ADDIS ABABA INSTITUTE OF TECHNOLOGY
SCHOOL OF CIVIL AND ENVIRONMENTAL ENGINEERING

Numerical Simulation of Moment and Lateral Load Resisting Capacity of Rectangular Reinforced Concrete Column with Bundle Bars

By
Serkalem Argaw

October, 2018

ADDIS ABABA

ADDIS ABABA INSTITUTE OF TECHNOLOGY
SCHOOL OF CIVIL AND ENVIRONMENTAL ENGINEERING

Numerical Simulation of Moment and Lateral Load Resisting Capacity of Rectangular Reinforced Concrete Column with Bundle Bars

By
Serkalem Argaw

A Thesis in Structural Engineering Stream

October, 2018

Addis Ababa

The undersigned have examined the thesis entitled '**Numerical Simulation of Moment and Lateral Load Resisting Capacity of Rectangular Reinforced Concrete Column with Bundle Bars'** presented by **Serkalem Argaw**, a candidate for the degree of **Master of Science** and hereby certify that it is worthy of acceptance.

Dr.Ing.	Bedilu Habte	_______________	_____________
	Advisor	Signature	Date
Dr.Ing.	_______________	_____________	_____________
	External Examiner	Signature	Date
Dr.Ing.	_______________	_____________	_____________
	Internal Examiner	Signature	Date
Dr.Ing.	_______________	_____________	_____________
	Chairman Examiner	Signature	Date

Declaration

I, the undersigned, declare that this thesis is my work and all sources of materials used for the thesis have been duly acknowledged.

Name: Serkalem Argaw

Signature: _____________________

Place: Addis Ababa Institute of Technology

Addis Ababa University

Date of submission: June, 2018

Advisor: Dr.Ing. Bedilu Habte

Signature: _________________

Acknowledgement

First I would like to thank the Almighty God for giving me patience, strength, and determination to accomplish this work.

Next, I would like to express my sincere appreciation to my advisor, Dr.Ing. Bedilu Habte, for his support, assistance, understanding and motivation throughout this thesis work.

Finally, I have no words to express my great appreciation and thanks to my family and friends for their lovely encouragement.

Serkalem Argaw

Abstract

This study aims at modeling and analyze moment and lateral load resisting capacity of rectangular reinforced concrete column with bundled bars. Four different arrangements of reinforcing bars are considered in this research: the first case was column with uniformly disturbed longitudinal bars along with the four faces 20Ø16 (conventional one), the second case, column with 2Ø16 bars bundled between the corner bars on four faces and 3Ø16 bars bundled at the corners, the third case, column with 1Ø16 bar between the corner bars on four faces and 4Ø16 bars bundled at the corners, and the fourth case for this paper was column with one bar at the center of the faces and 1Ø32 bar at the corner equivalent to the 4Ø16 bundled bars. For all cases axial load vs. moment capacity were determined using interaction curve for different strain profile and also lateral deformation by applying static lateral load.

Finite Element modeling of such analysis requires the determination of the nonlinear properties of each component in the structure, quantified by nodal displacement, global forces, section forces, and deformation capacities. A pushover analysis is performed on two dimensional cantilevered reinforced concrete column using Opensees and the results of the four different cases the inelastic behavior of the structural model are extracted and compared respectively. It was found that column with bundled bars have higher moment resisting capacity; and from the comparison of lateral force vs. displacement and axial force vs deformation relationship it is observed that columns with bundled bars have higher lateral load resisting capacity.

Table of Content

List of Tables

List of Figures

List of abbreviations and symbols

A_g	Areas of the concrete section
A_{st}	Total area of the steel in the section
A_{si}	Area of the steel in each row
BSec	Column width
b	Strain hardening ratio
barAreaSec	Area of longitudinal-reinforcement bars
C_c	Coefficient for area for the stress block
ColSec	Column Section
coreY	Distance of the section from z-axis to the inner edge of the cover concrete
coreZ	Distance of the section from y-axis to the inner edge of the cover concrete
CoverSec	Column covers to reinforcing steel NA.
coverY	Distance of the section from z-axis to the outer edge cover concrete
coverZ	Distance of the section from y-axis to the outer edge cover concrete
CrdTransf	Coordinate transformation
C_{R1}	Control the transition from elastic to plastic branches
C_{R2}	Control the transition from elastic to plastic branches
DB	Displacement based
Disp	Displacement
dispBeamColumn	Distributed-Plasticity, Displacement–Based beam column element
d_i	Depth of the steel for each row
ε_{c2}	Strain at maximum strength
ε_{cu2}	Ultimate strain
E_c	Modulus of Elasticity of the concrete
E_s	Modulus of Elasticity of the steel
E_{ts}	Tension softening stiffness
e_{sci}	Strain in the steel for each row
e_y	Yield strain of the steel

ele	Element
eleTag	Element tag
eps1C:	Concrete strain at maximum stress for confined
eps1U	Maximum strain for unconfined concrete
eps2C:	Concrete strain at ultimate stress for confined
eps2U	Ultimate strain for unconfined concrete
FB	Force based
f_{bd}	Bond stress
f_c	Stress in the concrete
f_{cd}	Design compressive strength of the concrete
f_{ck}	Compressive strength of concrete
f_{si}	Stress in the steel
f_{yk}	Tensile strength of steel
f_{c1C}	Confined concrete (mander model), maximum stress
f_{c1U}	Unconfined concrete (todeschini parabolic model), maximum stress
f_{c2C}	Ultimate stress for confined concrete
f_{c2U}	Ultimate stress for unconfined concrete
FEA	Finite Element Analysis
f_{tC}	Tensile strength for confined concrete
f_{tU}	Tensile strength for unconfined concrete
f_y	Steel yield stress
geomTransf	Geometric transfer
HSec	Depth of the section
I	Moment of inertias of the cross section
IDcolTransf	Identity of column transformation
IDconcCore	Material Identity number of confined core concrete
IDconcCover	Material Identity number of confined core concrete
IDreinf	Material Identity number of reinforcement
K_{fc}	Ratio of confined to unconfined concrete strength
KN	Kilo newton
l_b	Development length
M	Meter
m	Number of row
M_c	Ultimate moment carrying capacity of the concrete

M_s	Ultimate moment carrying capacity of the steel
M_u	Ultimate moment
M_o	Pure bending point
maxU	Maximum displacement
n_b	Number of bar in a bundle
ndf	Number of degree of freedom
ndm	Dimension of problem (1, 2, 3)
nfCoreY	Number of fibers for core concrete in y-direction
nfCoreZ	Number of fibers for core concrete in z-direction
nfCoverY	Number of fibers for cover concrete in y-direction
nfCoverZ	Number of fibers for cover concrete in z-direction
nfCoverY1	Number of fibers for cover concrete in y-direction
nfCoverZ1	Number of fibers for cover concrete in z-direction
np	Number of integration points
numBarsSec	Number of longitudinal-reinforcement bars in steel layer.
ɳ	Exponent/ denotation of EBCS EN 1992-1-1
P	Axial load
p_{si}	Axial load in steel
P_u	Total axial load
P_c	Compression force in the concrete
Po	Pure axial compression point
Quadr	Quadrilateral
RC	Reinforced concrete
R_o	Control the transition from elastic to plastic branches
Sec	Second
SecTag	Tag for symmetric section
Tcl	Tool command language
V	Shear force
x	Depth of equivalent rectangular block
α_{cc}	Coefficient taking account of long term effects
γ_c	The partial safety factor for concrete
γ_s	The partial safety factor for steel
σ_{sd}	Design stress of the bar at the position from where the anchorage is measured

$Ø_n$	Equivalent diameter of bundle
Ø	Diameter of bars
#	Comment
2D	Two dimension

Chapter 1: Introduction

1.1 Background

Columns are the primary structural elements that transfer the loads of a building vertically to the foundation. Reinforced concrete is one of the main material types used for columns construction. Numerous characteristics of a reinforced-concrete column affect the overall strength of the column: length, load eccentricity, cross-sectional area, end connections, and concrete and steel strength.

In the construction of reinforced concrete structures, it is sometimes necessary to place reinforcement in bundle. Bundling of bars becomes necessary when large numbers of bars are required to be accommodated in a structural member. Concrete codes dictate minimum criteria for spacing of the reinforcing bars to ensure that during construction fresh concrete can be placed easily in between and around the bars. Therefore, when there are large numbers of bars required to be provided based on design, it may not be possible to place the bars separately with necessary clearance. In such cases, there are two options:

1. Increase the size of the member (columns, beams).
2. Bundle the bars in groups of two, three or four bars.

However, option 1 means unnecessary cost implications due to the increase in the volume of concrete. Hence, engineers mostly resort to bundling of bars. Current codes allow as many as four bars to be placed in a group or bundle. There are provisions for increasing the length of anchorage based on the size of bundle, but in general there is insufficient guidance in the code to aid the designer using bar bundle. Currently in our country most designs and constructions are done by distributing large number of single bars on the faces of member which results in large cross section of the member very little information is available on the subject. This research contributes to the greater understanding of moment resisting capacity and lateral load resistance of column with bundle bars. For a reliable simulation of the nonlinear response of RC structures a proper material model for reinforcing bars with compressive axial and lateral loads would be essential.

