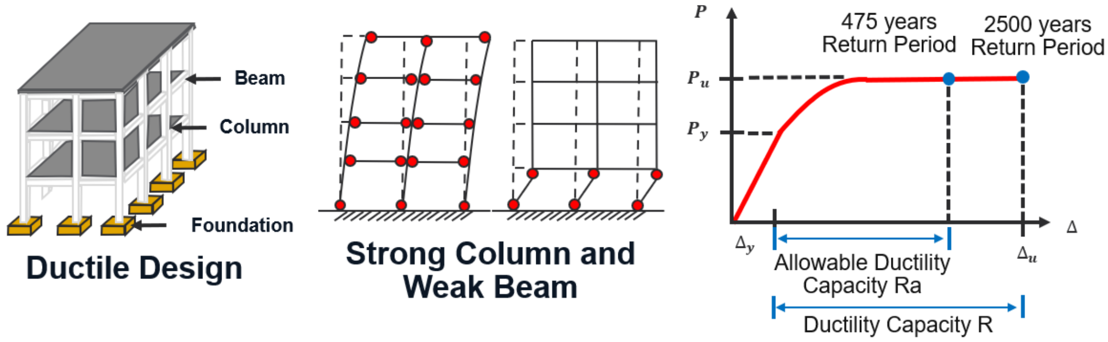


Figure 6 Architectural design flow chart

In The PBEE, the Ground Motion Intensity Measure (IM) is a crucial role as a parameter used to describe the characteristics of earthquake motion. It is typically associated with the strength and spectral properties of seismic waves.

By means of measuring parameters such as acceleration, velocity, displacement, and other characteristics of seismic waves, ground motion intensity measures (IM) are used to quantify the strength or energy of earthquakes.

These measures serve as indicators for assessing earthquake hazards and structural responses in the context of evaluating seismic risk(Figure 2).

Finally, in conjunction with the non-linear simulation model, which describes the structural response to seismic motion, is utilized.

- To evaluate the seismic risk of building :
  - Ground motion intensity measure (IM)
  - Nonlinear simulation model
  - Deterioration of Stiffness and Strength

Human casualties & Dollar losses

Figure 7 To evaluate the seismic risk of building

The common non-linear simulation model used for describing structural plastic deformation is the plastic hinge model. The plastic hinge is typically a specific region of the structural component that allows the structure to undergo reversible deformation when subjected to external forces or seismic loads, instead of experiencing irreversible damage. Designing structures to undergo plastic deformation under extreme loads enhances seismic performance, reduces the extent of structural damage during earthquakes, and improves overall safety. Structures subjected to seismic loads may undergo deformation, and the process of returning to their initial state may involve some energy losses. This non-linear behavior is referred to as structural hysteresis. In order to depict the relationship between load and deformation in this range, it is common to plot a hysteresis loop, which is crucial for assessing the seismic performance and design of structures. The analysis approach involves dividing the structure into individual components and then separately analyzing the hysteresis behavior of plastic hinges within each component.(Figure 3)

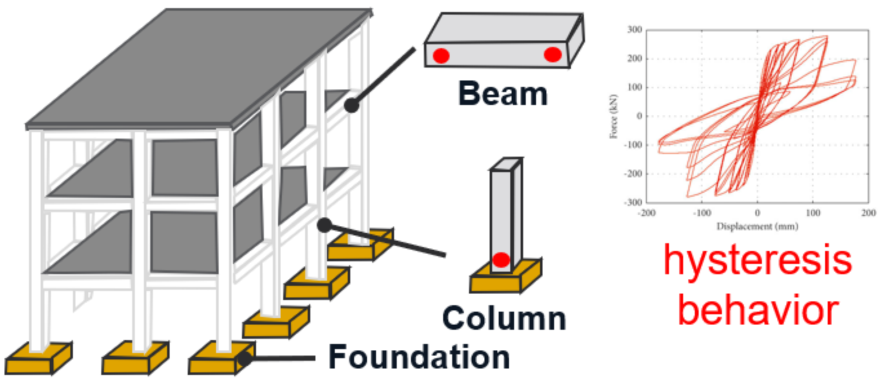


Figure 8 The analysis architectural design approach

By employing experiments and Finite Element Analysis (FEA) Software, it is possible to accurately analyze and determine the hysteresis loop for a given structure. (Figure 4)

However, in practical applications, it is often not feasible to repeat experiments or solely analyze the hysteresis loops of each column using finite element analysis software, considering the challenges of constructing extensive elements.

Therefore, nonlinear models are employed to simulate the behavior of beam-column structures under load, providing a more efficient way to capture the post-loading response without the need for detailed analysis of each individual component. Based on the distribution of plasticity, models can be classified into Concentrated Plasticity Model and Distributed Plasticity Model. The former is a linear analysis model commonly used in many commercial software applications, such as SAP2000, ETABS.

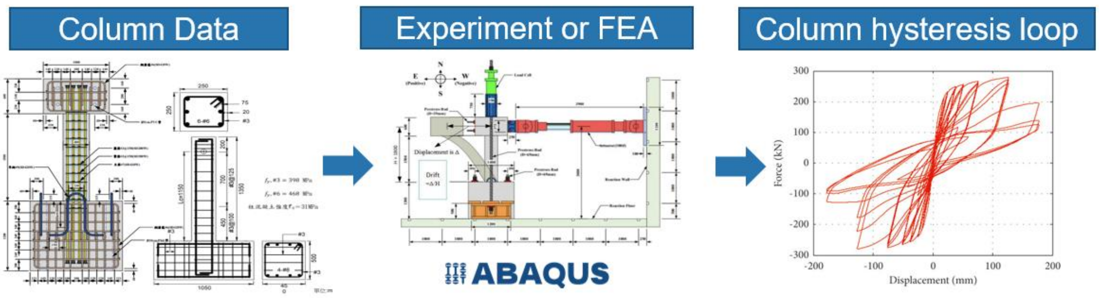


Figure 9 The ways to analyze hysteresis loops

Additionally, the analysis method for the former is simpler than the latter, and it is commonly applicable to component deterioration models and structural collapse analyses. Based on the described considerations, this report utilizes the Concentrated Plasticity Model for discussion.

Takeda model

Hysteresis refers to the dependence of a system on its past history. In a hysteretic system, the input–output relationship exhibits a multi-branch non-linearity, with transitions occurring between branches after input extrema [6].

Hysteresis is a natural phenomenon that is widely observed in various systems, and it has attracted the attention of researchers from different fields [7].

Hysteresis models play a vital role in connecting how structures behave when subjected to hysteresis with the analysis of nonlinear structures.

Many researchers have put forward various models to describe the behavior of different structural systems.

Some of these models are known as polynomial models because they use polynomials or segmented polynomials to represent the hysteresis loops.

One notable characteristic of these models is that their parameters typically have well-defined physical meanings, such as the relationships to stiffness, strength, or ductility of the structure.

For this report, we conducted simulations using both ABQUS and SAP2000 software. In SAP2000, we employed the Takeda hysteresis model for analysis. Therefore, this section will present an introduction to this specific model.

Takeda et al. [8] employed a trilinear curve to describe hysteresis and introduced a stiffness degradation rule. Backbone curve of Takeda model consists of three lines which represent three stages of concrete deformation. (Fig.I) In the first stage, structure is in the elastic stage. In the second and third stages, the structure is cracked and then yields. The stiffness degradation law of Takeda model can be expressed as:

$$k_r = \left(\frac{f_{s_{\mathrm{crack}}} + f_{s_{\mathrm{y}}}}{x_{\mathrm{crack}} + x_{\mathrm{y}}}\right) \left|\frac{x_{max}}{x_{\mathrm{y}}}\right|^{r_{\mathrm{T}}}$$

Where k_r is the unloading stiffness, x_{max} is the maximum displacement and x_y is the yielding displacement, x_{crack} and fs_{crack} are the displacement and restoring force when the structure is cracked, fs_y is the yielding restoring force and r_T is the stiffness degradation coefficient of Takeda model.

