

Jimma University

School of Graduate Studies

Jimma Institute of Technology

Faculty of Civil and Environmental Engineering

Structural Engineering Stream

**Investigation of Reinforced Concrete Column with Concealed Pipe for
Plumbing and Electric Installation using Finite Element Analysis**

A Research is Submitted to the School of Graduate Studies of Jimma University in Partial Fulfillment of the Requirements for the Degree of Master of Science in Civil Engineering (Structural Engineering)

By: Gebi Bula

Jimma, Ethiopia

February, 2018

Jimma University
School of Graduate Studies
Jimma Institute of Technology
Faculty of Civil and Environmental Engineering
Structural Engineering Stream

**Investigation of Reinforced Concrete Column with Concealed Pipe for
Plumbing and Electric Installation using Finite Element Analysis**

A Research is Submitted to the School of Graduate Studies of Jimma University in Partial fulfillment of the requirements for the Degree of Master of Science in Civil Engineering (Structural Engineering)

By: Gebi Bula

Advisor: Dr. Temsegen Wondimu

Co-Advisor: Engr. Elmer, C. Agon

February, 2018
Jimma, Ethiopia

DECLARATION

I declare that the thesis comprises my own work. In compliance with internationally accepted practices, I have acknowledged and refereed all materials used in this work. I understand that non-adherence to the principles of academic honesty and integrity, misrepresentation/fabrication of any idea/data/fact/source will constitute sufficient ground for disciplinary action by the University and can also evoke penal action from the sources which have not been properly cited or acknowledged.

Name of the student _____ Signature _____

Date of submission: _____

Place: Jimma

This thesis has been submitted for examination with my approval as a university advisor.

Advisor Name: _____

Advisor's Signature: _____

Co-Advisor Name: _____

Co-Advisor's Signature: _____

Jimma University
School of Graduate Studies
Jimma Institute of Technology
Faculty of Civil and Environmental Engineering
Structural Engineering Stream

THESIS APPROVAL SHEET

Student:

Name	Signature	Date
------	-----------	------

The following graduate faculty members certify that this student has successfully presented the necessary written final thesis and oral presentation for partial fulfillment of the thesis requirements for the Degree of Master of Science in Structural Engineering.

Approved by:

Advisor:

Name	Signature	Date
------	-----------	------

Co-Advisor:

Name	Signature	Date
------	-----------	------

External Examiner:

Name	Signature	Date
------	-----------	------

Internal Examiner:

Name	Signature	Date
------	-----------	------

Chair Holder:

Name	Signature	Date
------	-----------	------

Faculty Dean:

Name	Signature	Date
------	-----------	------

To my beloved family

Love you forever

ACKNOWLEDGEMENT

I have a host of people to grace of their involvement in this major work. The order I mentioned below does not necessarily represent the amount of work they did.

I am sincerely grateful to my advisor **Dr.Temesgen Wondimu**, for his continuous support, suggestions and immeasurable contribution to my thesis. He always provides me guidance and feedback in this thesis. I am also very thankful to my co-advisor **Engr.Elmer, C.Agon** for his advice, guidance and motivation. Without their help, I would not have completed my thesis.

I would also like to express my deepest gratitude to Jimma university particularly, Jimma institute of technology department of civil engineering for providing the necessary facilities and the staff in school of civil engineering for their help in the completion of this thesis. Besides, I would like to appreciate Ethiopian roads Authority/ERA that gave me the full scholarship of post graduate education .

ABSTRACT

It is known that the ultimate strength and carrying capacity of columns depends on pattern of loading, cross-sectional shape of columns, diameter of reinforcement, and cross-sectional area of the columns. Hollow-core systems are cost effective, allow construction to remain on budget and on schedule, require less maintenance, and create less environmental impact than conventional building materials. Hollow core columns having circular holes shall be better performance as it creates smooth configuration to cross-section of columns.

The behavior of reinforced concrete columns with holes under axial load is not understood, and researches in the subject are needed to help designers and structural expert officials. Holes drilled out to install additional services or equipment, such as ducts through columns, beams, or walls, can lead to loss of strength and possible structural failure. Until now little work has been done on effect of holes in columns and, hence, this study aims to examine the amount of capacity and strength lost due to the presence of holes in columns. Nonlinear finite element analysis on 14-columns (7-squares; 7-rectangulars) specimens is conducted by using ABAQUS software. The nonlinear finite element analysis program ABAQUS is used owing to its capabilities to predict either the response of reinforced concrete columns in the post-elastic range or in the ultimate strength range of reinforced concrete columns. An extensive set of parameters is investigated including; dimensions of the holes with diameter of (20mm, 50mm), and different relative position of the holes in the columns. The shapes of holes are circular for both columns.

The carrying capacity and ultimate strength of reinforced concrete short-braced columns with different position of longitudinal openings are discussed in the lesson of study. The effect of the electric wiring through the columns with help of ducts are adequately assessed under the effects of axial loading; because the structural response of the members with a hole under pure axial loading may be significantly different from that of solid column due to existence of avoid section.

In conclusion, the analysis result shows significant reduction in load carrying capacities for both square and rectangular columns with duct of wiring compare to their respective solid columns due to reduction of their effective cross-sectional area. The study also reveals that, square columns are sensitive in reduction of capacity than rectangular columns of the same size of the holes at the same position in the columns cross-section. For instance, the peak reduction for rectangular column is 5.55 % (case-d) and 7.75 % (case-d) for square columns compare to their respective control specimens.

Keywords: Columns capacity, ABAQUS, Nonlinear Finite Element Analysis, Openings, RC Columns, Stress Analysis, Ultimate strength.

TABLE OF CONTENTS

DECLARATION	iivi
DEDICATION	viii
ACKNOWLEDGEMENT	vi
ABSTRACT.....	vi
TABLE OF CONTENTS.....	viii
LIST OF TABLES	vi
LIST OF FIGURES	vi
LIST OF ABBREVIATIONS	xivi
LIST OF SYMBOLS	xvi
CHAPTER ONE	1
INTRODUCTION	1
1.1. Background	1
1.2. Problem Statement	2
1.3. Research questions	3
1.4. Objective of the Study	3
1.4.1 General Objective.....	3
1.4.2 Specific objectives.....	3
1.5. Significance of the study	3
1.6. Scopes of the Study	4
CHAPTER TWO	5
LITERATURE REVIEW	5
2.1. Theoretical review	5
2.2. Concrete	7
2.2.1 Concrete Damaged Plasticity	9
2.2.2 Elastic Modulus of Concrete	10
2.3. Steel Reinforcement	11
2.4. Reinforced Concrete.....	12
2.5. Reinforced Concrete Column.....	13
2.5.1 Types of Column and Failure Modes	13
2.5.2 Capacity of Short columns subjected to axial loads only.....	15
2.5.3 Buckling of columns	18
2.6. Finite element method	20
2.6.1 Brief history.....	20

2.6.2 Definition of FEM	20
2.6.3 Advantages of FEM	21
2.6.4 Methods of Formulating Finite Element Problems	21
2.6.5 Types of Finite Element	22
2.6.6 Basic Steps of Finite Element Analysis	22
2.6.7 Fundamental Requirements	24
2.6.8 Scopes of Types of Column Meshing	24
2.6.9 Verification of Results.....	24
2.7. ABAQUS	25
2.7.1 Characteristic of ABAQUS Software	25
2.7.2 Analysis Procedure According to ABAQUS Software.....	25
2.8. Non-linear Analysis.....	25
2.9. Critique of the existing literature relevant to the study	29
CHAPTER THREE.....	35
MATERIALS AND RESEARCH METHODOLOGY	35
3.1. Introduction	35
3.2. Finite Element Modeling.....	35
3.2.1 Geometry	35
3.2.2 Finite Element method (Discretization)	39
3.2.3 Abaqus modeling and analysis procedures	40
3.2.4 Material Properties	42
3.2.5 Boundary Condition	51
3.2.6 Loading.....	52
3.2.7 Meshing Design (technique)	53
3.3. Non-linear analysis.....	58
3.4. Classification of the models under study	63
3.5. Flowchart of the Finite Element Analysis.....	64
3.6. Experimental validation	65
3.6.1 General description on the genesis of data used in experiment	65
3.6.2 Comparison of the Results	68
CHAPTER FOUR.....	71
ANALYSIS, RESULT AND DISCUSSIONS	71
4.1. Introduction	71
4.2. The Results of Finite Element Analysis.....	71

4.3. Vertical Stress-Strain Curves of the Models Based on FEA Results	78
4.4. Reduction in Load Carrying Capacity of Reinforced Concrete Columns Concealing Hole	82
4.5. Discussions	88
4.5.1 Discussion of the results on rectangular columns	88
4.5.2 Discussion of the results on square columns	88
CHAPTER FIVE	89
CONCLUSION AND RECOMMENDATION	89
5.1. Conclusions	89
5.2. Recommendation	90
REFERENCES	91
Appendix A	93
Appendix B	95
Appendix C	97

LIST OF TABLES

3.1. The cases under study and dimension of the column models	38
3.2. Material data input (concrete properties)	45
3.3. Steel properties for the longitudinal reinforcements	49
3.4. Steel properties for Transverse reinforcements.....	50
3.5. The ratios and types of models.....	63
3.6. Concrete properties of validity parameters	65
3.7. Steel properties of validity parameters	66
3.8. Validity of the finite element result in terms of strain –stress.....	69
3.9. Validity of the finite element result in terms of load carrying capacity.....	70
4.1. Maximum vertical stress and strain of the columns under study	72
4.2. The percentages of reduction in load carrying capacity of rectangular columns with hole in 3D	82
4.3. The percentages of reduction in load carrying capacity of square columns	83

LIST OF FIGURES

2.1. Hole (rain water pipe) is positioned inside columns	5
2.2. Typical columns with hole at the center.....	6
2.3. Response of concrete to uniaxial loading in tension (a) and compression.....	8
2.4. Static modulus of concrete	10
2.5. Typical stress-strain curves for reinforcing steel	11
2.6. Simplified stress-strain curve for reinforcing steel	12
2.7. Effects on slender column	14
2.8. Columns failure modes.....	15
2.9. Axial load-strain response of steel and concrete in an axially loaded-column	16
2.10. Buckling of column	19
2.11. Bending moment in the column	19
2.12. Flow chart for the basic steps of FEA	23
2.13. $P - \delta$ effects on a column	27
2.14. Nonlinear boundary condition.....	27
2.15. Forms of modified Newton-Raphson iteration.....	28
2.16. Inappropriate positioning of PVC for wiring pipe in column's section	30
2.17. Configurations of lateral reinforcement in hollow columns	34
3.1. The position of the hole in the models for both rectangular and square column	36
3.2. Geometrical descriptions of plate, rebar, and stirrup	37
3.3. Nominal concrete stress-strains for (a) compression and (b) tension	48
3.4. Reinforcement bar stress-strain relation according to ES EN 1992.....	49
3.5. Typical model showing generated mesh, loading and boundary condition	52
3.6. Commonly used element families	54
3.7. Linear brick, quadratic brick, and modified tetrahedral elements.....	54
3.8. Mesh pattern in different cases.....	57
3.9. First iteration in an increment	59
3.10. Second iteration in an increment	60
3.11. Flowchart of finite element analysis	64
3.12. Positioning of the hole in center of column's cross-section.....	67
3.13. Experimental set-up.....	67
3.14. Model sample and strain-stress distribution of rectangular columns	68
3.15. Model sample and strain-stress distribution of square columns	68

4.1. Cases layout-descriptions in 2D	74
4.2. Cases layout- descriptions in 3D	74
4.3. Plot of undeformed and deformed shape of model-case a&b	75
4.4. Vertical stress distribution contour of square column model with the hole at Case-b, case-d, case-c, and case-a respectively in 2D and 3D layout.....	76
4.5. Vertical strain distribution contour of square column model with the hole at case-b case-d, case-c, and case-a respectively in 2D and 3D layout.....	76
4.6. Vertical stress distribution contour of rectangular column model with the hole at Case-c, case-a, case-b, and case-d respectively in 2D and 3D layout.....	77
4.7. Vertical strain distribution contour of rectangular column model with the hole at Case-c, case-a, case-b, and case-d respectively in 2D and 3D layout.....	77
4.8. Stress-strain curves of square columns for case-a, case-b, case-c, and case-d respectively.....	78
4.9. Stress-strain curve of square columns for case-a, case-b, case-c, and case-d respectively.....	79
4.10. Stress-strain curve of rectangular columns for case-a, case-b, case-c, and case-d respectively.....	80
4.11. Stress-strain curve of rectangular columns for case-a, case-b, case-c, and case-d respectively.....	81
4.12. Load-deflection curve of square columns for case-a, case-b, case-c, and case-d	84
4.13. Load-deflection curve of square columns for case-a, case-b, case-c, and case-d respectively.....	85
4.14. Load-deflection curve of square columns for case-a, case-b, case-c, and case-d respectively.....	86
4.15. Load-deflection curve of square columns for case-a, case-b, case-c, and case-d respectively.	87

LIST OF ABBREVIATIONS

ACI	American Concrete Institute
CDP	Concrete Damage Plasticity
EBCS	Ethiopian Building Code Standard
EN	European code standard
ES	Ethiopian building code standard
FEM	Finite Element Method
JU	Jimma University
PVC	Poly Vinyl Chloride
R-	Rectangle
RC	Reinforced Concrete
S-	Square

LIST OF SYMBOLS

C_r	-	Rectangular column
P	-	Vertical load to the Column
C_s	-	Square column
N	-	Column Design Load
M	-	Ultimate Moment
f_y	-	Characteristic yield strength of steel
e	-	Eccentricity of load
d	-	Effective depth
A_g	-	Gross area of the column section
$S-10$	-	Square column with radius of 10mm
h	-	Height of the column
b	-	Width of column
E	-	Young's Modulus
f_{cu}	-	Characteristic compressive strength of concrete
$50-75$	-	Position of the hole (pipe) at center from one side and
$R-25$	-	Rectangular column with radius of 25mm
$S-25$	-	Square column with radius of 25mm
A_{sc}	-	Area of the longitudinal reinforcement
$50-50$	-	Position of the hole (pipe) at the center
$R-10$	-	Rectangular column with radius of 10mm
A_g	-	Total area of longitudinal reinforcement
A_{nc}	-	Net concrete area of the cross-section of the model
$75-75$	-	Position of the hole (pipe) at quarter from both sides at quarter from another side

CHAPTER ONE

INTRODUCTION

1.1 Background

A column is a structural element that transmits, through compression, the weight of the structure above to other structural elements below. In other words, a column is a compression member. The global behavior of such members depends upon the strength of the material used, nature of end supports, the length/radius ratio and shape of its cross-section. The above mentioned parameters are also, sufficiently comprehensive in influencing load carrying capacity of the columns. Failure of a column in a critical location can cause the progressive collapse of the adjoining floors and the ultimate total collapse of the entire structure.

For design purposes, columns are divided into two types, short columns and slender columns. Considering lateral load action, columns are divided into two groups, which are braced columns and unbraced columns.

The construction of tall building using hollow columns is an attractive means by which the superstructure weight is minimized whilst maintaining column strength, and significantly reducing the construction cost.

It is timely sensitive to put the holes (pipes) vertically inside the reinforced concrete columns to accommodate the essential services such as, electric wiring from floor to floor, drainage of roof top rain water etc. The holes (pipes) are placed inside the columns, based on pretext to maintain the aesthetic of the buildings. The practice of embedding down pipes (holes) inside reinforced concrete columns is followed particularly in multistory buildings will exactly reduce the stability of structure so that, the structure load carrying capacity and its ultimate strength will be reduced significantly. The diameters of holes (pipes) vary, depending on the amount of cables required to pass through and the position of placing this pipe that will influence the capacity of structure as well.

The material properties specifically, the compressive strength of the concrete (grade) and tensile strength of steel (grade) is assumed.

Current design philosophy[13] for ductile reinforced concrete columns aims at providing sufficient transverse steel in potential plastic hinge regions to confine the compressed concrete, prevent lateral buckling of the longitudinal reinforcing steel, and act as shear reinforcements.

1.2 Statements of the problem

Building weight and cost will be increased with construction material increment rate. Efficient structural and material systems are needed to reduce weight and cost. Medium rise building may contain a column with a longitudinal hole for different service, which will result in cost and weight reduction. These types of columns are highly susceptible for distorting the stability of structure as a whole.

The practice of drilling hole (PVC pipe) inside reinforced concrete (RC) columns for electric wiring and for draining the rainwater are becoming an attractive activity nowadays for building aesthetic and to be safe from electric fire shock, however this method of draining and wiring could cause serious damage to the safety of the structure in sense that, the hole will affect the effective cross-sectional area of the column [18].

To the best of the knowledge of the authors, no significant investigations have been carried out to study the load carrying capacity of these types of columns. In addition to this no study has been conducted to indicate where the electric wiring should be exactly placed in column cross-section in terms of column load carrying capacity increment. On other hand, nothing has been said on the previous study about types of reinforced columns, rectangular or square, and about size of hole by exactly indicating perfect position of this hole in order of increasing columns carrying capacity. Most of the previous works in this regard have been limited to the study of the effects of constant axial load and eccentric load on the behavior of rectangular and circular hollow reinforced concrete columns and also no information and guidelines in codes of practice (ACI 318-05, BS 8110-97) on this problem are available and no other significant investigations have been carried out.

1.3 Research Questions

1. How much reduction in load carrying capacity of the RC columns due to piping installation?
2. Does the three-dimensional non-linear finite element analysis fit the estimation of load carrying capacity of columns affected by the reduction of cross-section?
3. How much deviation in strength of solid column compare to the columns with the conduit?

1.4 Objectives

1.4.1 General objective

The main objective of study is to investigate reinforced concrete columns with concealed pipe for plumbing and electric installation using finite element analysis.

1.4.2 Specific Objectives

1. To investigate the reduction in load carrying capacity of the reinforced concrete short braced columns having rectangular and square cross-sections with hole in different places
2. To model and analyze the RC column using three-dimensional non-linear finite element analysis.
3. To compare the research findings with the results of solid columns (control specimen) of the same dimensions under the same studying parameters.

1.5 Significance of the Study

This study was conducted to compare the compressive strength and load carrying capacity of solid RC columns to columns with holes (pipes) which allows electric wire through them. Wide societies particularly, city Administrations will benefit from the study as a source of information and foundation for the construction industry that can help to improve and control qualities of the construction regarding to standard and specifications. Owners, contractors and consultants will benefit from the study as reference point of view so that they will understand apparently, how to utilize the material for construction for building projects.

The study will provide lessons that will help the concerned body those desired to come up with appropriate measures to address problems resulting from using solid RC column and hollow columns with different pipe placement vertical through the column in considering different pipe diameter and its orientation, by apparently identifying their structural stability and strength. Other researchers will use the findings as a reference for further researches on structural stability and strength of columns with the pipe embedded inside it.

1.6 Scope of the Study

The study comprised 3-D finite element analyses of RC braced short columns with holes (embedded pipes) representing medium level building (i.e. medium rising building columns). The investigation concentrated on the modeling of axially loaded columns with hole (PVC drain pipes) positioned at different places in the cross-sections of the columns; at center, one quarter from both sides, and one quarter from one side and one half from another side . The ABAQUS finite element program was used to model the columns. The analysis covers material nonlinearity. The dimensions of the solid columns for comparative study were the same as that of columns with hole modeled by software. The columns analyzing and modeling carried out and the finite element analysis results were compared with the solid columns of the same dimensions as part of the study.

CHAPTER TWO

REVIEW OF RELATED LITERATURE

2.1 Theoretical review

In the construction of modern buildings, a network of pipes and ducts is necessary to accommodate the essential services like water supply, sewage, air-conditioning, electricity, telephone, and computer network. Usually, these pipes and ducts are placed underneath the beam soffit and, for aesthetic reasons, are covered by a suspended ceiling, thus creating a dead space. Passing these ducts through longitudinal and transverse openings in the columns leads to a reduction in the dead space and results in a more compact design. The practice of positioning hole (Poly Vinyl Chloride (PVC)) inside reinforced concrete (RC) columns for electric wiring, and drain the rainwater from the rooftop of the multi-story buildings and discharge it at the ground level has become a usual practice nowadays (Figure 2.1). The practice has been adopted under the impression that, exposing the pipes (holes) outside the columns will affect the appearance of the buildings.



Figure 2.1 Hole (rain water pipe) is positioned inside columns [13].

However, this method of drainage could cause serious damage to the safety of the structure. Columns constructed with holes (embedded PVC drainpipes), not only have reduced load carrying capacities but also, could be very dangerous to the safety of the entire building structure and can reduce its useful life significantly.

Some of the problems caused by the practice of positioning hole (PVC pipe) inside the column are as follows [13]:

- Placing the hole in the corner or edge of the column's section will reduce the effective cross-sectional area of the column significantly and also will affect to its shear capacity.
- Even in case the hole (PVC pipe) is positioned at the central part of the column's cross-section, the assessment of the effective depth of the column section might become inaccurate and hence, load carrying capacity of the columns is further reduced.
- Leakage from the joint lapping part of the pipe (hole) can cause corrosion and rusting in the reinforcement of the column, and hence loss of bond and reduction in the strength of the structural element.
- The huge reduction in the column's strength at ground level, where elbow part is used to discharge rainwater.

Figure 2.2 shows a typical column with hole positioned at the center of column cross-section. The cross section dimensions of the column are represented by h and b , where its height is l .

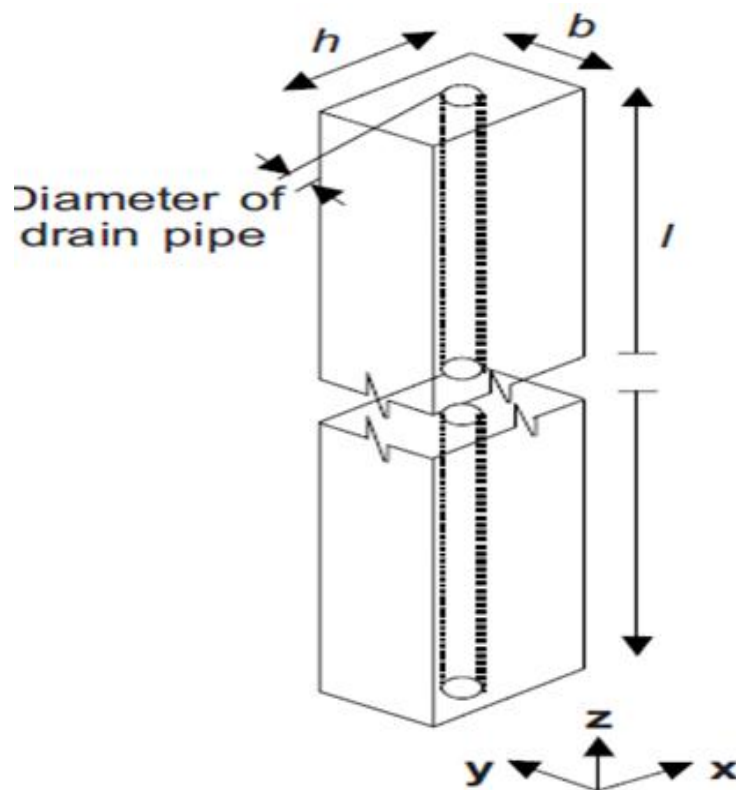


Figure 2.2 Typical columns with hole at the center [5].

Many formulations in codes of practice or recent research documents have been proposed and validated only for solid sections requiring a better understanding of the role played by the different resisting mechanisms and some adjustments. The most important parameters that govern the behavior of hollow columns are [11]:

- The shape of section: square, circular or rectangular,
- The amount of longitudinal and transverse reinforcement (steel ratio ρ),
- The numbers of layers of reinforcement placed near inside and outside faces of the section and tied through the wall thickness,
- The wall relative thickness (inside to outside diameter ratio D_i/D_e), and
- The axial load ratio

2.2 Concrete

Concrete materials have a sufficient compressive strength, so are widely used in civil engineering, reactor buildings, bridges, irrigation works and blast resistant structures, etc. Concrete has an inconsistent stress-strain relation, depending on its respective strength. It assumes that the main two failure mechanisms are tensile cracking and compressive crushing of the concrete material. The evolution of the yield (or failure) surface is controlled by two hardening variables, $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$, linked to failure mechanisms under tension and compression loading, respectively. We refer to $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$ as tensile and compressive equivalent plastic strains, respectively. The main assumptions about the mechanical behavior of concrete for uniaxial tension and compression in Abaqus is discussed as below. The model assumes that the uniaxial tensile and compressive response of concrete is characterized by damaged plasticity, as shown in Figure 2.3

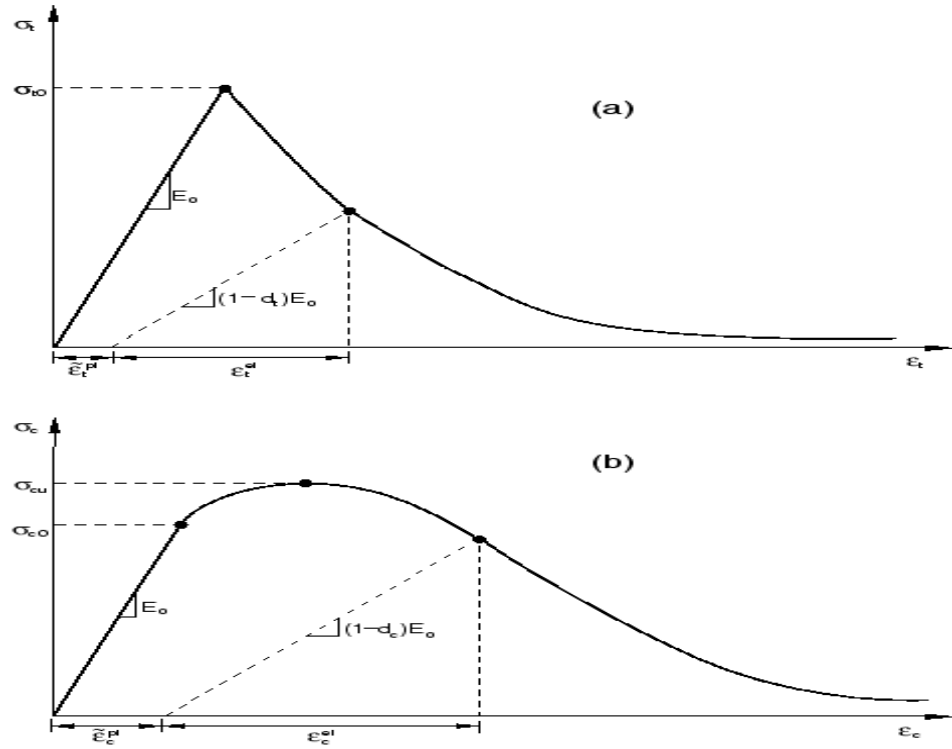


Figure 2.3 Response of concrete to uniaxial loading in tension (a) and compression (b) [14]

The behavior of concrete is almost elastically when the load is applied to the concrete. So according to the stress, the strain of the concrete is increasing approximately in a linear manner. Under uniaxial tension the stress-strain response follows a linear elastic relationship until the value of the failure stress to σ_{to} is reached. The failure stress corresponds to the onset of micro-cracking in the concrete material. Beyond the failure stress the formation of micro cracks is represented macroscopically with a softening stress-strain response, which induces strain localization in the concrete structure. Under uniaxial compression the response is linear until the value of initial yield σ_{co} . In the plastic regime the response is typically characterized by stress hardening followed by strain softening beyond the ultimate stress σ_{cu} . This representation, although somewhat simplified, captures the main features of the response of concrete.

It is assumed that the uniaxial stress-strain curves can be converted into stress versus plastic-strain curves. This conversion is performed automatically by Abaqus from the user-provided stress versus inelastic strain data, as:

$$\begin{aligned} \sigma_t &= \sigma_t(\epsilon_t^{pl}, f_i) \\ \sigma_c &= \sigma_c(\epsilon_c^{pl}, f_i) \end{aligned} \quad [2.1]$$

Where the subscripts t and c refer to tension and compression, respectively; $\varepsilon_t^{\sim pl}$ and $\varepsilon_c^{\sim pl}$ are the equivalent plastic strains, and f_i are other predefined field variables.

2.2.1 Concrete Damaged Plasticity

The concrete damaged plasticity model uses the concept of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete. In contrast to the brittle cracking model, it allows the definition of strain hardening in compression and can be defined to be sensitive to the straining rate, which resembles the behavior of concrete more realistically. When the concrete specimen is unloaded from any point on the strain softening branch of the stress-strain curves, the unloading response is weakened: the elastic stiffness of the material appears to be damaged (or degraded). The degradation of the elastic stiffness is characterized by two damage variables, d_t and d_c which are assumed to be functions of the plastic strains, temperature, and field variables:

$$\begin{aligned} d_t &= d_t(\varepsilon_t^{\sim pl}, f_i); 0 \leq d_t \leq 1 \\ d_c &= d_c(\varepsilon_c^{\sim pl}, f_i); 0 \leq d_c \leq 1 \end{aligned} \quad [2.2]$$

The damage variables can take values from zero, representing the undamaged material, to one, which represents total loss of strength.

If E_0 is the initial (undamaged) elastic stiffness of the material, the stress-strain relations under uniaxial tension and compression loading are, respectively

$$\begin{aligned} \sigma_t &= (1 - d_t)E_0(\varepsilon_t - \varepsilon_t^{\sim pl}); \\ \sigma_c &= (1 - d_c)E_0(\varepsilon_c - \varepsilon_c^{\sim pl}); \end{aligned} \quad [2.3]$$

We define the effective tensile and compressive cohesion stresses $\bar{\sigma}_t$ and $\bar{\sigma}_c$ as

$$\bar{\sigma}_t = \frac{\sigma_t}{1-d_t} = E_o(\varepsilon_t - \varepsilon_t^{\sim pl});$$

$$\bar{\sigma}_c = \frac{\sigma_c}{1-d_c} = E_o(\varepsilon_t - \varepsilon_c^{\sim pl});$$
[2.4]

The effective cohesion stresses determine the size of the yield (or failure) surface. Modified Hognestad model for uniaxial unconfined stress-strain relationship is universally accepted normal grade of concrete. According to Hognestad approach;

$$\text{For } 0 < \varepsilon < \varepsilon_o, \frac{\sigma}{\sigma_{cu}} = 2 \frac{\varepsilon}{\varepsilon_o} \left(1 - \frac{\varepsilon}{2\varepsilon_o}\right)$$

$$\text{For } \varepsilon_o < \varepsilon < \varepsilon_{cu}, \frac{\sigma}{\sigma_{cu}} = 1 - 0.15 \left(\frac{\varepsilon - \varepsilon_o}{\varepsilon_{cu} - \varepsilon_o}\right)$$
[2.5]

2.2.2 Elastic Modulus of Concrete

The stress-strain relationship for concrete is almost linear provided that the stress applied is not greater than one third of the ultimate compressive strength. A number of alternative definitions are able to describe the elasticity of the concrete, but the most commonly accepted is $E = E_c$, where E_c is known as secant or static modulus (see Figure 2.4).