1.2 Statement of the Problem

The present code for the structural design of RC structures lacks detailed information regarding bundling of reinforcing bars. Furthermore, very little information is available in published literature on tests of structures with bundled bars capacity, but not clearly stated which bundle size more safe and economical. Up to now, the design of column is done based on uniformly distributed longitudinal bars along the faces of column. When large number of bars is needed according to the design the designers choose increasing of cross section to provide the intended minimum clear space between longitudinal bars. Alternatively, they may choose larger diameter bars, which may increase the total cost of construction. Even if in practical situation this recommendation of the code is violated, when the longitudinal bar of column and the main bar of beams come at the same point of intersection at beam-column connections, or due to poor workmanship the column bars may shift from the position towards the center of the column and it results in the reduction of load caring capacity of the column and disturbance of the necessary clearance. Using FEA software tools to model and understand the response of RC structures with bundled bars under static lateral loading is very helpful in terms of time and cost compared to experimental approaches.

1.3 Objective

The objective of this thesis is to study and numerically simulate the moment and static lateral load capacity of column with bundled bars. The following objectives are set to achieve the aim of the study:

1. Examine the effect of rectangular column with bundle bars and comparing moment resisting capacity of the column with that of uniformly distributed single bars with the same area as a bundle.
2. Model and perform a non-linear analysis to simulate a reinforced concrete structures under gravity and static lateral loads using Opensees FE software package.
3. Determine the effect of size of bundled bars on the response of RC structures.

1.4 Scope of the study

This study deals with the moment and lateral load capacity for different bar arrangements. The work does not include laboratory investigation. The scope of this study is limited to moment capacity determination analytically using interaction curve and lateral displacements of column using Opensees model with bundled bars.

1.5 Methodology

To complete this work a five-phased methodology can be described as follows:

1. Literature review- related works will be reviewed to get a deeper understanding about column bar arrangements, capacity determination and different works done by different researchers using Opensees software.

2. Designing the model based on the parameters to check moment carrying capacity and presentation of result.
3. Modeled and analyzed, employing nonlinear Finite Element Modeling using Opensees software
4. Discuss the obtained result
5. State conclusion and recommendations

1.6 Application of the Research

1. The developed model using Opensees finite element software will be applicable for analysis of two dimensional reinforced concrete column with static load only.
2. The object oriented program is flexible to modify dimensions, material strength, geometry, loading, analysis type etc.
3. Students may use this for future research in line with upgrading the object oriented program to handle many other types of structural analysis.

1.7 Contents of thesis

This thesis consists of six chapters, which are arranged in a logical sequence for the reader to follow. Chapter one contains introduction, objectives, scope and methodology of the work. Chapter two discusses the work done by various researchers regarding bundling of bars in reinforced concrete structural members and code recommendations. The third chapter deals with the determination of moment capacity of column with different number of bundled bars using strain profile and comparison of the result by constructing interaction curve. The fourth chapter deals with the details of the structure modeling in Opensees and theory related to the Opensees, material modeling, and analytical programming procedure involved in modeling of the column. In the fifth chapter, the result of non-linear analysis of column in different reinforcement arrangement is discussed in details. In the sixth chapter, conclusions and recommendations of the study are given followed by the references.

Chapter 2: Literature Review

2.1 Bundled bars

Bars may be arranged singly, or in pairs in contact, or in groups of three or four bars bundled in contact. Bundled bars shall be enclosed within stirrups or ties. Bundled bars shall be tied together to ensure the bars remaining together. Bars larger than 32 mm diameter shall not be bundled, except in columns based on [1,2,].

ES EN 1992-1-1 code specifies factors to be used as multipliers of the basic equation, accounting for clear spacing, and cover, which may increase the development length and provides size of bundles and formulas to determine equivalent area of bundles. A maximum of two bundled bars in any one plane is implied (three or four adjacent bars in one plane are not considered as bundled bars) [1,2].

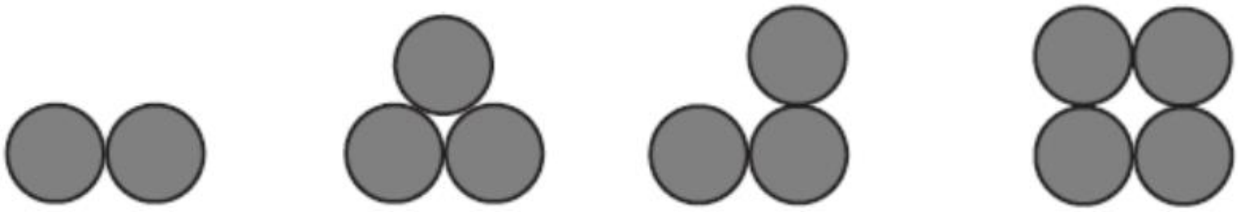


Fig 2.1. Possible Reinforcing Bars Bundling Schemes

2.1.1 Bundled bar development length

The anchorages of the bars of a bundle can only be straight anchorages. The anchorage of a bundle is dependent upon the anchorage of the individual bars. The basic development length for diameter less than 32mm is given by:

$$l_b = \frac{\phi}{4}\left(\frac{\sigma_{sd}}{f_{bd}}\right) \qquad \text{……………………………………………… Equation 2.1}$$

The anchorages shall be staggered; for bundles of 2, 3, or 4 bars the staggering shall be respectively. 1.2, 1.3 or 1.4 times the anchorage length of the individual bars. Joints can be made on only one bar at a time but at any section there shall be no more than four bars in bundle. Bars in a bundle shall terminate at different points spaced apart by not less than 40 times the bar diameter except for bundles stopping at a support based on [1,2,3].

2.1.2 Bundle bar equivalent area

For design bundles of bars containing n bars having the same diameter are replaced by a single notional bar having the same center of gravity. And an equivalent diameter: [1]

$$\phi_n = \phi\sqrt{n_b} \leq 55 \quad \text{...Equation 2.2}$$

$n_b \leq 4$ for vertical bars in compression and for bars of a lapped joint

$n_b \leq 3$ for all other cases.

The equivalent diameter ϕ_n is taken in to account in evaluating the minimum cover. However, the cover provided shall be measured from the actual outside contour of the bundle.

In [3] Hanson and Rieffenstahl reported the results of an investigation of the feasibility of using bundle bar details in beams & columns. The first half of the program consisted of tests of part of columns. The first column of a pair had conventionally spaced reinforcement the second bundled reinforcement. The same number of bars used in each set of tests. After the test authors state that "…when only external bar perimeters were used to calculate bond stress, there was no systematic difference in ultimate bond stress developed by spaced and bundled bars." The authors' conclusion was that the bundling reinforcement is a safe detailing practice, as long as each bar is" individually well anchored". They also recommend that bond stress for the bars be computed on the basis of the bundle in direct contact with the concrete.

Also [2] state bundles of standard bar size can save space & reduce congestion for placement & compaction. Bundling of parallel reinforcement bars in contact is permitted but only if ties enclose such bundles, group of parallel reinforcing bars bundled in contact as one unit are limited to four in any bundle.

2.1.3 Crack Control for Bundled Reinforcement

The literature search located two other papers, dealing with applying code provisions for crack control when detailing bundled bars. The discussions are still interesting in that they involve questions about bundled bar behavior.

Daniel [3] presented Nawy's proposed a method for applying the crack control provision of ACI 318-71 to bundled bars. The concern was that there was no specific instruction in the code for interpreting the equation for bundled geometries. Nawy proposed that designers modify the equation with parameters which account for the change in exposed bar area when the bars grouped together.

Daniel [3] presented a similar modification in (Lutz 1975). He felt that Nawy's modification was confusing. Instead, he presented a different method of modifying the code equations, based on slightly different assumptions about how grouping the bars changed their effective perimeter. The argument over which perimeter reflects behavior most accurately is particularly interesting in that it points to a good deal of confusion over the issue: Lutz states, "There is very little experimental information that could be used to aid in evaluating the expression presented."

2.2 Moment capacity of column

A column is a vertical structural member supporting axial compressive loads, with or without moments. Almost all compression members in concrete structures are subjected to moments in addition to axial loads. These may be due to misalignment of the load on the column, or may result from the column resisting a portion of the unbalanced moments at the ends of the beams supported by the columns.

When determining the ultimate moment resistance of reinforced or pre-stressed concrete cross sections assume: [1]

- Plane section remain plane
- The tensile strength of the concrete is ignored
- The maximum strain in concrete of the outermost compassion fibber is taken as 0.0035.
- The maximum compression strain in concrete in axial compression is taken as 0.002.
- The stress in the concrete in compression are derived from the design stress /strain relationship.
- The maximum compressive strain at the highly compressed extreme fibber in concrete subjected to axial compression and bending & when there is no tension on the section shall be 0.0035-0.75 times the strain at the lest compressed fibber. [5]
- The maximum Strain in the tension reinforcement in the section of failure shall not be less than $\left(\frac{fg}{1.15Es}\right) + 0.002$ at the least compressed fibber.