The hysteresis loop varies with applied loads at different loading stages and is influenced by the structural stiffness and strength. In Takeda's paper, specific hysteresis rules were defined within this model to describe the situation where stiffness decreases as the load increases. Within the model, unloading in the elastic stage of the structure is similar to the dynamic model, following an elastic path.

During reloading, the curve moves along the backbone curve in an opposite direction to the previous load, aiming at a point representing the maximum deformation from prior loading cycles. This approach leads to a gradual reduction in energy dissipation as deformation increases. This simple model requires no additional parameters, and is more appropriate for reinforced concrete than for metals.

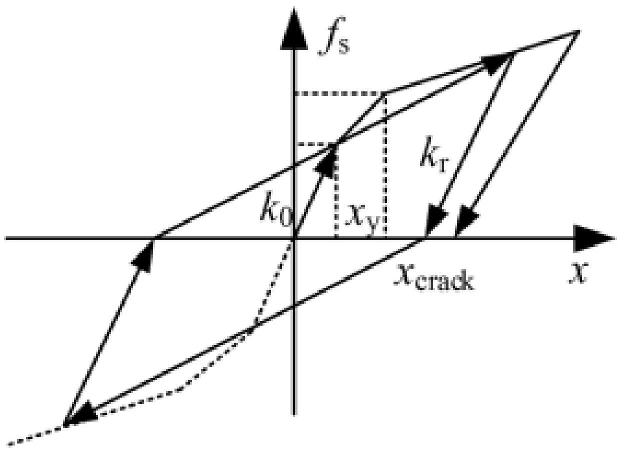


Figure 10 The hysteretic curves of Takeda model

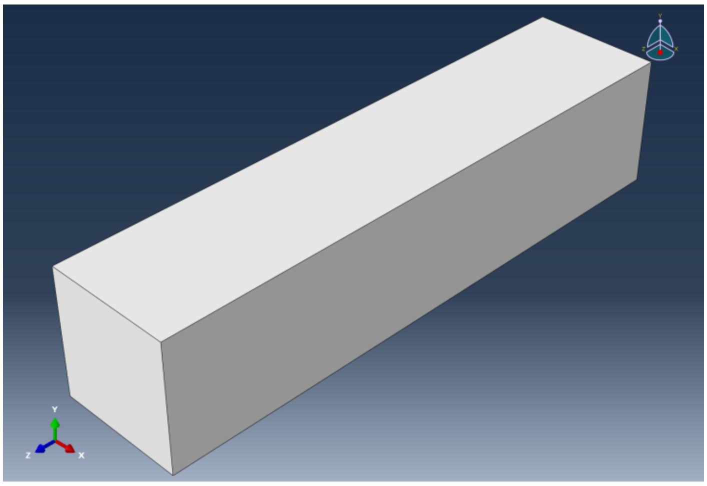
IV. Description of model

- **ABAQUS**

1. Modeling

Part Module :

Draw the concrete column, vertical reinforcement, inner stirrup and outer stirrup in the Part module as shown below.



Concrete column

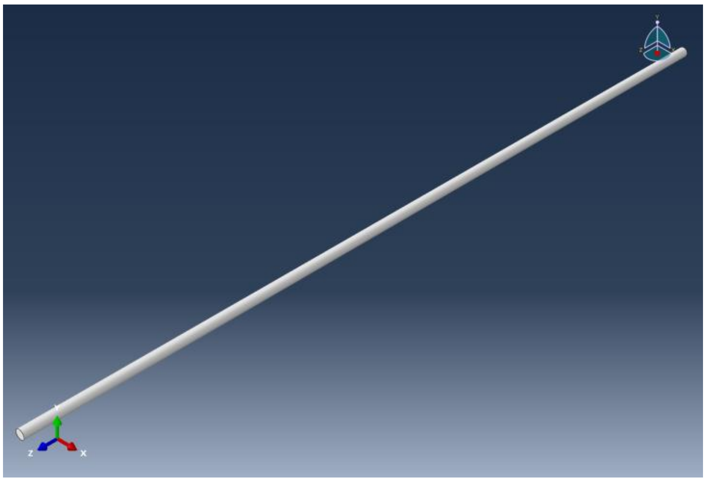


Figure 12 Vertical Rebar

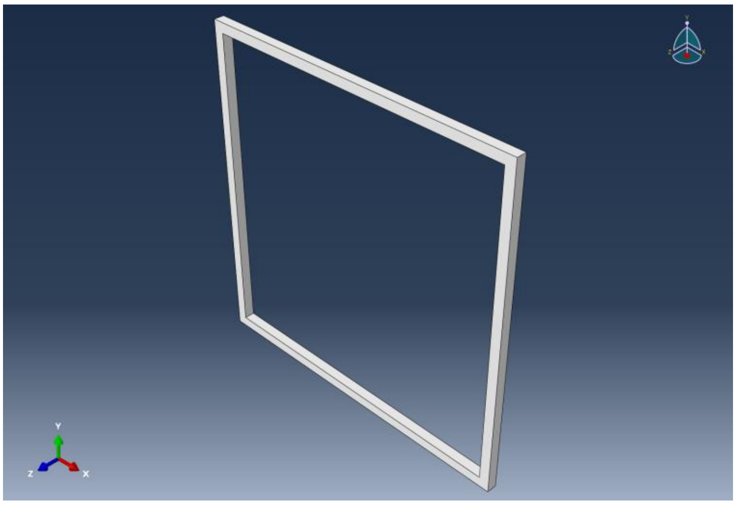


Figure 13 Inner stirrup

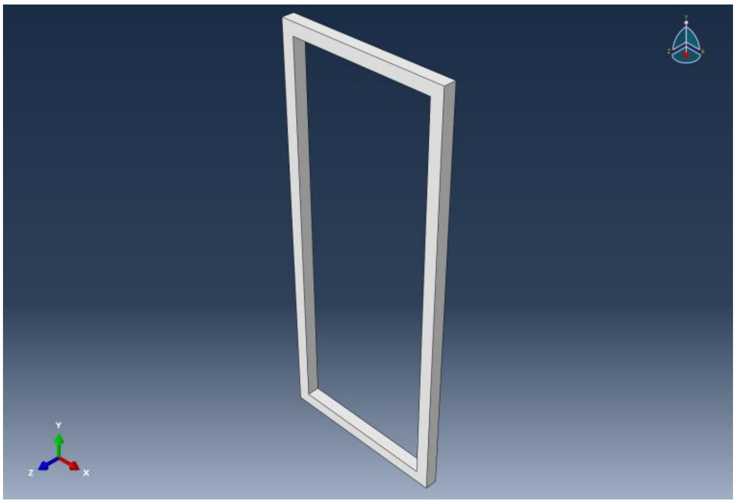


Figure 14 Outer stirrup

Property Module:

The material properties of the concrete include density 2.4E-9(ton/mm^3), Young's modulus 28321.156(MPa), Poisson's ratio 0.25, and the plastic portion of the concrete is adopted to start from 36.31(MPa), and gradually increase in a non-linear manner, as shown in the following figure.

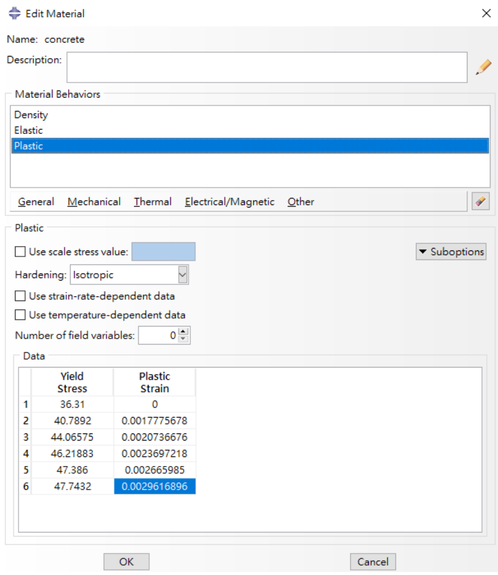
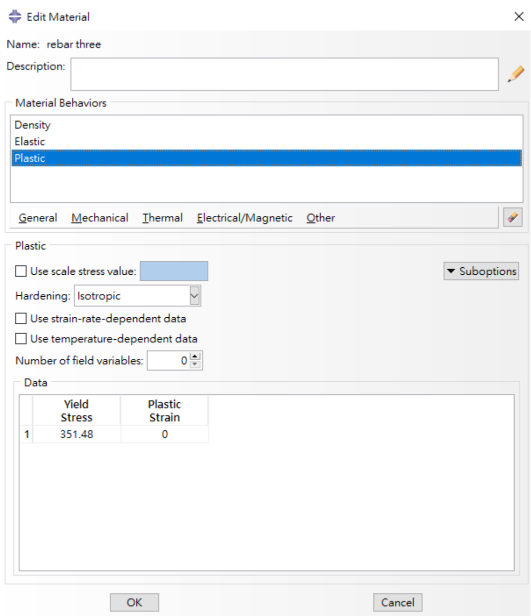


Figure 15 Concrete Plasticity Parameters

Rebar density using 7.9E-9 (ton/mm^3), Young's modulus 210,000 (MPa), Poisson's ratio 0.3, and yield stress is 484.7121 (MPa) and 351.48 (MPa), respectively, are shown below.