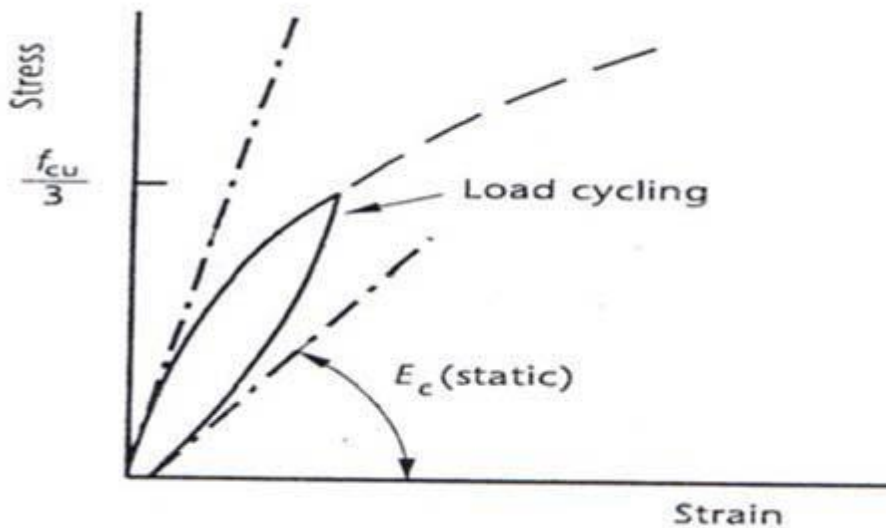


Figure 2.4 Static modulus of concrete [20]

The modulus of elasticity of concrete is not constant and highly depends on the compressive strength of concrete.

BS 1881 has recommended a series of procedure to acquire the static modulus. In brief, concrete samples in standard cylindrical shape will be loaded just above one third of its compressive strength, and then cycled back to zero stress in order to remove the effect of initial ‘bedding-in’ and minor redistribution of stress in the concrete under the load. Eventually the concrete strain will react almost linearly to the stress and the average slope of the graph will be the static modulus of elasticity [8].

2.3 Steel Reinforcement

Steel has great tensile strength and use in the concrete because concrete does not act in tension well alone; also there is good bond between concrete and reinforcement. The reinforcing steel has a wide range of strength. It has more consistent properties and quality compared to the concrete, because it is manufactured in a controlled environment. There are many types of steel reinforcement. The most common are plain round mild steel bars and high-yield stress deformed bars. The typical stress-strain relations of the reinforcing steel can be described in the stress-strain curve as shown in Figure 2.5

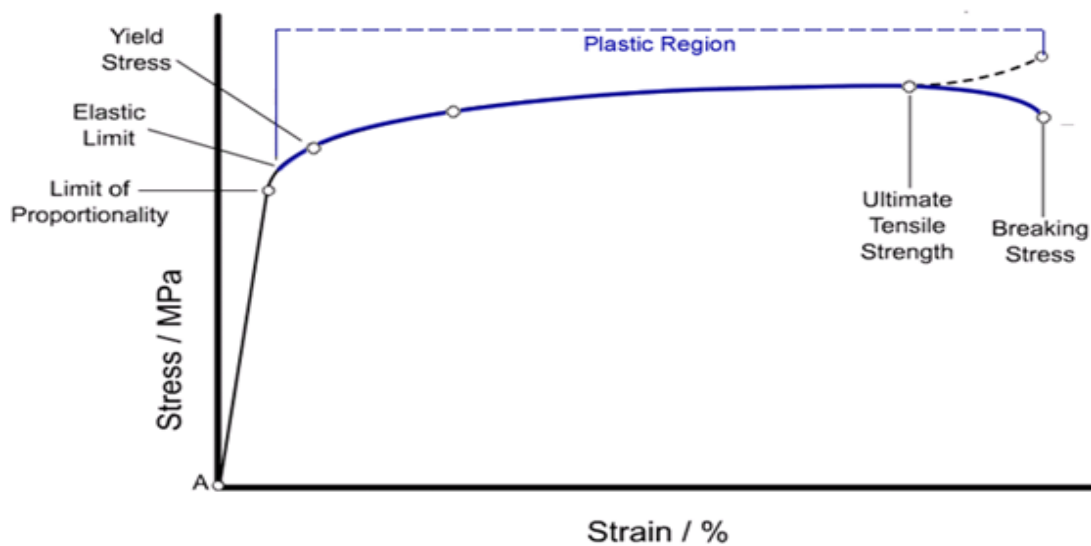


Figure 2.5 Typical stress-strain curves for reinforcing steel [20]

From the Figure 2.5, the mild steel behaves as an elastic material until it reaches its yield point, finally it will have a sudden increase in strain with minute changes in stress until it reaches the failure point. On the other hand, the high yield steel does not have a limited yield point but has a more gradual change from elastic to plastic behavior.

Reinforcing steels have a similar slope in the elastic region with $E_s = 200\text{kN/mm}^2$. The specific strength taken for the mild steel is the yield stress. For the high yield steel, the specific strength is taken as the 0.2% proof stress. BS 8110 has recommended an elastic-plastic model for stress-strain relationship, which the hardening effect is, neglected [7]. The stress-strain curve may be simplified bilinear as shown in Figure 2.6.

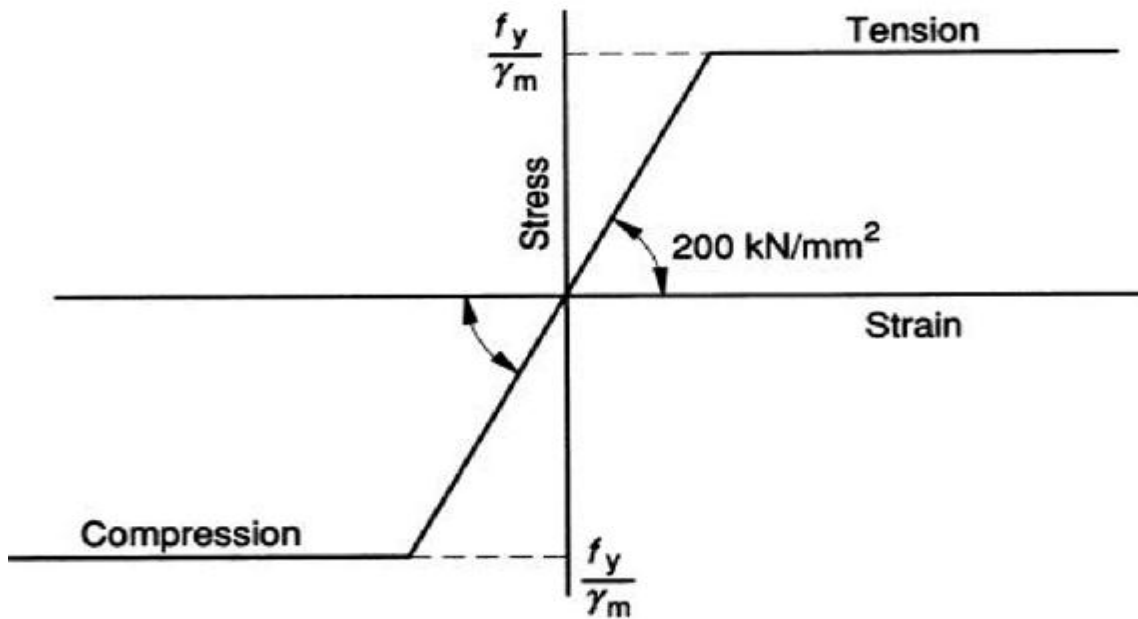


Figure 2.6 Simplified stress-strain curve for reinforcing steel [7]

2.4 Reinforced Concrete

Concrete is strong in compression, but weak in tension. As a result, cracks develop whenever loads, restrained shrinkage, or temperature changes give rise to tensile stresses in excess of the tensile strength of the concrete. The tensile cracking reduces the stiffness of the concrete and is usually the major contributor to the nonlinear behavior of reinforced concrete structures, such as panels and shells, where the stress is generally the biaxial tension-compression type [14]. Concrete and reinforcement can solidify because of good bond between them and also feature of concrete in compression and steel in tension. Reinforced concrete is a durable and ductile construction material that can be formed into many shapes and sizes from a simple rectangular column to a shell.

Tensile strength of concrete is neglected in the design of reinforced concrete because the tensile strength of concrete is just about 10% of its compressive strength. Therefore the tensile force is assumed to be resisted by the reinforcing steel completely.

The tensile stress is transferred to reinforcing steel through bonding between concrete and steel, thus insufficient bond will cause the reinforcement to slip within the concrete. Reinforcing steel can only develop its strength in concrete on the condition that it is anchored well to the concrete [14].

2.5 Reinforced Concrete Column

Columns are compression members, although they may have to resist bending forces due to the eccentricity. A column in a structure transfers loads from beams and slabs down to foundations. Design of the column is governed by the ultimate limit state, and the service limit state is seldom to be considered [21].

2.5.1 Types of Column and Failure Modes

For considering lateral load action, columns are divided into two types which are braced and unbraced. A column is a braced column when the load is resisted by the bracing members like shear wall. In this case, the column does not resist lateral load. An unbraced column is the column that is subjected to lateral loads.

For design purposes, column can be classified as short or slender by a ratio of effective height (l_e) in the bending axis considered to the column depth in the respective axis (h). BS 8110 has recommended that a slender column can be determined by Equation 2.6(for braced structure) or equation 2.7(for unbraced structure) [7].The column failure mode can be predicted when knowing the column is short or slender,

- Braced columns:

$$\frac{L}{H} \geq 15, \quad [2.6]$$

- unbraced columns:

$$\frac{L}{H} \geq 10, \quad [2.7]$$

Lateral loading acts on a building can cause lateral deflection (δ) to the column. Consequently, gravity load (p) in the column with eccentricity (δ) will induce additional moment ($M = p.\delta$), and increase the column moment.

Buckling unlikely occurs to a short column. So when the axial load exceeded its material strength, it will fail. This will cause the column to bulge and eventually to crush. Buckling may cause a slender column to deflect sideways and therefore induces an additional moment ($M = p.\delta$), as illustrated ($p - \delta$ effect) in Figure 2.7

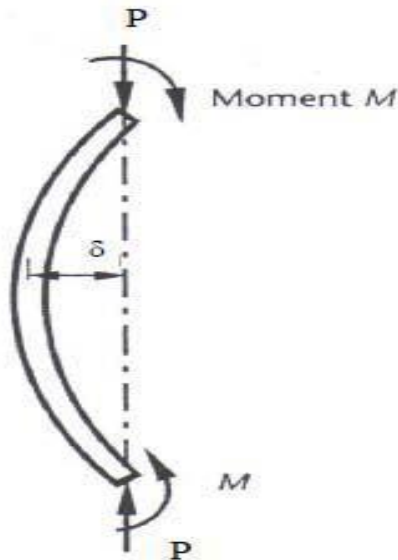


Figure 2.7 $p - \delta$ effects on slender column [20]

In heavily clad low and medium rise buildings, the $p - \delta$ effects may be small. However for lightly clad tall buildings with greater lateral flexibility, the $p - \delta$ effects may become significant. The $p - \delta$ effects may be large enough and require an increase in the column size.

There are three modes of column failure, crushing, intermediate and buckling.

- Crushing usually occurs in short columns when there is material failure with negligible lateral deflection. When there are large end moments acted on a column with an intermediate slenderness ratio, it is possible to occur. Figure 2.8 (a) illustrates the column failure mode in crushing.
- Intermediate is typical of intermediate columns and occurs when material failure intensified by the additional moment and the lateral deflection.
- In buckling, there is instability failure which occurs with slender columns. Figure 2.8 (b) illustrates the failure mode of column in buckling.

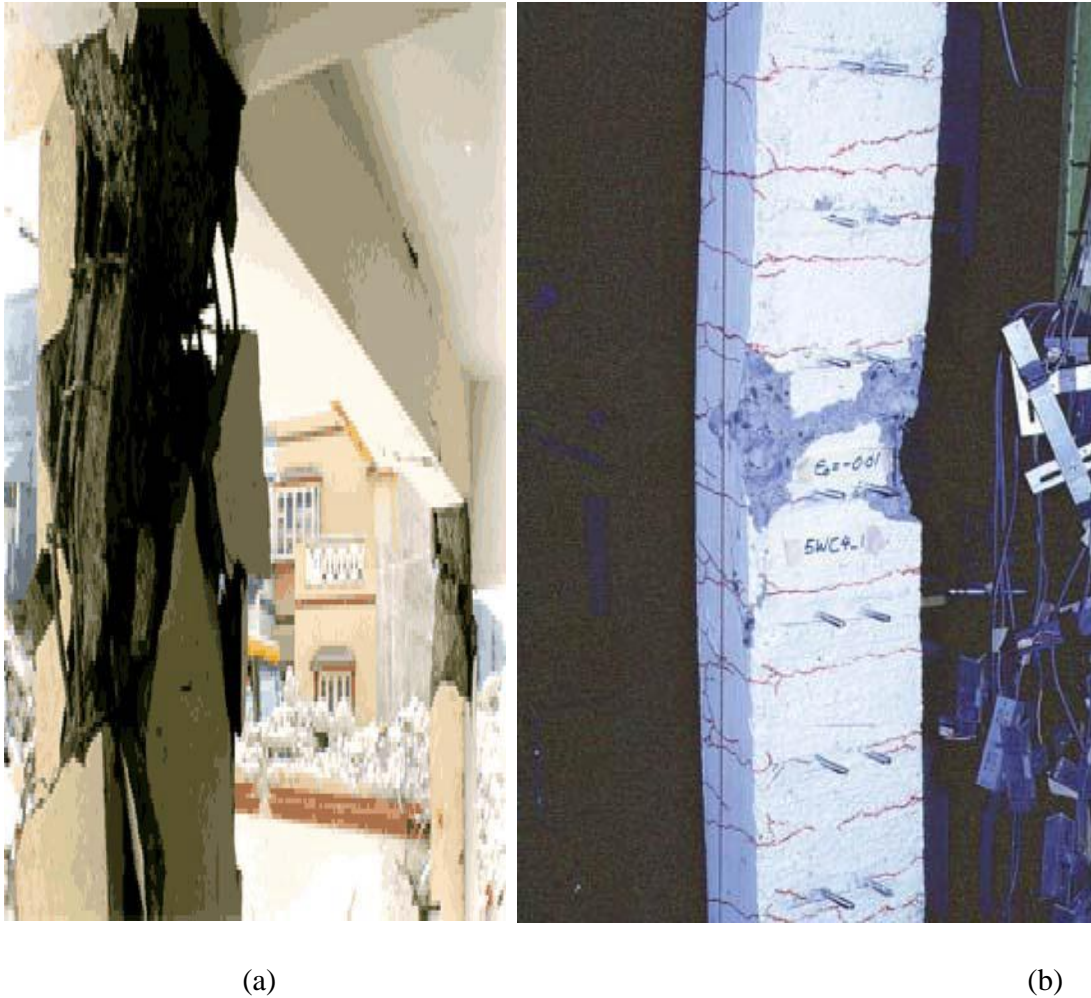


Figure 2.8 Columns failure modes [4]

2.5.2 Capacity of Short columns subjected to axial loads only

If a column is loaded to the failure the reinforcement is likely to reach its yield strength before the concrete fails in compression (see figure 2.9) this statements remains valid for the material strengths generally used in buildings as well as for higher concrete strengths where the strain coinciding with the maximum stress increases as the concrete strength increases.

If the higher strength reinforcement is used the concrete may reach its maximum strength before the reinforcement yields. However, increase in deformation in the concrete at maximum stress will allow the reinforcement to reach its yield strength (park and paulay, 1975) so that the total resistance of a reinforced concrete can be expressed as the following sum;

$$N_{uz} = Kf'_c(A_{cg} - A_{sc}) + f_y A_{sc} \quad [2.8]$$

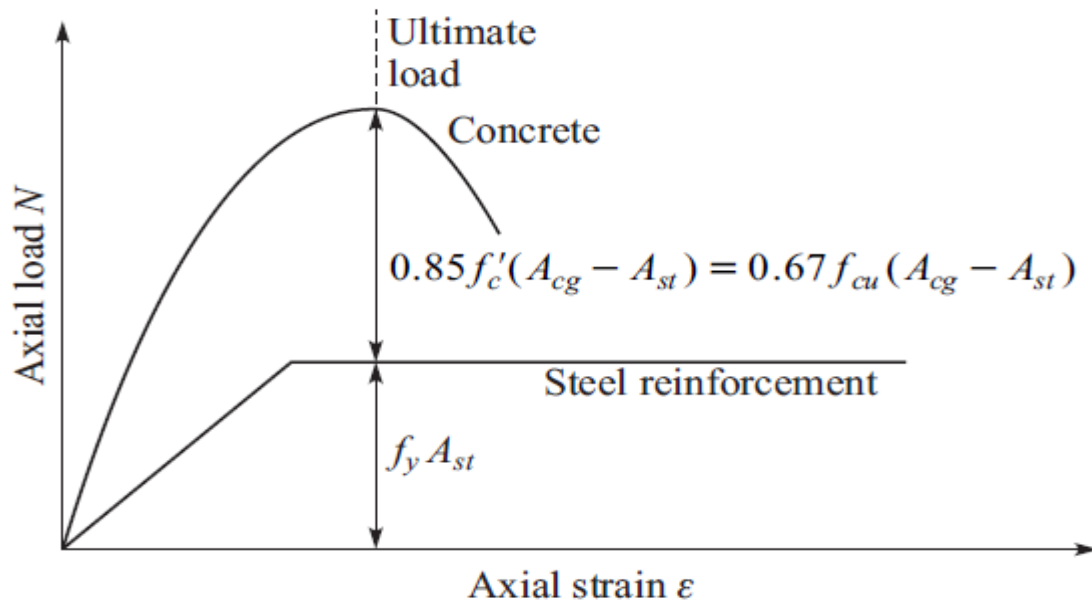


Figure 2.9 Axial load-strain response of steel and concrete in an axially loaded-column (Park and Paulay, 1984)

Experimental tests on a large number of columns (Richard and Brown, 1934; Hognestad, 1951) showed that the factor K that is multiplied to f'_c range from 0.55 to values greater than 1.0 with an average of 0.83. A value of $K = 0.83$ is recommended by the ACI code (ACI 318, 1989).

The following reasons may explain why $K < 1$ and the fully cylinder strength, as expected, is not reached in column (Kong and Even, 1987):

- The columns height to width ratio is much greater than that of a cylinder so that the influence of end restraints will be much less than in a cylinder.
- Columns are tested at a slower rate than cylinders and it is known that the measured strength increases with an increase in loading rate.
- The concrete in a column will be less compacted than a cylinder.
- In vertically cast columns bleed water in the top of column will lead to a lower cement/water ratio and hence lower strength.

It should be noted that it is coincidence that K is equal to a similar factor used in flexure since they originate from different testing conditions. (Park and Paulay, BS 8110) Expressing Eq. (2.8) in terms of the cube strength f_{cu} by assuming $f'_c = 0.8 f_{cu}$ and introducing material safety factors.

$$N_{uz} = \frac{0.67}{\gamma_{mc}} f_{cu} (A_{cg} - A_{sc}) + \frac{f_y}{\gamma_{ms}} A_{sc} \quad [2.9]$$

At the ultimate limit state with $\gamma_{mc}=1.5$ and $\gamma_{ms}=1.15$ this equation reduced to

$$N_{uz} = 0.45 f_{cu} (A_{cg} - A_{sc}) + 0.87 f_y A_{sc} \quad [2.10]$$

The recommendations in SABS0100 are based on the CP110 code, that proceeded BS8110, where it was believed that the reinforcement would buckle before the design yield strength compression of $0.87f_y$ would be reached. It was also considered uneconomical to provide the large quantity of ties necessary to prevent the bars from buckling so that the design yield strength in compression is reduced as follows,

$$N_{uz} = \frac{0.67}{\gamma_{mc}} f_{cu} (A_{cg} - A_{sc}) + \left(\frac{f_y}{\gamma_{ms} + f_y / 2000} \right) A_{sc} \quad [2.11]$$

For local reinforcement yield strengths ranging between 250 and 450 MPa, the design compression stress in the reinforcement will range between $0.784f_y$ and $0.729f_y$, it appears as if an average value of $0.750 f_y$ is recommended by SABS 0100.

$$N_{uz} = 0.45 f_{cu} (A_{cg} - A_{sc}) + 0.75 f_y A_{sc} \quad [2.12]$$

The concrete in a reinforced concrete column will creep over a time, reducing the stress in the concrete and increasing those in the reinforcement. However, neither creep nor load history will have any significant effect on the ultimate load carrying capacity on the reinforced concrete column.

The above equations apply to column with no imperfections where the load is applied at the centroid of the cross-section. These conditions are seldom encountered in practice and it is customary to allow small eccentricity ($\approx 0.05h$) of loading,

For BS, 8110;

$$N_{uz} = 0.40 f_{cu} (A_{cg} - A_{sc}) + 0.75 f_y A_{sc} \quad [2.13]$$

For SABS, 0100

$$N_{uz} = 0.40 f_{cu} (A_{cg} - A_{sc}) + 0.67 f_y A_{sc} \quad [2.14]$$

2.5.3 Buckling of Column

Buckling is a failure mode characterized by a sudden failure of a structural member subjected to high compressive stresses, where the actual compressive stress is less than the ultimate compressive stresses at the point of failure that the material is capable of withstanding.

A short column will fail under the action of an axial load by direct compression before it buckles, but a slender column will fail by buckling in the same manner. When a compression member becomes longer, the role of the geometry and stiffness (Young 'modulus) becomes more and more important.

When axial load applies at the column, the critical load for a slender column can be determined by the Euler Buckling Load,

$$P_c = \frac{\Pi^2 EA}{\left(\frac{KL}{r}\right)^2} \quad [2.15]$$

In reality, loads are seldom loaded on the column centric, and the moment may react due to the small eccentricity. The initial column shape can be approximated before the application of load by the sine function, suggested by Meyer [18],

$$y_o = \delta \sin \frac{\pi x}{L} \quad [2.16]$$

Where δ_o is the maximum deviation from the straightness at mid-height (see Figure 2.10),

Also the maximum displacement due to buckling at the mid-height can be expressed as,

$$y_{MAX} = \frac{\delta_o}{1 - \frac{P}{P_c}} \quad [2.17]$$

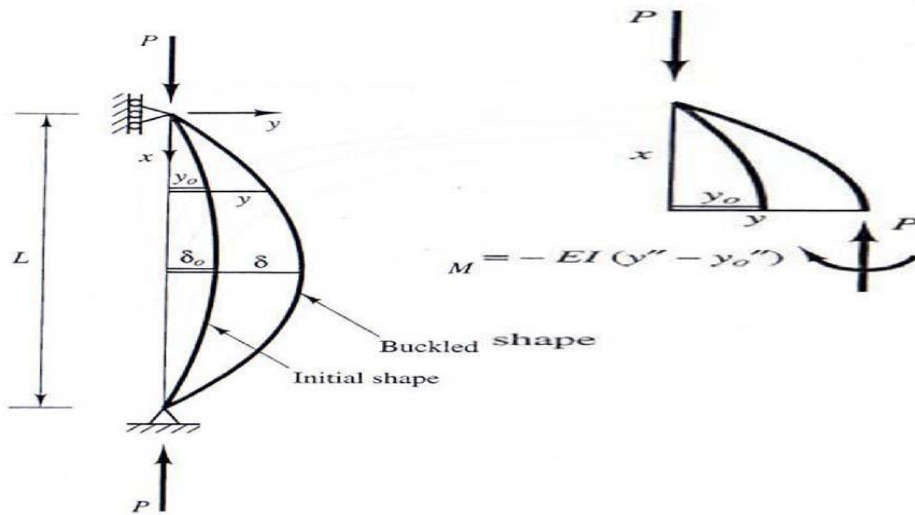


Figure 2.10 Buckling of column [18]

It is important that consider the effect of $P - \delta$ in the buckling. Hence, the bending moment taking place in the column, when the column reaches its instability state, as illustrated in Figure 2.11

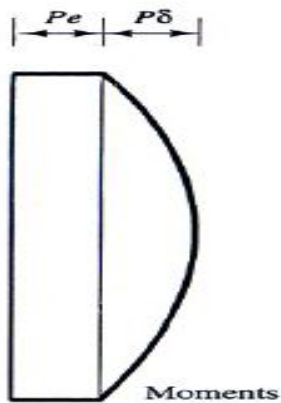


Figure 2.11 Bending moment in the column [18]

Therefore, the maximum moment is,

$$M_o = \frac{M_o}{1 - \frac{P}{P_c}} \quad [2.18]$$

Where $M_o = P.e$ is the moment due to the load eccentricity (e), and also the factor $\frac{1}{[1 - (P / Pc)]}$ is the moment magnification factor. When the column load reaches the buckling load, the buckling moment will grow unbounded.

2.6 Finite Element Method

The Finite Element Method (FEM) of analysis is a very powerful, modern computational tool. This method has been used successfully to solve very complex structural engineering problems. FEM has also been widely used in other fields such as thermal analysis, fluid mechanics, and electromagnetic fields.

Since the method involves a large number of computations, therefore it requires computer to solve a problem.

The finite element method represents the extension of matrix method for skeletal structures to the analysis of continuum structures. In finite element method, the continuum is idealized as a structure consisting of a number of individual elements connected only at nodal points [12].

2.6.1 Brief History

Courant has been credited with being the first person to develop the finite element method in 1943. He used piecewise polynomial interpolation over sub regions to investigate torsion problems.

The next significant step in the utilization of finite element method was taken by Boeing in the 1950s, in which Boeing used triangular stress elements to model airplane wings. At 1956, Turner et al. had presented their findings on the stiffness matrices for beam, truss and other elements. Also at 1960, Clough made the term “Finite Element popular. During 1960s, investigators began to apply the finite element method to other areas of engineering, such as heat transfer and seepage flow problems.

The very first attempt to analyze the reinforced concrete by finite element was done by Ngo and Scordelis in 1967. And finally, Zienkiewicz and Cheung wrote the first book entirely devoted to the finite element method in 1967 [12].

2.6.2 Definition of FEM

The finite element method (FEM) is a general numerical technique for approximating the behavior of continua by assembly of small parts (elements). Each element is of simple geometry and therefore is much easier to analyze than the actual structure.

In essence approximate a complicated solution by a model that consists of piecewise continuous simple solutions. The elements are called “Finite” to distinguish them from differential elements used in calculus [12].

2.6.3 Advantages of FEM

When numerical analyses were first introduced in engineering practice in the 1960s, many analysis methods such as boundary element method and finite difference method, were in use. Over time, these methods were dominated by the FEM because of its inherent generality and numerical efficiency. Although other methods retain advantages in certain niche applications, they are difficult or impossible to employ to other types of analyses. At the same time, the FEM has wider applicability. The versatile analysis tool can be applied to almost any types of engineering problems. For this reason, the FEM has widespread adoption for increasingly diverse problems and dominated the market of commercial analysis software [2].

One of the main advantages of FEM over most other analysis methods is the fact that FEM can handle irregular geometries routinely and implement higher order elements with relative ease. Besides, very little extra effort is required in the FE formulation when anisotropic or heterogeneous are to be modeled. In FE modeling, elements with different properties and geometries are used to cater for structures with different types and behaviors. Another advantage of FEM is the ease to handle mixed boundary condition. All the various types of boundary conditions that may encounter in problem can be included in the formulation of FE [2].

2.6.4 Methods of Formulating Finite Element Problems

The finite element method consists primarily of replacing a set of differential equations in terms of unknown variables with an equivalent but approximate set of algebraic equations where each of the unknown variables is evaluated at a nodal point. Generally there are three approaches to formulate the finite element problems [12].

1. Direct Method

In direct method individual structural members, such as bars, are analyzed with techniques similar to those used in analysis of simple trusses and frames. The displacement caused by applied forces is expressed by a set of equations convertible into a stiffness matrix for each of the structural member. The assembly of the element stiffness matrices together will form a large (global) matrix, which represents the stiffness of the entire structure. Direct method is difficult to apply to two-and three dimensional problems.

2. The Minimum Total Potential Energy Method

The minimum total potential energy method is a common approach in generating finite element model in solid mechanics. External load applied to the body will cause the body to deform. During deformation, the work done by external forces is stored in the material in the form of elastic energy, called “Strain Energy”. The minimum total potential energy principle states that for a stable system, the displacement at the equilibrium position occurs such that the value of the system’s total potential energy is a minimum.

2. Weighted Residual Method

The weighted residual methods are based on assuming an approximate solution for the governing differential equations. The assumed solution must satisfy the initial and boundary conditions of the given problem, because the assumed solution is not exact, substitution of the solution into the differential equation will lead to some residual or error. Simply stated, each residual method requires the error to vanish over some selected intervals or at some points.

2.6.5 Types of Finite Element

Classification according to the way the element represents the displacement field in three dimensions distinguishes among solid, shell, membrane and beam elements.

I. Solid element

The solid element fully represents all three dimensions. The solid element models the 3-D displacement field with three variables.

II. Shell element

The shell elements are used when the thickness of the shell is considered small relative to the other dimensions. Stresses normal to the shell cross section are usually assumed to have linear distribution; consequently the shell element can model bending. The shell element models the displacement field with two variables.

III. Membrane element

The membrane element is visually similar to the shell element, but stresses normal to the shell cross section are usually assumed to be constant. The membrane element can model only membrane stresses but not bending stresses.

Iv. Beam element

The cross section is small in comparison with the length. The beam element models the 3-D displacement field with one variable.