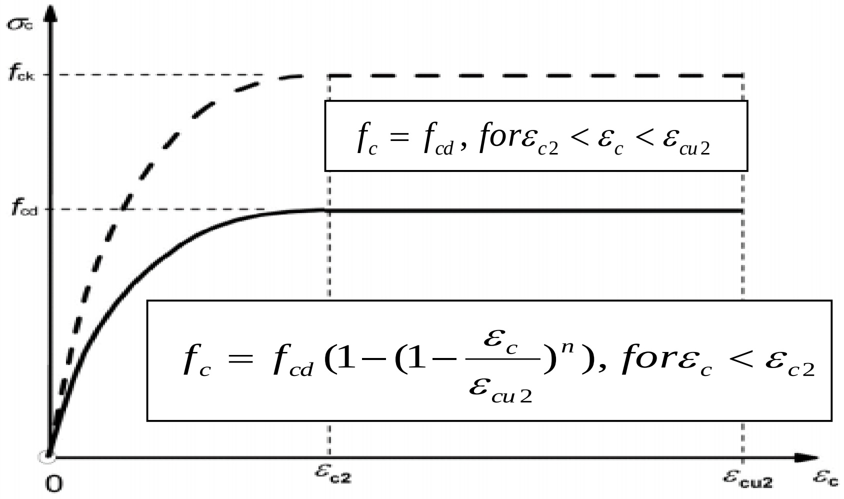


Fig 2.2. Parabola-rectangle for concrete under compression

2.2.1 Limiting strain diagram for different depth of neutral axis

To study the behavior of rectangular column can be explained with the help of limiting strain diagram & interaction curve. The limiting strain diagram shows the different strain profile depend on the position of neutral axis while the interaction curve shows ultimate load bearing capacity & resisting moment of column depend on the position neutral axis.

Compression failure mode

Fig 2.3 shows strain profile for neutral axis at infinite distance. When neutral axis is at infinite distance then it attains maximum axial compression strain of 0.002 at highly compressed edge as well as least compressed edge [5].

The point E in the Fig. 2.9 shows the region of pure axial compression.

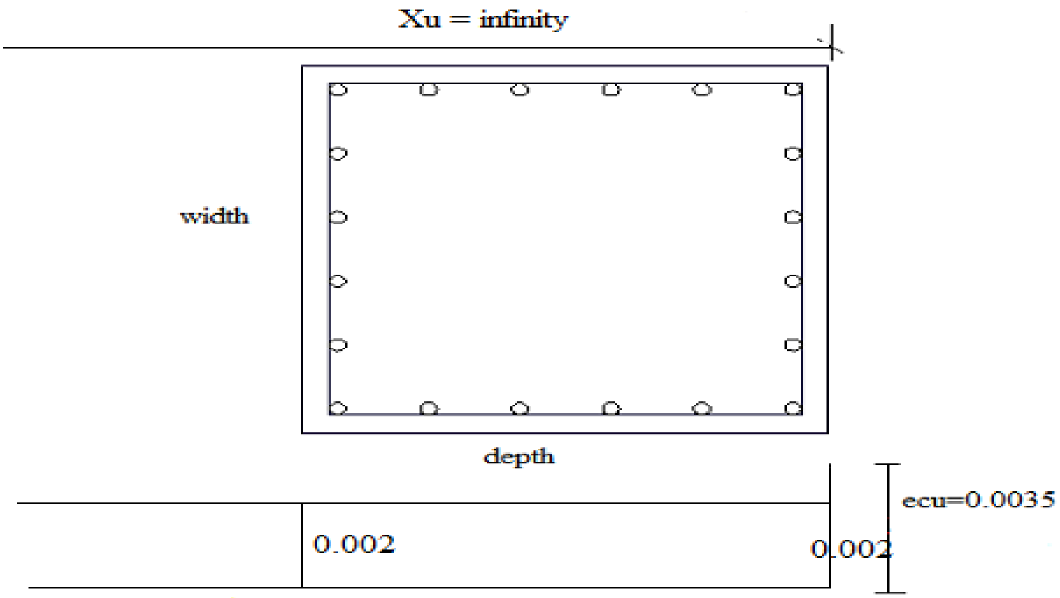


Fig 2.3 Limiting Strain Diagram for Neutral Axis at Infinite Distance

Fig. 2.4 shows the limiting strain profile for neutral axis outside the section. Here the strain at the maximum compressive edge is taken as 0.0035-0.75esc. The axial compressive stain in least compressed edge will be less than 0.002.

The region AE in the Fig. 2.9 shows the axial compression region where the strain decreases from 0.002 to 0.

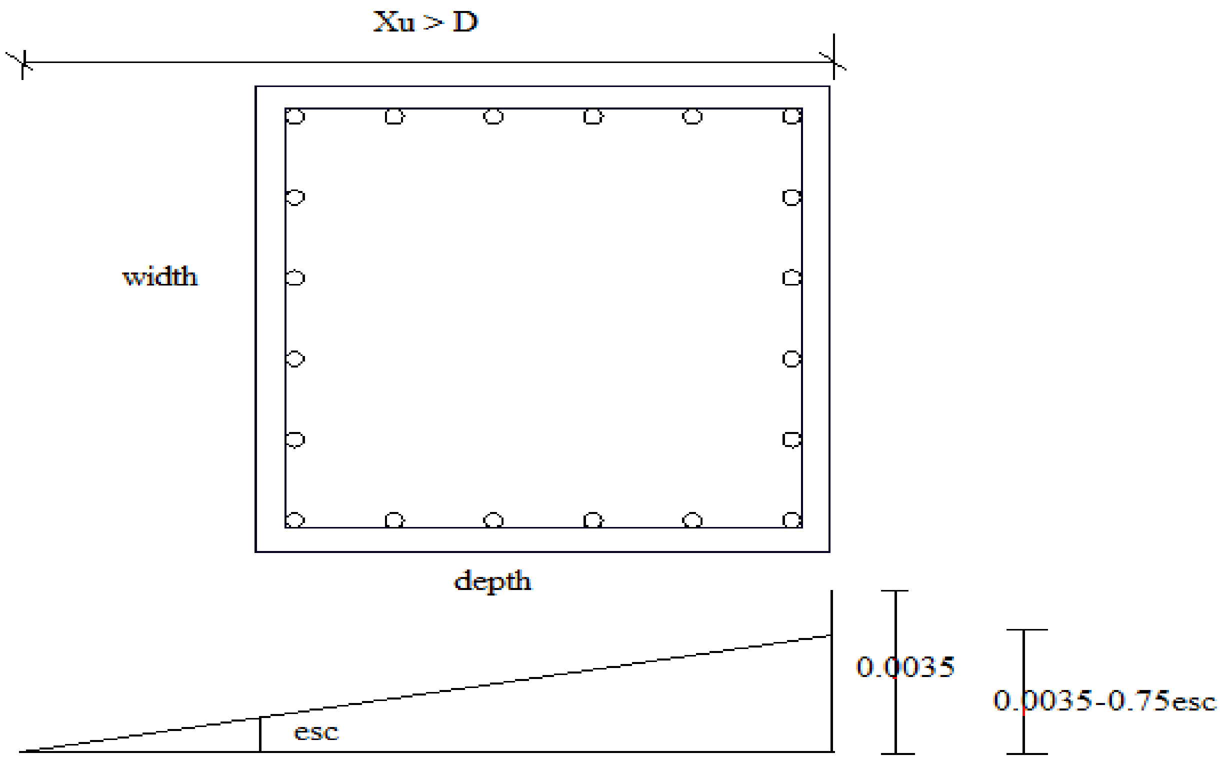


Fig. 2.4 Limiting Strain Diagram for Neutral Axis Outside the Section

Fig. 2.5 shows the limiting strain diagram which shows the attainment of maximum compressive strain in concrete at the highly compressed edge is 0.0035 and zero strain at the least compressed edge.

The point A in the Fig. 2.9 of interaction curve is the point where the axial compressive strain becomes zero. It is the point where there will be no compressive strain or tensile strain. As per the assumptions of limit state of collapse it is also the point due to which the compressive strain in the concrete will reaches its maximum compressive strain of 0.0035. This strain remains same for all the neutral axis positions inside the section.

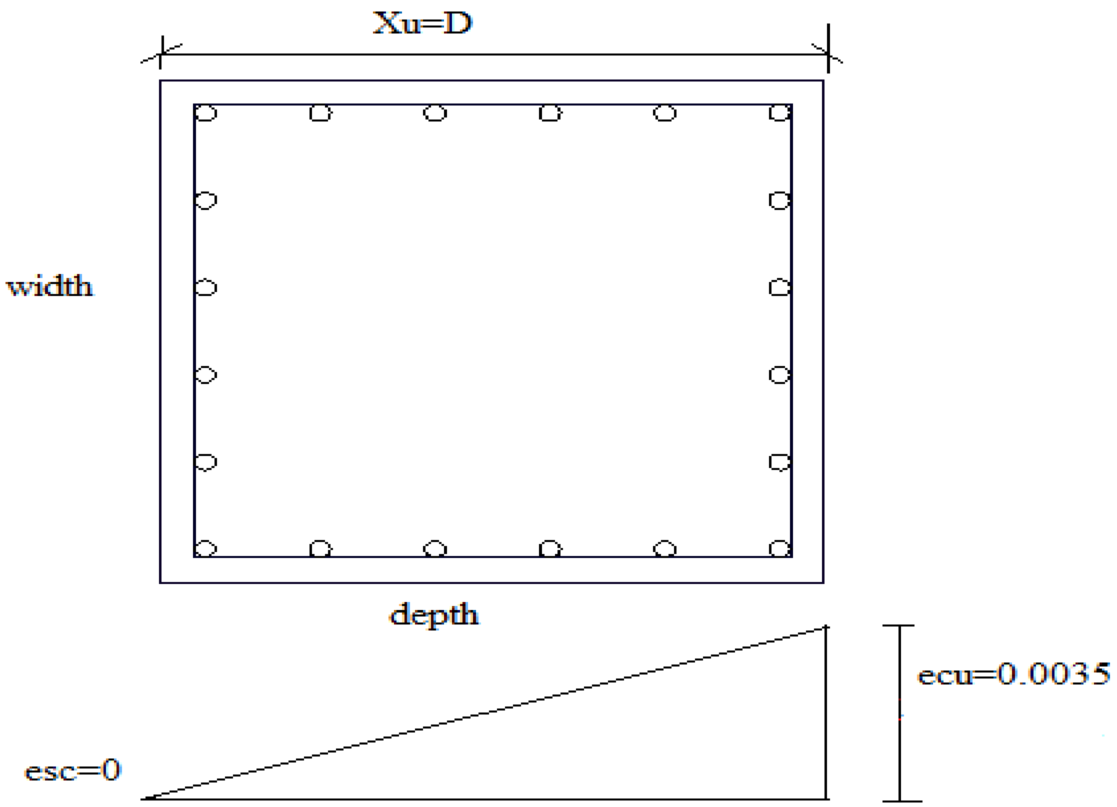


Fig. 2.5 Limiting Strain Diagram for Neutral Axis at Distance Equal to the Depth

Fig. 2.6 shows the limiting strain diagrams for the compressive failure zone. Here the tensile strain in steel is attained at the least compressed edge of the steel. The tensile strain will range from zero to 0.0035. The values Pu & Mu in compressive failure zone is taken to design a column because compressive failure zone gives under reinforced section. The steel fails before concrete developing cracks in the concrete giving enough indications of failure.
The region between B to A in Fig. 2.9 of interaction curve shows the compression failure region because the compressive strain dominates.