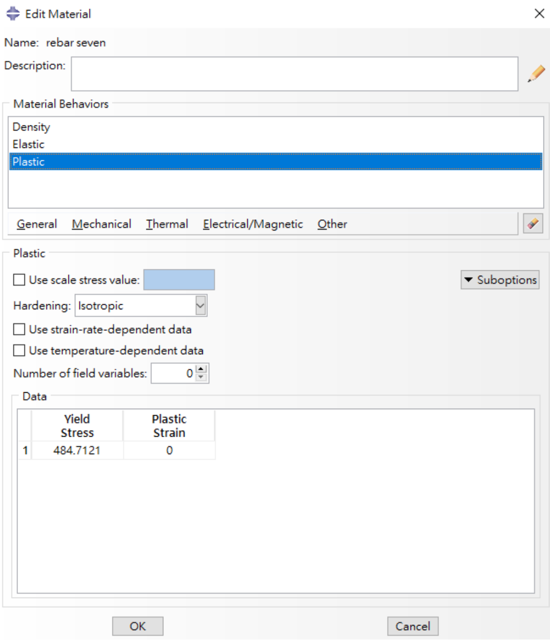


Figure 16、17 Plasticity Parameters of Reinforcing Steel

Assembly Module:

Assemble the previously built model into a reinforced concrete column in the assembly module, as shown in the figure.

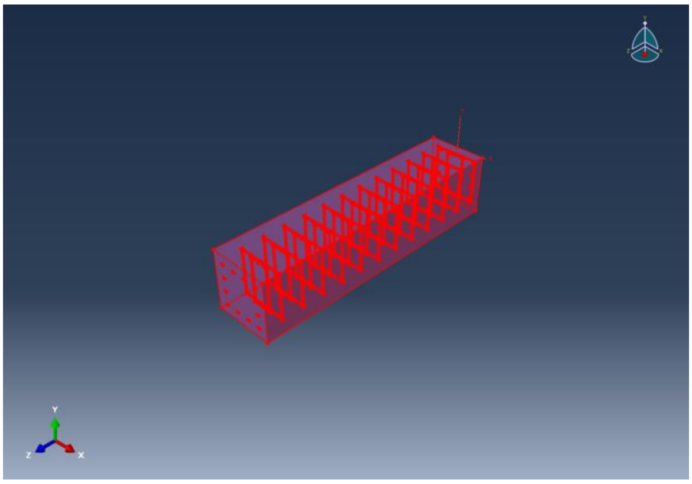


Figure 18 Assembled RC Column

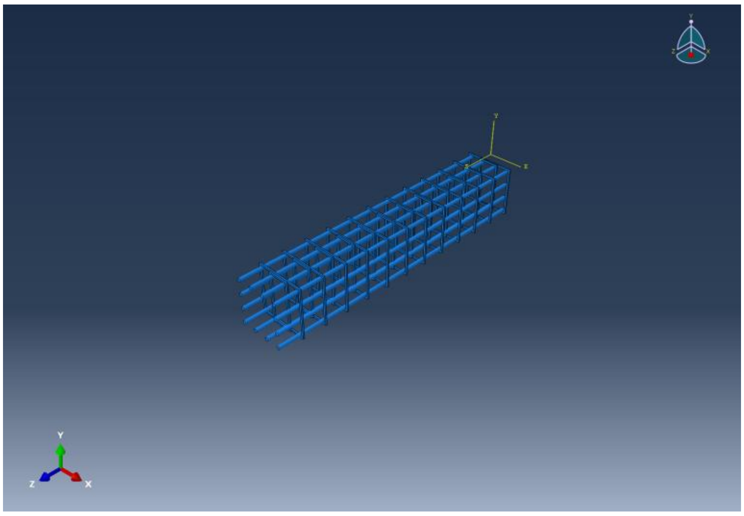


Figure 19 Inner Reinforcement

Step Module:

The total length of the lateral thrust experiment was 3168 seconds, and the initial and minimum incrementations were set to 0.01 seconds, as shown in the following figure.

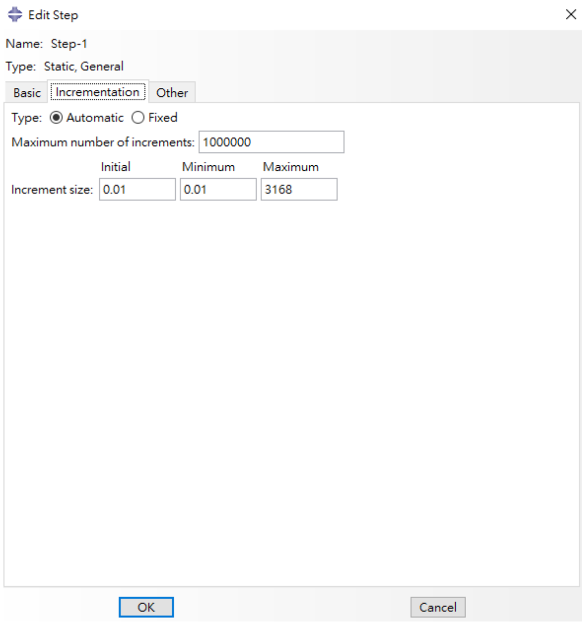


Figure 20 Specify the time step

Interaction Module:

Set the interaction between the concrete columns and the reinforcement bars, set the concrete as the embedded surface and the reinforcement bars as the host surface to avoid slipping during the process and to enhance the convergence, as shown in the following figure.

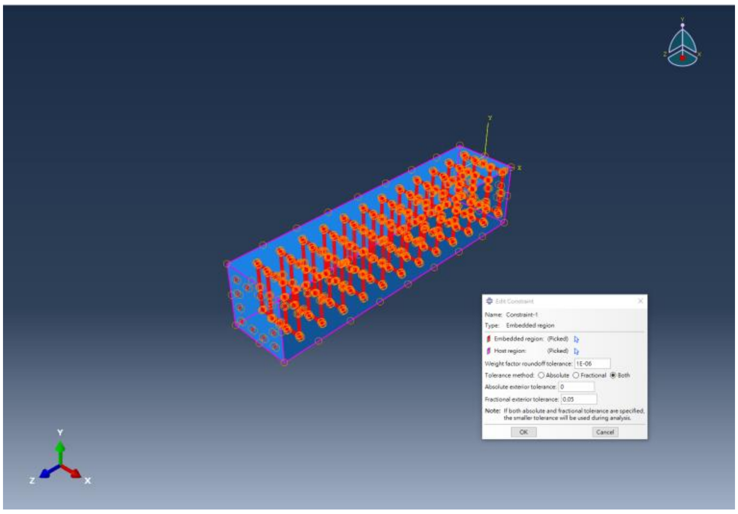


Figure 21 Interaction

Load Module:

Set the fixed end and the top push displacement in the Load module as shown below.