2.6.6 Basic Steps of Finite Element Analysis

In general, there are several approaches to formulating finite element problems:

1. Direct formulation
2. The minimum total potential energy formulation
3. Weighted residual formulation

Again, it is important to note that the basic steps involved in any finite element analysis, regardless of how generate the finite element model, will be the same as those listed in Figure 2.12

Basic steps for the finite element analyses are illustrated as a flow chart in Figure 2.12

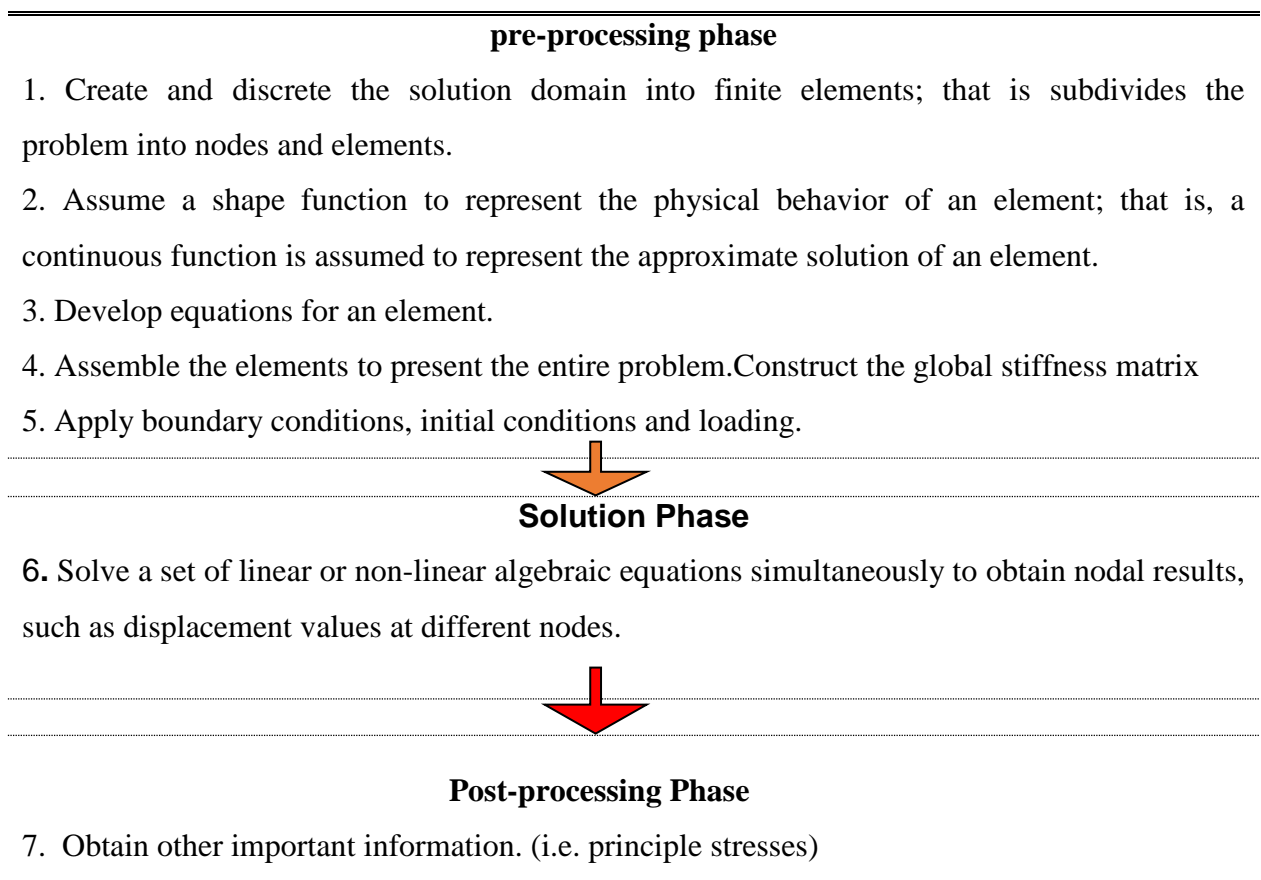


Figure 2.12 Flow chart for the basic steps of FEA [12]

2.6.7 Fundamental Requirements

Three basic conditions must be observed, which are:

1. The equilibrium of forces
2. The compatibility of displacements
3. The material behavior law

The first condition requires that the internal forces balance the external applied loads. Compatibility condition requires that the deformed structure fits together that the deformations of the member are compatible. It is also necessary to know the relationship between load and deformation for each component of the structure (material behavior law). This relationship in linear elasticity is the Hook's law.

2.6.8 Meshing

The most important requirement of the FEM is the need to split the solution domain (model geometry) into simply shaped subdomains called "finite elements." This is a discretization process commonly called "meshing" and elements are called "finite" because of their finite, rather than infinitesimally small size having infinite number of degrees of freedom. Thus the continuous model with an infinite number of degrees of freedom (DOF) is approximated by a discretized FE model with a finite DOF. This allows for reasonably simple polynomial functions to be used to approximate the field variables in each element. Meshing the model geometry also discretizes the original continuous boundary condition. The loads and restraints are represented by discrete loads and supports applied to element nodes [16].

There are many ways in turning a mathematical model into an FE model by meshing. The three major factors which define the choice of discretization are element size, element order (order of the element shape function) and element mapping (element shape may distort from the ideal shape after mapped to the actual shape in the FE mesh).

2.6.9 Verification of Results

Finite element analysis software has become a common tool in the hands of design engineers. The results of the finite element analysis have to be verified so that it does not contain errors such as applying wrong boundary conditions and loads, poor element shape and size after meshing, wrong input data, selecting inappropriate types of elements.

Experimental testing of the model is one of the best ways for checking the results, but it may be time consuming and expensive. Hence, it is always a good practice to start by applying equilibrium conditions and energy balance to different portions of a model to ensure that the physical laws are not violated [4].

2.7 ABAQUS

ABAQUS is one of the world's leading structural analysis systems. The ABAQUS system uses finite element analysis techniques to provide accurate solutions for all types of linear and nonlinear stress, dynamic, and thermal problems. It is an associative feature-based modeler. The model geometry is entered in terms of features which are sub-divided (discretized) into finite element in order to perform the analysis. Increasing the discretization of the features will usually result in an increase in accuracy of the solution, but with a corresponding increase in solution time and disk space required. The features in ABAQUS form a hierarchy that is volumes are comprised of surfaces, which in turn are made up of lines or combined lines, which are defined by points [2].

2.7.1 Characteristic of ABAQUS Software

ABAQUS software can analysis and organize complex structure problems and shapes including 3 dimensional structures. This software also can be used in dynamic structural analyses with temperature changes. ABAQUS software can solve problems up to 5000 number of elements.

2.7.2 Analysis Procedure According to ABAQUS Software

1. Pre-processing phase

Pre-processing involves creating a geometric representation of the structure, then assigning properties, then outputting the information as a formatted data file (.dat) suitable for processing by ABAQUS [2].

2. Finite Element Solver

Sets of linear or nonlinear algebra equations are solved simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in heat transfer problems.

3. Result-Processing

In this process, the results can be processed to show the contour of displacements, stresses, strains, reactions and other important information. Graphs as well as the deformed shapes of a model can be plotted.

2.8 Non-linear Analysis

In a linear finite element analysis, all materials are assumed to have linear elastic behavior and deformations are small enough not to significantly affect the overall behavior of the structure. However, nonlinear finite element analysis is required in situations such as gross changes in structural geometry, permanent deformations and structural cracks.

The nonlinear analysis generally can be divided into three types:

1. Material nonlinearity
2. Geometric nonlinearity
3. Boundary nonlinearity

1. Material nonlinearity; Material nonlinearity effects occur from a nonlinear constitutive model which has disproportionate stresses and strains. A nonlinear material model is one that does not follow a linear relation between strain and stress and thus does not have a constant modulus of elasticity. Using such a material, model stiffness changes during the loading process and the stiffness matrix must be recalculated during the solution process. The nonlinear material properties require more complex definition, which depends on what stress-strain relationship is assumed in the model [16]. Common examples of nonlinear material behavior are the plastic yielding of metals, the ductile fracture of granular composites such as concrete or time-dependent behavior such as creep.

2. Geometric nonlinearity; Geometric nonlinearity occurs when there is significant change in the structural configuration during loading. In nonlinear geometry analysis, the shape deformation (rather than the change in material properties, as was the case in the nonlinear material model) changes the structure matrix. Consequently, stiffness does not remain constant throughout the process of deformation due to the applied load, and the stiffness matrix must be recalculated during the process of load application. Common examples of geometric nonlinearity are plate structures which develop membrane behavior, and the geometric split of truss or shell structures. The geometric non-linearity is the main factor that causes the second order effect ($P - \delta$ effect). In Figure 2.14, it is necessary to consider geometrical non-linearity for knowing the actual response of the column to the load, because the linear solution would fail to consider the bending moment due to the eccentricity (δ) of the vertical load (P). If the effects of nonlinear in geometry are neglected, errors could be occurred depend on the deflections.



Figure 2.13 P- δ effects on a column [2]

3. Boundary nonlinearity; In boundary nonlinearity, the modifications to the external restraints resulting from deformation process such as lift-off, or smooth or frictional contact are taken into account within an analysis. In the Figure 2.15, the structure and its supporting surface can resist being pushed together, but not being pulled apart. The required contact condition may be imposed by using joint elements to connect between the structure and the rigid support, and specifying a nonlinear contact joint model incorporating large and zero local stiffness in compression and tension respectively.

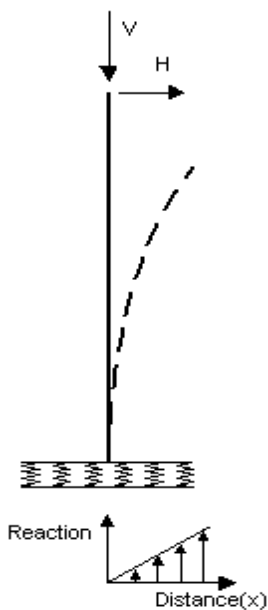


Figure 2.14 Nonlinear boundary condition [2]

ABAQUS applies the modified Newton-Raphson iteration in the non-linear static analysis. Newton-Raphson iteration is stable and converges quadratically, provided an initial estimation which is close enough to the solution is available. Besides that, the tangent stiffness matrix needs to be inverted during each iteration. It may fail to converge when extreme material nonlinearities are present in a structure, because of the complex iteration procedure.

Therefore, modification was done on Newton-Raphson iteration. With the modified Newton iteration, the current tangent stiffness matrix is replaced with a previous stiffness matrix. This reduces the number of iteration as the factorization of the tangent stiffness matrix is not required for every iteration.

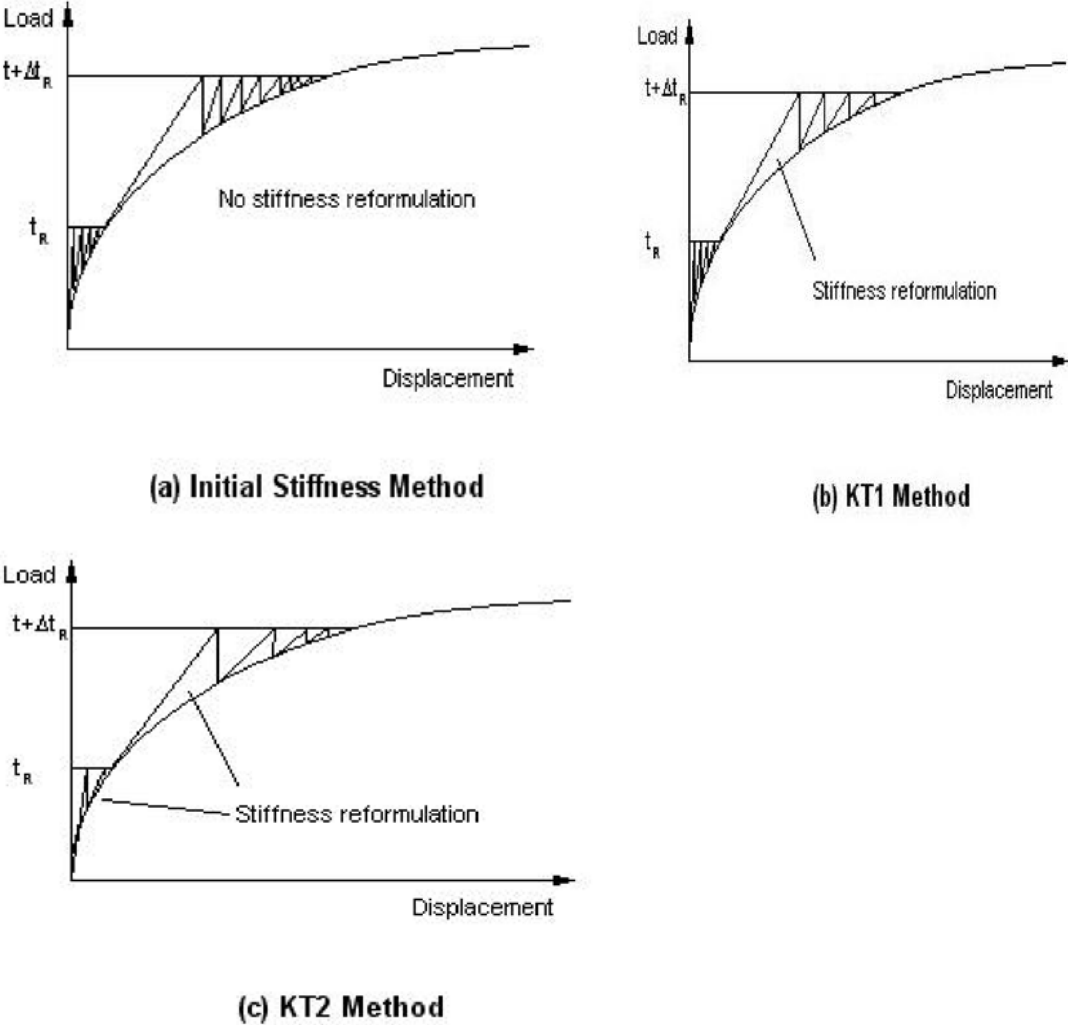


Figure 2.15 Forms of modified Newton-Raphson iteration [2]

2.9 Critique of the existing literature relevant to the study

There are few studies conducted by researchers to comprehend the behavior of the reinforced concrete column with hole. The study of column with hole for variety of service is not yet finished; it is developed throughout the year with the enhancement of technology. The previous study's shows that different position of pipe layout (inside or outside) the column with different diameter will be influence the carrying capacity of column as well disturbs the stability and also affect the aesthetic values of building. The literature which is reviewed in this chapter is following; Advantages of using columns with the holes over the solid columns, especially for bridge with high elevations in terms of reduction in mass, which will reduce the inertial response of the bridge piers, and the vulnerabilities of RC bridge columns with the duration of its service time.

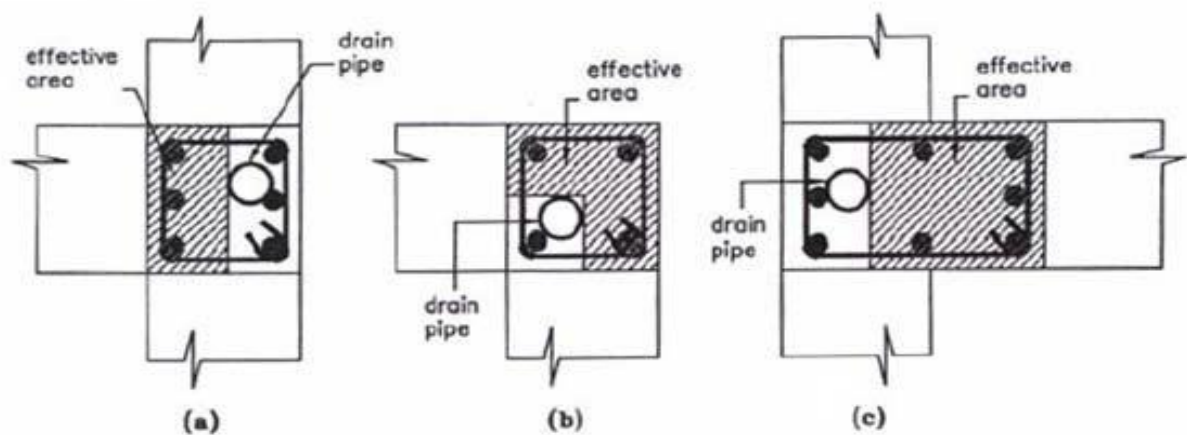
- ✚ Discussion of most important parameters that govern the behavior of hollow columns such as, the shape of section: circular or rectangular, the axial load ratio, the amount of longitudinal and transverse reinforcement (steel ratio ρ), and the numbers of layers of reinforcement placed near inside and outside faces of the section and tied through the wall thickness.
- ✚ The main role of transverse reinforcement shall be discussed as to provide shear resistance, to prevent premature buckling of longitudinal reinforcement, and to confine the compressed concrete.
- ✚ Discussion how critical section will be influenced under application of bending moment, axial load, and shear force for both rectangular and circular columns with a hole at different vertical placement in terms of carrying capacity of whole cross-section.

Jahangir Bakhteri, Wahid Omar and Ahmad Mahir Makhtar [7], presented a critical review of the reinforced concrete columns and walls concealing rain water pipe in multistory building. This paper showed that providing the hole inside columns not only reduces the load carrying capacity of the columns, also cause several damage to the buildings safety: the few are listed below;

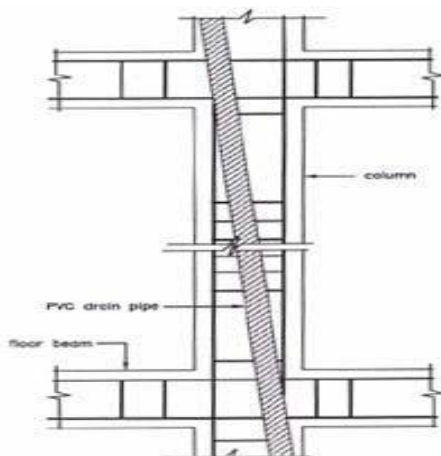
- Placing the hole in the corner or edge of the column's section will reduce the effective cross-sectional area of the column significantly and also will affect to its shear capacity.
- Even in case the hole (PVC pipe) is positioned at the central part of the column's cross-section, the assessment of the effective depth of the column section might become inaccurate and hence, load carrying capacity of the columns is further reduced.

- Leakage from the joint lapping part of the pipe (hole) can cause corrosion and rusting in the reinforcement of the column, and hence loss of bond and reduction in the strength of the structural element.

In Figure 2.18 (a), (b) and (c). The pipe may not be held at central position in the column/wall, because during casting and vibrating of the concrete there are chances that the pipe may get an inclined position which will cause further decrease in load carrying capacity of the column /wall.(Figure 2.18 (d))



(a), (b) & (c): Column section in plan showing probable effective area [7]



(d): vertical section of column showing inclined position of the PVC pipe

Figure 2.16 Inappropriate positioning of PVC for wiring pipe in column's section [7]

In case of the column with higher flexural stresses (i.e. moment), the assumption made in the effective depths of the column which contains drain pipe is usually inappropriate.

This is because, for the design purposes, the practicing engineers assume an approximate effective depth for the column's section and a rational formula for the calculation of the effective depth of such type of columns is lacking. An experimental study has been carried out (Jahangir Bakhteri & Ahmad Iskandar,[14]) to consider the effect of concealing Poly Vinyl Chloride (PVC) pipe inside reinforced concrete (RC) columns in multistory buildings. Compressive strength and load carrying capacity of rectangular reinforced concrete short columns with embedded rain water pipe have been investigated and the results have been presented. The models have been loaded axially until their failure and the collapse load and maximum axial strain of samples are recorded.

However, Bakhteri [7] studies revealed, as this method of drainage could cause serious damage to the safety of the structure. Columns constructed with holes (embedded PVC drainpipes), not only have reduced load carrying capacities but also, could be very dangerous to the safety of the entire building structure and can reduce its useful life significantly. The study also discussed some results of a research carried out to investigate the compressive strength and load carrying capacity of rectangular reinforced concrete short columns (models) having rain water pipes inside it. Figure 2.2 shows a typical column with drain pipe positioned at the center of its cross-section. The cross-sectional dimensions of the column are represented by h and b , where its height is represented by l .

To summarize this paper all the results show reduction in load carrying capacities of the columns by positioning drain pipes inside them. It is clear that; the hole locations at critical section, the hole dimensions (sizes), and the types of the columns have predominant effect on the load carrying capacity and strength of the columns. But nothing has been said about above mentioned parameters of study on capacity of the columns. In this paper the extensive study has been under taken based on location of ducts at critical sections and size of the ducts for electric wiring other than drain pipes. It has also been mentioned that in the case of structure subjected to lateral load, this practice should not be used because appropriate determination of the effective depth of the column concealing rain water pipe is difficult.

Mander [18] investigated the flexural strength and ductility of rectangular and circular hollow RC columns. The flexural strength and ductility available from circular hollow reinforced concrete column sections with two layers of longitudinal and transverse reinforcement placed near both the inside and outside faces of the section and tied through the wall thickness by transverse reinforcement.

The core of the tube walls was well confined by the reinforcement, when properly detailed, and the columns performed in a ductile manner during cyclic lateral loading. Such columns, when extensively studied, were shown to perform an inelastic behavior in cyclic lateral loading. Inoue et al. [14] focused on the deterioration of concrete shear resistance for hollow column. They concluded that the reduction in concrete shear resistance should be considered in the design of RC columns having hollow sections.

But they don't extend their finding to columns under purely axial loading rather than lateral loading cyclical. In this study, lateral loading is not considered as parameters of study, because the columns under study are short braced columns types, in which lateral loads is resisted by additional structures like shear walls. In addition to this, One layer of longitudinal and transverse reinforcement placed near outside face of the section and the inside face of the wall tube is supported by PVC conduit which does not allow the leak of concrete pieces inside the wall tube, indirectly fostering the shear capacity of the columns by making smooth configuration between aggregates and confine the wall thickness properly.

Yeh et al. (2001) [25] conducted a test for circular hollow columns of which main variables were the aspect ratio (column height/diameter of section), connection details of longitudinal steel, amount of transverse reinforcement, and arrangement of cross ties. They tested three specimens with the following dimensions: column lengths of 5,500 mm and 3,500 mm, outside diameter of 1,500 mm, and inside diameter of 900 mm, and hollow ratio of 0.6. The experiment showed that the failure behavior and ductility capacity varied according to the aspect ratio, presence of lap-splice of the longitudinal reinforcement, amount of transverse reinforcement, and shape of cross-ties.

As per to this study, nothing has been discussed about location of the holes through the columns and the effect of cross-sectional dimensions change on columns capacity. And also the study is limited to the one type of the columns (circular) only.

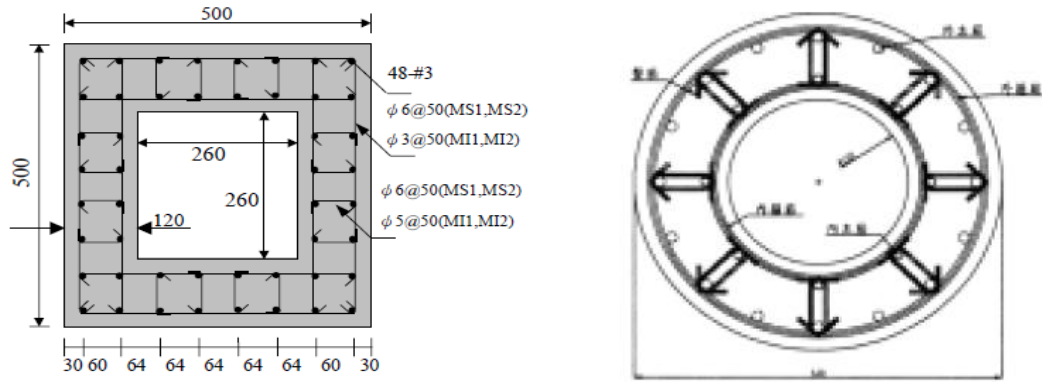
Hoshikuma et al. (2000) [12] performed an experiment of circular hollow columns with single layer of longitudinal reinforcement. They also describe as hollow section columns take the advantage of reduced concrete mass to reduce the inertial response of the structures, which act as vertical beams spanning between foundation and superstructure, and also reduce the tendency for thermally- induced cracking at an early age resulting from heat-of-hydration temperature variations.

They tested two specimens with the following dimensions: a column length of 6,528 mm, outside diameter of 1,524 mm, inside diameter of 1,245 mm, and hollow ratio of 0.82. They defined column failure as the state when the concrete strain reached 0.005 at the inside face of the wall section, and concluded that the neutral axis was located inside the hollow when the hollow ratio and longitudinal reinforcement ratio were large enough.

In reality, the position of the neutral axis directly influences the strain distribution of the columns with the holes, but it is not enough criterions to say something about capacity of the columns of this types. The study concentrates one the one type column it doesn't discuss the different types of the columns like rectangular and square columns. In the context of this paper square columns and rectangular with the circular holes is discussed in detail by comparing their compressive strength and their load carrying capacity with their respective control specimen.

Y. L. Mo, Y.-K. Yeh, C.-T.Cheng, I. C. Tsai and C. C. Kao [24], worked on hollow bridge column. To maximize structural efficiency in terms of the strength/mass and stiffness/mass ratios and to reduce the mass contribution of the column to seismic response, it is desirable to use a hollow section with high-strength concrete for the columns. To consider both ductility and workability, the configuration of lateral steel in the hollow columns used in the high-speed rail project of Taiwan. The purpose of this integrated project is five points:

- I.** To study experimentally the flexural, shear, and retrofit performances of hollow rectangular and circular reinforced concrete bridge columns.
- II.** To investigate the constitutive models of confined concrete as affected by the configuration, spacing and steel ratio of confinement reinforcement, as well as the compression/shear ratio
- III.** To evaluate the reduction of shear strength with increasing ductility factor and develop a set of guidelines for seismic design, retrofit, and repair of such columns
- IV.** To extend the applicability of the truss model for shear element to include such columns to develop a computer program that can predict the shear stress-shear strain relationship of hollow bridge columns with confined reinforcement. Find a good method for seismic repair of such columns and develop a computer program that can perform seismic design, retrofit, and repair of such columns.



(a) Hollow bridge columns with circular section (b) hollow bridge columns with Rectangular-section

Figure 2.17 Configurations of lateral reinforcement in hollow columns [24]

The above mentioned investigation shows that, when a specimen with insufficient shear reinforcement was FRP retrofitted, the failure mode of the specimen could change from brittle shear failure to ductile flexural failure.

The specimen, with both insufficient lateral reinforcement and lap splice at the plastic hinge region, should not be used in design because it has much lower ductility due to premature bond failure. The specimen, with insufficient lateral reinforcement, has flexure and shear failures. Its ductility is less than that of specimen failed by flexure because its plastic hinge could not be fully developed. The ultimate displacements and flexural capacities of FRP-repaired columns were about 50% to 60% and 10% to 20%, respectively, greater than those of original columns.

In the column analyses there were three cases, namely, Case 1 with code required confinement reinforcement, Case 2 with 50% of code required confinement reinforcement, and Case 3 with 50% of code required confinement reinforcement and the remaining 50% replaced by steel plate. In Case 1, the ductility capacity was very good, and increased with increasing confinement reinforcement. The shear strength was less than that calculated by Priestley's formula. In Case 2, both the ductility capacity and the shear strength were reduced dramatically. In Case 3, both the ductility capacity and the shear strength were not less than those for Case 1.

CHAPTER THREE

MATERIALS, AND RESEARCH METHODOLOGY.

3.1 Introduction

The implementation of nonlinear material laws in finite element analysis codes is generally tackled by the software development industry in one or more ways. A number of powerful finite element software packages have been becoming commercially available. They include ABAQUS, ANSYS, COSMOSM, MARC and LUSAS. In order to meet the objectives of this research, the reinforced concrete column with hole was modeled using ABAQUS version 6.13-1. The research significantly reveals the effect of introducing longitudinal holes of different size at different position through the columns.

This chapter explains the process to build the model and the process to run the nonlinear analysis using ABAQUS software.

3.2 Finite Element Modeling

The technique of finite element modeling lies in the development of a suitable mesh arrangement. Thus, many trial models were created during the early stage of the research. Different element types and different methods of modeling were created before the final arrangement of mesh and the element was determined. In the following sections, the procedures for the implementation of a FEM for nonlinear analysis are described.

3.2.1 Geometry

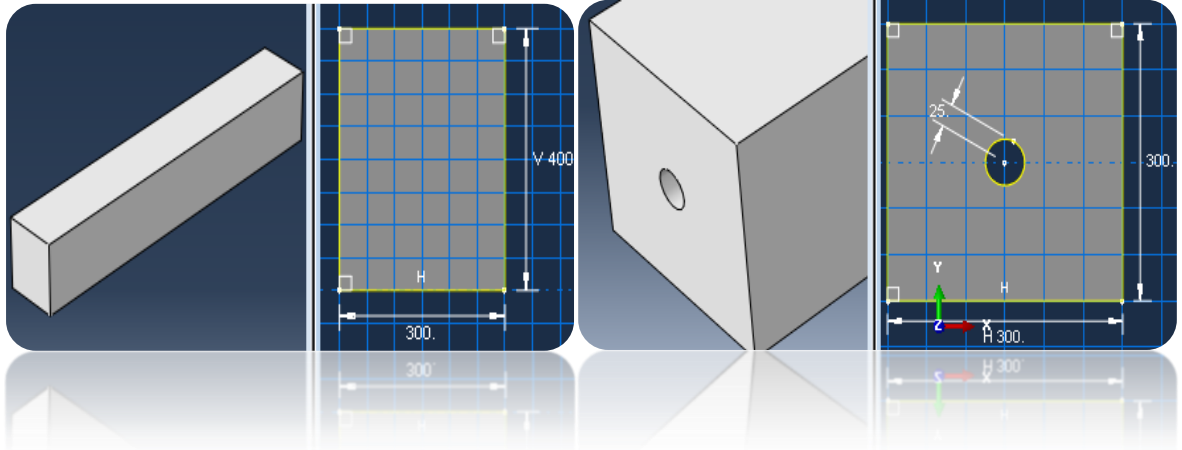
Reinforced concrete square columns of different hole size with different placement of the longitudinal hole through the columns and reinforced concrete rectangular columns of the same types are modeled using ABAQUS software. Two sets of columns are modeled with the same size of hole and reinforcement.

Each set contains seven columns with their respective controller column (solid). The first sets are square columns while the second sets are rectangular columns. In addition to this all column types contains two sub-models.

- ✓ Type (a): Model without hole (control model)
- ✓ Type (b): Model with hole

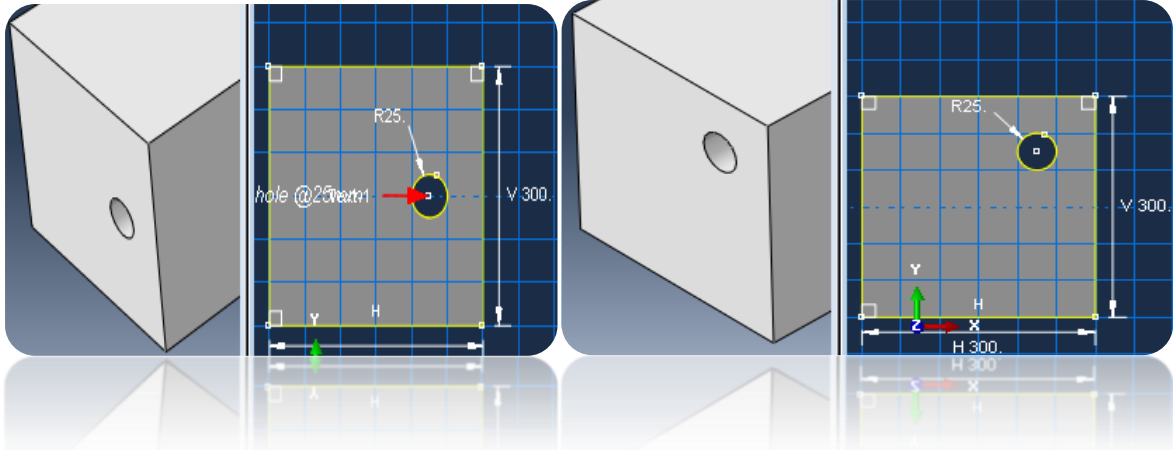
The dimensions of the column's cross-section are (300×300) mm for square columns and (300×400) mm for rectangular columns. All models were placed with 30 mm concrete cover to reinforcement. The longitudinal reinforcement rebar's comprised 4#14mm in diameter, and the transverse reinforcement consisted of shear links, 8mm in diameter@200mm.