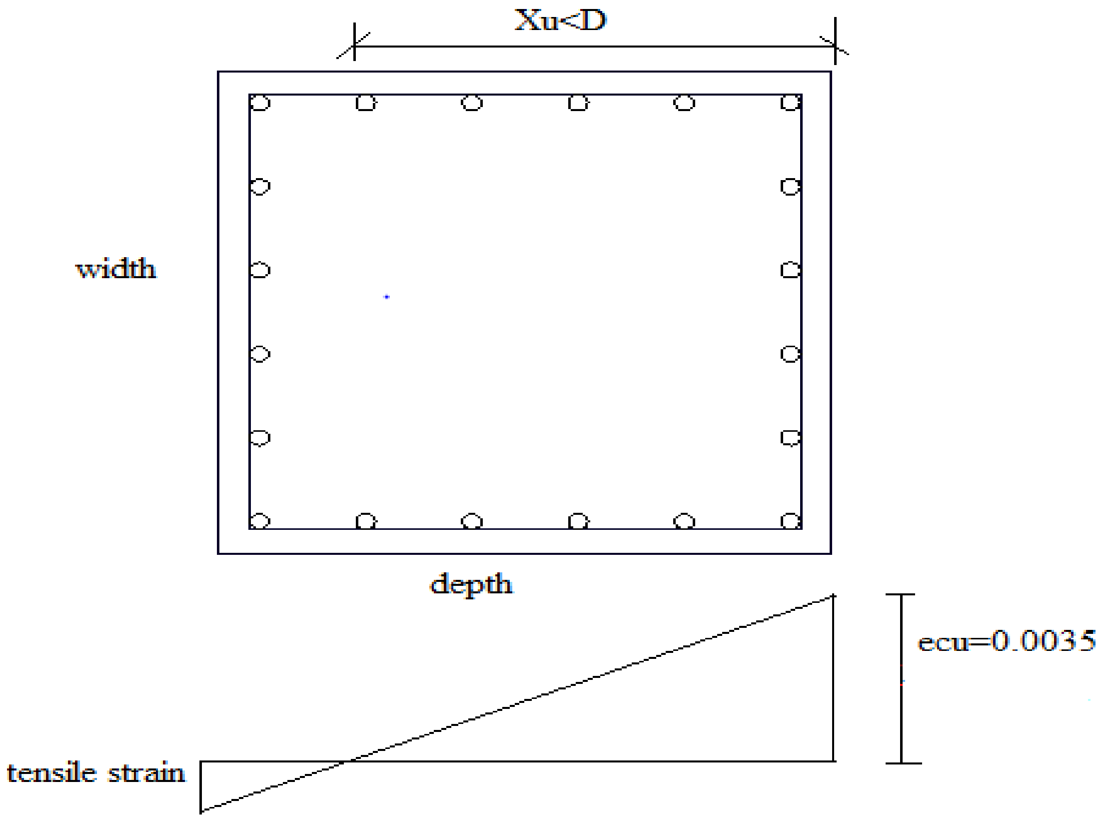


Fig. 2.6 Limiting Strain Diagram for Neutral Axis Inside the Section

Fig. 2.7 shows the limiting strain diagram for the balanced failure of the section. It is the point where both compressive strain at the most compressed edge and tension strain at the least compressed edge fiber becomes same. That is both ends will attain the strain of 0.0035. The point B in Fig 2.9 of interaction curve shows the balanced failure zone.

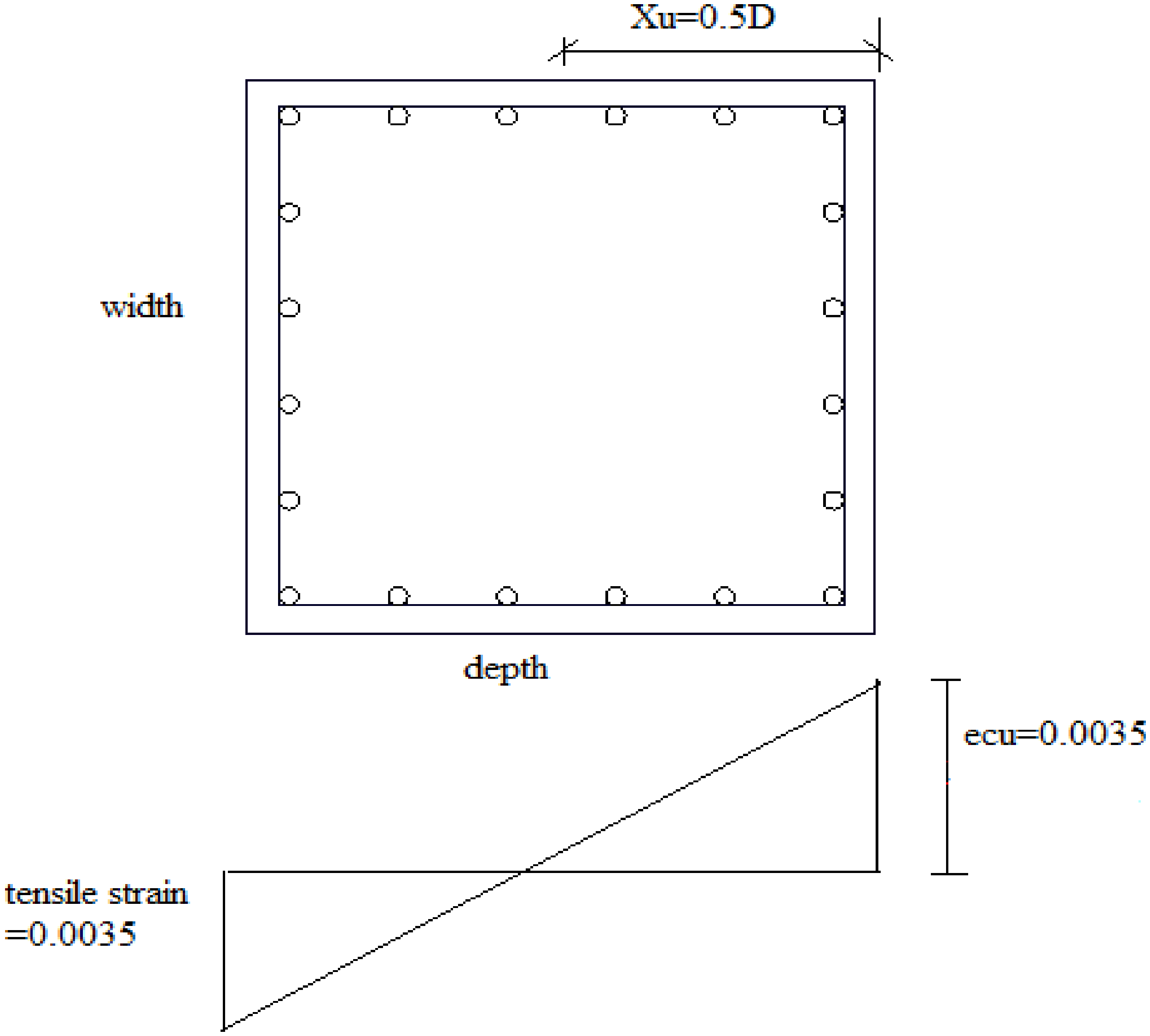


Fig. 2.7 Limiting Strain Diagram for Balanced Section

Tension failure mode

Fig. 2.8 shows limiting strain diagram for the tension failure zone. In this zone the tension failure dominates. In this zone the tensile stain exceeds 0.0035. Tensile failure zone gives over reinforced section which will results in the failure of concrete before steel in the column. This will result in catastrophic results. So to the design column the values of ultimate load and ultimate moment are not taken from this region.

The region between C to D from Fig. 2.9 of interaction curve shows the tension failure region. In this region flexure failure take place by bending. The point D shows the point of pure bending which is Mo.