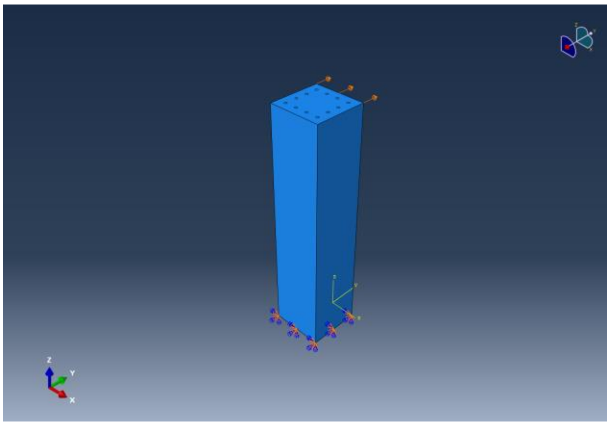


Figure 22 Fixed end & side push displacement direction

Mesh Module:

Cut the specimen into elements in the mesh module and simulate it using C3D8R elements as shown below.

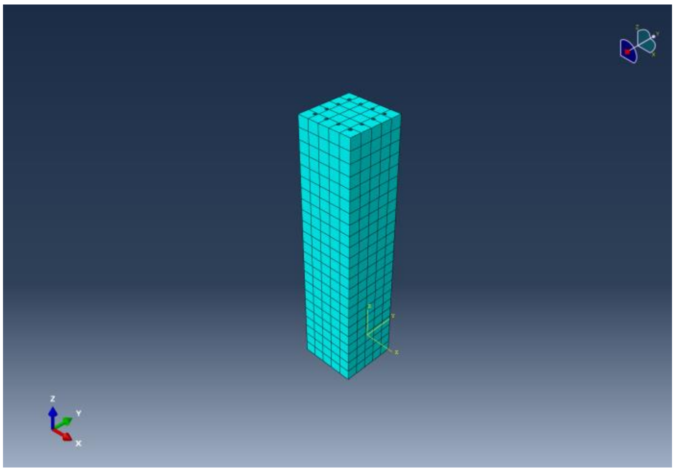


Figure 23 Concrete column mesh

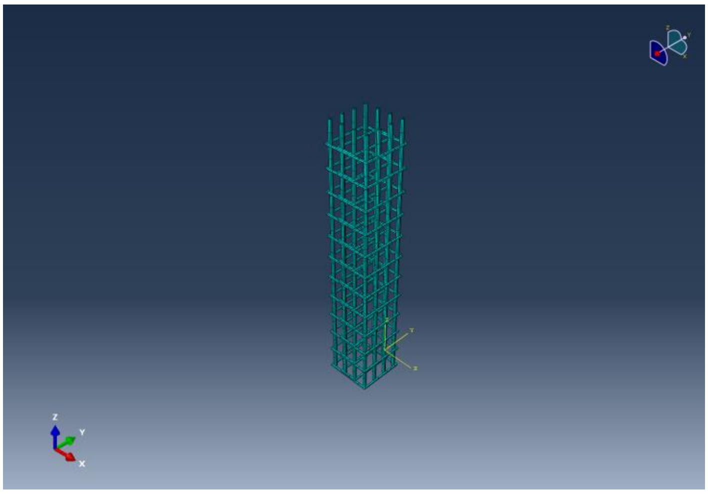


Figure 24 Reinforce mesh

Job Module:

The analysis was performed in the job module, and there were a total of 278 analysis steps in this analysis.

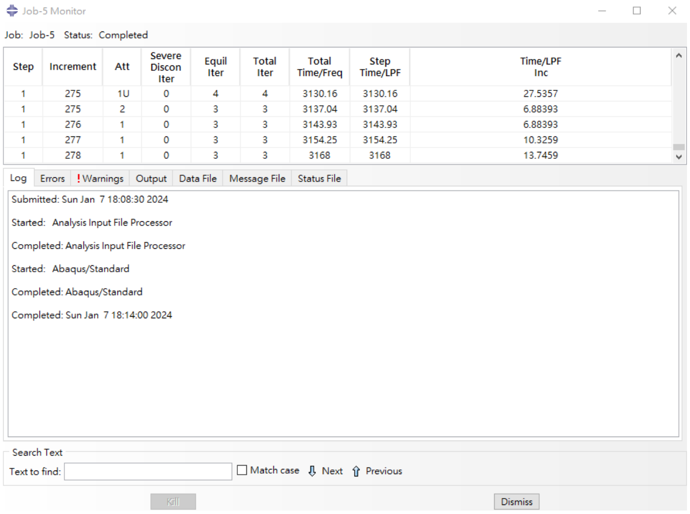


Figure 25 Job

2. Discussion of results

Bottom stress:

From the stress results (Fig. 26), it can be found that the stress is concentrated at the bottom two sides, and from the stress time plot (Fig. 27), it can be found that the stress keeps coming back and forth between positive and negative, in which the maximum value of the stress is 56.0588 (MPa) (Fig. 28), and the bottom node N565 is used here.

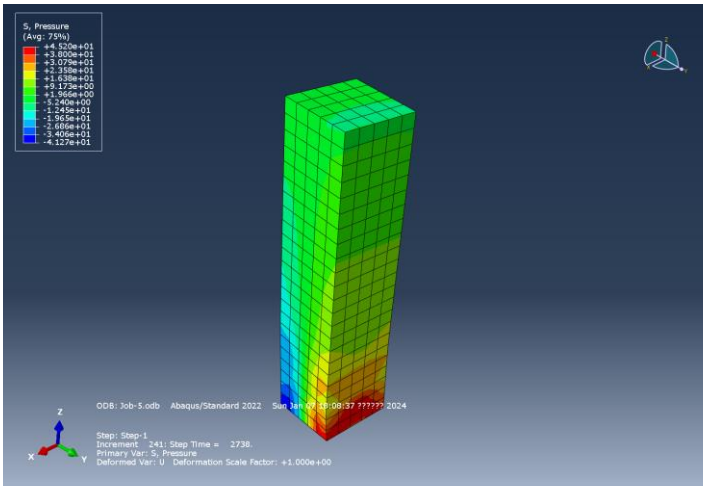


Figure 26 Stress result

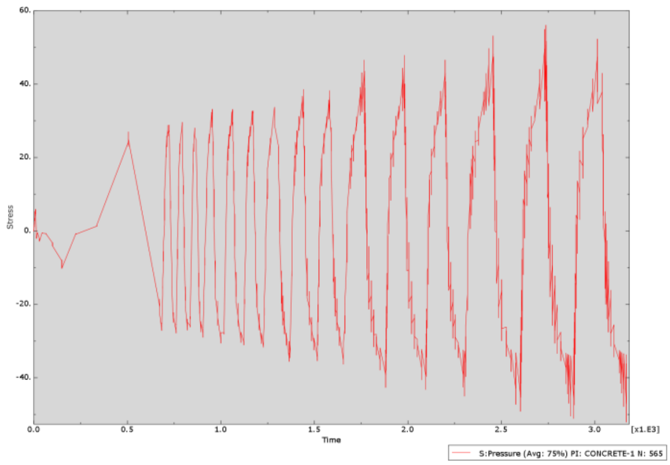


Figure 27 Node N565 Stress Historical Plot

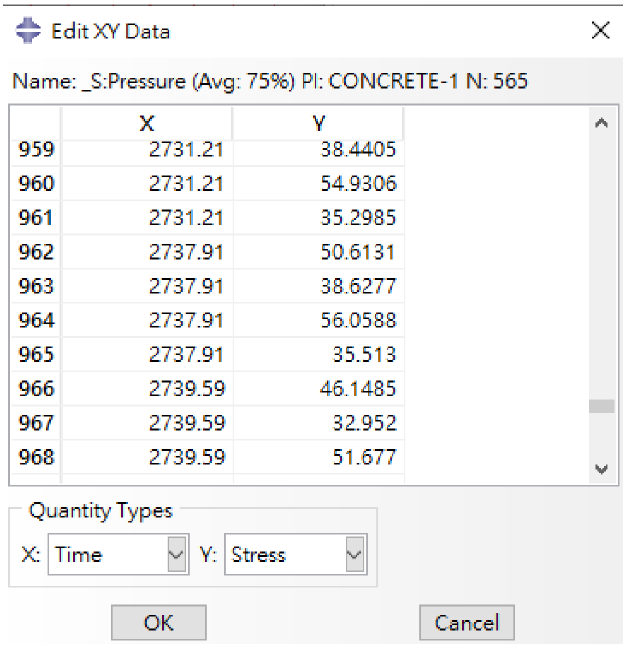


Figure 28 Node N565 Stress History

Rebar Stress Discussion:

The maximum von Mises stress criterion is based on the von Mises-Hencky theory, which is also known as the shear energy theory or the maximum distortion energy theory. The principal stresses σ1, σ2, σ3 can be expressed as von Mises stresses:

σ vonMises= {[(σ1 - σ2)2 + (σ2 - σ3)2 + (σ1 - σ3)2] / 2}1/2

This theory states that when the von Mises stress equals the stress limit, the ductile material begins to yield at a certain point. In most cases, the yield strength can be used as a stress limit [9].