In addition to above motioned geometers ,the analytically rigid plate is introduced to both end of the column to increase the moment in whole models so that ,the capacity of the column is more enhanced. The dimensions of the plate are (400×400) mm for the square columns, and (400×500) mm for rectangular columns. The holes having two different diameters (20, and 50 mm) were modeled. The following figure clearly depicts the exact drawing of the model by indicating the exact position of the holes in the models.



a) Solid column with no hole

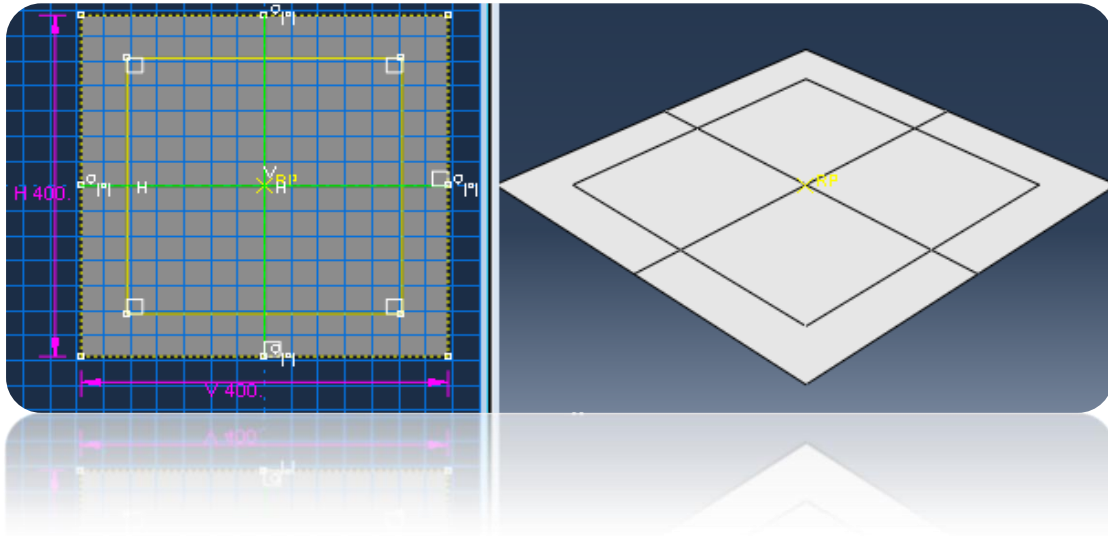
b) column with the hole @center



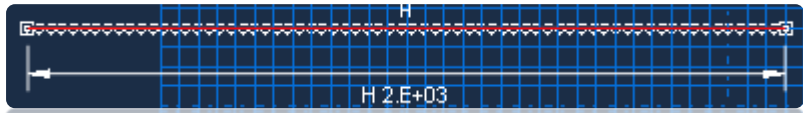
c)hole@one quarter from one side and one half from another sides

d) hole@quarter from both sides

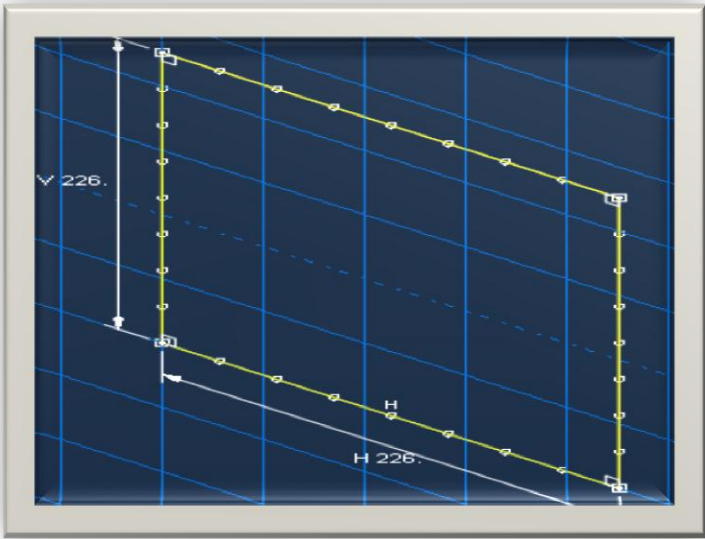
figure 3.1 the position of the hole in the models for both rectangular and square columns



e) Geometer of the plate with its dimension



f) Geometry of longitudinal reinforcement



g) Geometry of stirrup

Figure 3.2 geometrical descriptions of plate, rebar, and stirrup

The geometry of the models specification and dimensions of each set are shown in table 3.1.

Table 3.1: The **cases** under study and dimension of the column models

position of the hole and cases of study	Typical columns	H(mm)	B(mm)	Gross Area (mm ²)	Hole Diameter (mm)	Hole Area (mm ²)	Bar size (mm)	Stirrup size (mm)
Solid square columns	solid	300	300	90000	0	0	4Y14	10Y8
@center of columns	Cs-1	300	300	90000	20	314.159	4Y14	10Y8
	Cs-3	300	300	90000	50	1963.495	4Y14	10Y8
@0.25*B from H and 0.5*H from B	Cs-1	300	300	90000	20	314.159	4Y14	10Y8
	Cs-3	300	300	90000	50	1963.495	4Y14	10Y8
@0.25*B from H and 0.25*H from B	Cs-1	300	300	90000	20	314.159	4Y14	10Y8
	Cs-3	300	300	90000	50	1963.495	4Y14	10Y8
Solid rectangular columns	solid	300	400	120000	0	0	4Y14	10Y8
@center of columns	Cs-1	300	400	120000	20	314.159	4Y14	10Y8
	Cs-3	300	400	120000	50	1963.495	4Y14	10Y8
@0.25*B from H and 0.5*H from B	Cs-1	300	400	120000	20	314.159	4Y14	10Y8
	Cs-3	300	400	120000	50	1963.495	4Y14	10Y8
@0.25*B from H and 0.25*H from B	Cs-1	300	400	120000	20	314.159	4Y14	10Y8
	Cs-3	300	400	120000	50	1963.495	4Y14	10Y8

Geometry of the reinforcement bars are greatly influence column's geometry for design. The reinforcement bars have a circular cross section and hence, they are modeled with a circular cross-section. Even though, they have miniscule effect on the studied behavior of the column stirrups are also modeled with circular cross-section.

3.2.2 Finite Element method (Discretization)

Finite elements and nodes define the basic geometry of the physical structures being modeled in ABAQUS. Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements; that is, which nodes belong to which elements included the geometry of models. The collection of all the elements and nodes in a model is called the *mesh*. Generally, the mesh will be only an approximation of the actual geometry of the structure. The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation.

The Finite Element Method (FEM) is a numerical technique used to perform Finite Element Analysis (FEA) of any given physical phenomenon. The description of the laws of physics for space- and time-dependent problems are usually expressed in terms of nonlinear relationships which cannot be solved with analytical methods, instead, an approximation can be constructed, typically based upon different types of discretization. These discretization methods approximate the real problem to the numerical models and the finite element method (FEM) is used to compute such approximations.

Reasons of nonlinear analysis in structural engineering include:- Strength analysis, Stability analysis, Progressive failure analysis, Safety and serviceability assessment of existing infrastructure, A shift towards high performance materials and more efficient utilization of structural components, Direct use of NFEA in design for both ultimate load and serviceability limit states, Simulation of materials processing and manufacturing, to understand basic structural behavior, to test the validity of proposed „material models“.

In solving the Nonlinear Problems; external and internal forces have to be in balance to satisfy equilibrium, consideration to the solution of nonlinear equilibrium equations for prescribed values of the load or time parameter have to be taken, the problem consists of finding the displacement vector which produces an internal force vector.

The most frequently used solution procedures for NFEA consists of a predictor step involving forward Euler load incrementation and a corrector step in which some kind of Newton iterations are used to enforce equilibrium. The incremental-iterative procedure that advances the solution while satisfying the global equilibrium equations at each iteration „i“, within each time (load) step „n+1“, is governed by the incremental equations. Newton's method is the most rapidly convergent process for solution of problems in which only one evaluation of the residual is made in each iteration. Indeed, it is the only method, provided that the initial solution is within the ball of convergence, in which the asymptotic rate of convergence is quadratic.

Convergence criteria; a convergence criterion measures how well the obtained solution satisfies equilibrium. In NFEA of the convergence criteria are usually based on some norm of the Displacements (total or incremental), Residuals, and Energy (product of residual and displacement).

Divergence criteria; if the equilibrium path includes critical points negative load increments must be prescribed to go beyond limit points. If the load increments are too large such that the solution falls outside the ball of convergence analysis may fail to converge.

3.2.3 Abaqus modeling and analysis procedures

Overview

Abaqus/CAE is the Complete Abaqus Environment that provides a simple, consistent interface for creating Abaqus models, interactively submitting and monitoring Abaqus jobs, and evaluating results from Abaqus simulations. A complete Abaqus analysis usually consists of three distinct stages: preprocessing, simulation, and post processing.

Preprocessing (Abaqus/CAE): - In this stage we must define the model of the physical problem and create an Abaqus input file. The model is usually created graphically using Abaqus/CAE or another preprocessor, although the Abaqus input file for a simple analysis can be created directly using a text editor.

Simulation (Abaqus/Standard): - The simulation, which normally is run as a background process, is the stage in which Abaqus/Standard solves the numerical problem defined in the model. Examples of output from a stress analysis include displacements and stresses that are

stored in binary files ready for post processing. Depending on the complexity of the problem being analyzed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run.

Post processing (Abaqus/CAE): - In this stage we can evaluate the results once the simulation has been completed and the displacements, stresses, or other fundamental variables have been calculated. The evaluation is generally done interactively using the Visualization module of Abaqus/CAE or another postprocessor. The Visualization module, which reads the neutral binary output database file, has a variety of options for displaying the results, including color contour plots, animations, deformed shape plots, and X - Y plots. Abaqus/CAE is divided into modules, where each module defines a logical aspect of the modeling process. The order of the modules in the menu corresponds to a logical sequence we may follow to create a model. In many circumstances we must follow this natural progression to complete a modeling task. A completed model contains everything that Abaqus needs to start the analysis.

Abaqus/CAE uses a model database to store our models. The following list of the modules available within Abaqus/CAE briefly describes the modeling tasks we can perform in each module. The order of the modules in the list corresponds to the order of the modules in the context bar's **Module** list. Each module contains only those tools that are relevant to a specific portion of the modeling task.

Creating parts; The Part module allows you to create individual parts by sketching their geometry directly in Abaqus/CAE or by importing their geometry from other geometric modeling programs.

Defining property; a section definition contains information about the properties of a part or a region of a part, such as a region's associated material definition and cross-sectional geometry. In the Property module we have to create section and material definitions and assign them to regions of parts. Material modeling for reinforce concrete in this study is defined following the Concrete Damage-Plasticity Model.

In the study, the concrete damaged plasticity model (CDP) is used which is implemented in this program. It is because the CDP is currently one of the most popular concrete models used for simulation of concrete behavior prediction for general stress cases in Abaqus.

Some parameters of CDP are decisive to obtain proper and realistic results. These parameters are: the viscosity parameter, the dilation angle in p-q plane, the flow potential eccentricity, the ratio of initial biaxial compressive yield stress to initial uniaxial compressive yield stress.

The concrete damaged plasticity model in Abaqus: provides a general capability for modeling concrete in all types of structures, uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete; used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures; can be used with rebar to model concrete reinforcement; is designed for applications in which concrete is subjected to monotonic, cyclic, and/or dynamic loading under low confining pressures;

consists of the combination of non-associated multi-hardening plasticity and scalar (isotropic) damaged elasticity to describe the irreversible damage that occurs during the fracturing process; can be used in conjunction with a viscoelastic regularization of the constitutive equations in Abaqus/Standard to improve the convergence rate in the softening regime; and requires that the elastic behavior of the material be isotropic and linear.

3.2.4 Material Properties

Material properties for all parameters of study must be specified. High-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the abaqus results are limited by the accuracy and extent of the material data. Material nonlinearity occurs when the stress- strain relationship ceases to be linear by plastic yielding and strain hardening. Materially nonlinear effects arise from nonlinear constitutive model (that is, progressively disproportionate stresses and strains). Common examples of nonlinear material behavior are the plastic yielding of metals, the ductile fracture of granular composites such as concrete or time-dependent behavior such as creep ABAQUS incorporates a variety of nonlinear constitutive models, covering the behavior of the more common engineering materials.

Material natural properties under load and their combinations are accurately modeled. Both the codes considered here for comparison material modeling is, however, with full of assumptions for simplicity of manual analysis. Accordingly concrete and reinforcing steel materials mechanical properties are modeled as follows

Numerical Model for Stress-Strain Curve in Compression

Modified Hognestad model for uniaxial unconfined stress-strain relationship is universally accepted for normal grade of concrete. However, several models are proposed by various researchers for high strength concrete but none of these is universally accepted as standard model. This is mainly due to the variation in test conditions and concrete grades. In present paper, applicability of few of these models is evaluated by comparing experimental values and behavior predicted by these models.

According to Hognestad approach;

$$\begin{aligned} \text{For } 0 < \varepsilon < \varepsilon_o, \frac{\sigma}{\sigma_{cu}} &= 2 \frac{\varepsilon}{\varepsilon_o} \left(1 - \frac{\varepsilon}{2\varepsilon_o}\right) \\ \text{For } \varepsilon_o < \varepsilon < \varepsilon_{cu}, \frac{\sigma}{\sigma_{cu}} &= 1 - 0.15 \left(\frac{\varepsilon - \varepsilon_o}{\varepsilon_{cu} - \varepsilon_o}\right) \end{aligned} \quad [3.1]$$

For low grade concrete the ES EN 1992:2015 material model works resemble with the Abaqus requirement (-1-1 2004).

$$\frac{\sigma_c}{f_{cm}} = \frac{k\eta - \eta^2}{1 + (k - 2)\eta} \quad [3.2]$$

Where,

$$k = 1.05 E_{cm} | \varepsilon_{c1} | / f_{cm}, \text{ and } \eta = \frac{\varepsilon_c}{\varepsilon_{c1}}$$

It is assumed that the uniaxial stress-strain curves can be converted into stress versus plastic-strain Curves in Abaqus for concrete damage plasticity model. Compressive stress data are provided as a tabular function of inelastic (or crushing) strain.

Positive (absolute) values should be given for the compressive stress and strain. The stress-strain curve can be defined beyond the ultimate stress, into the strain-softening regime. Hardening data are given in terms of an inelastic strain, instead of plastic strain.

The compressive inelastic strain is defined as the total strain minus the elastic strain corresponding to the undamaged material, where, Dc and Dt determination in CDPM is not included in the manual.

It is additionally supposed, that a range of concrete elasticity is 40% of the ultimate compression capacity and 70% of the tension. Unloading data are provided to Abaqus in terms of compressive damage curves, as discussed below. Abaqus automatically converts the inelastic strain values to plastic strain values using the relationship.

$$\varepsilon_c^{\sim pl} = \varepsilon_c^{\sim in} - \frac{d_c}{(1-d_c)} \frac{\sigma_c}{E_0} \quad [3.3]$$

Abaqus will issue an error message if the calculated plastic strain values are negative and/or decreasing with increasing inelastic strain, which typically indicates that the compressive damage curves are incorrect. In the absence of compressive damage inelastic strain=plastic strain.

The nominal (engineering) stresses from experimental results, Hognestad model in our case, shall be converted into effective (true) stress according to ABAQUS plastic material modeling manual.

$$\sigma_{true} = \sigma_{nom} (1 + \varepsilon_{nom}) \quad [3.4]$$

$$\sigma_{true} = \ln(1 + \varepsilon_{nom}) \quad [3.5]$$

$$\varepsilon_{ln}^{pl} = \ln(1 + \varepsilon_{nom}) - \frac{\sigma_{true}}{E} \quad [3.6]$$

$$\text{Where, } E = 500\sqrt{\sigma_{cu}} \quad [3.7]$$

Numerical Model for Tensile Behavior of Concrete

Concrete is assumed to follow a linear ascending branch up to the tensile strength σ_{tu} . The strain at cracking stress is given as (Teng, Jiang et al. 2009):

$$\varepsilon_{cr} = \frac{\sigma_{tu}}{E} \quad [3.8]$$

Cracks are considered smeared within the reinforced concrete, average tensile stresses

after reaching σ_{tu} can be calculated due to tension stiffening

Tensile stress-strain relationship was assumed to be linear up to the uniaxial tensile strength and then determined using the exponential function in.

$$\sigma = f_t \left(\frac{\varepsilon_t}{\varepsilon} \right)^{(0.7+1000\varepsilon)} \quad [3.9]$$

Where, $\varepsilon_t = \frac{f_t}{E_c}$, which approximately match with ABAQUS graph,

Table 3.2 Material data input (concrete properties)

Concrete grade		C25	Concrete Plasticity Parameters	
Concrete density		2.4E-009		
Concrete elasticity			γ (Eccentricity)	0.1
E(GPa)		29.00	β (dilation angle)	36°
ν		0.20	K	0.667
			$\sigma_{bo} / \sigma_{co}$	1.16
			Viscosity parameters	0.00001

concrete damaged plasticity

Compression damage		Tension damage	
Damage Parameter(dc)	Inelastic Strain	Damage Parameter(dt)	Displacement
0.000000000	0.000000000	0.000000000	0.00
0.000000000	0.000224404	0.115065799	0.01
0.000000000	0.000502312	0.230131597	0.02
0.000000000	0.000866458	0.345197396	0.03
0.000000000	0.001035466	0.460263195	0.04
0.013483333	0.001303392	0.575328994	0.05
0.092174414	0.001802322	0.800000000	0.07

0.232536864	0.002354489	0.800341287	0.07
0.426249807	0.002952712	0.807532899	0.08
0.452885733	0.003027705	0.814724512	0.09
		0.821916124	0.10
		0.829107737	0.11
		0.836299349	0.12
		0.843490961	0.13
		0.850682574	0.14
		0.857874186	0.15
		0.865065799	0.16
		0.872257411	0.17
		0.879449024	0.18
		0.886640636	0.19
		0.893832248	0.20
		0.901023861	0.21
		0.908215473	0.22
		0.915407086	0.23
		0.922598698	0.24
		0.929790311	0.25

compressive behavior

Yield stress	Inelastic strain
11.042148	0.000431
18.465033	0.000862
22.898635	0.001293
24.834751	0.001724
25.000000	0.001898
24.662917	0.002155
22.695636	0.002586
19.186574	0.003017
14.343758	0.003448
13.677857	0.003500

tensile behavior

Yield stress	Cracking strain
--------------	-----------------

1.983446706	0.00
1.755219826	0.01
1.526992947	0.02
1.298766067	0.03
1.070539188	0.04
0.842312309	0.05
0.396689341	0.07
0.396012417	0.07
0.381748237	0.08
0.367484057	0.09
0.353219877	0.10
0.338955697	0.11
0.324691517	0.12
0.310427337	0.13
0.296163157	0.14
0.281898977	0.15
0.267634797	0.16
0.253370617	0.17
0.239106437	0.18
0.224842257	0.19
0.210578077	0.20
0.196313897	0.21
0.182049717	0.22
0.167785537	0.23
0.153521357	0.24
0.139257177	0.25

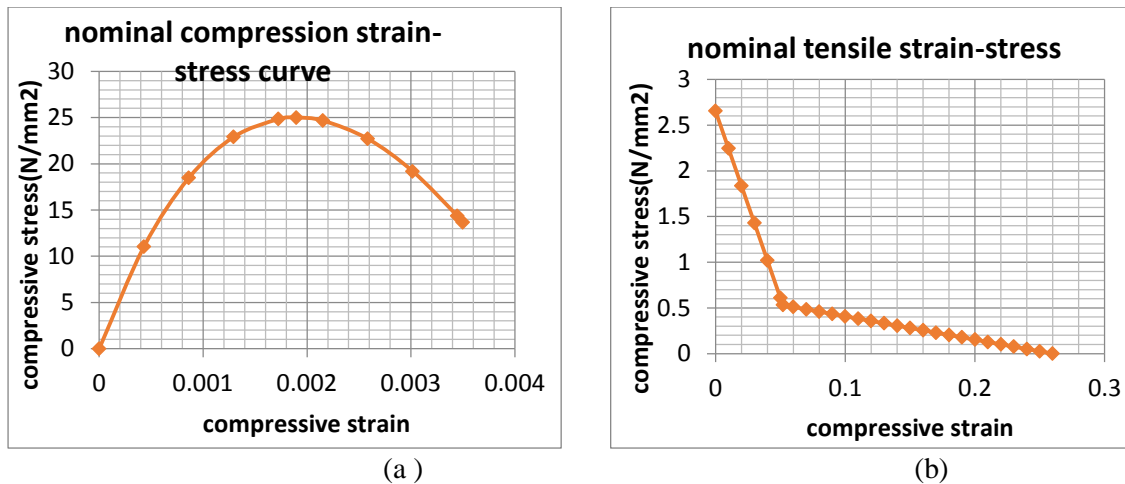


Figure 3.3 Nominal concrete stress-strains for (a) compression and (b) tension

Uniaxial stress-strain model for steel reinforcement in tension

The reinforcing steel is defined as an isotropic material. In the plastic properties, the stress potential model was selected and Von Mises yield surface was tried for the plastic behavior of the steel. In Abaqus reinforcement in concrete structures is typically provided by means of rebars, which are one-dimensional rods that can be defined singly or embedded in oriented surfaces. Rebars are typically used with metal plasticity models to describe the behavior of the rebar material and are superposed on a mesh of standard element types used to model the concrete (Simulia 2013)).

There are many different ways of modeling reinforcement in Abaqus; for example as smeared reinforcement in the concrete, as cohesive elements, with built-in rebar layer which is available in certain element types and as discrete truss or beam elements with the embedded region constraint. All the analyses in this study used the embedded region alternative and therefore this section will only explain how to model reinforcement in this manner. The material is defined as an ideal elasto-plastic material, i.e. no hardening behavior for those experiments that doesn't have the fracture tensile strength and including the hardening behavior if ultimate tensile strength was available. This is done by using the elastic and plastic material models in Abaqus. The input parameters should be set according to the reinforcement used in the structure. The required input parameters are Young's modulus, Poisson's ratio, yield strength and the ultimate strength with ultimate strain recommended according to ES EN 1992. Normally it is sufficient to model the reinforcement with truss elements, for which the only input parameter needed is the cross-sectional area of the reinforcement.

The benefit of beam elements is that they can account for the dowel effect, which only gives a small increase of the load capacity in the structure. As the beam elements require more input parameters and also more computational power this alternative is not recommended and not selected for this study.

Reinforcement data which are not fully available on literatures referred for validation in their experiment I refer ES EN 1992:2015 reinforcement material model, metal plasticity theory, seems either like metal hardening or perfectly plastic after elastic behavior. The value of K, ratio of fracture stress and yield stress, determine the ultimate strain capacity of the reinforcement.

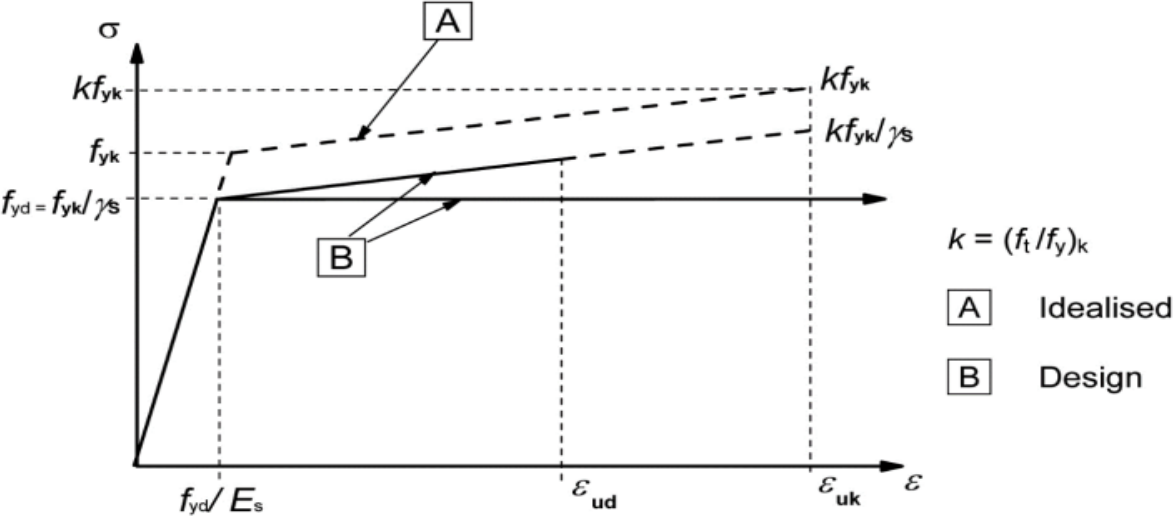


Figure 3.4 Reinforcement bar stress-strain relation according to ES EN 1992

Table 3.3 below shows the steel properties for the longitudinal reinforcements

Density steel bar	= 2.4E-008	
Elastic Properties	Young's Modulus=200000 N/mm ²	
	Poisson's Ratio=0.2	
Plastic Properties	Yield stress(N/mm ²)	Plastic strain
	300	0.0
	300	0.2

Table 3.4: Transverse reinforcements

Density of stirrup	= 7.8E-009	
Elastic Properties	Young's Modulus= 200000 N/mm ²	
	Poisson's Ratio= 0.3	
Plastic Properties	Yield stress(N/mm ²)	Plastic strain
	300	0.0
	300	0.2

Analytical rigid plate; Parts of structural supporting model which does not need meshing and material properties to be assigned as that of discrete rigid plate, and function as base material for fostering the capacity of the column for the cases understudy.

Assembling parts; when a part is created, it exists in its own coordinate system, independent of other parts in the model. In the Assembly module, it is possible to create instances of the parts as well as instances of other models and to position the instances relative to each other in a global coordinate system, thus creating an assembly. An Abaqus model contains only one assembly.

Defining analysis step; The Step module is used to create and configure analysis steps and associated output requests. The step sequence provides a convenient way to capture changes in a model (such as loading and boundary condition changes); output requests can vary as necessary between steps. A basic concept in Abaqus is the division of the problem history into steps. In its simplest form a step can be just a static analysis in Abaqus/Standard of a load change from one magnitude to another. We can provide a description of each step that will appear in the data (.dat) file. The available analysis procedures are listed below and are described in the sections that follow. Only one procedure is allowed per step.

General analysis steps; an analysis step during which the response can be either linear or nonlinear is called a general analysis step. An analysis step during which the response can be linear only is called a linear perturbation analysis step and has not been utilized in the study because of nonlinearity consideration. A general analysis step is one in which the effects of any nonlinearities present in the model can be included.

Sources of nonlinearity; Nonlinear stress analysis problems can contain up to three sources of nonlinearity: material nonlinearity, geometric nonlinearity, and boundary nonlinearity.

Material nonlinearity: Abaqus offers models for a wide range of nonlinear material behaviors.

Many of the materials are history dependent: the material's response at any time depends on what has happened to it at previous times. Thus, the solution must be obtained by following the actual loading sequence. The general analysis procedures are designed with this in view.

Geometric nonlinearity; It is possible in Abaqus to define a problem as a small-displacement analysis, which means that geometric nonlinearity is ignored in the element calculations the kinematic relationships are linearized. By default, large displacements and rotations are accounted for in contact constraints even if the small-displacement element formulations are used for the analysis; i.e., a large-sliding contact tracking algorithm is used. The elements in a small-displacement analysis are formulated in the reference (original) configuration, using original nodal coordinates. The errors in such an approximation are of the order of the strains and rotations compared to unity.

The alternative to a small-displacement analysis in Abaqus is to include large-displacement effects. In this case most elements are formulated in the current configuration using current nodal positions. Elements therefore distort from their original shapes as the deformation increases. Almost all of the elements in Abaqus use a fully nonlinear formulation.

The calculated stress is the true (Cauchy) stress. For all elements the stress components are given in the global directions unless a local orientation is used at a point. For small-displacement analysis the infinitesimal strain measure is used, which is output with the strain output variable E, strain output specified with output variables LE and NE is the same as with E. Incorporating the effects of geometric nonlinearity in an analysis requires only minor changes to an Abaqus/Standard model. Geometrically nonlinear effects need to make sure that whether **Nlgeom** is turned on or off in step definition.

Boundary nonlinearity; Contact problems are a common source of nonlinearity in stress analysis, other sources of boundary nonlinearity are nonlinear elastic springs, films, radiation, multi-point constraints, etc.

3.2.5 Boundary Condition

The bottom of the column was supported by pin connection in a way that it will restrain the movement in x, y and z directions. The top of the column, which was supported by pin connection, allowed movement in vertical y direction. Thus the restraint at the top of the column was fixed from movement in x and z directions.

3.2.6 Loading

The Load module in Abaqus allows specifying loads, boundary conditions, and predefined fields. Loads and boundary conditions are step-dependent objects, which mean that we must specify the analysis steps in which they are active; some predefined fields are step-dependent, while others are applied only at the beginning of the analysis.

The axial compression load was assigned at the top face of the column. The results of the software analysis were showed that the amount of loading is constant in each column. However, ABAQUS would apply the load in the incremental manner until the analysis stopped. The generated mesh, applied load and boundary condition of the model are shown in Figure 3.5

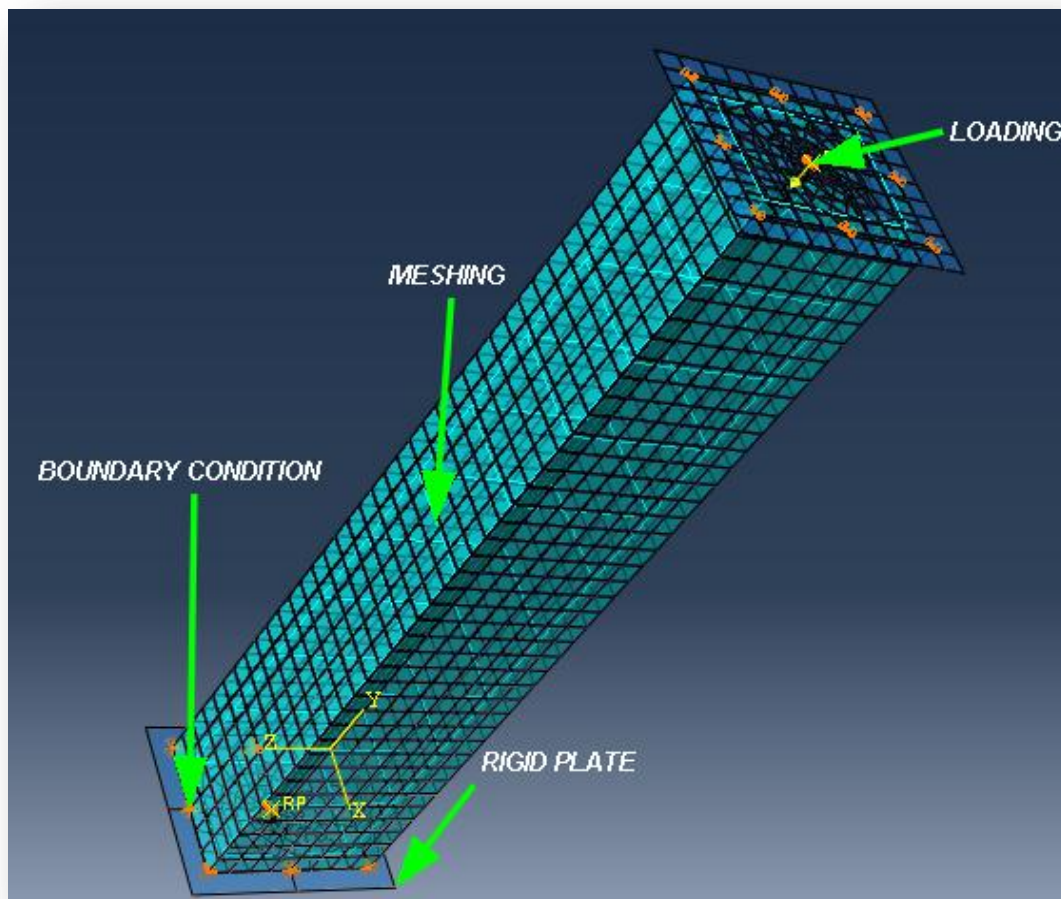


Figure 3.5 Typical model showing generated mesh, loading and boundary condition

3.2.7 Mesh design (techniques)

The Mesh module contains tools that allow us to generate a finite element mesh on an assembly created within Abaqus/CAE. Various levels of automation and control are available so that we can create a mesh that meets the needs of our analysis.