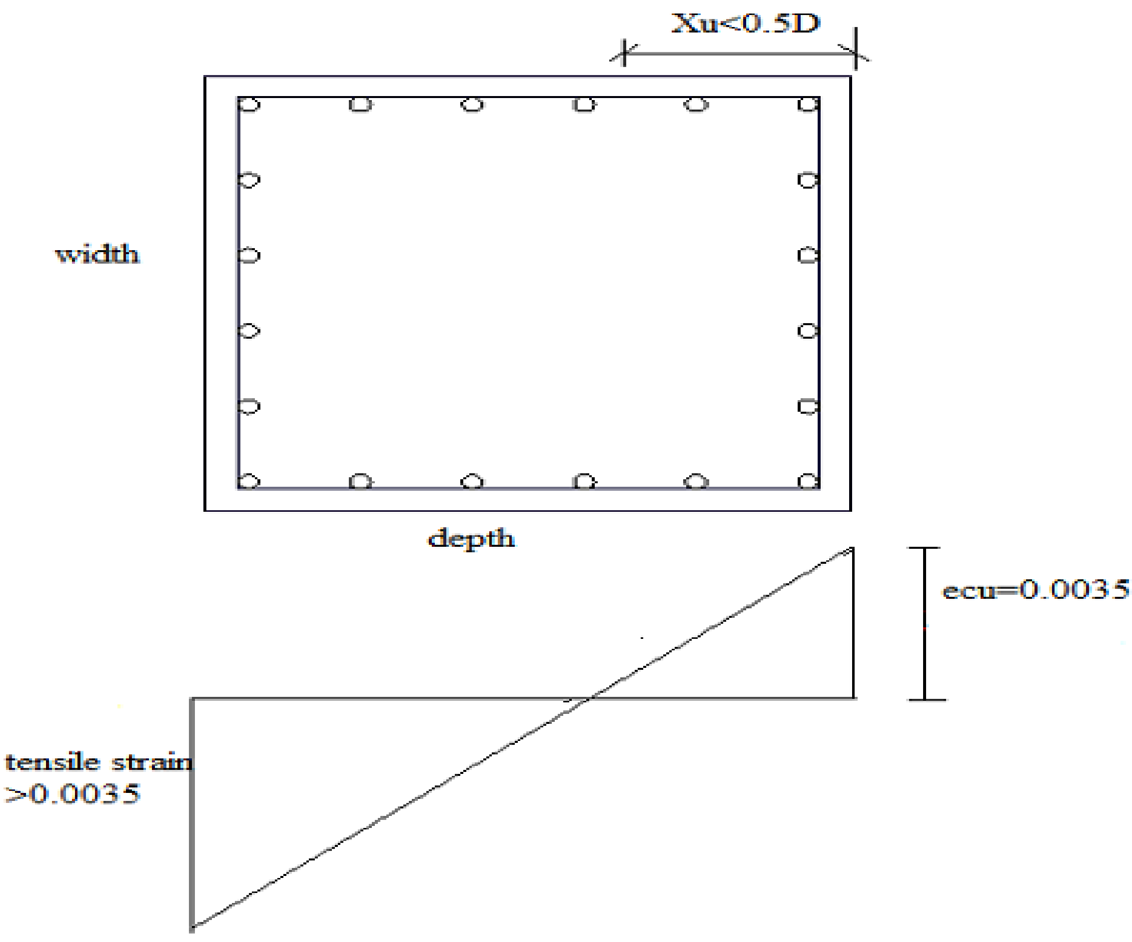


Fig. 2.8 Limiting Strain Diagram for Tension Failure Zone

2.2.2 Column Interaction Diagram. The plot of axial capacity (P_n) vs. moment capacity (M_n) is called an interaction diagram. Each point on the interaction diagram is associated with a unique strain profile for the column cross-section. An interaction diagram has three key points, as shown in the figure below. Each point and each region between the points is discussed below. [5]

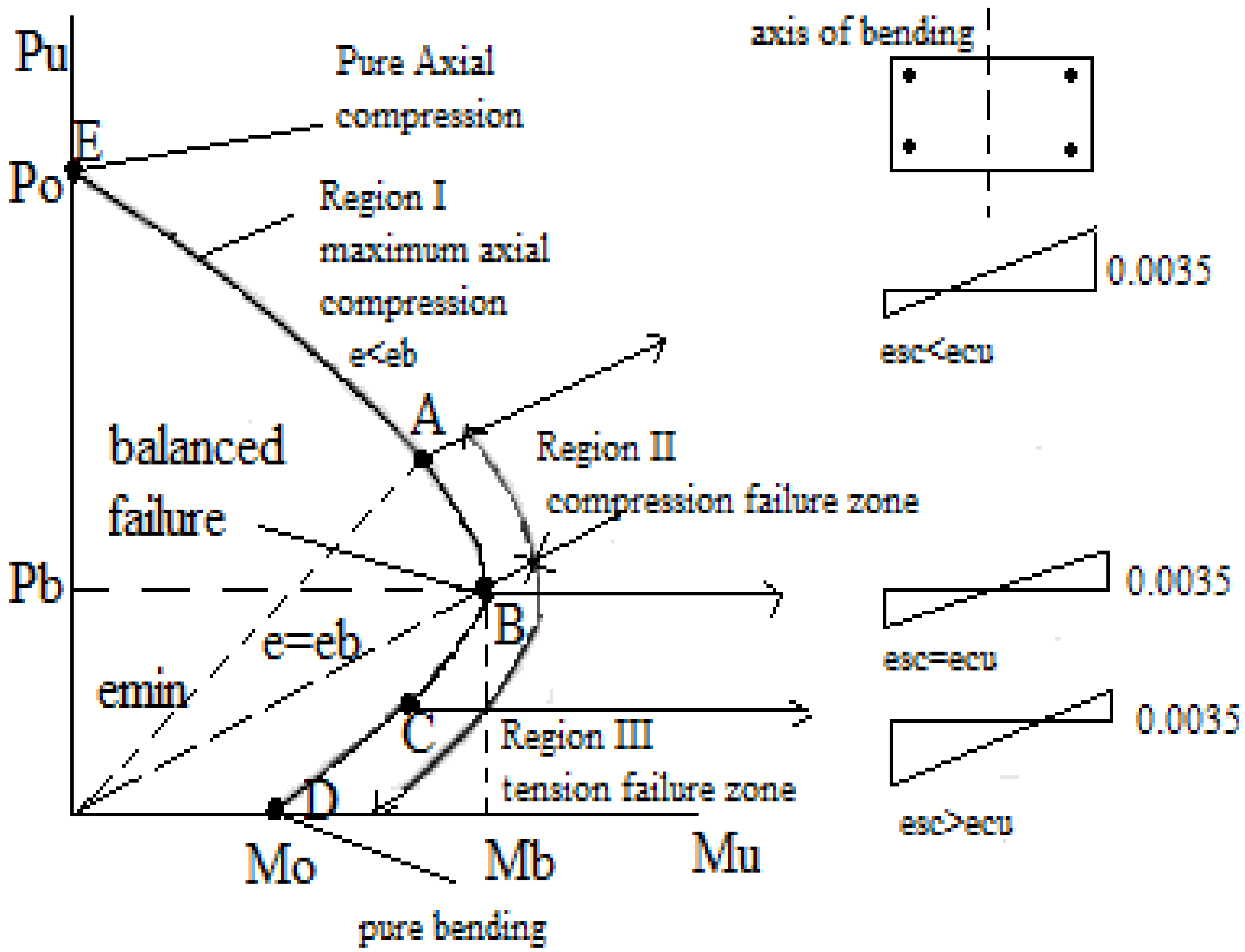


Fig 2.9. P_u-M_u Interaction Curve Diagram

Point E: The column is in pure compression. The maximum axial capacity of the column occurs in this state.

Point A to Point B (compression-controlled failure): The concrete crushes before the tension steel (layer furthest from the compression face) yields. Moment capacity decreases because the steel does not reach its full strength.

Point B (Balanced failure): A so-called "balanced" failure occurs when the concrete crushes (ε_c = -0.0035) at the same the tension steel yields (ε_s = 0.002).

Point B to Point C (tension-controlled failure): As compression force is applied to the section, the compression area can increase beyond the area balanced by the tension steel. Larger compression force leads to larger moment.

Point C: The column behaves as a beam. The compression area is limited by the area balanced by the tension steel.

In [5] reported that moment resisting capacity of column with unequally spaced longitudinal reinforcement compared to the column with equally spaced longitudinal reinforcement for neutral axis laying both at inside & outside the section. Unequally spaced longitudinal reinforcement gradual increase in resisting moments at different points of column starting from pure bending point (Mo) of the curve till the pure axial compression point (Po) as compared to the column section with equally spaced longitudinal reinforcement, especially in balanced failure zone and compression failure zone including axial compression zone.

2.3 Opensees

Opensees (Open system for earthquake engineering simulation) is object-oriented open source software which allows users to implement finite element methods to model the structural and geotechnical systems and simulate the response under earthquake loading. It has been under development by the Pacific Earthquake Engineering Research Center since 1997. Because Opensees is object-oriented framework software, in a finite element application, mainly four types of objects; model builder object, domain object, recorder object, and analysis object need to be constructed. In Opensees, the interpretation is accomplished by adding commands into Tcl script for finite element analysis. Each command is associated with a C++ procedure that is built inside and is called by the interpreter to analyze the command [12]. In Opensees both linear and nonlinear structural and geotechnical models can be built; various simulations: static push-over analysis, static reversed cyclic analysis, dynamic time-series analysis, uniform-support excitation, multi-support excitation can be effectively conducted; Opensees provides a library of various materials, elements and analysis which is powerful for numerical simulation of nonlinear systems. However, since Opensees is research developed software which denotes that it is not maturely developed, the simulations conducted using opensees need to be testified with results obtained from other software.in this work it is compared with SAP2000.