From the rebar stress result, it can be seen that the bar enters the yield stage at the bottom at about 500 s (Fig. 29), and the upper part of the bar does not enter the buckling stage even at the end of the simulation (Fig. 30).

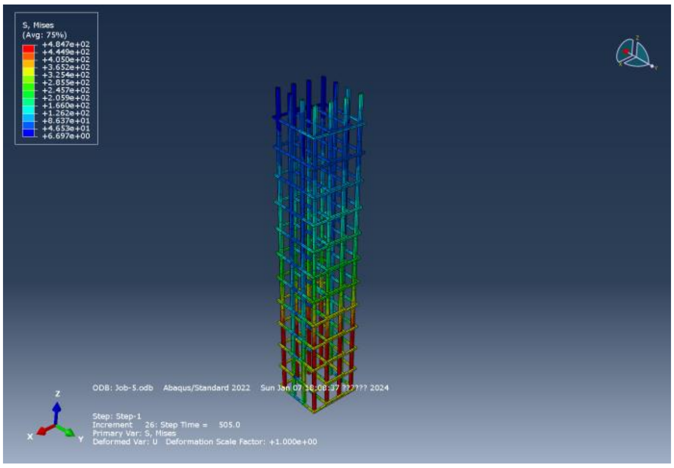


Figure 29 Stress result of steel bar (505 sec)

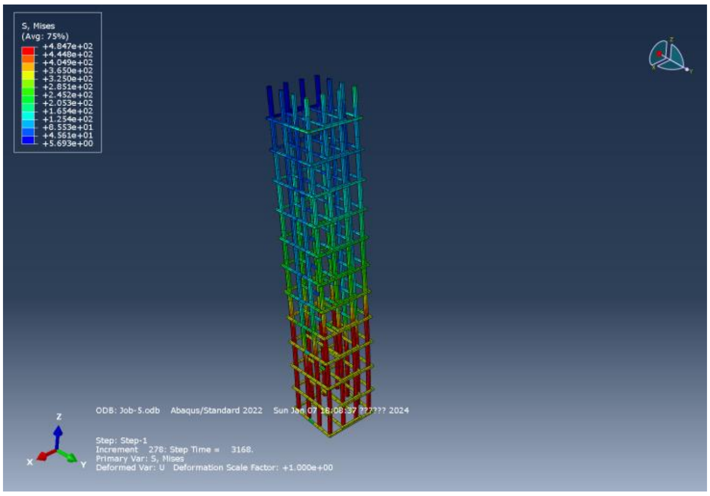


Figure 30 End of simulation (3168 seconds)

Top part shifted:

This section discusses the top node N433, which can be seen in the displacement time plot (Fig. 31) as it oscillates with the side thrust.

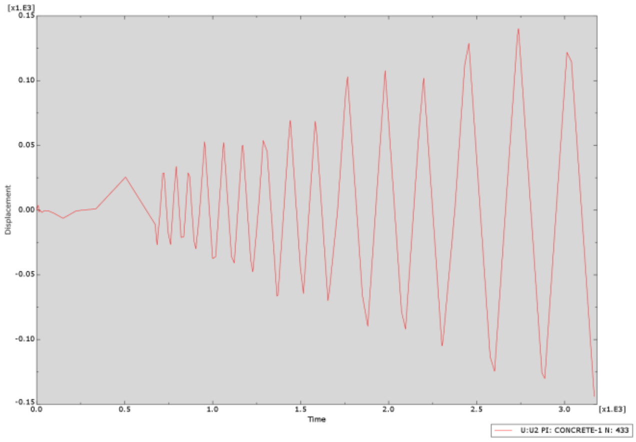


Figure 31 Node N433 Displacement Historical Chart

Hysteresis loop:

The hysteresis loop (Fig.33) can be plotted using the displacement histories of the top node N433 and the shear histories of the bottom node N571 (Fig.32).

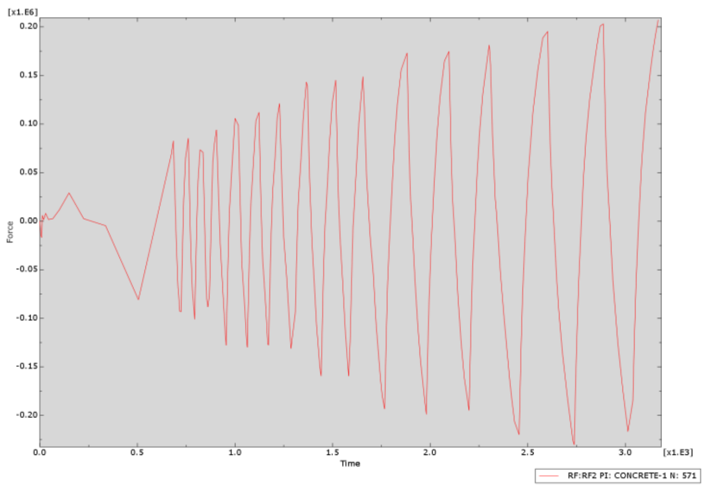


Figure 32 Node N571 shear time plot

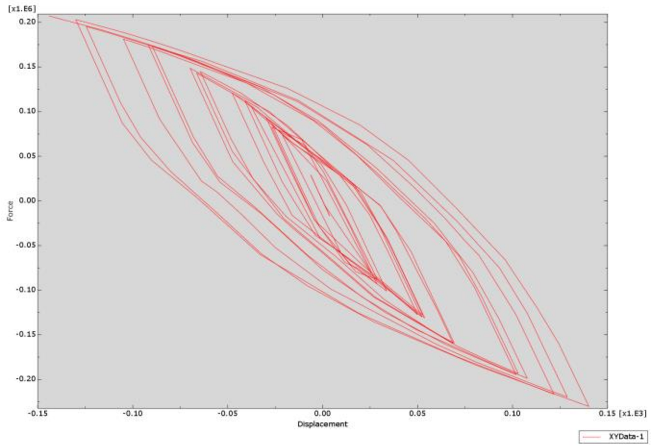


Figure 33 Hysteresis loop

- **SAP2000**

1. Introduction

SAP2000 is a powerful structural analysis and design software developed by Computers and Structures. It is used to design almost any kind of civil engineering structure. The software is equipped with a range of features that allow for the modeling and analysis of structures, with an emphasis on buildings.

2. Modeling

We used SAP2000 to simulate the RC columns.

It is divided into the following steps.

1. Define the material parameters

2. Define the cross section

3. Define cyclic load

4. Define the plastic hinge and simulate the structural response.

5. Export hysteresis loops

a. Define the material parameters

Define concrete parameters by using the concrete parameters provided in the handout.