Finite elements and nodes define the basic geometry of the physical structure being modeled in Abaqus. Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements, that is, which nodes belong to which elements comprise the model geometry. The collection of all the elements and nodes in a model is called the mesh. Generally, the mesh will be only an approximation of the actual geometry of the structure.

The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation. The greater the mesh density (i.e., the greater the number of elements in the mesh), the more accurate the results. As the mesh density increases, the analysis results converge to a unique solution, and the computer time required for the analysis increases. The solution obtained from the numerical model is generally an approximation to the solution of the physical problem being simulated.

Abaqus has an extensive element library to provide a powerful set of tools for solving many different problems. Five aspects of an element characterize its behavior: Family, Degrees of freedom (directly related to the element family), Number of nodes, Formulation and Integration. Each element in Abaqus has a unique name, such as T3D2, C3D8I, or C3D20R. The element name identifies each of the five aspects of an element. Figure 3.6 shows the element families that are used most commonly in a stress analysis. One of the major distinctions between different element families is the geometry type that each family assumes.

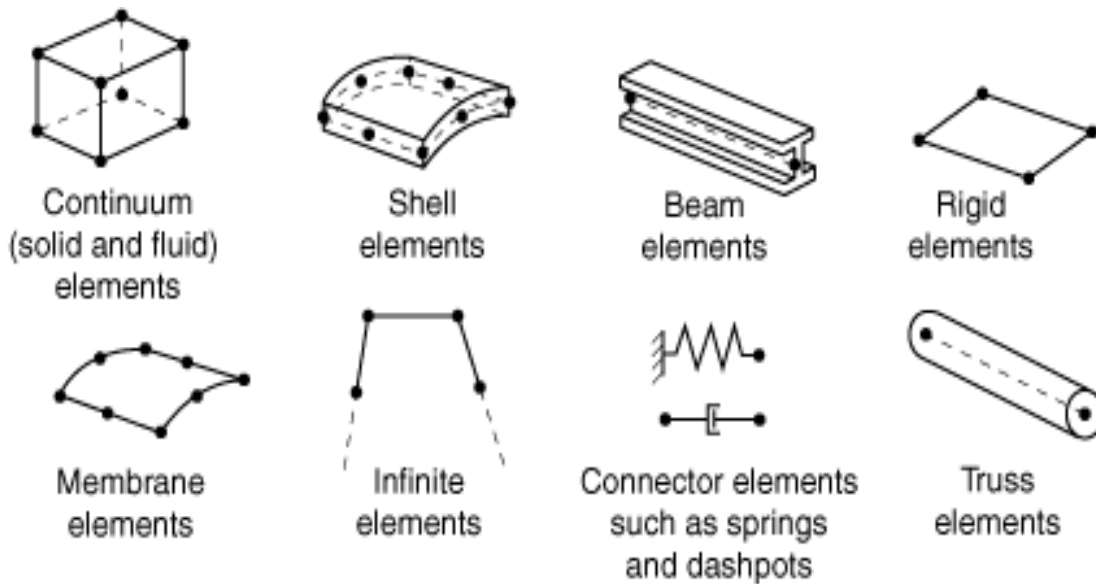


Figure 3.6 commonly used element families.

The degrees of freedom are the fundamental variables calculated during the analysis. Displacements or other degrees of freedom are calculated at the nodes of the element. At any other point in the element, the displacements are obtained by interpolating from the nodal displacements. In Abaqus/Standard elements with midside nodes, such as the 20-node brick shown in Figure 3.7(b) which is implemented for torsion test, use quadratic interpolation and are often called quadratic elements or second-order elements. Modified triangular or tetrahedral elements with midside nodes, such as the 10-node tetrahedron shown in Figure 3.7(c), use a modified second-order interpolation and are often called modified or modified second-order elements.

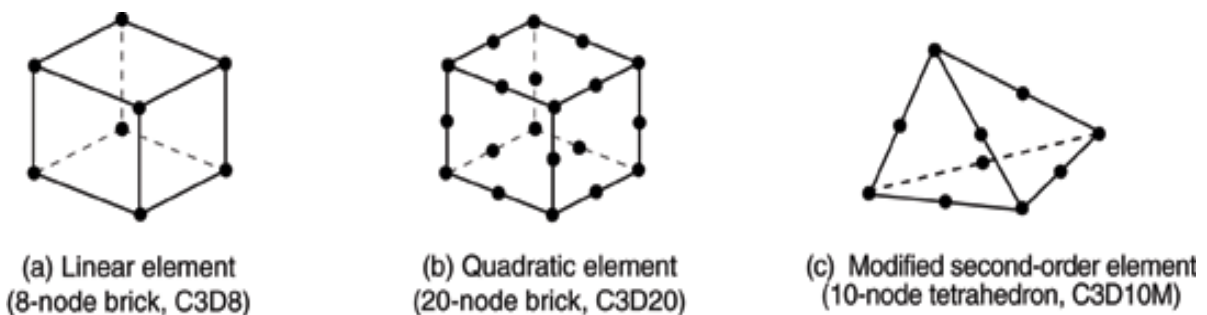


Figure 3.7 linear brick, quadratic brick, and modified tetrahedral elements.

An element's formulation refers to the mathematical theory used to define the element's behavior. In the Lagrangian, or material, description of behavior the element deforms with the material. All stress/displacement elements in Abaqus are based on the Lagrangian formulation.

Some Abaqus/Standard element families have a standard formulation as well as some alternative formulations. Elements with alternative formulations are identified by an additional character at the end of the element name. For example, the continuum, beam, and truss element families include members with a hybrid formulation (to deal with incompressible or inextensible behavior); these elements are identified by the letter H at the end of the name (C3D8H or B31H).

Abaqus uses numerical techniques to integrate various quantities over the volume of each element, thus allowing complete generality in material behavior. Using Gaussian quadrature for most elements, Abaqus evaluates the material response at each integration point in each element. Some continuum elements in Abaqus can use full or reduced integration, a choice that can have a significant effect on the accuracy of the element for a given problem. Abaqus uses the letter R at the end of the element name to label reduced-integration elements.

Using Continuum Elements; the continuum (solid) family of stress/displacement elements is the most comprehensive of the element libraries in Abaqus. The Abaqus/Standard solid element library includes first-order (linear) interpolation elements and second-order (quadratic) interpolation elements in two or three dimensions using either full or reduced integration. Triangles and quadrilaterals are available in two dimensions; and tetrahedral, triangular wedges, and hexahedra (“bricks”) are provided in three dimensions. Modified second-order triangular and tetrahedral elements are also provided. In addition, hybrid and incompatible mode elements are available in Abaqus/Standard. The thought of choosing which of these elements is best for the model may seem daunting, especially at first.

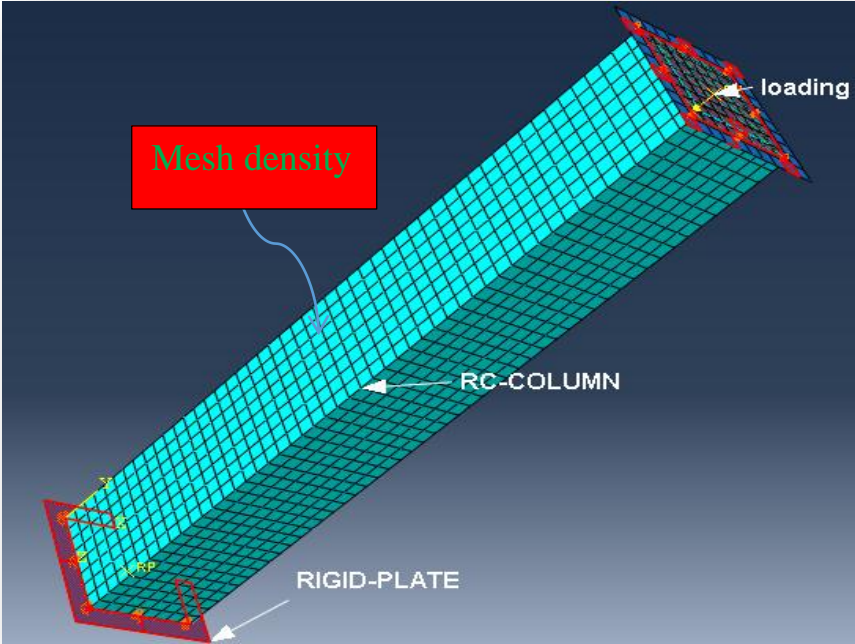
Selecting continuum elements; the correct choice of element for a particular simulation is vital, if accurate results are to be obtained at a reasonable cost. The following recommendations apply to Abaqus/Standard (Abaqus Manual 2013): Minimize the mesh distortion as much as possible. Use a fine mesh of linear, reduced-integration elements for simulations involving very large mesh distortions (large-strain analysis). In three dimensions use hexahedral (brick-shaped) elements wherever possible. They give the best results for the minimum cost. Complex geometries can be difficult to mesh completely with hexahedrons; therefore, wedge and tetrahedral elements may be necessary. The linear versions of these elements, C3D4 and C3D6, are poor elements (fine meshes are needed to obtain accurate results); as a result, these elements should generally be used only when necessary to complete a mesh, and, even then, they should be far from any areas where accurate results are needed.

The quadratic tetrahedral elements in Abaqus/Standard (C3D10 or C3D10HS) are suitable for general usage; but when used with contact, they should be used only with the surface-to-surface contact discretization. An alternative to these elements is the modified quadratic tetrahedral element (C3D10M) available in both analysis products. This element is robust for large-deformation problems and contact problems using either the traditional node-to-surface or the surface-to-surface contact discretization and exhibits minimal shear and volumetric locking. With either type of element, however, the analysis will take longer to run than an equivalent mesh of hexahedral elements. You should not use a mesh containing only linear tetrahedral elements (C3D4): the results will be inaccurate unless you use an extremely large number of elements. Abaqus provides an advanced feature, called sub modeling that allows to obtain more detailed (and accurate) results in a region of interest in the structure. The solution from a coarse mesh of the entire structure is used to drive a detailed local analysis that uses a fine mesh in this region of interest.

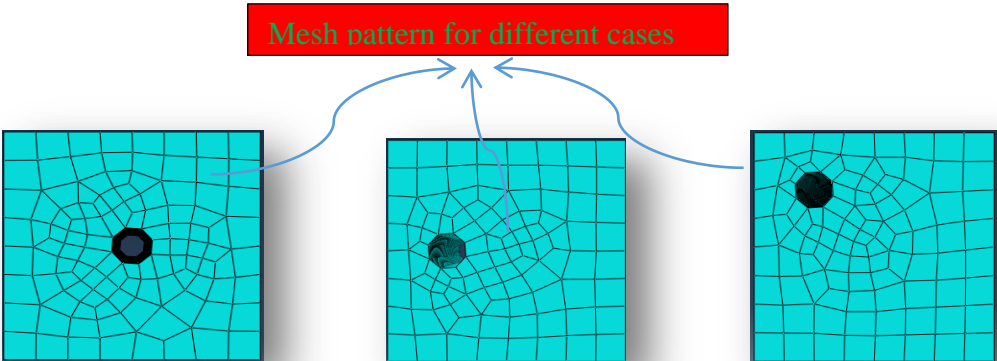
Truss elements; Truss elements are rods that can carry only tensile or compressive loads. They have no resistance to bending; therefore, they are useful for modeling pin-jointed frames. Moreover, truss elements can be used as an approximation for cables or strings (for example, in a tennis racket). Trusses are also sometimes used to represent reinforcement within other elements like concrete. All truss element names begin with the letter “T.” The next two characters indicate the dimensionality of the element “2D” for two-dimensional trusses and “3D” for three-dimensional trusses. The final character represents the number of nodes in the element. Truss elements have only translational degrees of freedom at each node. Three-dimensional truss elements have degrees of freedom 1, 2, and 3, while two-dimensional truss elements have degrees of freedom 1 and 2. All truss elements must refer to a truss section property that associates a material property definition with the element and specifies its cross-sectional area. In addition to the standard formulation, a hybrid truss element formulation is available in Abaqus/Standard. It is useful for modeling very stiff links whose stiffness is much greater than that of the overall structure. Axial stress and strain are available as output for truss elements. Based on Abaqus manual recommendations and many trials of using different mesh convergence issues the following elements were selected for analysis.

In order to obtain accurate results from the FE model, all the elements in the model were purposely assigned the same mesh size to ensure that each of two different materials shares the same node. The type of mesh selected in the model is structured.

The mesh element for modeling of concrete and steel loading and bearing plates is 8-node brick element with three translation degrees of freedom at each node with reduced integration (C3D8R). The reinforcement bar is being defined by truss element which is called T3D2 (3Dimensional-2Node truss element) for analysis. And the mesh element for modeling of concrete is 8-node brick element with three translation degrees of freedom at each node with reduced integration (C3D8R). The reinforcement bar is being defined by truss element which is called T3D2 (3Dimensional-2Node truss element) for analysis. The figure 3.8 depicts the mesh pattern and mesh density of the the columns under studies.



Case (a)-solid column in 3D



Case (b)

Case(c)

Case (d)

Figure 3.8 mesh pattern in different cases

3.3 Nonlinear analysis

The analytical investigation carried out here is conducted on 14-RC columns; all columns are raised in vertical position with a vertical load on top surface. Analytical rigid plate is provided at both ends of the column to decrease the rotational effect of the column so that it will enhance the capacity of the column. In nonlinear analysis, the load applied to a finite element model is divided into a series of load increments called load step. Nonlinear analysis control properties were defined as properties of a load case. At the completion of each load increment, the stiffness matrix of the model is adjusted to reflect the nonlinear changes in the structural stiffness before proceeding to the next load increment.

Abaqus/Standard uses the Newton-Raphson method to obtain solutions for nonlinear problems. In a nonlinear analysis the solution usually cannot be calculated by solving a single system of equations, as would be done in a linear problem. Instead, the solution is found by applying the specified loads gradually and incrementally working toward the final solution. Therefore, Abaqus/Standard breaks the simulation into a number of load increments and finds the approximate equilibrium configuration at the end of each load increment. It often takes Abaqus/Standard several iterations to determine an acceptable solution to a given load increment. The sum of all of the incremental responses is the approximate solution for the nonlinear analysis. Thus, Abaqus/Standard combines incremental and iterative procedures for solving nonlinear problems.

Steps, increments, and iterations

It is important to clearly understand the difference between an analysis step, a load increment, and iteration. The load history for a simulation consists of one or more steps. Define the steps, which generally consist of an analysis procedure option, loading options, and output request options.

An increment is part of a step. In nonlinear analyses the total load applied in a step is broken into smaller increments so that the nonlinear solution path can be followed. At the end of each increment the structure is in (approximate) equilibrium and results are available for writing to the output database, restart, data, or results files. The increments at which we select results to be written to the output database file are called frames.

Iteration is an attempt at finding an equilibrium solution in an increment when solving with an implicit method.

If the model is not in equilibrium at the end of the iteration, Abaqus/Standard tries another iteration. With every iteration the solution Abaqus/Standard obtains should be closer to equilibrium; sometimes Abaqus/Standard may need much iteration to obtain an equilibrium solution. When an equilibrium solution has been obtained, the increment is complete. Results can be requested only at the end of an increment.

Equilibrium iterations and convergence in Abaqus/Standard

The nonlinear response of a structure to a small load increment, Δp is shown in Figure 3.9. Abaqus/Standard uses the structure's initial stiffness K_0 which is based on its configuration at U_0 and Δp to calculate a displacement correction c_a for the structure. Using c_a the structure's configuration is updated to u_a .

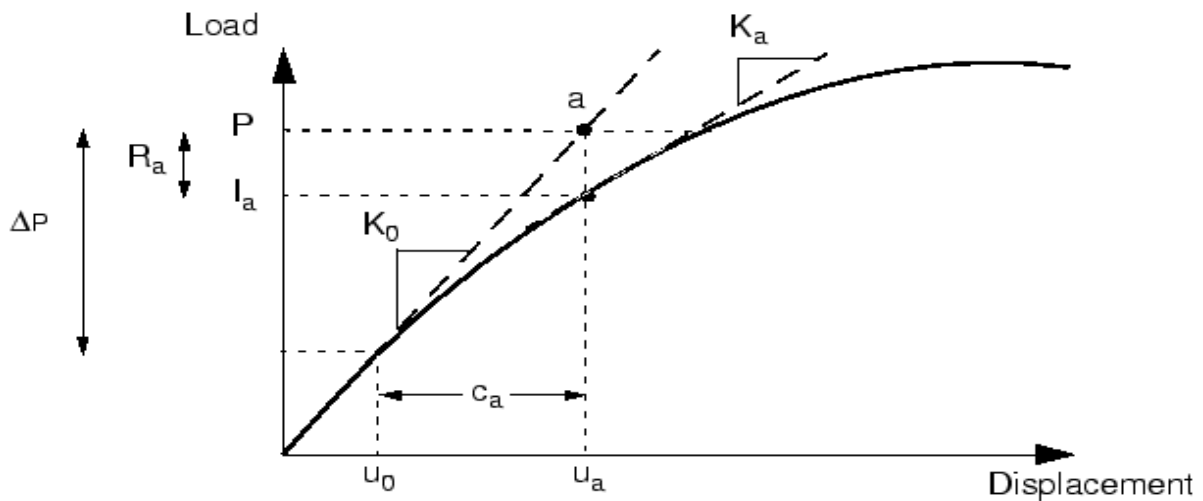


Figure 3.9 First iteration in an increment.

Convergence

Abaqus/Standard forms a new stiffness k_a for the structure, based on its updated configuration u_a . Abaqus/Standard also calculates I_a in this updated configuration. The difference between the total applied load, p , and I_a can now be calculated as:

$$R_a = p - I_a \text{ Where, } R_a \text{ is the force residual for the iteration.}$$

If R_a is zero at every degree of freedom in the model, point a in Figure 3.9 would lie on the load-deflection curve, and the structure would be in equilibrium. In a nonlinear problem it is almost impossible to have R_a equal zero, so Abaqus/Standard compares it to a tolerance value. If R_a is less than this force residual tolerance, Abaqus/Standard accepts the structure's updated configuration as the equilibrium solution.

By default, this tolerance value is set to 0.5% of an average force in the structure, averaged over time. Abaqus/Standard automatically calculates this spatially and time-averaged force throughout the simulation.

If R_a is less than the current tolerance value, p and I_a are in equilibrium, and u_a is a valid equilibrium configuration for the structure under the applied load. However, before Abaqus/Standard accepts the solution, it also checks that the displacement correction c_a is small relative to the total incremental displacement, $\Delta u = u_a - u_o$ if c_a is greater than 1% of the incremental displacement, Abaqus/Standard performs another iteration. Both convergence checks must be satisfied before a solution is said to have *converged* for that load increment. The exception to this rule is the case of a *linear* increment, which is defined as any increment in which the largest force residual is less than 10^{-8} times the time-averaged force. Any case that passes such a stringent comparison of the largest force residual with the time-averaged force is considered linear and does not require further iteration. The solution is accepted without any check on the size of the displacement correction. If the solution from iteration is not converged, Abaqus/Standard performs another iteration to try to bring the internal and external forces into balance. This second iteration uses the stiffness k_a calculated at the end of the previous iteration together with R_a to determine another displacement correction c_b that brings the system closer to equilibrium (point b in Figure 3.10).

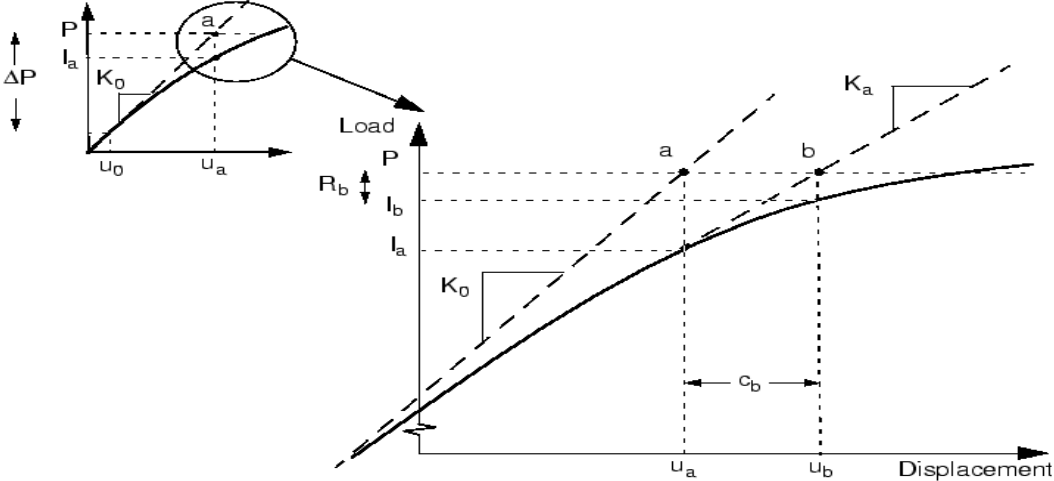


Figure 3.10 Second iteration.

Abaqus/Standard calculates a new force residual using the internal forces from the structure's new configuration u_b . Again, the largest force residual at any degree of freedom, R_b , is compared against the force residual tolerance, and the displacement correction for the second iteration, c_b is compared to the increment of displacement. $\Delta u_b = u_b - u_o$. If necessary, Abaqus/Standard performs further iterations. For each iteration in a nonlinear analysis Abaqus/Standard forms the model's stiffness matrix and solves a system of equations. This means that each iteration is equivalent, in computational cost, to conducting a complete linear analysis. It should now be clear that the computational expense of a nonlinear analysis in Abaqus/Standard can be many times greater than for a linear one.

When modeling and analyzing reinforced concrete, convergence problems often occur in Abaqus during the solution. There are many different ways to handle these problems and some of them are: The time increment parameters should always, when analyzing concrete with the Static/General time step, be set to lower values than the default values in Abaqus. As an effect of this, the maximum number of time increments has to be increased. The reason to this is the need to allow a more accurate solution at points when cracking occurs. Convergence problems can arise if the reinforcement nodes coincide with the concrete element nodes. These convergence problems occur because the high stresses in the reinforcement heavily distort the less stiff material elements. To avoid this, reinforcement should be placed where it does not coincide with any other material element node. Another way of handle convergence problems is to introduce automatic stabilization in the time steps. According to results in Malm and Ansell (2008) this stabilization does not interfere with the concrete behavior and is therefore an effective way of handling convergence problems. The automatic stabilization can be turned on in the "Edit step" window in Abaqus. In this study the "Specify dissipated energy fraction" option with default values is used. The default value for the automatic stabilization is 0.0002 and for the adaptive stabilization with maximum ratio of stabilization to strain energy is 0.05.

Automatic incrementation control in Abaqus/Standard

Abaqus/Standard automatically adjusts the size of the load increments so that it solves nonlinear problems easily and efficiently. It only needs to suggest the size of the first increment in each step of your simulation. Thereafter, Abaqus/Standard automatically adjusts the size of the increments.

Generally, it is to your advantage to provide a reasonable initial increment size only in very mildly nonlinear problems can all of the loads in a step be applied in a single increment.

The number of iterations needed to find a converged solution for a load increment will vary depending on the degree of nonlinearity in the system. By default, if the solution appears to diverge, Abaqus/Standard abandons the increment and starts again with the increment size set to 25% of its previous value. An attempt is then made at finding a converged solution with this smaller load increment. If the increment still fails to converge, Abaqus/Standard reduces the increment size again. By default, Abaqus/Standard allows a maximum of five cutbacks of increment size in an increment before stopping the analysis.

In a nonlinear analysis a step takes place over a finite period of time, although this time has no physical meaning unless inertial effects or rate-dependent behavior are present. In Abaqus/Standard you specify the initial time increment $\Delta T_{initial}$ and the total step time T_{total} . The ratio of the initial time increment to the step time specifies the proportion of load applied in the first increment. The initial load increment is given by

$$\frac{\Delta T_{initial}}{T_{total}} * loadmagnitudo$$

The choice of initial time increment can be critical in certain nonlinear simulations in Abaqus/Standard, but for most analyses an initial increment size that is 5% to 10% of the total step time is usually sufficient. In static simulations the total step time is usually set to 1.0 for convenience, unless, for example, rate-dependent material effects or dashpots are included in the model. With a total step time of 1.0 the proportion of load applied is always equal to the current step time; i.e. 50% of the total load is applied when the step time is 0.5.

Abaqus/Standard has no upper limit on the increment size, ΔT_{max} other than the total step time.

If the increment converges in fewer than five iterations, this indicates that the solution is being found fairly easily. Therefore, Abaqus/Standard automatically increases the increment size by 50% if two consecutive increments require fewer than five iterations to obtain a converged solution.

3.4 The Classification of the models under study

According to the loading arrangement and the analysis of a structure, the Columns are classified into one of the following types based on BS 8110:

1. A braced column: where the lateral loads are resisted by walls or some other Form of bracing.
 2. An unbraced column: where the lateral load are resisted by bending action of the columns.
- All models were assumed to be braced column, which means no bending action considered in this study. Also a column is classified as short if the L_{ex} / h and L_{ey} / b

- ✓ Less than 15 for a braced column.
- ✓ Less than 10 for an unbraced column.

Where,

L_{ex} / h And L_{ey} / b = the effective height relative to XX and YY axis.

H= the overall depth of the section in the plane of bending about the XX axis.

B= the dimension perpendicular to the XX axis.

All columns were assumed to be braced column. So they were designed as short braced column. The L_{ey} / b and L_{ex} / h ratios and types of models are shown in the table 3.5

Table 3.5 the L_{ey} / b and L_{ex} / h ratios and types of models

Column-sets	L_{ey} / b	L_{ex} / h	Column types
Square columns	9.33	9.33	short
Rectangular columns	9.33	7.00	short

3.5 Flowchart of the Finite Element Analysis

In order to achieve the stated objectives, this FEA study is carried out in few Stages. The flowchart of all stage is indicated in Figure 3.11 which provides a general Overview about finite element analysis of this research and its completion procedure.

Draw the Geometry of the Model and Make the Hole along Model(b)

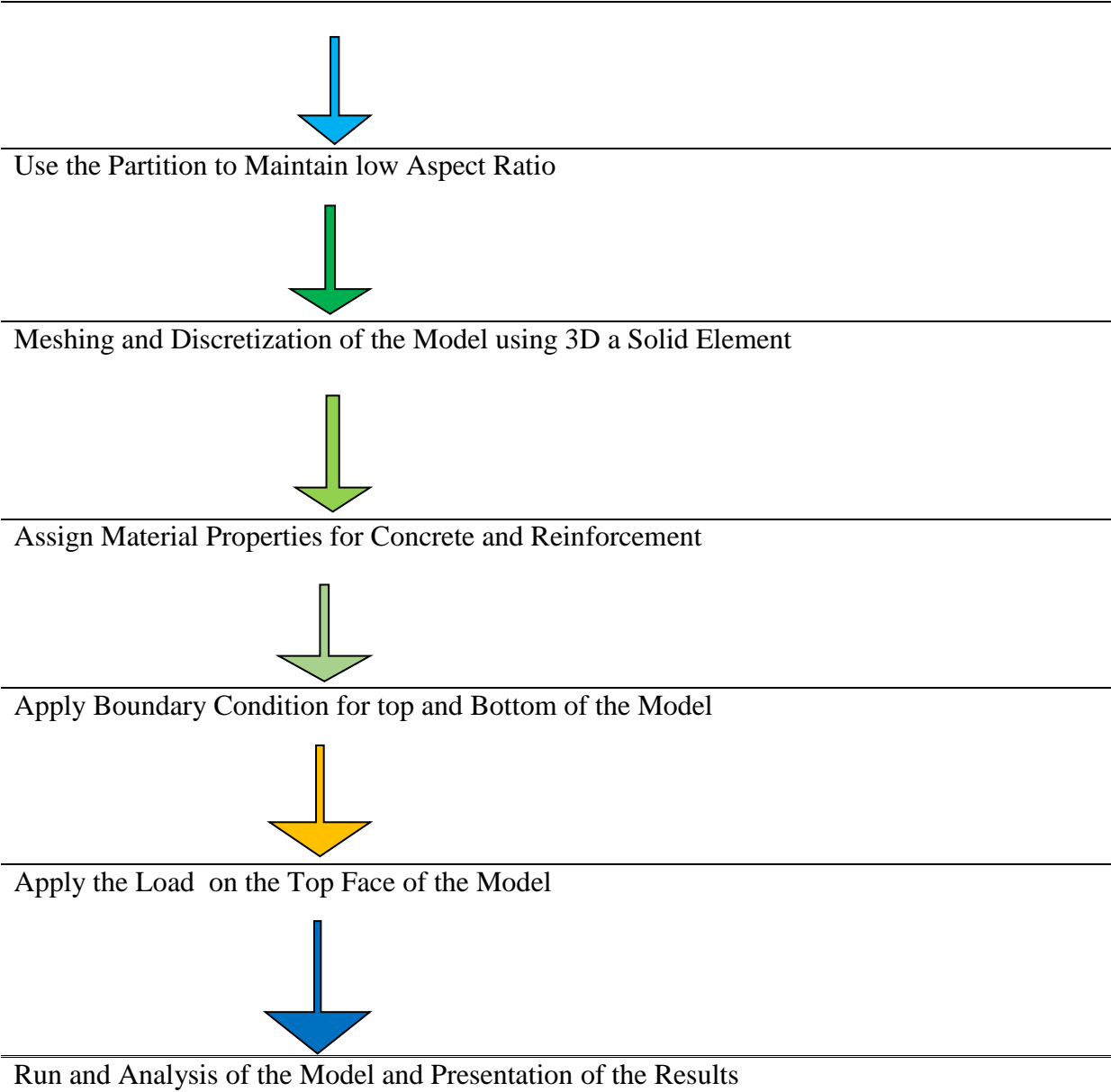


Figure 3.11 Flowchart of finite element analysis

3.6 Experimental validation

The validity of the proposed analytical model is checked through extensive comparisons between analytical and experimental results of RC columns under compression load. This should apparently reveal the load carrying capacity of the column in theoretical and experimental using tabular format for two sets (rectangular, square)of columns under study specifically, for columns C1,C2 and C3 for square and C4 and C5 for rectangular columns. The theoretical results from finite element analysis showed in general a good agreement with the experimental values.