2.3.1 Modeling Limitation

- Three dimensional elements are modeled with 1-D line elements and 2-D cross section and it is assumed that plane section remain planes.
- No dependable productive model on flexure and shear interaction due to the reason that it is not yet proved.
- For many existing structures it is unknown exactly what is within the cover
- Quality control in construction is good enough
- Material property over time

- Concrete strength with curing
- Creep and Fatigue
- Corrosion in reinforcement

2.3.2 Consideration in Modeling

- **Materials** (Confined Concrete, Unconfined Concrete, Reinforcing Steel)
- **Sections** (Elastic Section, Unaxal Section-uncoupled axial and flexure, Fiber Section-coupled P-M-M Interaction)
- **System** (2D/3D, Rigid/Flexible Diaphragm)
- **Elements** (Structural Elements, Beams-no axial load, Columns P-M Interaction)
- **Plastic-Hinge behavior** (confinement Hinge length & growth, Yield Penetration, Bond Stress/Strength, Bar Pull-out, Anchorage loss, Bar elongation and buckling)
- **Element Type** (Continuum model, Distributed plasticity, Lumped plasticity, Displacement-based)

2.3.2.1 Beam-Column Elements

Beam-column elements are the most common models of behavior in the computational simulation of columns. The formulation of the beam column elements in Opensees takes place in a basic system, free or rigid-body displacement modes. At cross-sections along the element are the sections deformations and forces. The compatibility relationship between the element and section deformations, and the equilibrium relationship between the section and basic forces, depends on the beam-column element formulation, for which there are two approaches: displacement-based and force-based. Regardless of the element formulation, the element response depends on the response at each of its sections. The use of a software abstraction to represent the force-deformation response of a section facilitates the implementation of the beam-column element models in Opensees. Each section object encapsulates the force deformation response by either a resultant plasticity model or by the numerical integration of the material stress-strain response over the section area to describe the interaction of section forces. Therefore, the software design proceeds hierarchically from element to section and in turn from section to material, which follows directly from the equations of structural mechanics. The displacement-based formulation follows the standard finite element procedure of specifying an approximate displacement field over the element domain, from which compatible deformations are computed at each section along the element. The equilibrium relationship between the section forces and the basic forces is satisfied in an average sense or weak form under the displacement-based formulation. In contrast, the force-based formulation exactly satisfies equilibrium between the basic forces and section forces in a strong form, whereas the compatibility relationship between section deformations and basic element deformations is stated in integral form (using the principle of virtual forces). Like in typical finite element analysis, mesh refinement (h-refinement) is needed to capture higher order distributions. The accuracy of the solution can be improved by increasing the number of elements (not by increasing number of integration points). This is due to the fact that displacement based element uses displacement interpolation functions that approximate the exact solution and thus, involve both discretization and numerical error.

The force-based element state determination procedure, where the section deformations are computed under the condition such that section equilibrium is always satisfied, as well as the advantages of the force-based formulation over the standard displacement-based formulation, are described by Neuenhofer and Filippou. One advantage of the force-based beam-column element formulation is the ease with which the shear force formation behavior is taken into account. The shear forces at each section along the element are computed from the basic forces in the same manner as the axial force and the bending moments, and the relationship between the shear deformations and the element deformations come from the average compatibility inherent in the force-based formulation. The hysteretic models in Opensees are built from simpler components that represent the backbone, or envelope of the cyclic response, and the cyclic degradation of stiffness and strength. The accuracy of the solution can be improved by either increasing the number of integration points (preferable from a computational mechanics stand point) or number of elements. This is due to the fact that force based element uses the exact force interpolation functions and thus, doesn't involve discretization error but only numerical error. [15]

2.3.2.2 Beam-Column Models

Modeling of the element formulations used for the beam-columns in structural earthquake analysis in Opensees are primarily divided into two categories; distributed inelasticity, where plasticity can form anywhere along the member length, and concentrated plasticity, where the formation of plastic hinges is restrained to the member ends. The distributed inelasticity members are modeled with the fiber approach, which consists of discretizing the member section into several material fibers, in addition to discretization along the element length; Its two main formulations are the displacement-based (DB) method, which is the classical textbook finite element formulation, and the force-based (FB) method [15]. The DB formulation is based on displacement shape functions, due to FB elements not having restrains on their displacement fields they are able to approximate inelastic structural response with a greater accuracy than DB elements. While these are the most known finite element formulations, there has also been presented others that either improve or do not possess their disadvantages [15]

Physical localization is defined as structural properties being dependent on the member's size when experiencing strain-softening. Strains, deformations, and ductility will differ for constant concrete section depending on its length. However, the pre peak strain-stress response will remain identical for all height-diameter ratios. Research on specimen has resulted in several proposals on how to model localization response. Another localization issue arises for the numerical analysis of softening behavior, due to strains concentrating at section integration points with highest bending moment values. This yields non-objective response, which means that the calculated response differs based on finer mesh discretization and does not converge into one single solution. Regularization procedures have been developed to correct these issues, and thus obtain objective response. Some of these methods are adjusting extremity element length of DB members, adding a damage variable to the constitutive relation, and the fracture energy criterion and post-processing of local response [15].

Concentrated plasticity elements are designed to lump plasticity at their member ends, which is expected to occur for beam-columns subject to strong lateral forces. These elements range from simple one- or two-component models, often with nonlinear springs at member ends, to more sophisticated elements with FB fiber modeling at predetermined plastic hinge lengths.

2.3.2.2.1 Fiber model

Distributed plasticity models have the advantage that there is no predetermined length that concentrates the inelastic behavior. Thus, inelasticity can form at any section point along the element length; where the points are determined by the numerical integration method used. The drawback of this property is that the computational effort is greatly increased compared to concentrated inelasticity models, as the interior span of the elements is not assumed linear elastic. Naturally the distributed inelasticity models will give a closer to exact solution and thus give an improved approximation of the experimental response.

Structural members of reinforced concrete do not consist of homogenous material sections. A similar case applies for steel sections with initial stresses at the welding points, which will affect the overall response. This renders it impossible to model the exact element behavior without discretizing the sections, and thus taking into account the different properties of the materials. This approach is called the fiber model, and is the commonly used distributed inelasticity element model. A fiber model of a reinforced concrete beam is made for the unconfined cover concrete, the confined core concrete, and the steel reinforcement bars. Thus, the element model will need to include three different material stress-strain relationships to render the correct response quantities. Computational effort can however be reduced by adjusting the level of the section discretization. For instance, the number of fibers in the z-direction of a beam can often be far less than in the y-direction. This is because the bending moment about the z-axis usually is significantly larger than about the y-axis, which will result in increased inelasticity and need for discretization. Although the fiber element is considered more "exact" than other element formulations, it has its numerical difficulties. One of the most significant is the interaction between the flexural and shear response, which still is under research.

The most common finite element formulations for the fiber element is the displacement based stiffness method, and the force based flexibility method. Their advantages and disadvantages will be discussed as follows. There exist several formulations that resolve some of the difficulties with these two methods [15].

2.3.2.2.2 DB elements

The main numerical localization issue in DB elements is explained by Zeris & Mahin for a simple cantilever column subjected to a lateral displacement at its tip. As the applied displacement increases, the sections near the column's base begin softening and lose their capacity. In order to maintain force equilibrium, the remaining sections have to unload elastically. The reason for the numerical failure is that while the base sections detects the softening behavior through the defined constitutive relation, the rest of the column do not know that it should unload when assembling the element stiffness matrix. These results in the unrealistic case where the base sections unload while the top sections continue to take more bending moment. [14,15]

2.3.2.2.3 FB elements

Localization issues in FB elements have been under wide research, as it is gradually being accepted as having clear advantages over DB elements. Unlike the DB elements, there is not an issue with force equilibrium in the FB elements after the onset of softening in a structural member; simply because equilibrium is strictly enforced in the formulation. In the strain-hardening case, both the global force-displacement and local moment curvature responses are objective, and converge into a single solution as the number of integration points along the member is increased. Strain-softening is often experienced in reinforced concrete columns that are subjected to large dead loads and lateral forces [15]. Similarly, to the elastic-perfectly plastic member section, the curvature localizes at the base section. But in contrast to the former case the base section begins softening, and is thus not able to carry the same loading as the section strains increase. Again, as the number of integration points increase and the plastic hinge region gets smaller, the resulting growing strains will inflict even lower material stiffness. Increasing the number of integration points yields larger curvature and lower ultimate base shear. Even though the applied displacement is prescribed, the different amounts of material stiffness degradation affect the load-carrying capacity, and thus make the global response nonobjective as well. This paper focuses on the development of software program which models and analyzes reinforced concrete column using Opensees software with bundled bars under lateral load on displacement based beam-column element formulation and with Fiber model of section. The program has incorporated Tcl command which is used to detect softening problem when the applied displacement at the top of the column increase. The outputs of the developed program are nodal displacement, local and global forces, section deformation, section forces, reaction forces.

2.3.3 Evaluations of section and Fiber points in Fiber Model

Since the elastoplastic damages occur under severe earthquake, reliable prediction of nonlinear behavior of structures in the earthquake is important to assess the seismic safety of the structures. At present, there are mainly four kinds of analytical models, namely fiber model, plastic hinge model, micro-model, and hybrid-model. The most advantage of fiber model is that it adopts uniaxialmaterial constitutive and is able to consider the coupling of the axial force and bending. For its concise and clear physical conception and reliable analysis result, researchers pay more and more attention to fiber model. The formulation of the fiber model is as following: [10]

1. First, the structure is divided to a discrete number of elements which includes beams or columns.
2. Then the elements are discretized into a number of sections. The section deformations are then used to determine the element displacements based on the assumed force and deformation interpolation functions.