Specimen number	Sampling Number	Specimen size(cm)		Age (days)	Compressive Strength(MPa)	Average Compressive Strength(MPa)	Axial pressure(0.1AgF'c)(KN
		Diameter	Height				
FF-15S-0.1	1	10	20	428	33.93	36.31	581
	2				35.74		
	3				39.25		

Table 5 Concrete properties

Material	Application	Yield strength (fy)	Tensile strength (fu)
		MPa	MPa
#3(SD280W)	Hoops	351.48	462.356
#7(SD420W)	Vertical (longitudinal) reinf.	484.7121	695.02

Table 6 Rebar properties

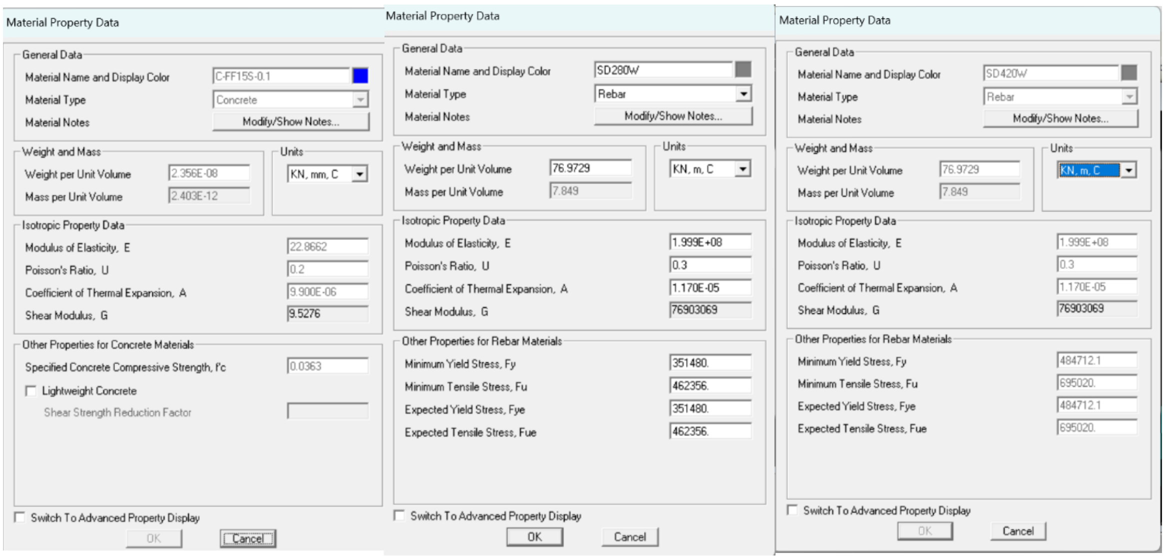


Figure 34 Definition of concrete and reinforcement material parameters

In particular, the Young's coefficient E for concrete is calculated here using the new specification $Ec=12000\times\sqrt{(fc)}$.

b. Define the cross section

Define the cross section based on the data given in the handout.

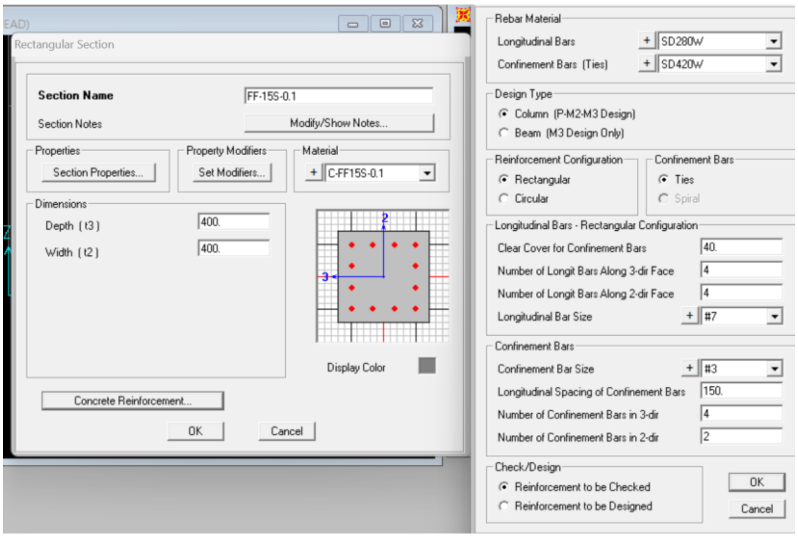


Figure 35 Section definition

c. Define cyclic load

To apply a Cyclic load using the Time History Function in SAP2000, first manually enter the displacement and time as required by the question.

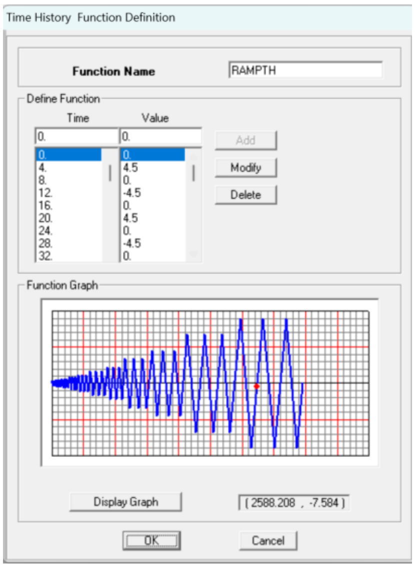


Figure 36 Time History Function

Then apply it to the model.

It is important to note that if the load is applied directly to a point, it will not be possible to apply the displacement to the node in the expected way. Restraints need to be added to the applied node first, and they need to be added to the dimension where the displacement is applied. In this way, the displacement can be applied to the node smoothly.

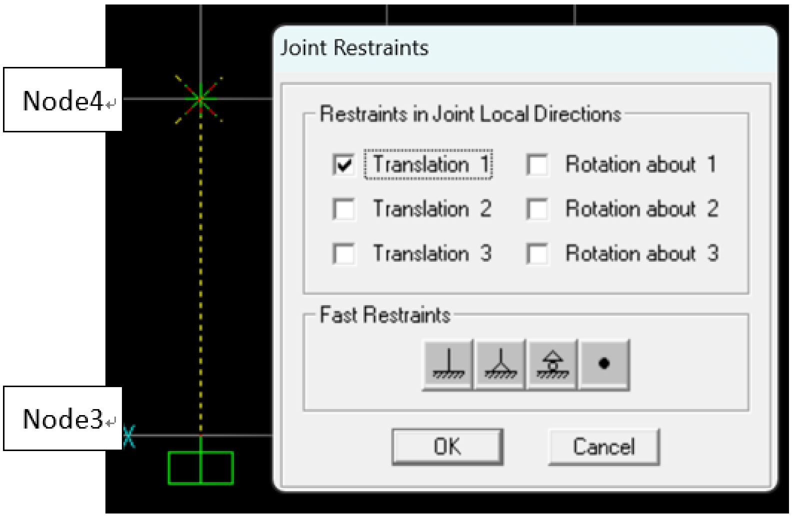


Figure 37 Applying Restraints to the Node

d. Define the plastic hinge and simulate the structural response.

Here we follow the flowchart to analyze the simulation.

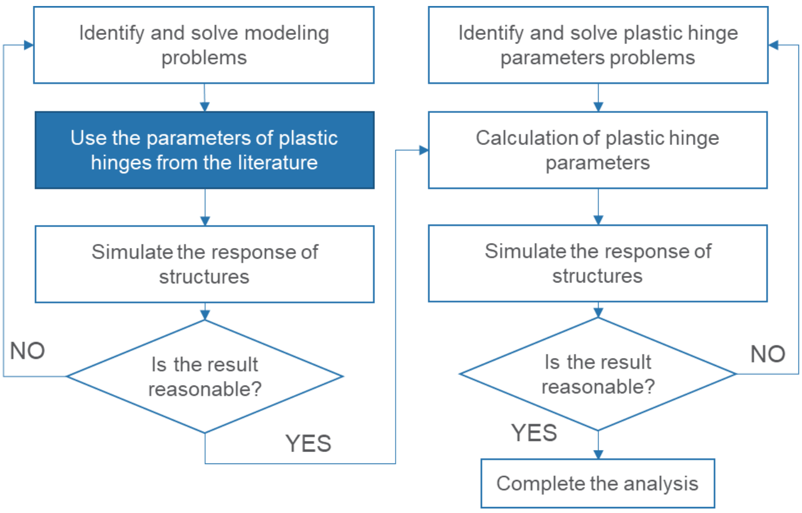


Figure 38 Flow chart

We first change the definition of M3 plastic hinge in SAP2000 as suggested in the literature[10]. Originally, point B in SAP2000 was the point of entry into the plastic zone, however, in the literature[10], it is mentioned that the stiffness of the RC column also changes after concrete cracking.