3.6.1 General description on the genesis of data used in experiment

a. Material Properties

Material nonlinearity occurs when the stress- strain relationship ceases to be linear by plastic yielding and strain hardening. Materially nonlinear effects arise from a nonlinear constitutive model (that is, progressively disproportionate stresses and strains). Common examples of nonlinear material behavior are the plastic yielding of metals, the ductile fracture of granular composites such as concrete or time-dependent behavior such as creep. FEM incorporates a variety of nonlinear constitutive models, covering the behavior of the more common engineering materials.

I. Concrete

The concrete is defined as an isotropic material in the modeling, and should be deliberately expressed in verification genesis.

Table 3.6: Concrete properties of validity parameters

Elastic Properties	Young's Modulus = 20130 N/mm ²
	Poisson's Ratio = 0.2
Plastic Properties	Compressive Strength = 37.12 N/mm ²
	Tensile Strength = 3.712 N/mm ²
	Strain at end of softening curve = 0.003

II. Steel Bar

In the plastic properties, the stress potential model was selected and Von Misses yield surface was tried for the plastic behavior of the steel.

Table 3.7: Steel properties of validity parameters

Elastic Properties	Young's Modulus = 200000 N/mm ²
	Poisson's Ratio = 0.3
Plastic Properties	Yield Stress = 460 N/mm ²

b. Loading Procedure

In order to represent the restraint provided by the beams and slab to the column in each floor level, in the testing set up, each end of the model were fixed to the testing machine to simulate the actual condition in the columns. Before positioning the models in testing machine the center of the columns were marked, then the center of the columns were positioned in center of testing machine loading surface to achieve an axial compressive loading condition without eccentricity. In the next stage, the model was checked to be aligned vertically by the spirit level readings on two opposite face of the column. In order to prevent local failure at the loaded ends of the columns, two pieces of plywood with 10 mm thickness were located in top and bottom of the model. In final stage before testing, the strain gauges wires were connected to the data logger. The models have been tested using the 5000 kN capacity universal testing machine. The columns were loaded under a monotonically increasing axial compressive load until their collapse. The loading and the vertical axial strain readings were recorded after every increment of 20 kN load. Positioning of the hole in center of column's cross-section, and The test setup is shown in Figure 3.12 and figure 3.13 respectively.



Figure 3.12 Positioning of the hole in center of column's cross-section



Figure 3.13 Experimental set-up

3.6.2 Comparison of the Results

The results of the reduced scale laboratory test were used as comparison with the results from the finite element analysis. Hence, the finite element model was implemented based on the parameters and conditions in the laboratory test. The model from experimental investigation, misses stress, and stress-strain distribution of the columns taken from experimental is showed in the figure below for both rectangular and square columns respectively.

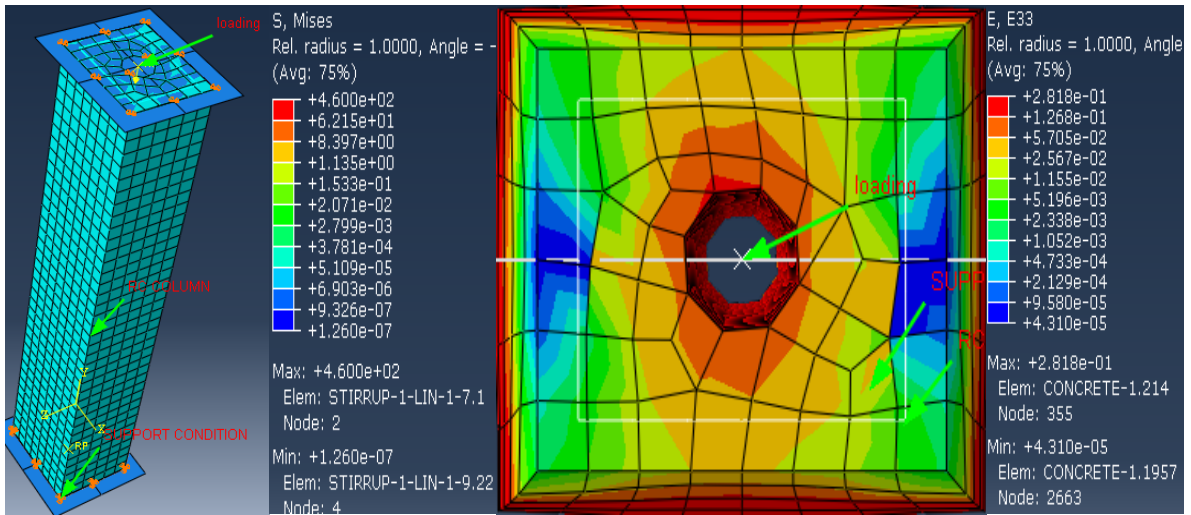


Figure 3.14 model sample and strain-stress distribution of **rectangular** columns

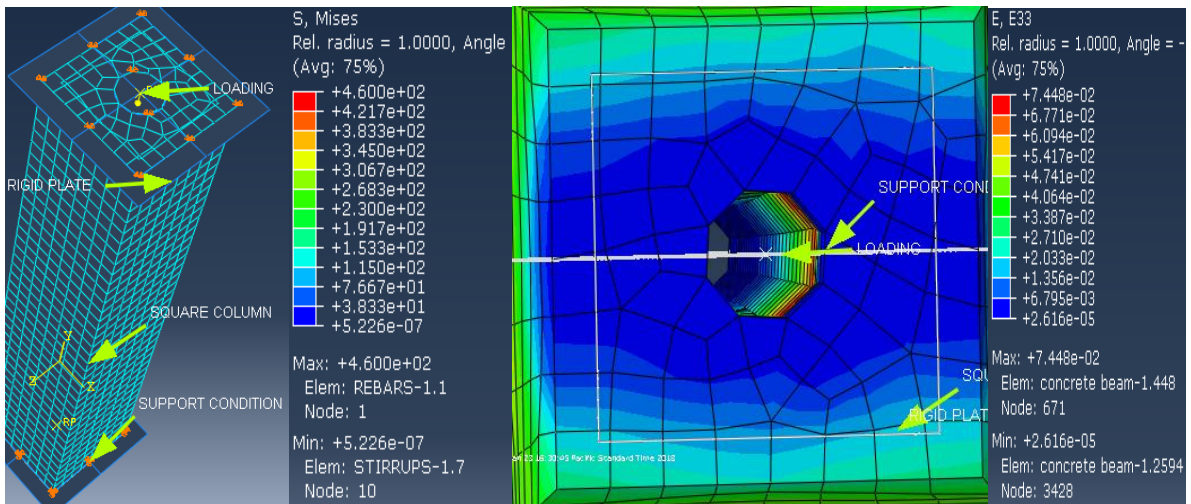


Figure 3.15 model sample and strain-stress distribution of **square** columns

The table below shows the comparative results of maximum vertical compressive stresses and strains from experimental investigation and finite element analysis of the models for both rectangular and square columns.

Table 3.8 Validity of the finite element result in terms of strain –stress

Column-sets (models)	Column-sets (types)with its hole size(mm)	stress $(\sigma)N/mm^2$ experimental	strain $(\varepsilon)*10^{-3}$ experimental	stress $(\sigma)N/mm^2$ FEA	strain $(\varepsilon)*10^{-3}$ FEA
Column-sets (C1,C2,C3) 300*300 With the hole at the center	C1(solid)	36.733	2.218	37.89	3.14
	C2(89)	35.011	2.341	37.585	2.24
	C3(114)	36.983	2.616	38.94	3.35
Column-sets (C4,C5) 300*250 With the hole at the center	C4(solid)	37.25	2.78	37.54	3.23
	C5(89)	36.78	2.16	37.32	2.28

The table below reveals the percentages ultimate load carrying capacity of the columns from experimental investigation and finite element analysis of the models. This validation aspect shows that there is good agreement between theoretical investigation and experimental values. From comparison of results it can be noticed all out puts seems to be appeared in the tolerable ranges of validation. These results are well discussed in the table 3.9.

Table 3.9 Validity of the finite element result in terms of load carrying capacity

Columns-sets (models)	Columns-sets with its hole size(mm) in diameter	Ultimate load (KN) Experimental	Ultimate load (KN) FEA	Load-variation percentage (%)
Columns-sets (C1,C2,C3) 300*300 With the hole at the center	C1(solid)	3116	3981.571	21.74
	C2(89)	2826	3680.600	23.21
	C3(114)	2951	3529.250	16.38
Columns-sets (C4,C5) 300*250 With the hole at the center	C4(solid)	3006	3945.23	23.74
	C5(89)	2407	2584.26	6.86

CHAPTER 4

ANALYSIS, RESULT AND DISCUSSIONS

4.1 Introduction

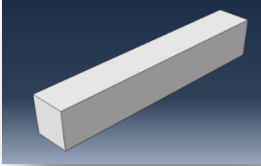
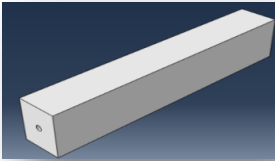
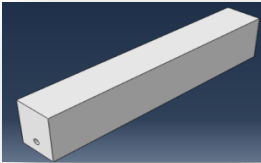
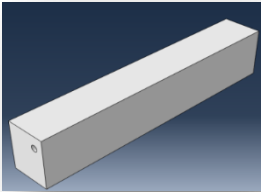
This chapter presents the results of comparative study between solid column of rectangular and square shapes and with their respective columns with the hole at different position to meet the objective of the study using finite element analysis. The comparative study was undertaken to determine the capacity of the solid columns with the columns of the same type, but with inclusive of the hole at different position in predicting the nonlinear behavior of the reinforced concrete column with hole and also inferring the perfect position of the hole in enhancing capacity of the structure. Comparison was made on the aspects of stress-strain curve, load-deflection curve, and mode of failure with the aid of graphs, tables and diagrams. Finally, the results of the comparative study between solid columns and columns with the holes using finite element analysis is depicted with aid of above discussed aspect of illustration.

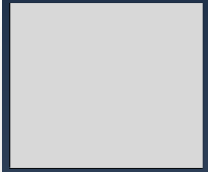
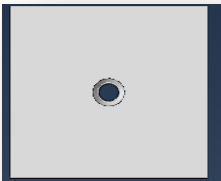
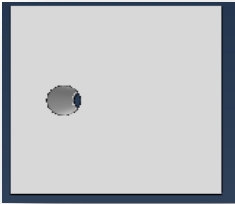
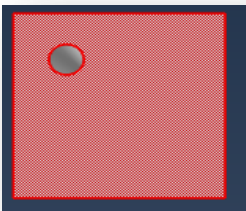
Verifications of the finite element analysis results were done during the modeling, to ensure the results obtained are reliable. In this part of the investigation, a preliminary study was conducted for having an early understanding on the column behavior. Subsequently, the results were inferred, leading to the outcomes that achieve the aims of the study.

4.2 The Results of Finite Element Analysis

The axial compression load was assigned at the top of the column, so due to the axial compression loading, the stress and strain is induced in the columns, and vertically direction is considered for study. The load carrying capacities of the columns were determined at specified point of element in the mesh. The maximum vertical compressive stress and strain in all models using finite element analysis are shown in table 4.1

Table 4.1: Maximum vertical stress and strain of the columns under study

Position Of The Hole	Column-sets	Column types	Max. vertical stress $\sigma(N/mm^2)$	Max. vertical strain $\varepsilon(\times 10^{-3})$
Solid rectangular column 	Cr-solid	Short-braced	24.83	4.89
Case-a @center of the columns 	Cr-10	Short-braced	24.36	5.53
	Cr-25	Short-braced	24.769	5.917
Case-b @0.25*B from H and 0.5*H from B 	Cr-10	Short-braced	24.933	5.109
	Cr-25	Short-braced	24.26	6.12
Case-c @0.25*B from H and 0.25*H from B 	Cr-10	Short-braced	24.648	5.10
	Cr-25	Short-braced	23.67	6.11
Case-d				

Position Of The Hole	Column sets	Column types	Max. vertical stress $\sigma(N/mm^2)$	Max. vertical strain $\varepsilon(\times 10^{-3})$
Solid square column 	Cs-solid	Short-braced	22.57	7.69
Case-a @center of the columns 	Cs-10	Short-braced	22.89	6.98
	Cs-25	Short-braced	22.84	7.07
case -b @0.25*B from H and 0.5*H from B 	Cs-10	Short-braced	22.16	6.93
	Cs-25	Short-braced	22.32	7.30
@0.25*B from H and 0.25*H from B 	Cs-10	Short-braced	23.78	6.71
	Cs-25	Short-braced	22.27	6.87

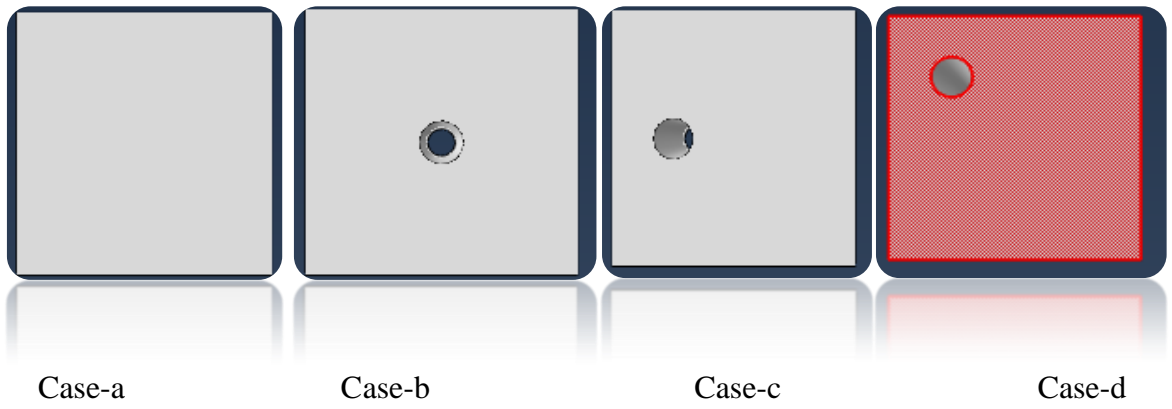
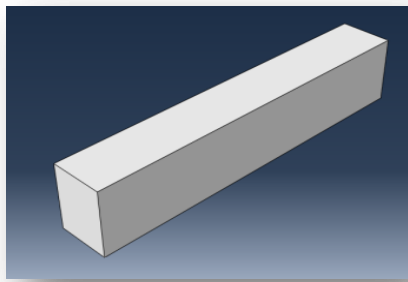
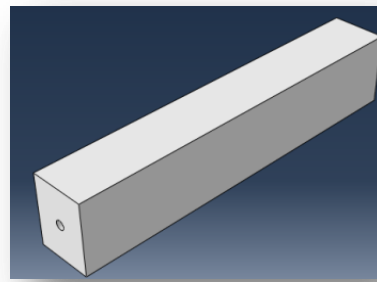


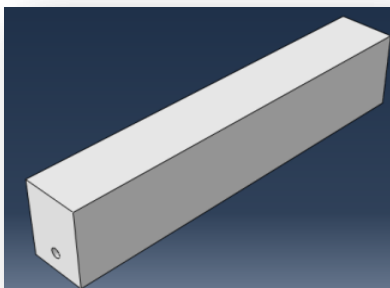
Figure 4.1 Cases layout-descriptions in 2D



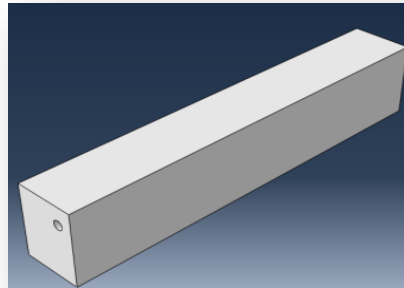
Case-a



Case-b



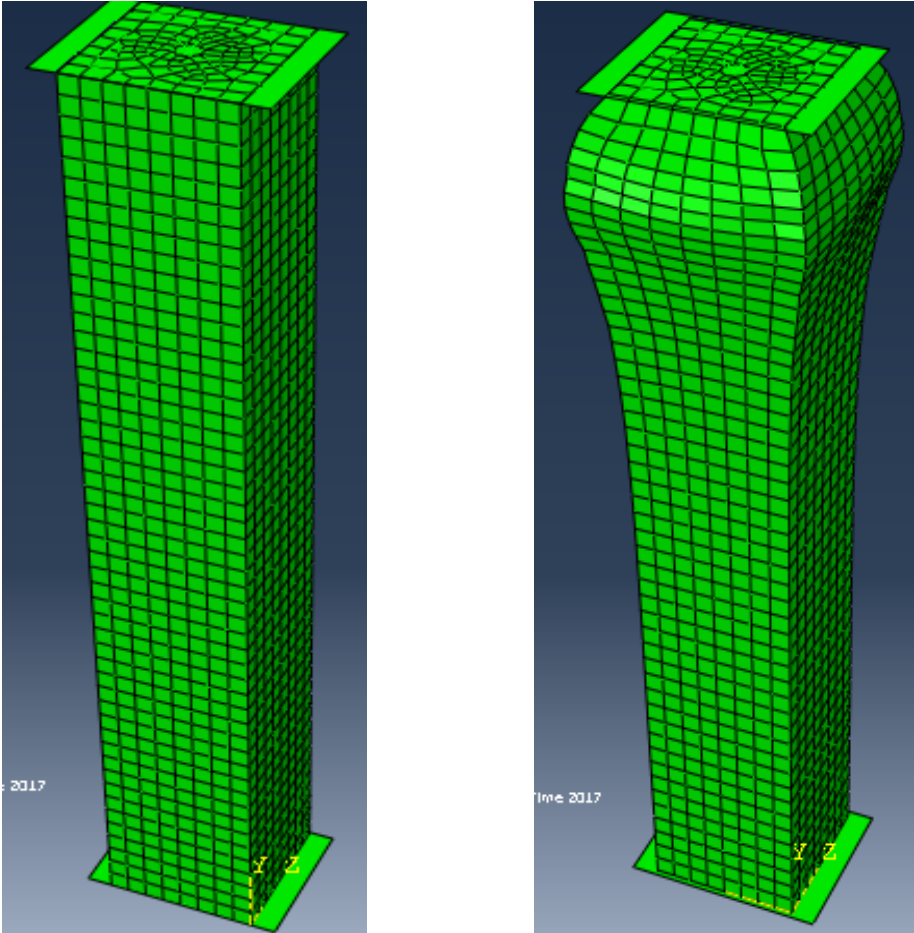
Case-c



Case-d

Figure 4.2 Cases layout- descriptions in 3D

The result of software analysis, by considering studying parameters can be depicted by use of graphs, descriptive tables, and self-expressing figures, so that one can notice the effects of the studying variables easily. The following figure describes the columns global behavior before the loading and under influence of the loading. This influence is termed as undeformed and deformed shape of the structures under investigation.it is clearly revealed in figure 4.3.



i) Plot of undeformed shape of model-case (a) **ii)** plot of deformed shape of model-case (b)
Figure 4.3 Plot of undeformed and deformed shape of model-case a&b

Vertical stress distribution contour of model of **square** columns with the longitudinal hole at the center is showed in Figure 4.4. According to the finite element analysis results, the maximum stress in y-direction is occurred near the hole area at bottom of the column because of the axial compressive loading at the top of the columns.

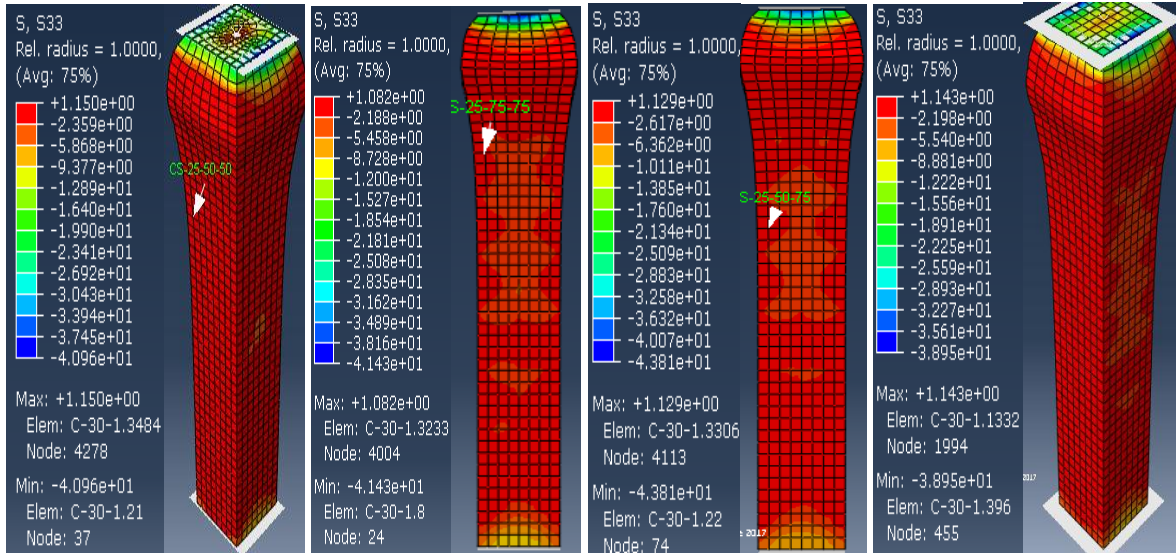


Figure 4.4 Vertical stress distribution contour of **square** column model with the hole at Case-b, case-d, case-c, and case-a respectively in 2D and 3D layout.

In Figures 4.5, the vertical strain distribution counter of **square** columns in the longitudinal direction is presented. It can be seen that large strain, i.e. deformations due to compressive loading and supports, is Concentrated near the loading area. The vertical strain contour of model Cs-25 in two-dimensional and in three-dimensional is shown in Figure 4.5.

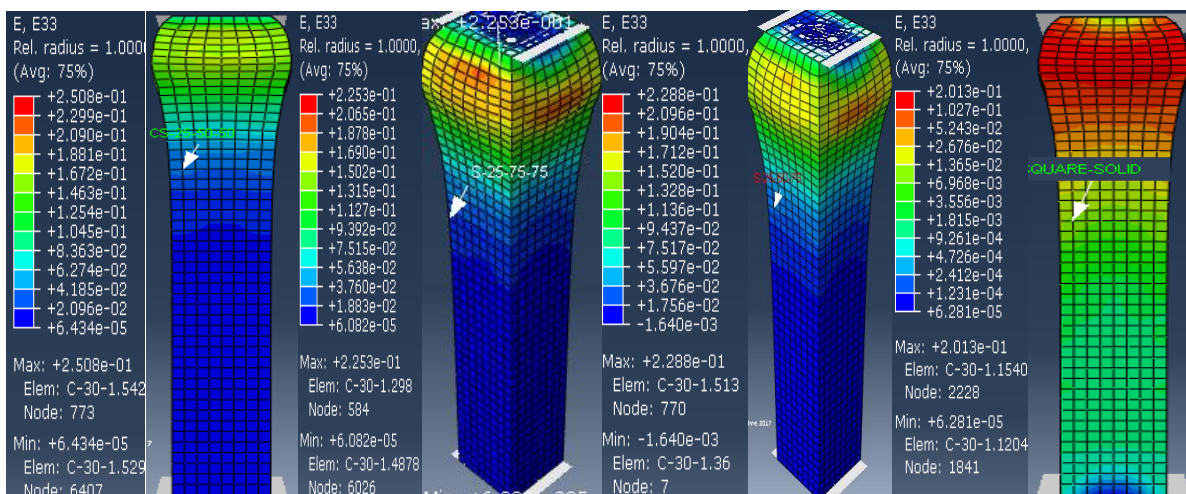


Figure 4.5 Vertical strain distribution contour of **square** column model with the hole at case-b case-d, case-c, and case-a respectively in 2D and 3D layout.

Vertical stress distribution contour of model of **rectangular** columns with the longitudinal hole at the center is showed in Figure 4.6. According to the finite element analysis results, the maximum stress in y-direction is occurred near the hole area at bottom of the column because of the axial compressive loading at the top of the columns.

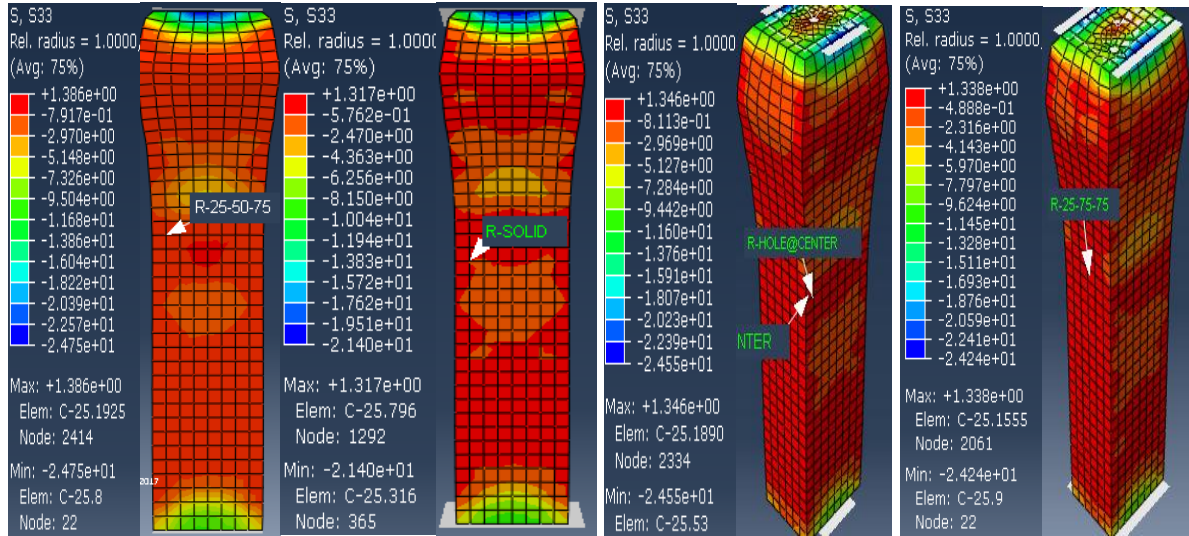


Figure 4.6 Vertical stress distribution contour of **rectangular** column model with the hole at Case-c, case-a, case-b, and case-d respectively in 2D and 3D layout.

In Figures 4.7, the vertical strain distribution counter of **rectangular** columns in the longitudinal direction is presented. It can be seen that large strain, i.e. deformations due to compressive loading and supports, is Concentrated near the loading area. The vertical strain contour of model Cr-25 in two-dimensional and in three-dimensional is shown in Figure 4.4.

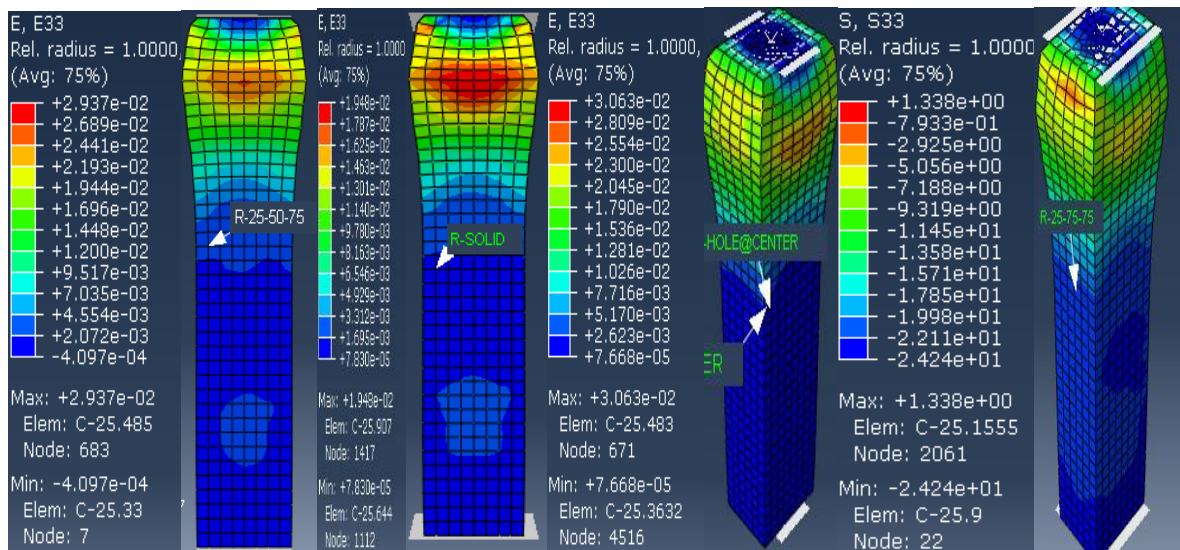


Figure 4.7 Vertical strain distribution contour of **rectangular** column model with the hole at Case-c, case-a, case-b, and case-d respectively in 2D and 3D layout.

4.3 Vertical Stress-Strain Curves of the Models Based on FEA Results

The typical stress-strain curves which are plotted in Figures 4.8 and 4.9 clearly shows the effect of hole positions and hole dimensions for **square** columns. This reveals the post-peak segment of the stress-strain curves are affected by using hole inside the column and by changing the hole dimensions. Also according to the stress-strain graph, the columns without the hole have a greater ultimate point of compressive strength in comparison with those with hole inside.