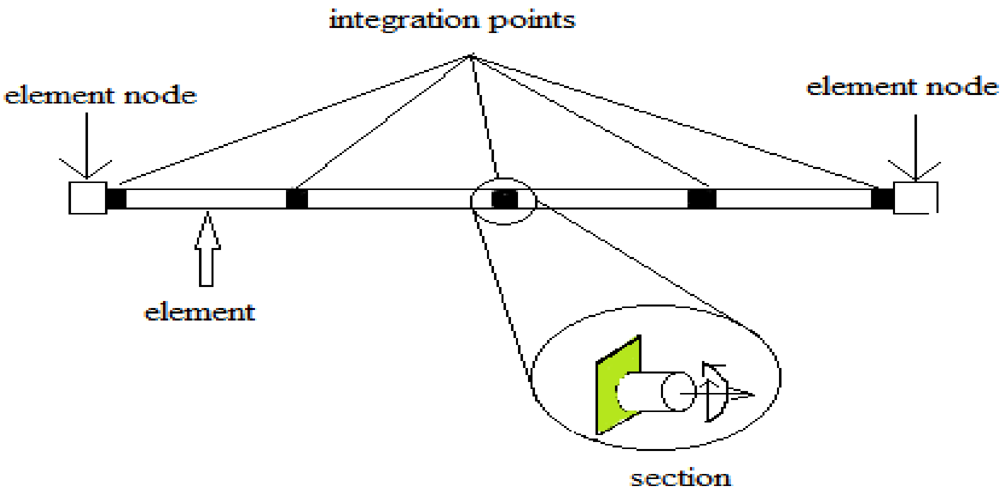


Fig.2.10 Element Integration Points

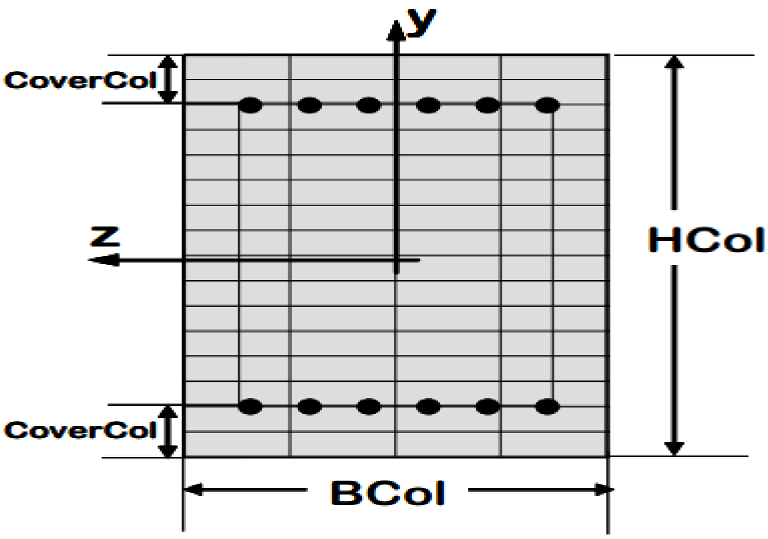


Fig 2.11 Fiber Section

3. Finally, the sections are divided into a number of fibers. The fiber strains are then used to determine the section deformations based on the plane section assumption. How to determine the appropriate number of section integration points of an element and the appropriate number fiber integration points of a section during the discretization? Theoretically, of course the more the better. Through the study of reasonable division in section integration points and fiber integration points, computation efficiency and accuracy are both ensured. In order to improve computation efficiency and accuracy, both the section integration points discretized in an element and fiber integration points divided in a section are evaluated by using three nonlinear beam-column elements (DB, FB, and PH elements) in finite element programming Opensees.

From the results, it is indicated that for the DB element, six section integration points and 6×6 fiber integration points are suggested to use; for the FB element, four section integration points and 6×6 fiber integration points are suggested to use; for the PH element, 0.2 of hinge length ratios and 6×6 fiber integration points are suggested to use [10]. For this paper Displaced based method was used with integration point of 5 and 30x30 fiber integration points are used.

2.3.4 Materials Once the nodes have been defined, the next step towards defining elements is the material (nDMaterial Command, uniaxialMaterial Command) definition. This step may not be necessary when using elastic element or sections, as the materials are defined with the element or section. There are two types of materials currently available in Opensees, uniaxial materials and nDmaterials. The different types of concrete and steel materials are among the uniaxial materials.

Concrete02 will be used for the structure under consideration of this paper, as the tensile strength of the concrete is of interest in the elastic range, and modeling linear tension softening is considered. The cover and core concrete will be modeled as different materials, using the same material type, but different stress and strain characteristics and different material tags. Steel02 will be used for the reinforcing steel. Because some material characteristics are dependent on others, it is recommended that the user define the material properties using variables. It is also a good idea to use variables for IDtags of materials, sections, elements, etc. This is done to ensure that the same IDtag is not used when defining the input.

2.3.5 Concrete Material Models

2.3.5.1 Unconfined and Confined concrete

Kent and Park (1971) proposed a stress strain equation for both unconfined and confined concrete. In their model they generalized Hognestad's (1951) equation to more completely describe the post peak stress strain behavior. In this model the ascending branch is represented by modifying the Hongnetad second Degree parabola by replacing 0.85f'c by f' and eco by 0.002.

$$\left[\frac{2\varepsilon_c}{\varepsilon_0}-\left(\frac{\varepsilon_c}{\varepsilon_0}\right)^2\right] \qquad \text{Equation 2.3}$$

The post-peak branch was assumed to be straight line whose slope was defined primarily as a function of concrete strength.

$$f_c = f_c^{'}\left[1-Z(\varepsilon_c-\varepsilon_0)\right] \qquad \text{Equation 2.4}$$

In which $Z = \dfrac{0.5}{\varepsilon_{50u} - \varepsilon_{c0}}$ …………………………………………………………….. Equation 2.5

Where ε_{50u} is the strain corresponding to the stress equal to 50% of the maximum concrete strength for unconfined concrete.

$\varepsilon_{50u} = \dfrac{3 + 0.29 f_c^{'}}{145 f_c^{'} - 1000}$; $f_c^{'}$ in MPa………………………………………………Equation 2.6

The Kent and Park model is represented in Fig 2.12

Popovics (1973) proposed a single equation to describe unconfined concrete stress-strain behavior. A major appeal of this model is that only requires three parameters to control the entire pre and post peak behavior, specifically f_c' ε_{c0} and E_c

$$\frac{f_c}{f_c^{'}} = \frac{n \dfrac{\varepsilon_c}{\varepsilon_{c0}}}{(n-1) + (\dfrac{\varepsilon_c}{\varepsilon_0})}$$ ……………………………………………………………… Equation 2.7

Popovics equation works well for most normal strength concrete (f_c'=55Mpa), but it lacks the necessary control over the slope of the post peak branch for high strength concrete.

Thorenfeldt et al. (1987) made modification to the Popovics (1973) relation to adjust the descending branch of the concrete stress strain relation. The authors proposed the following equation for the unconfined concrete stress-strain relation.

$$\frac{f_c}{f_c^{'}} = \frac{n \dfrac{\varepsilon_c}{\varepsilon_{c0}}}{(n-1) + (\dfrac{\varepsilon_c}{\varepsilon_0})^{nk}}$$ ………………………………………………………… Equation 2.8

In the above equation 'k' takes a value of 1 for values of $\varepsilon_c / \varepsilon_0$ <1 and values greater than 1 for $\varepsilon_c / \varepsilon_0$ >1. Thus by adjusting the values of "k" the post peak branches of the stress strain relation can be made steeper. This approach can be used for high-strength concrete where the post peak branch becomes steeper with increase in the concrete compressive strength.

Tsai (1988) proposed a generalized form of the Popovics (1973) equation with has greater control over the post-peak branch of the stress-strain relation. Tsai's equation consists of two additional parameters, one to control the post-peak behavior of the stress-strain curve. The proposed stress-strain relation for unconfined concrete by Tsai is

$$y = \frac{mx}{1 + (m - \dfrac{n}{n-1}) + \dfrac{x^n}{n-1}}$$ ………………………………………………………….... Equation 2.9