Therefore, point B is defined as the cracking bending moment, Mcr, instead of the yielding moment, My, and the yielding angle, θy, which are defined in SAP2000.

After passing through point B, the stiffness of the RC columns changes according to the BC diagonal line, and point C is defined as the yielding moment and the yielding angle, as suggested in the literature[10].

After passing point C, the stiffness varies along the straight line CD. According to the literature[10], point D is the starting point of the post-yielding segment, and point E is the Mu,UL and θu,UL in the limit state.

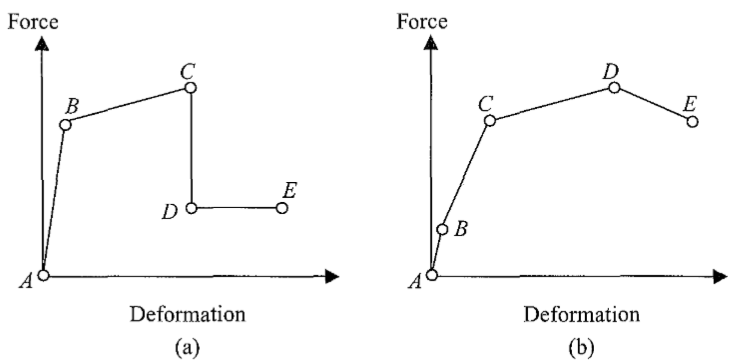


Figure 39

(a)Default Plastic Hinges (b) the plastic hinge curves proposed in this paper

Before calculating the plastic hinges for this RC column, use the plastic hinge parameters from the literature[10] to test if the model is OK.

After confirming that the model is OK, the plastic hinge parameters for this RC column are calculated.

The simulation is divided into two parts to identify the source of the problem, to see whether the problem is from model or plastic hinge parameters.

Here we use the backbone curve of JSCE-4 to define the parameters of plastic hinges.

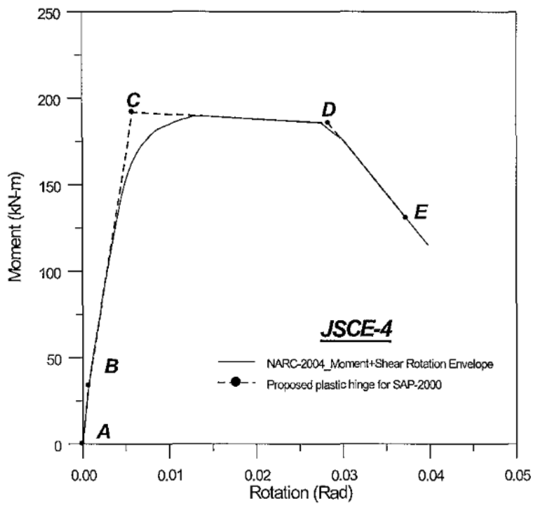


Figure 40

JSCE-4 Column Bending Moment and Angle Analysis and SAP2000 Input Method Definition

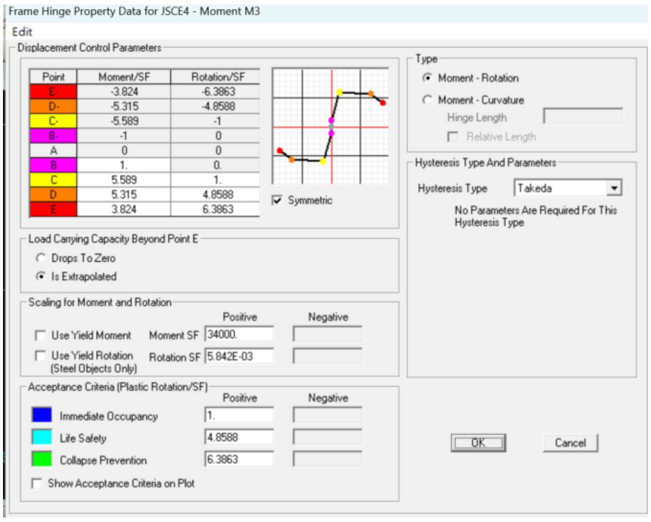


Figure 41 Plastic Hinge Definition

Then the structural analysis was carried out and the bending moments and angles of the plastic hinge and the backbone curve were plotted and it was observed that the stiffness of the RC column did not decay according to the dorsal curvature. Therefore, there is a problem with the analysis.

However, the problem cannot be solved by changing the integration method, changing the time step size, or changing the input parameters of the plastic hinge. At this point, the deadline is approaching. Therefore, the flow chart stops at the first part and does not go into the area of the actual calculation of the plastic hinge.

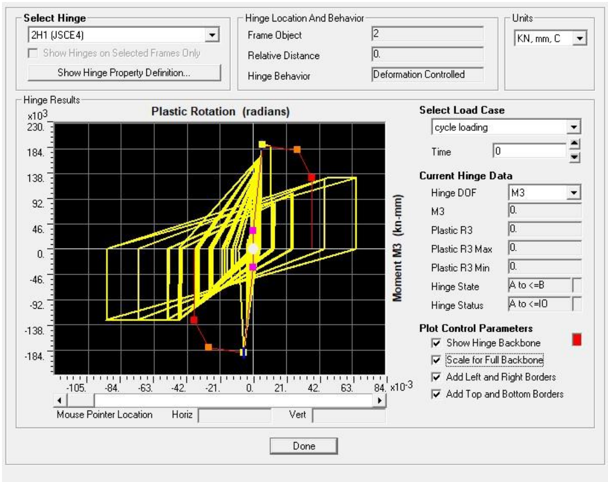


Figure 42 The bending moments and angles of the plastic hinge

e. Export hysteresis loops

The Hysteresis loop is plotted using the Show plot function of SAP2000.

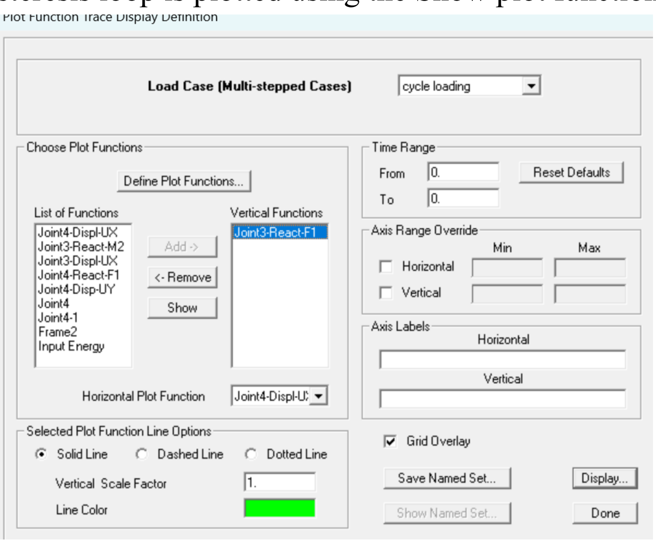


Figure 43 Show plot function

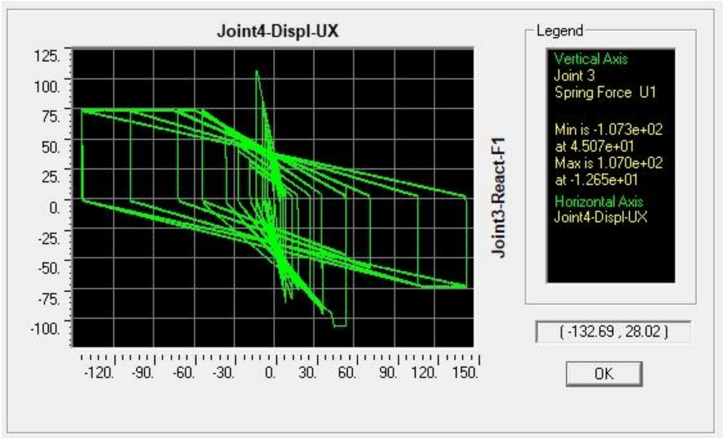


Figure 44 Hysteresis loop

It can be seen that the hysteresis loop has a problem with the deformation and there is a big jump in the middle. Therefore, the result of the analysis is problematic and needs to be corrected with additional time.