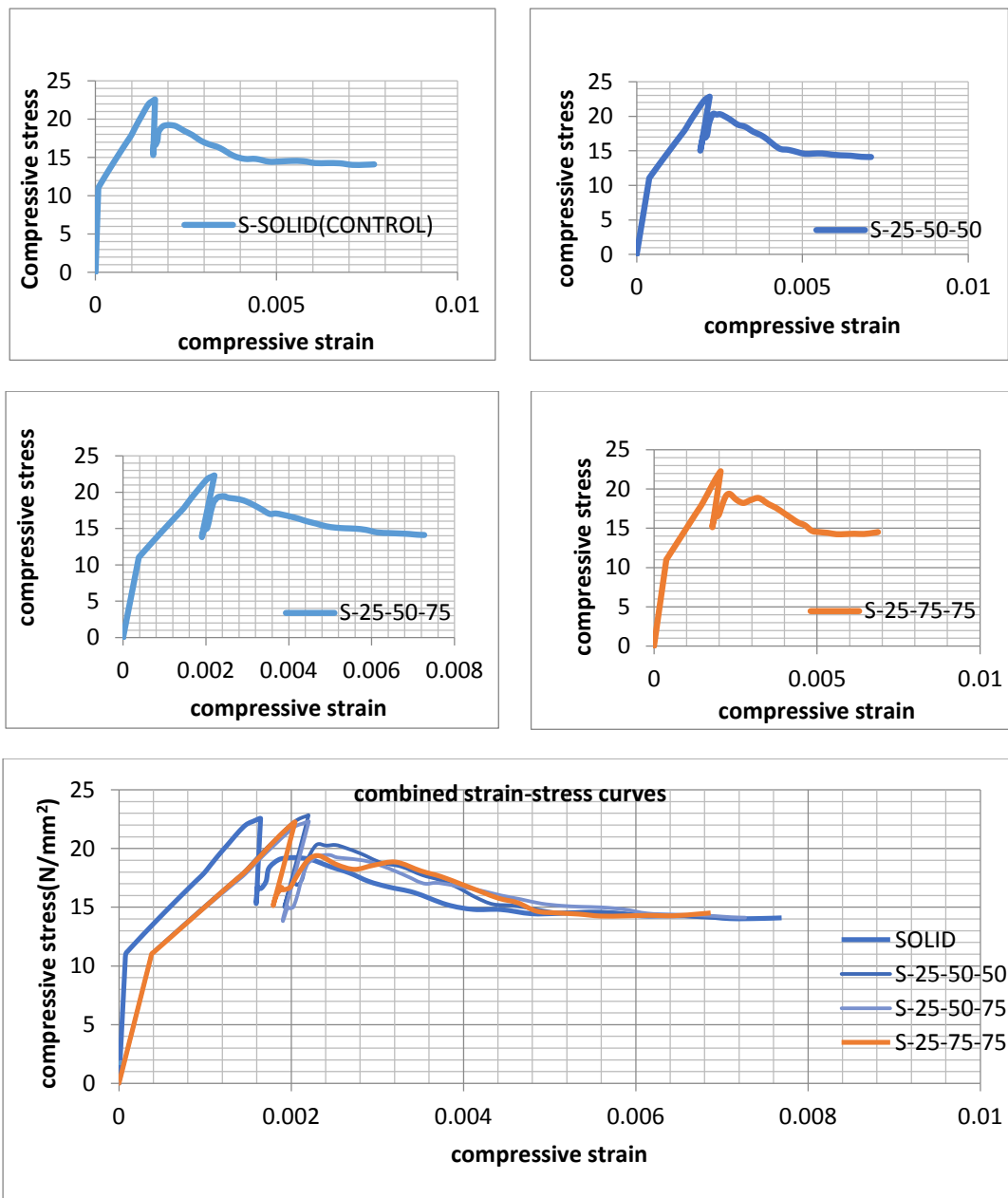


Figure 4.8 stress-strain curves of **square** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

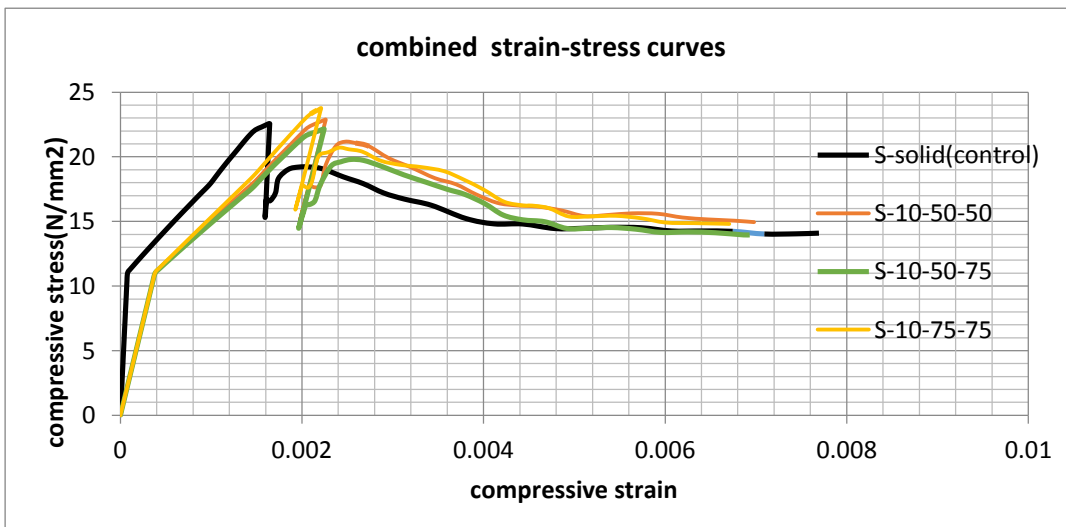
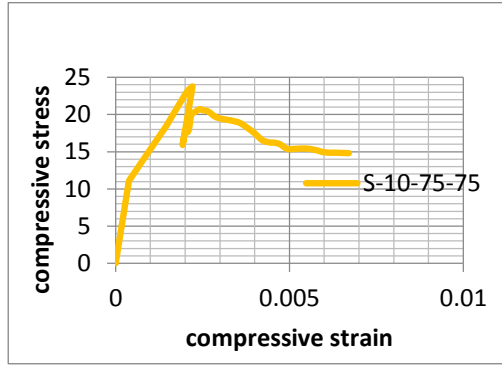
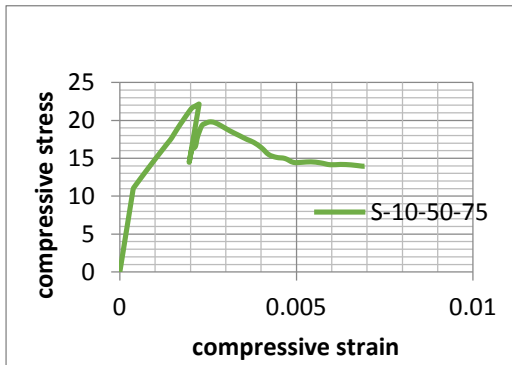
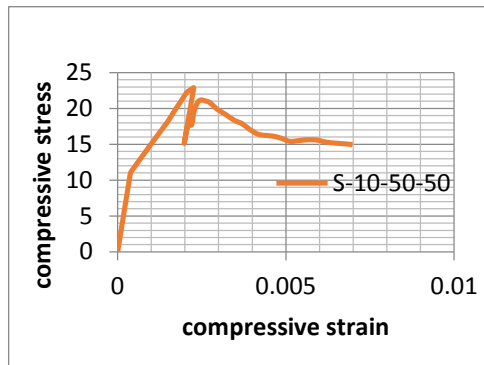
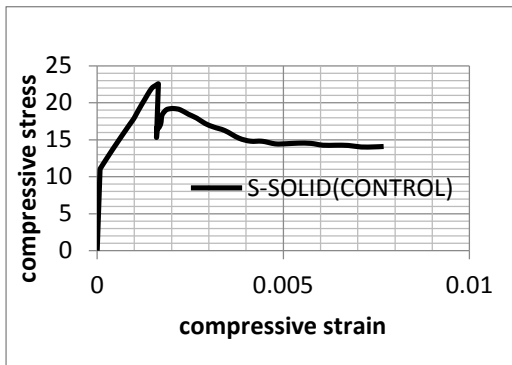


Figure 4.9 stress-strain curve of **square** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

The typical stress-strain curves which are plotted in Figures 4.10 and 4.11 clearly shows the effect of hole positions and hole dimensions for **rectangular** columns. This reveals the post-peak segment of the stress-strain curves are affected by using hole inside the column and by changing the hole dimensions. From the graphs drawn below one can notice that the stress-strain curves for each cases understudy represents comparative results, and depicts the peak point of compressive strength of columns without a hole and with hole inside.

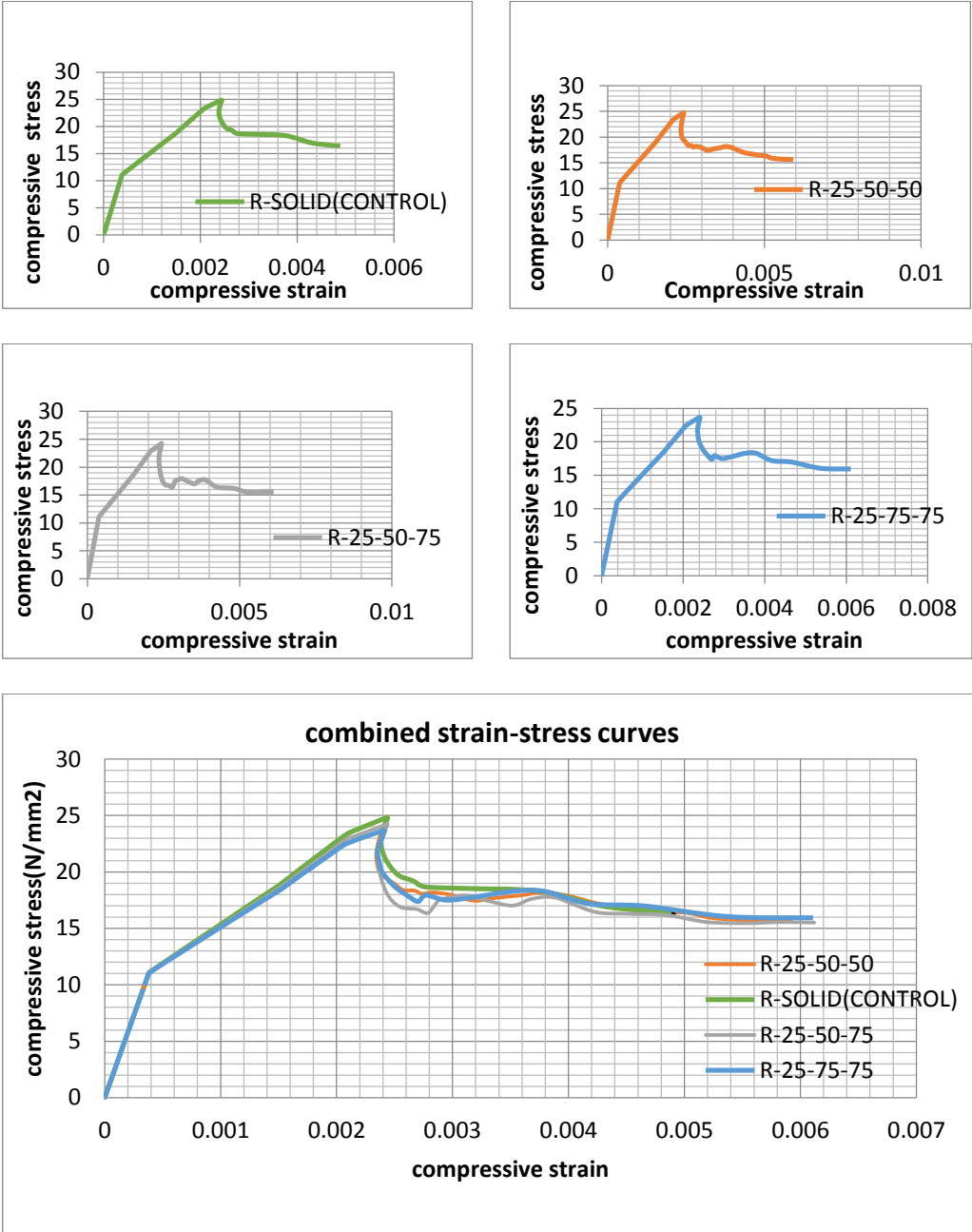


Figure 4.10 stress-strain curve of **rectangular** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

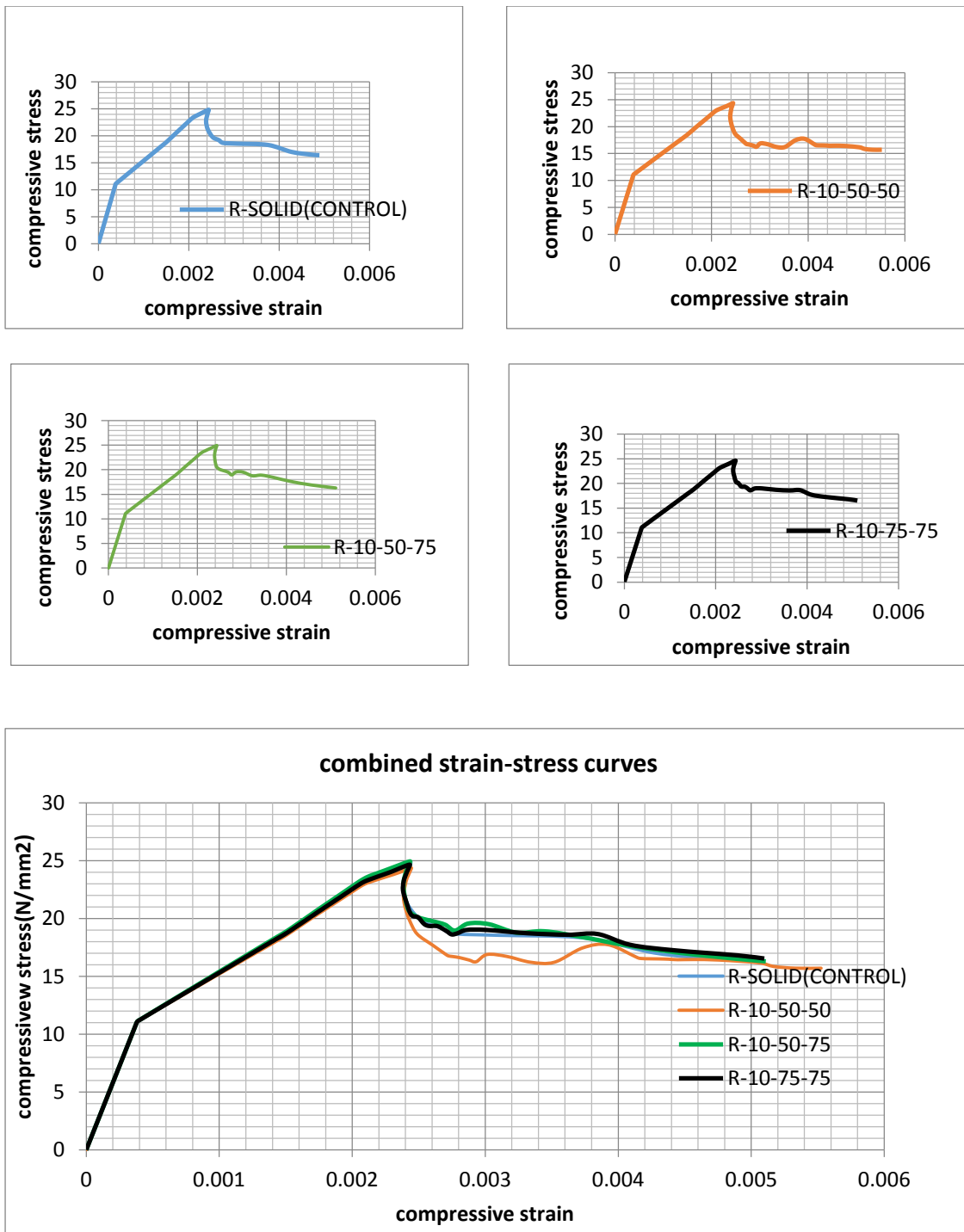


Figure 4.11 stress-strain curve of **rectangular** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

4.4 Reduction in Load Carrying Capacity of Columns

The percentages of the reduction in load carrying capacity of reinforced concrete short braced column having hole inside them based on finite element analysis are shown in Table 4.2. The current study showed a huge reduction in the load bearing capacity of the columns with hole (containing wiring pipes) and also the reduction load carrying capacity of columns with hole vary from **3.89%** to **5.55 %**.for rectangular and **4.5%** to **7.75%**.

Table 4.2: The percentages of reduction in load carrying capacity of **rectangular** columns with hole in 3D

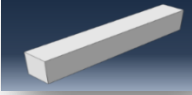
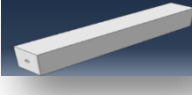
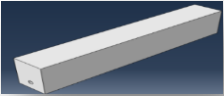
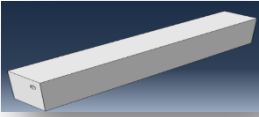

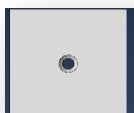
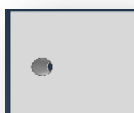
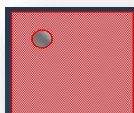
Position Of The Hole(model)	Column sets(types)	Ultimate Def. (mm)	Ultimate force(KN)	Reduction in load carrying Capacity of columns (%)
Solid rectangular column 	Cr-solid(control)	2.74	3192.34	0.00
Case-a @ center of the columns 	Cr-10	2.824	3057.60	4.00
	Cr-25	2.891	3026.31	5.20
Case-b @0.25*B from H and 0.5*H from B 	Cr-10	2.761	3068.0	3.89
	Cr-25	2.92	3023.60	5.29
Case-c @0.25*B from H and 0.5*H from B 	Cr-10	2.764	3061.62	4.10
	Cr-25	2.897	3015.20	5.55
case-d				

Table 4.3: The percentages of reduction in load carrying capacity of **square** columns with hole in 2D

Position Of The Hole(model)	Column sets(types)	Ultimate Def. (mm)	Ultimate force(KN)	Reduction in load carrying Capacity of columns (%)
<p>Solid square column</p>  <p>case-a</p>	Cs-solid (control) specimen	2.533	2230.70	0.00
<p>@ the center of the column</p>  <p>case-b</p>	Cs-10	2.403	2130.30	4.50
	Cs-25	2.461	2088.33	6.38
<p>@ 0.25*B from H and 0.5*H from B</p>  <p>case-c</p>	Cs-10	2.460	2124.23	4.77
	Cs-25	2.491	2072.70	7.08
<p>@ 0.25*B from H and 0.5*H from B</p>  <p>case-d</p>	Cs-10	2.401	2129.40	4.54
	Cs-25	2.463	2057.70	7.75

Figures 4.12 and 4.13 show the theoretical load-deformation of **square** columns (Cs2, Cs3,andCs4) with size of hole **25mm** in radius and (Cs2, Cs3,andCs4) with size of hole **10mm** in radius respectively; which have position of holes case-b, case-c, case-d, and control specimen(Cs1); holes in the **case-d** for both holes size, has significant effect on the ultimate loads and deformations of columns understudy, hence affect the toughness of tested specimens, but holes in has **case-b** for both holes size, limited effect on the ultimate loads and deformations of tested columns. From Table 4.2, it can be seen that, ultimate loads of (Cs2, Cs3,andCs4) with size of hole **25mm** in radius are (93.62, 92.92 and 92.25%), and (95.5,95.23 and 95.46%) respectively.

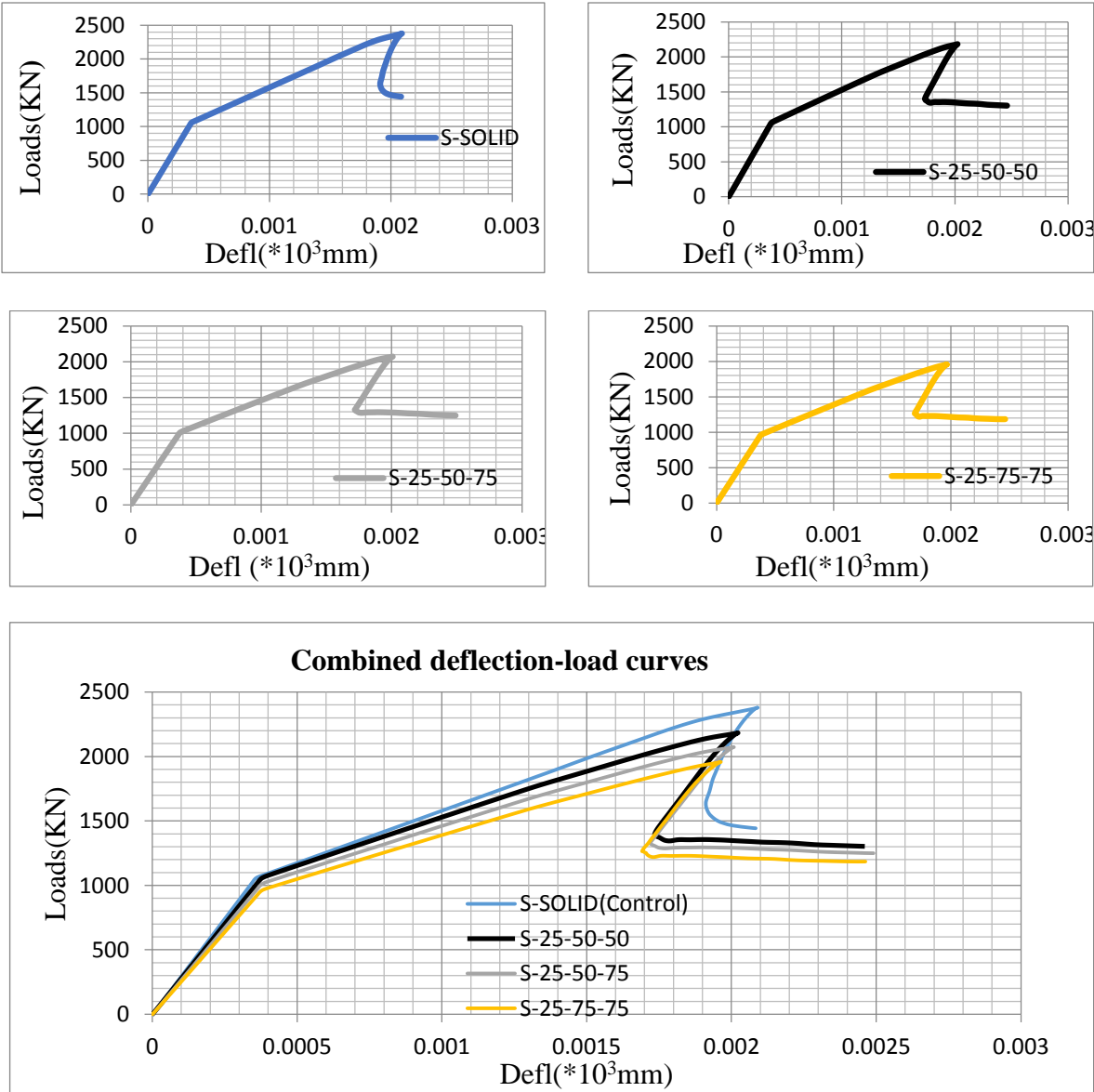


Figure 4.12 load-deflection curve of **square** columns for case-a, case-b, case-c, and case-d respectively

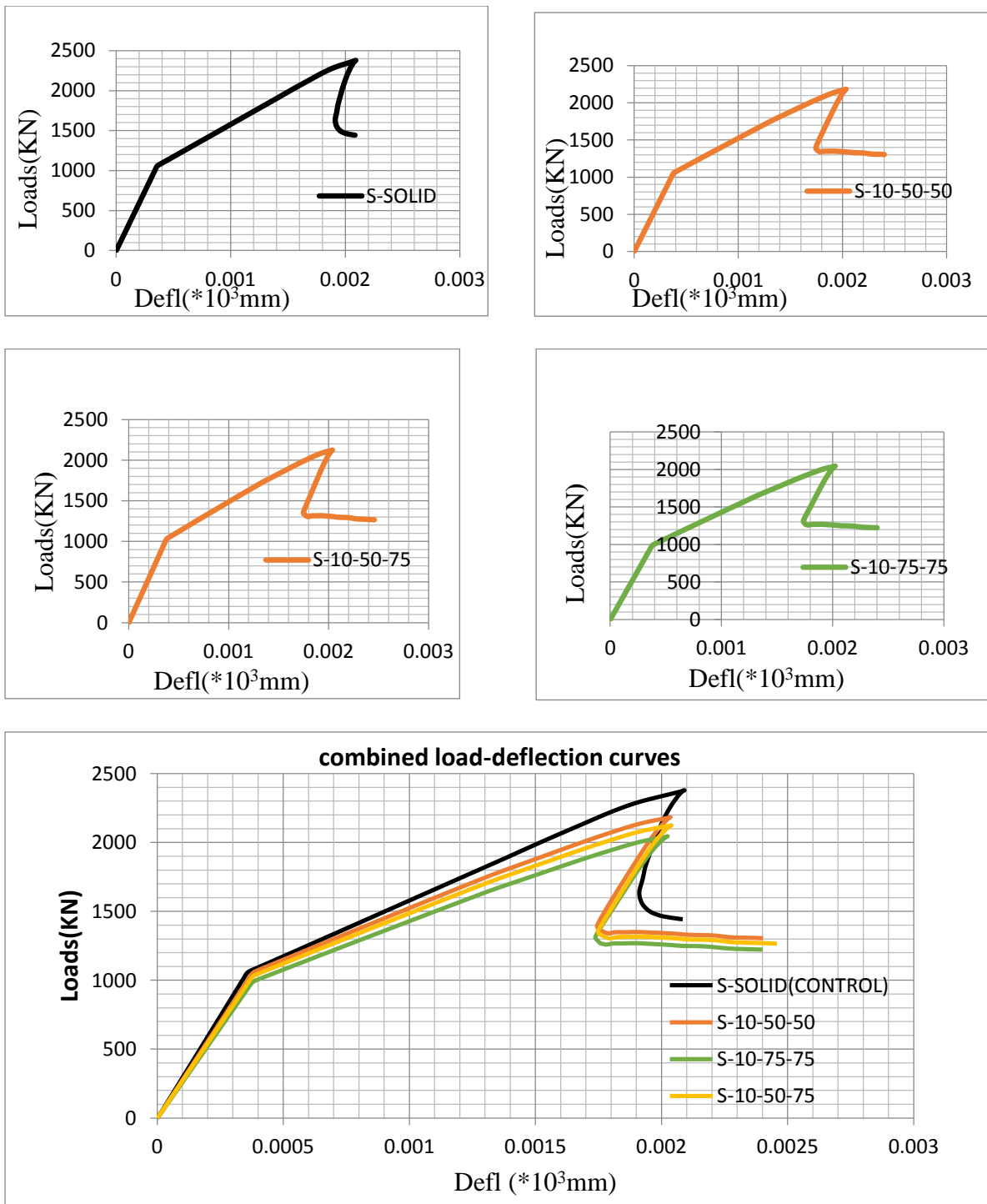


Figure 4.13 load-deflection curve of **square** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

Figures 4.14 and 4.15 show the theoretical load-deformation of **rectangular** columns (Cs2, Cs3, and Cs4) with size of holes **25mm** in radius and (Cs2, Cs3, and Cs4) with size of hole **10mm** in radius respectively; which have position of holes case-b, case-c, case-d, and control specimen (Cs1); holes in the **case-d** for both holes size, has significant effect on the ultimate loads and deformations of columns under study, hence affect the toughness of tested specimens, but holes in **case-b** for holes size 25mm, and **case-c** for holes size 10mm limited effect on the ultimate loads and deformations of tested columns. From Table 4.3, it can be seen that, ultimate loads of (Cs2, Cs3, and Cs4) with size of hole **25mm** in radius are (94.8, 94.71 and 94.45%), and (96, 96.11 and 95.9%) respectively.

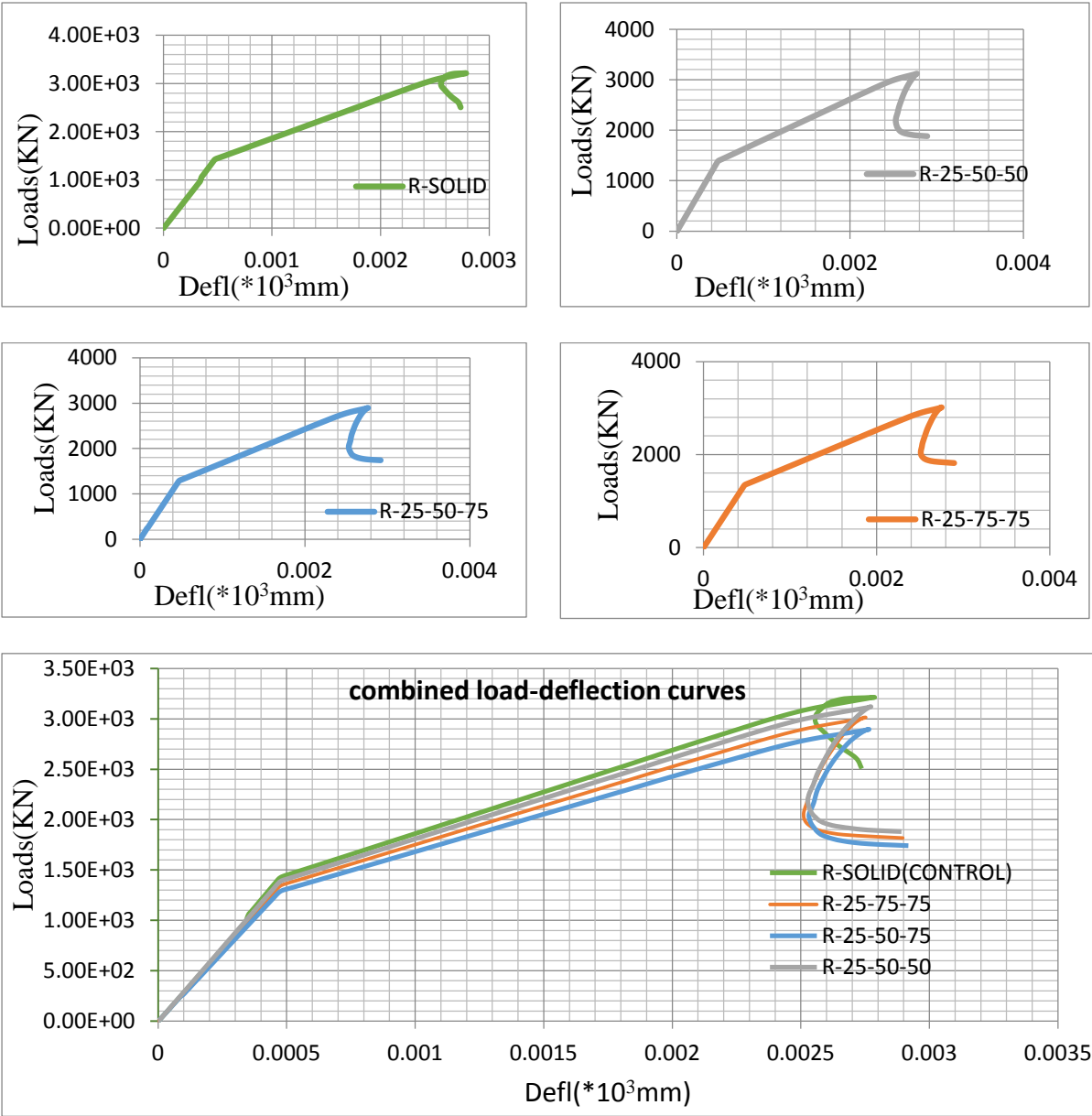


Figure 4.14 load-deflection curve of square columns for case-a, case-b, case-c, and case-d respectively.