V. Results and discussion

● ABAQUS

Due to the uncertainty of the given plasticity parameters of concrete, it is more difficult to use the more accurate Concrete Damaged Plasticity model for simulating concrete in ABAQUS. In this simulation, the general plastic behavior is used and the corresponding stress strain variations are given one by one (Fig. 34), and therefore, there will be a discrepancy between the simulation and the actual results.

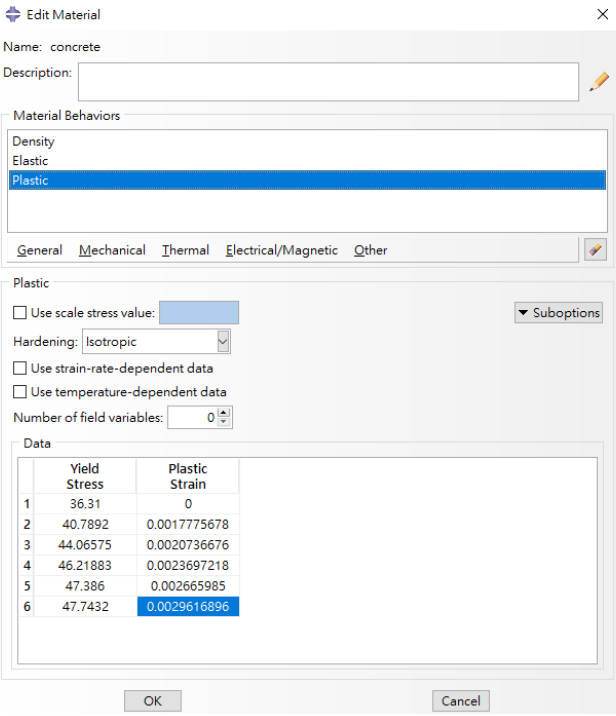


Figure 45 Plasticity parameters of concrete

- **SAP2000**

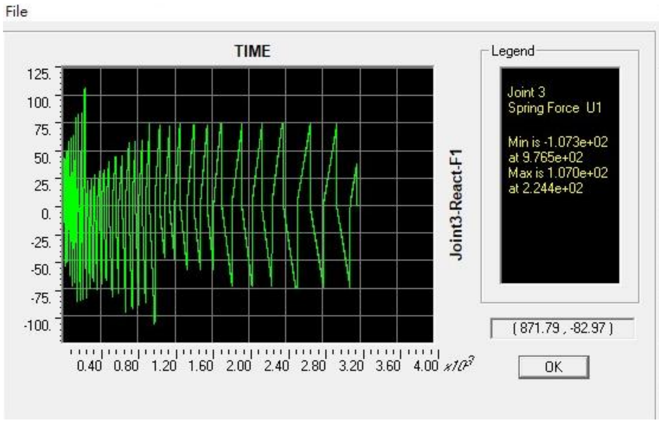


Figure 46 The base-shear force at the fixed-end

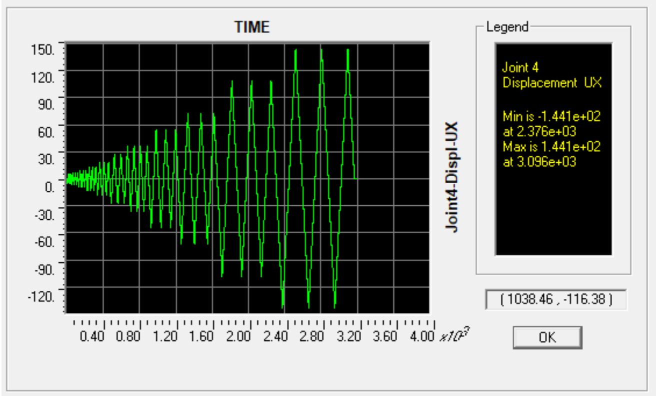


Figure 47 The displacement at the free-end

There were many problems with the plastic hinge setup this time, the first being that the SAP2000 version in the literature was different and the parameters could not be used directly. Even if the parameters were entered successfully, there were still problems with the model. Therefore, the plastic hinge parameters for this RC column were not actually calculated this time, and therefore the simulation results were only borrowed from the experimental results of other people's research, and strictly speaking, the simulation was not to the point.

In the future, it is necessary to have a deeper understanding of the plastic hinge parameters and to find out what the problem of this model is, so that the simulation experiments can be done more smoothly in the future.

VI. Conclusions

In the early days of earthquake engineering, conventional models such as linear elastic or simplified plasticity models were commonly employed. But these models didn't quite capture how structures really respond during non-linear moments or weren't accurate enough under seismic conditions.

The emergence of the Takeda model aims to address these limitations. It provides a more accurate simulation and prediction of the behavior of structures subjected to cyclic seismic loading and it has the following advantages, suitable for reinforced concrete, requires fewer parameters, and has low energy consumption.

Consequently, the Takeda model is now being utilized to better represent the structural response under seismic conditions.[8]

We want to compare the advantages and disadvantages of SAP2000 and ABAQUS, please look this table.

	SAP2000[10]	ABAQUS[11]
ADVANTAGES	1. Easy to use and learn 2. In some case, SAP2000's analysis speed might be faster than ABAQUS	1. The most reliable analysis results 2. The strongest problem-solving capability
DISADVANTAGES	1. It might not be as useful as ABAQUS in handling complex problems 2. Hard to solve the Nonlinear and nonstructural dynamics	1. When dealing with complex problems, it requires more time 2. Maybe hard to use and learn

Table 7 Comparison betweenSAP2000 and ABAQUS

So, we believe that ABAQUS is a better choice than SAP2000 for analyzing this project.

VII. REFERENCES

[1]台灣地震：
http://gis.geo.ncu.edu.tw/gis/eq/tweqharz.htm

[2]中央氣象署地震測報中心：
https://scweb.cwa.gov.tw/zh-tw/earthquake/data

[3]Perform pushover analysis ：
How to Perform Pushover Analysis? (midasstructure.com)

[4]Taiwan seismic zone
https://earthjay.com/?p=10450

[5] Lignos, D. G., & Krawinkler, H. (2013). Development and Utilization of Structural Component Databases for Performance-Based Earthquake Engineering. Journal of Structural Engineering, 139(8), 1382-1394. https://doi.org/10.1061/(asce)st.1943-541x.0000646

[6] I.D. Mayergoyz, Introduction, in: I.D. Mayergoyz (Eds.), Mathematical Models of Hysteresis and Their Applications, Elsevier Science, New York, 2003, pp. xiii xxv

[7] Tianyu Wang, Mohammad Noori, Wael A. Altabey, Zhishen Wu, Ramin Ghiasi, Sin-Chi Kuok, Ahmed Silik, Nabeel S.D. Farhan, Vasilis Sarhosis, Ehsan Noroozinejad Farsangi,
From model-driven to data-driven: A review of hysteresis modeling in structural and mechanical systems,
Mechanical Systems and Signal Processing,
Volume 204, 2023, 110785, ISSN 0888-3270

[8] T. Takeda, M.A. Sozen, N.N. Nielsen, Reinforced concrete response to simulated earthquakes, J. Struct Div. 12 (1970) 2557–2573

[9] Maximum von Mises stress
https://help.solidworks.com/2019/chinese/SolidWorks/cworks/r_maximum_von_mises_stress_criterion.htm

[10]宋裕祺(Yu-Chi Sung), 劉光晏(Kuang-Yen Liu), 蘇進國(Chin-Kuo Su), 蔡益超(I-Chau Tsai)及張國鎮(Kuo-Chen Chang), 作者, 「鋼筋混凝土柱側推分析之研究與探討」, 建築學報, 期 46, 頁 47–65, 6 月 2004.

[11]交大論文--內縮型式框筒結構側向勁度加強分析(SAP2000 相關說明)
https://ir.nctu.edu.tw/bitstream/11536/72914/1/122301.pdf

[12]交大論文--高壓氣瓶的高溫旋壓縮口之有限元素分析研究(ABAQUS)
https://ir.nctu.edu.tw/bitstream/11536/47107/1/450101.pdf