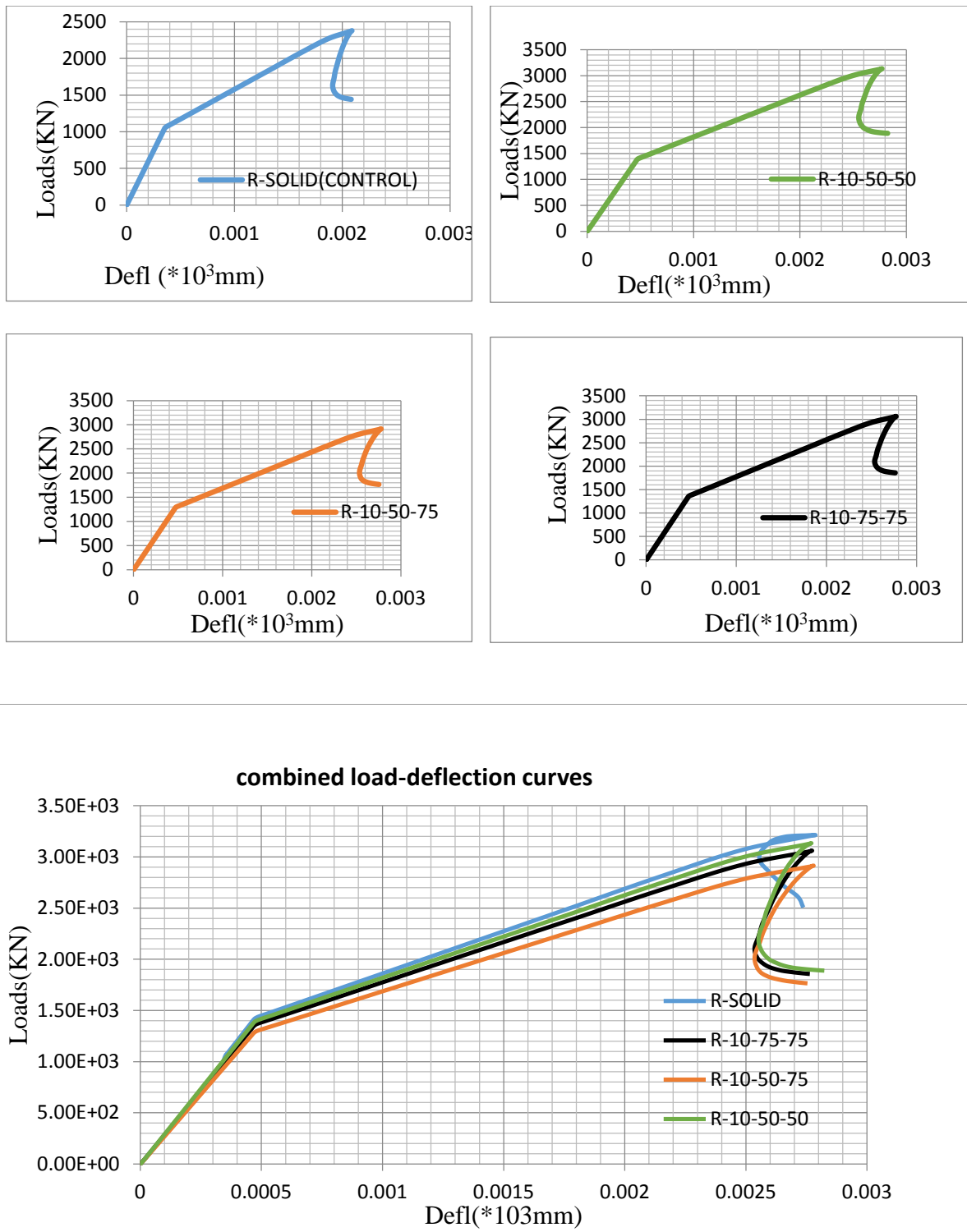


Figure 4.15. load-deflection curve of **square** columns for case-**a**, case-**b**, case-**c**, and case-**d** respectively.

4.5 Discussion

Overview

According to the finite element analysis result, the maximum stress in vertical direction is occurred near the hole at bottom of the columns. Also the maximum strain in vertical direction is concentrated near the loading area. The net concrete area of the columns cross-section is reduced because the area of the hole has already being deducted which directly affect the load carrying capacity of the columns. The failure loads of the columns with hole (embedded pipes) were less than the load without the hole.

4.5.1 Discussion on rectangular columns

Comparison of the results implemented by finite element analysis shows that among the models of rectangular columns with the holes of radius 10mm and 25mm the model in different cases (case-c, case-a, case-d, and case-b) for **10mm** radius and;(case-a, case-b, case-c, case-d) for **25mm** predicts much higher vertical **compressive strength** in respective orders.

The analysis reveals that, the position of the hole in the column also matters the load carrying capacity of the reinforced concrete columns so that one can exactly point out where he/she should put the wiring duct in the column from this analysis. Here is the order of percentage increment in **load carrying capacity** in different placement of duct for electric wiring;(case-a, case-c, case-d, case-b) for **10mm** radius and;(case-a, case-b, case-c, case-d) for **25mm** respectively for both cases of hole size.

4.5.2 Discussion on square columns

The results of the square column models implemented by finite element analysis shows that among the models of square columns with the holes of radius 10mm and 25mm the model in different cases (case-d, case-b, case-a, case-c) for **10mm** radius and;(case-b, case-a, case-c, case-d) for **25mm** predicts much higher vertical **compressive strength** in respective orders.

The analysis also reveals that, the position of the hole in the column also matters the capacity of the Reinforced concrete columns so that one can exactly point out where he/she should put the wiring duct in the column from this analysis. Here is the order of percentage increment in **load carrying capacity** in different placement of duct for electric wiring;(case-a, case-b, case-d, case-c) for **10mm** radius and;(case-a, case-b, case-c, case-d) for **25mm** respectively for both cases of hole size.

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

5.1 Conclusions

The inelastic behavior of 14 columns are investigated in the current study under the effect of increasing loading by employing the inelastic FE analysis program ABAQUS. Several parameters were investigated including; the hole location, types of the column and the hole dimensions with diameters of (20mm, 50mm), for the whole study with equal longitudinal length of column. The shape of columns was square and rectangular. The relative position of the holes in columns; in the middle for both rectangular and square with a diameter of 20mm and 50mm once at a time, and secondly, the same dimension of the hole is under taken for both types columns at position of hole $0.25*B$ and $0.25*H$ from H and B respectively, and at last stage the hole position is taken at $0.25*B$ from H, and $0.5*H$ from B respectively. The shape of hole is circle for both rectangular columns and square columns. The study focuses on the consequences of the investigated parameters on the deformation and ultimate resisting load. The conclusions made from this investigation are:

1. The theoretical results from Finite Element Analysis showed in general a good agreement with the experimental values.
2. The hole with larger diameter has significant effect on the column behavior; reducing the ductility and toughness of tested columns, in response, its remedial solution is selecting the position of the hole in a way that it will remain in the tolerable range of resistance.
3. To increase the resistance of the columns with the holes, rectangular column is better than square columns with the same size of the hole.
4. Among rectangular columns under investigation using the hole of duct for wiring as **case-d** of large size of hole is no advised and **case-c** of small size of hole is more favored, and among square columns under investigation using the hole of duct for wiring as **case-d** of large size of hole is no advised and **case-b** of small size of hole is more favored.

5.2 Recommendation

1. Further investigation is needed to study the effect of hole on the load carrying Capacity of slender column.
2. Further research is needed to be carried out to study the unbraced column with hole using steel pipe.
3. Investigating the same effect by changing the shape of the hole to square or rectangular, and extend the study for the other shapes of columns for regular and irregular shapes.

REFERENCES

1. ABAQUS Documentation. Version 6.10-1
2. ABAQUS/Standard Version 6.6 Analysis User's manual, 2006.
3. American Concrete Institutes. Building Code Requirements for Reinforced Concrete. USA, ACI 318. 2008.
4. Ashouf A.F. and Rishi G., (1999), Tests of reinforced concrete continuous deep beams with web openings, *ACI structural journal*, 97(3), pp 418-426.
5. Bakhteri, J., W. Omar and A. M. Makhtar. A Critical Review of the Reinforced Concrete Columns Concealing Rain Water Pipes in Multistorey Buildings. *Journal of Civil Engineering*. 2002. 15(2): 39-52.
6. Boukais, Said. A Three Dimensional Finite Element Analysis of Damages Reinforced Concrete Columns. M.Sc. Thesis. University of Dundee; 1989.
7. British Standards Institution. *Structural Use of Concrete, Part 1: Code of Practice for Design and Construction, Part 2: Code of Practice for Special Circumstances*. London, BS 8110. 1997.
8. British Standards Institution. *Testing Concrete, Part 121*. London, BS 1881.1983.
9. Campione, G., Fossetti, M., Papia, M., "Behavior Fiber-Reinforced Concrete Columns under Eccentrically Compressive Loads", *ACI Structural Journal*, Vol. 107, No. 3, May-June 2010, pp. 272-280.
10. Chandrupatla, Tirupathi R. and Belegundu, Ashok D. *Introduction to Finite Elements in Engineering*. 3rd. ed. USA: Prentice Hall. 2002.
11. Hoshikuma, J., and Priestly, M. J. N., "Flexural Behavior of Circular Hollow Columns with A Single Layer of Reinforcement under Seismic Loading", University of California, Structural Engineering Report no. SSRP-2000/13, 2000, pp. 149.
12. Inoue, S., and N. Egawa. 1996. Flexural and Shear Behavior of Reinforced Concrete Hollow Beams under Reversed Cyclic Loads. *Proceedings of 11th World Conference on Earthquake Engineering*. Paper No.1359.
13. Jahangir Bakhteri and Syed Ahmad Iskandar. Experimental Study of Reinforced Concrete Columns Concealing Rain Water Pipes. *Journal of Civil Engineering*. 2005. 43(B): 13-26.
14. James G. Macgregor and James K. Wight. Reinforced concrete mechanics and design. Upper Saddle River, NJ: Prentice Hall. 2009.

15. Kurowski P. M. *Finite Element Analysis for Design Engineers*. SAE International. 2004.
16. Lee, J., and Son, H., "Failure and Strength of High-Strength Concrete Columns Subjected to Eccentric Loads", *ACI Structural Journal*, Vol.1 97, No. 1, January-February 2000, pp. 75-85.
17. Mander, J. B., M. J. N. Priestley, and R. Park. 1983. Behavior of Ductile Hollow Reinforced Concrete Columns. *Bulletin of the New Zealand National Society for Earthquake Engineering*. 97(7): 1969-1990.
18. Meyer, Christian. *Design of Concrete Structures*. New Jersey: Prentice-Hall.1996.
19. Moaveni, Saeed. *Finite Element Analysis – Theory and Application with ANSYS*. 2th. ed. USA: Prentice Hall. 2003.
20. **Mosley**, W. H. and Bungey, J. H. and Hulse, R. *Reinforced Concrete Design*. 5th. ed. Bristol: J. W. Arrowsmith Ltd. 1999.
21. Mousavian, S. H. *Experimental Study of Reinforced Concrete Columns with + Pipes*. M.Sc. Thesis. University Technology Malaysia; 2010.
22. Simpson D., (2003), the provision of holes in reinforced concrete beams, *Concrete* (London), 37(3), pp 24-25.
23. William, K.J. and Warnke, E.D., (1975), Constitutive model for the Triaxial behavior of concrete, *Proceedings of the international association for bridge and structural engineering*, 19, p 174, ISMES, Bergamo, Italy
24. Yeh Y.-K., Mo Y. L., Yang "Seismic performance of hollow circular bridge piers" *ACI structural Journal*, 2001, 98(6), PP: 862-871
25. Y. L. Mo, Y. K. Yeh, C. T. Cheng, C. Tsai and C. C. Kao. *Seismic Performance and Retrofit of Hollow Bridge Columns*. *Earthquake Engineering and Engineering Seismology*. 2001. 3(1): 59-66.

APPENDIX

Appendix A, Material modeling calculation for Abaqus input

Concrete grade	C25	Concrete Plasticity Parameters	
Concrete density	2.4E-009		
Concrete elasticity		γ (Eccentricity)	0.1
E(GPa)	29.00	β (dilation angle)	36°
ν	0.20	K	0.667
		$\sigma_{bo} / \sigma_{co}$	1.16
		Viscosity parameters	0.00001

concrete damaged plasticity

Compression damage		Tension damage	
Damage Parameter(dc)	Inelastic Strain	Damage Parameter(dt)	Displacement
0.000000000	0.000000000	0.000000000	0.00
0.000000000	0.000224404	0.115065799	0.01
0.000000000	0.000502312	0.230131597	0.02
0.000000000	0.000866458	0.345197396	0.03
0.000000000	0.001035466	0.460263195	0.04
0.000000000	0.001303392	0.575328994	0.05
0.013483333	0.001802322	0.800000000	0.07
0.092174414	0.002354489	0.800341287	0.07
0.232536864	0.002952712	0.807532899	0.08
0.426249807	0.003027705	0.814724512	0.09
0.452885733		0.821916124	0.10
		0.829107737	0.11
		0.836299349	0.12
		0.843490961	0.13
		0.850682574	0.14
		0.857874186	0.15
		0.865065799	0.16
		0.872257411	0.17
		0.879449024	0.18
		0.886640636	0.19
		0.893832248	0.20
		0.901023861	0.21
		0.908215473	0.22
		0.915407086	0.23
		0.922598698	0.24
		0.929790311	0.25

$$E = 500\sqrt{\sigma_{cu}}$$

$$\frac{\sigma_c}{f_{cm}} = \frac{k\eta - \eta^2}{1 + (k - 2)\eta}$$

$$k = 1.05E_{cm} |\varepsilon_{c1}| / f_{cm}, \text{ and } \eta = \frac{\varepsilon_c}{\varepsilon_{c1}}$$

$$\sigma = f_t \left(\frac{\varepsilon_t}{\varepsilon} \right)^{(0.7+1000\varepsilon)}$$

$$\varepsilon_t = \frac{f_t}{E_c}$$

$$\eta = \frac{\varepsilon_c}{\varepsilon_{c1}}$$

compressive behavior

Yield stress	Inelastic strain
11.042148	0.000431
18.465033	0.000862
22.898635	0.001293
24.834751	0.001724
25.000000	0.001898
24.662917	0.002155
22.695636	0.002586
19.186574	0.003017
14.343758	0.003448
13.677857	0.003500

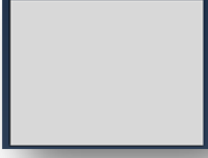


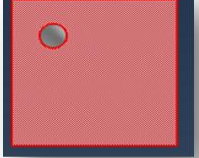
tensile behavior

Yield stress	Cracking strain
1.983446706	0.00
1.755219826	0.01
1.526992947	0.02
1.298766067	0.03
1.070539188	0.04
0.842312309	0.05
0.396689341	0.07
0.396012417	0.07
0.381748237	0.08
0.367484057	0.09
0.353219877	0.10
0.338955697	0.11
0.324691517	0.12
0.310427337	0.13
0.296163157	0.14
0.281898977	0.15
0.267634797	0.16
0.253370617	0.17
0.239106437	0.18
0.224842257	0.19
0.210578077	0.20
0.196313897	0.21
0.182049717	0.22
0.167785537	0.23
0.153521357	0.24
0.139257177	0.25

Appendix B,

Typical Column load carrying capacity descriptive data from software analysis

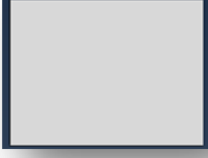


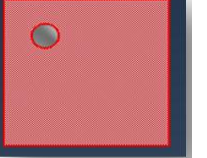
(For $\phi = 50mm$)

[For square columns; as a sample]: only some values are considered from whole analysis							
Case-a		Case-b		Case-c		Case-d	
							
Deff(mm)	LCC(N)	Deff(mm)	LCC(N)	Deff(mm)	LCC(N)	Deff(mm)	LCC(N)
0.977502401	1.56E+06	0.978646	1.51E+06	0.980239	1.45E+06	0.980277	1.38E+06
1.010744268	1.59E+06	1.02615	1.55E+06	1.02791	1.48E+06	1.02795	1.41E+06
1.044038912	1.61E+06	1.07395	1.58E+06	1.07590	1.51E+06	1.07593	1.44E+06
1.077378794	1.64E+06	1.12201	1.62E+06	1.12415	1.55E+06	1.12418	1.47E+06
1.110763912	1.67E+06	1.17028	1.66E+06	1.17262	1.58E+06	1.17263	1.51E+06
1.144179189	1.70E+06	1.21872	1.69E+06	1.22126	1.62E+06	1.22121	1.54E+06
1.177647243	1.72E+06	1.26714	1.73E+06	1.26985	1.65E+06	1.26963	1.57E+06
1.211205771	1.75E+06	1.31549	1.76E+06	1.31846	1.68E+06	1.31815	1.60E+06
1.244801997	1.78E+06	1.36474	1.80E+06	1.36809	1.72E+06	1.36786	1.63E+06
1.278390684	1.80E+06	1.41535	1.83E+06	1.419	1.75E+06	1.41883	1.66E+06
1.312130161	1.83E+06	1.46677	1.86E+06	1.47072	1.78E+06	1.47058	1.69E+06
1.346027969	1.86E+06	1.51882	1.90E+06	1.52313	1.81E+06	1.52291	1.72E+06
1.379790065	1.89E+06	1.57127	1.93E+06	1.57593	1.85E+06	1.57555	1.76E+06
1.413197802	1.91E+06	1.62398	1.97E+06	1.62907	1.88E+06	1.62853	1.79E+06
1.446447209	1.94E+06	1.67697	2.00E+06	1.68242	1.91E+06	1.68205	1.82E+06
1.47962122	1.97E+06	1.72987	2.03E+06	1.73572	1.94E+06	1.73549	1.85E+06
1.512795232	2.00E+06	1.78266	2.07E+06	1.78885	1.97E+06	1.78855	1.88E+06
1.545999401	2.02E+06	1.83481	2.10E+06	1.84103	2.00E+06	1.8405	1.90E+06
1.579346822	2.05E+06	1.88604	2.13E+06	1.89247	2.03E+06	1.89111	1.93E+06
1.61291289	2.08E+06	1.93805	2.15E+06	1.94494	2.05E+06	1.93761	1.95E+06
1.646614669	2.10E+06	1.98897	2.17E+06	1.9913	2.07E+06	1.9485	1.95E+06

1.68030891	2.13E+06	2.00114	2.17E+06	2.00128	2.07E+06	1.96015	1.96E+06
1.713890056	2.16E+06	2.01585	2.18E+06	2.0047	2.07E+06	1.96279	1.96E+06
1.747320412	2.18E+06	2.0205	2.18E+06	2.00861	2.07E+06	1.96346	1.96E+06
1.780705531	2.21E+06	2.0228	2.18E+06	2.00828	2.07E+06	1.96348	1.96E+06
1.81425652	2.23E+06	2.02277	2.18E+06	2.00683	2.07E+06	1.96025	1.96E+06
1.848131709	2.26E+06	2.01925	2.18E+06	1.99950	2.06E+06	1.94475	1.93E+06
1.883167989	2.28E+06	1.99602	2.14E+06	1.98487	2.04E+06	1.91801	1.89E+06
1.920262565	2.30E+06	1.96278	2.07E+06	1.96436	2.00E+06	1.88799	1.82E+06
1.959860271	2.32E+06	1.92983	1.99E+06	1.93125	1.93E+06	1.86017	1.75E+06
2.001184534	2.34E+06	1.90062	1.91E+06	1.88862	1.81E+06	1.82596	1.66E+06
2.042342927	2.35E+06	1.86569	1.80E+06	1.85459	1.72E+06	1.79946	1.58E+06
2.077303811	2.37E+06	1.83755	1.72E+06	1.82665	1.64E+06	1.77801	1.52E+06
2.084451302	2.37E+06	1.81473	1.65E+06	1.8033	1.57E+06	1.75911	1.46E+06
2.086833799	2.38E+06	1.79587	1.59E+06	1.78457	1.52E+06	1.74358	1.42E+06
2.089548037	2.38E+06	1.78071	1.55E+06	1.76846	1.47E+06	1.73004	1.38E+06
2.091063486	2.38E+06	1.76678	1.51E+06	1.75561	1.43E+06	1.71966	1.35E+06
2.090196438	2.38E+06	1.75654	1.48E+06	1.74559	1.41E+06	1.71173	1.32E+06
2.084730266	2.37E+06	1.74808	1.45E+06	1.73438	1.37E+06	1.70136	1.30E+06
2.072697147	2.35E+06	1.74007	1.42E+06	1.7264	1.35E+06	1.69595	1.28E+06
2.053765342	2.31E+06	1.73632	1.41E+06	1.72413	1.34E+06	1.69365	1.28E+06
2.030822497	2.25E+06	1.73488	1.40E+06	1.72166	1.34E+06	1.6927	1.27E+06
2.008528054	2.17E+06	1.73559	1.39E+06	1.71896	1.33E+06	1.69138	1.27E+06
1.988872452	2.09E+06	1.73712	1.39E+06	1.71822	1.33E+06	1.6917	1.26E+06
1.971508873	2.01E+06	1.73896	1.39E+06	1.71807	1.33E+06	1.69561	1.26E+06
1.958194031	1.94E+06	1.74257	1.38E+06	1.7181	1.33E+06	1.69818	1.26E+06
1.946168452	1.88E+06	1.75113	1.37E+06	1.71845	1.33E+06	1.70413	1.25E+06
1.935816653	1.82E+06	1.76377	1.36E+06	1.71949	1.33E+06	1.7055	1.25E+06
1.929664381	1.78E+06	1.77599	1.35E+06	1.71994	1.32E+06	1.70648	1.25E+06
1.924673201	1.73E+06	1.79098	1.35E+06	1.72105	1.32E+06	1.70925	1.24E+06
1.918641562	1.69E+06	1.80471	1.35E+06	1.72355	1.32E+06	1.7155	1.23E+06

Appendix C,
Typical Column compressive strength descriptive data from software
analysis (for $\phi = 50mm$)

[For square columns; as a sample]: only some values are considered from whole analysis

Case-a		Case-b		Case-c		Case-d	
							
Strain	Stress (N/mm ²)	strain	Stress (N/mm ²)	strain	Stress (N/mm ²)	strain	Stress (N/mm ²)
3.40E-05	4.88731	0.000171679	4.98147	0.000184705	5.35867	0.000185839	5.39044
3.66E-05	5.25838	0.000184713	5.35967	0.000197739	5.73681	0.000199053	5.77372
3.92E-05	5.6294	0.000197748	5.73788	0.000210773	6.11495	0.000212276	6.15726
4.17E-05	6.00054	0.000210783	6.11609	0.000223808	6.4931	0.000225508	6.54106
4.43E-05	6.37163	0.000223818	6.49431	0.000236842	6.87124	0.000238748	6.9251
4.69E-05	6.74271	0.000236853	6.87252	0.000249877	7.24938	0.000251996	7.30935
4.95E-05	7.1138	0.000249888	7.25074	0.000262911	7.62753	0.000265251	7.69382
5.21E-05	7.48488	0.000262923	7.62895	0.000275945	8.00567	0.000278512	8.07847
5.46E-05	7.85597	0.000275958	8.00717	0.00028898	8.38382	0.00029178	8.46332
5.72E-05	8.22706	0.000288993	8.38538	0.000302014	8.76196	0.000305054	8.84833
5.98E-05	8.59814	0.000302028	8.7636	0.000315049	9.14011	0.000318333	9.2335
6.24E-05	8.96923	0.000315063	9.14181	0.000328083	9.51825	0.000331617	9.61882
6.50E-05	9.34032	0.000328098	9.52003	0.000341118	9.8964	0.000344906	10.0043
6.75E-05	9.7114	0.000341133	9.89824	0.000354152	10.2745	0.000358184	10.3894
7.01E-05	10.0825	0.000354167	10.2764	0.000367126	10.651	0.00037091	10.763
7.27E-05	10.4534	0.000367137	10.6528	0.000379172	11.0069	0.000381318	11.0349
7.52E-05	10.818	0.000379123	11.0076	0.000401635	11.1763	0.000402239	11.158
8.32E-05	11.0925	0.000401464	11.1949	0.0004303	11.3644	0.000430199	11.3398
0.000103499	11.2627	0.000430006	11.3876	0.000463397	11.5798	0.000462911	11.552
0.000128329	11.4636	0.00046295	11.6083	0.000499828	11.8155	0.000499105	11.7863

0.000156452	11.6887	0.000499216	11.8499	0.000538914	12.0669	0.000537978	12.0378
0.00018708	11.9317	0.000538126	12.1081	0.000580115	12.3308	0.000578989	12.3031
0.000219726	12.1888	0.000579125	12.379	0.000623081	12.6049	0.000621719	12.5796
0.000254062	12.4573	0.000621878	12.6607	0.00066749	12.887	0.00066585	12.8653
0.000289821	12.7352	0.000666066	12.9508	0.000713039	13.1753	0.000711109	13.1584
0.00032677	13.0205	0.000711387	13.2474	0.000759527	13.4683	0.000757269	13.4575
0.000364686	13.3115	0.000757613	13.549	0.000806782	13.7651	0.000804176	13.7616
0.000403426	13.6071	0.00080458	13.8545	0.000854688	14.0649	0.000851709	14.0699
0.000442842	13.9061	0.000852186	14.1633	0.000903158	14.3671	0.000899782	14.3818
0.00048285	14.208	0.000900342	14.4748	0.000952112	14.6714	0.000948313	14.6969
0.000523391	14.5123	0.000948965	14.7885	0.00100148	14.9771	0.000997233	15.0146
0.000564403	14.8184	0.000997985	15.1038	0.00105121	15.284	0.00104649	15.3347
0.000605832	15.1261	0.00104734	15.4204	0.00110124	15.5917	0.00109602	15.6567
0.000647639	15.4351	0.00109699	15.738	0.00115153	15.9	0.00114579	15.9805
0.000689778	15.7449	0.00114689	16.0564	0.00120204	16.2086	0.00119575	16.3058
0.000732211	16.0554	0.00119698	16.3751	0.00125275	16.5177	0.00124589	16.6325
0.000774897	16.3663	0.00124723	16.694	0.00130387	16.829	0.00129617	16.953
0.000817884	16.6782	0.00129783	17.0155	0.00135547	17.1408	0.00134638	17.2682
0.00086135	16.9927	0.00134884	17.3398	0.00140854	17.4547	0.00139778	17.5854
0.000905567	17.308	0.00140111	17.6639	0.00146444	17.8122	0.00145195	17.939
0.000951574	17.6275	0.00145609	18.0273	0.00152161	18.2452	0.00150739	18.3351
0.000999218	17.9963	0.00151234	18.4646	0.00157928	18.6872	0.00156285	18.7583
0.00104599	18.431	0.00156893	18.9253	0.0016372	19.1206	0.00161808	19.1729
0.00109322	18.8656	0.0016257	19.3737	0.00169498	19.5428	0.00167257	19.5976
0.00114091	19.294	0.00168229	19.801	0.00175275	19.9448	0.00172672	20.0424
0.0011885	19.7118	0.00173895	20.2176	0.00181128	20.3479	0.00178183	20.4668
0.00123608	20.1125	0.00179633	20.6263	0.00187022	20.7602	0.00183636	20.8814
0.00128387	20.514	0.00185396	21.0482	0.00192878	21.1617	0.00189001	21.2747
0.00133165	20.9269	0.00191114	21.4593	0.00198784	21.546	0.00194303	21.6358
0.0013795	21.3296	0.00196829	21.8521	0.00205251	21.9175	0.00199515	21.986
0.00142901	21.7178	0.00202902	22.2385	0.00212675	22.1166	0.00203673	22.2296
0.00148635	22.0846	0.00209817	22.5512	0.00218947	22.2866	0.00204454	22.2695
0.00155457	22.3075	0.00216766	22.745	0.00220078	22.3122	0.0020434	22.2366

0.00161984	22.5225	0.00218347	22.7924	0.00220407	22.3175	0.00203914	22.1205
0.00163439	22.5653	0.00219856	22.8332	0.00220524	22.3097	0.00203679	22.0558
0.0016446	22.5731	0.00220193	22.8364	0.00219757	22.0973	0.00203133	21.9034
0.00164449	22.5442	0.00219755	22.715	0.00219253	21.9573	0.00201768	21.5176
0.00164189	22.1756	0.00219456	22.6321	0.00217631	21.5046	0.0019828	20.5238
0.00164034	21.9554	0.0021849	22.3611	0.00214897	20.7404	0.0019414	19.3407
0.00163577	21.3202	0.00214058	21.1174	0.00211703	19.8512	0.00191018	18.4554
0.0016268	20.0255	0.00209334	19.7977	0.0020778	18.7655	0.00188808	17.8347
0.00161969	19.0325	0.00205796	18.8189	0.00203819	17.6763	0.00186493	17.1897
0.00161444	18.2716	0.00203168	18.0974	0.00201041	16.9158	0.00184968	16.7721
0.00160917	17.4646	0.00200369	17.3294	0.00198739	16.289	0.00183686	16.4259
0.00160364	16.4912	0.0019813	16.7179	0.00196731	15.7395	0.00182216	16.0235
0.00159998	15.8431	0.00196265	16.2117	0.0019519	15.3211	0.00181221	15.7541
0.0015973	15.5066	0.00194806	15.8166	0.0019379	14.9447	0.00180283	15.4959
0.00159449	15.3192	0.0019373	15.5234	0.00192766	14.669	0.00179806	15.3624
0.00159249	15.3134	0.00192821	15.2692	0.00191946	14.454	0.00179679	15.3232
0.00159311	15.4801	0.00192343	15.1473	0.00190729	14.1136	0.00179106	15.1472
0.0015978	15.9525	0.00191887	15.0268	0.0019034	13.9667	0.00179123	15.1409
0.00160277	16.3734	0.00192025	15.0612	0.00190214	13.9144	0.001796	15.2756

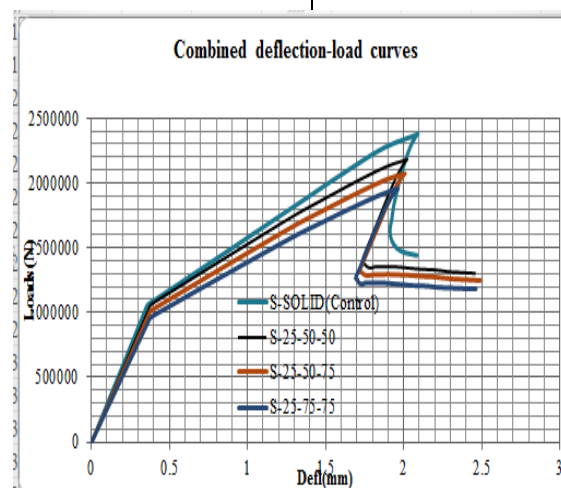
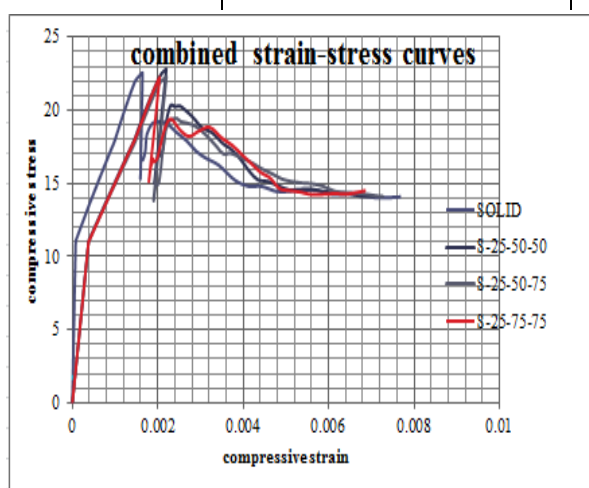


Figure 5.1 graphs which represents the sample data taken from software analysis