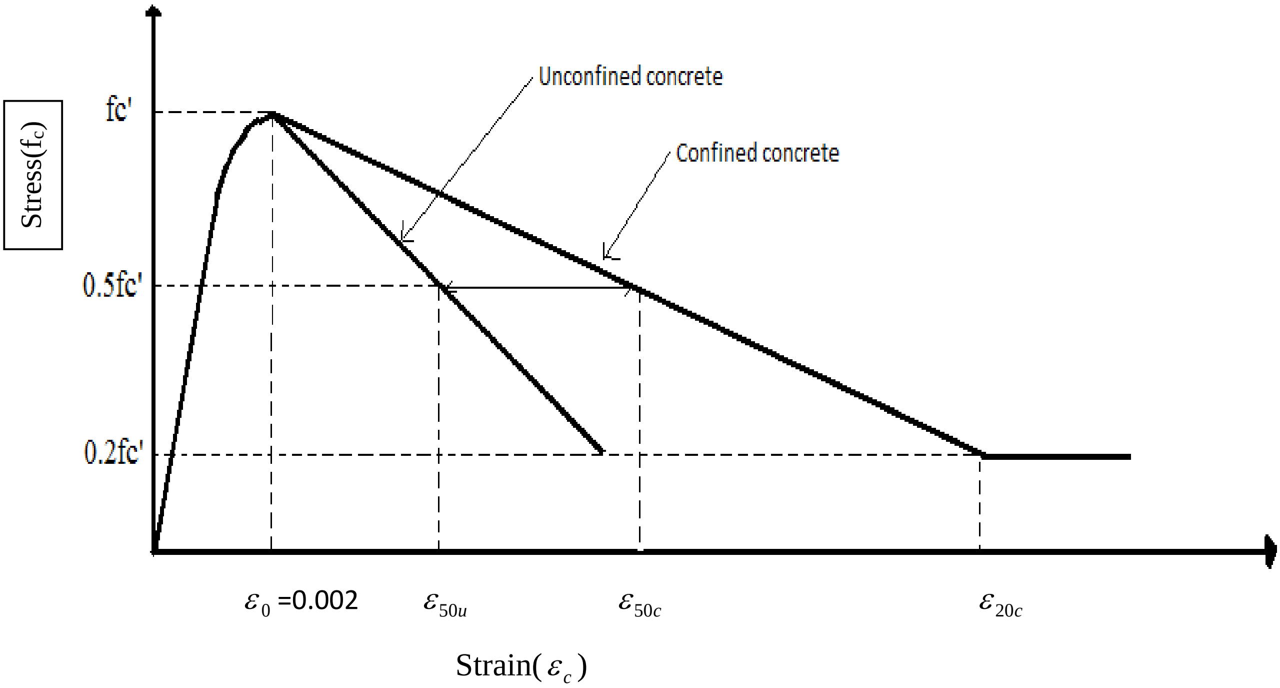


Fig.2.12 Proposed stress-strain model for confined and unconfined concrete Kent and Park (1971) model

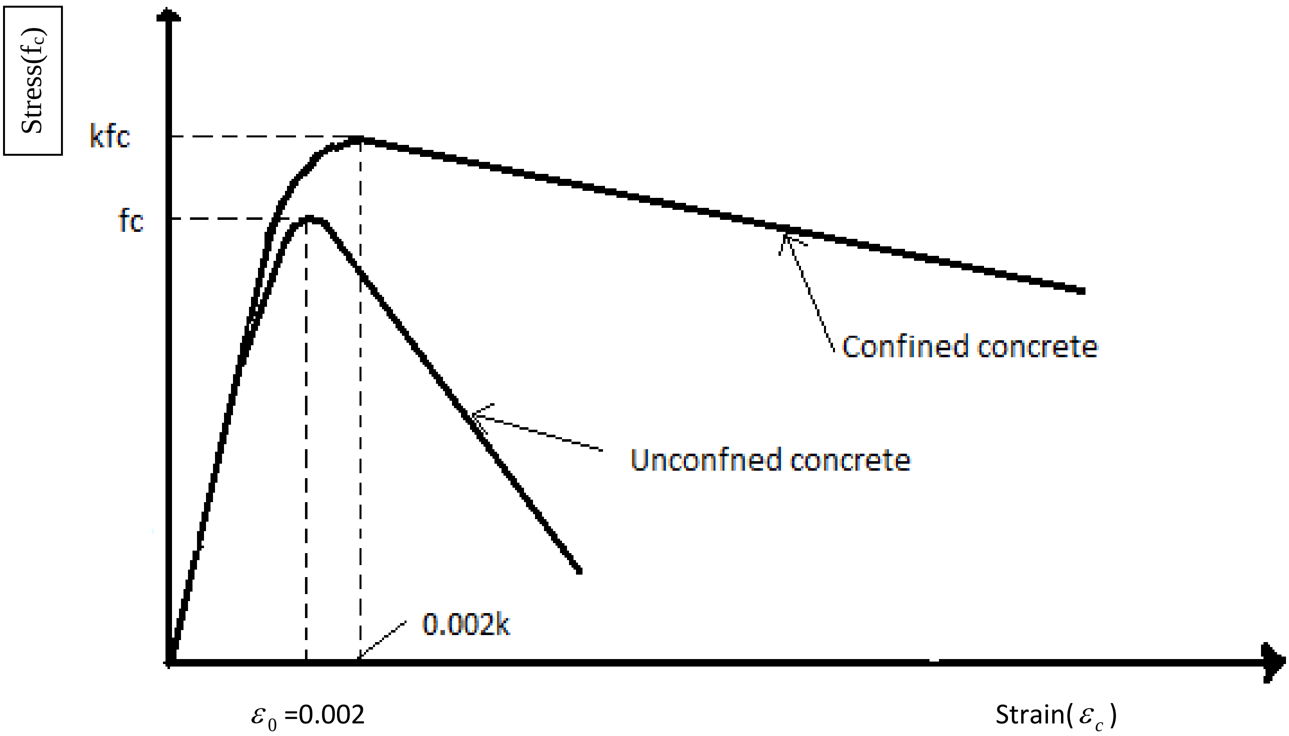


Fig.2.13 stress-strain behavior of compressed concrete confined by rectangular steel hoops-modified Kent and Park (scott et al. 1982) model

2.3.6 Preparation of Input data for the Program

The first task in developing a program is to clearly define the way data is organized within the Program [13]. The input data for analysis of two dimensional reinforced concrete column for lateral loading are organized as follows: A series of tcl scripts store data defining the element geometry and material properties. Lists and arrays are the simplest data structures. An array is a tcl variable with a string-valued index. The array elements are defined using the set command in tcl. A list on the other hand is simply a string with list elements separated by a space. Once the data has been stored into the lists and the arrays, OpenSees commands are called upon to use this data to define the model. Model defining progresses through, definition of the nodes (using the command Node in OpenSees), definition of the material models (using the command UniaxialMaterial in OpenSees), definition of the sections (using the patch, layer and fiber commands in OpenSees) and definition of the elements (using the element command in OpenSees). Once the whole model has been set up the analyze.tcl sets up the various analysis tools in OpenSees and performs the required analysis specified in setAnalysisParameters.tcl. Fig. 2.14 shows the model building process, data transfer, and tasks required to accomplish the analysis of the model.

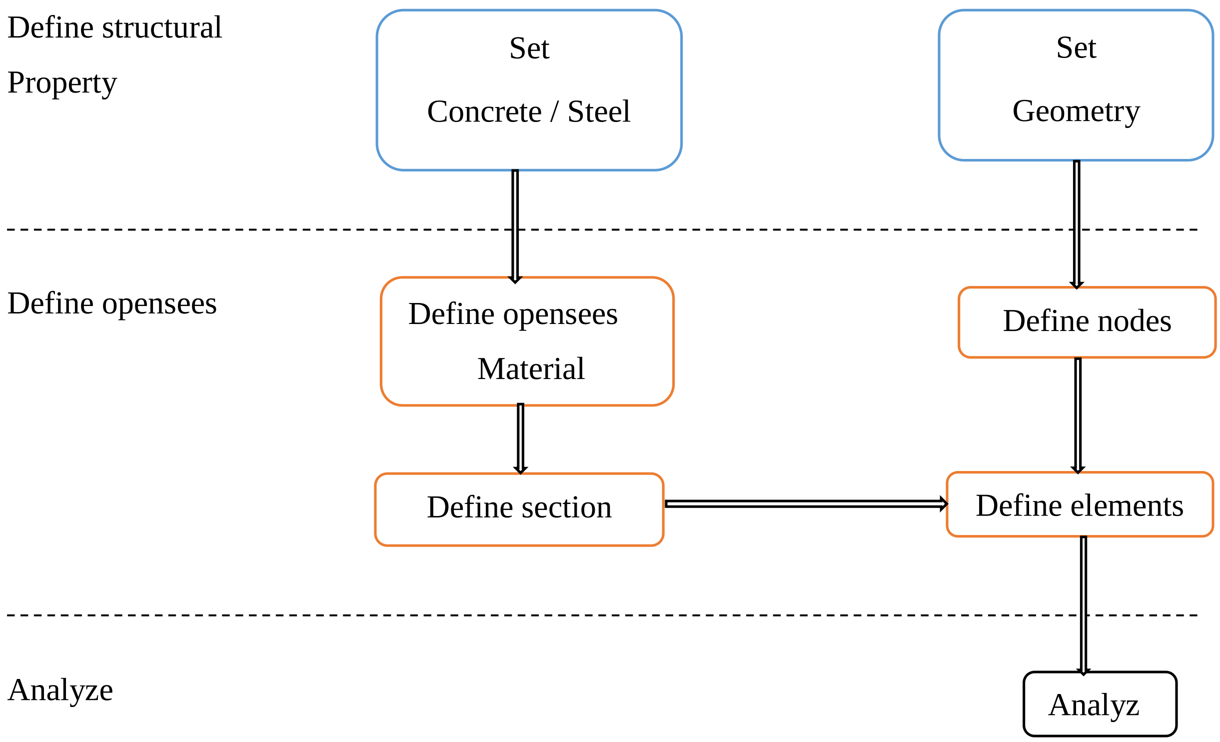


Fig.2.14 Component of Analysis in Opensees

Opensees comprise of a set of modules to perform creation of the finite element model, specification of any analysis procedure, selection of quantities to be monitored during the analysis, and the output of the results. In each finite element analysis, an analysis is used to construct 4 main types of object, as shown in Fig 2.15.

2.3.7 Main abstraction in opensees framework

Domain Holds the state of the model at time ti and (ti +dt) and is responsible for storing the objects created by the ModelBuilder and for providing access of this objects to the Analysis and Recorder.

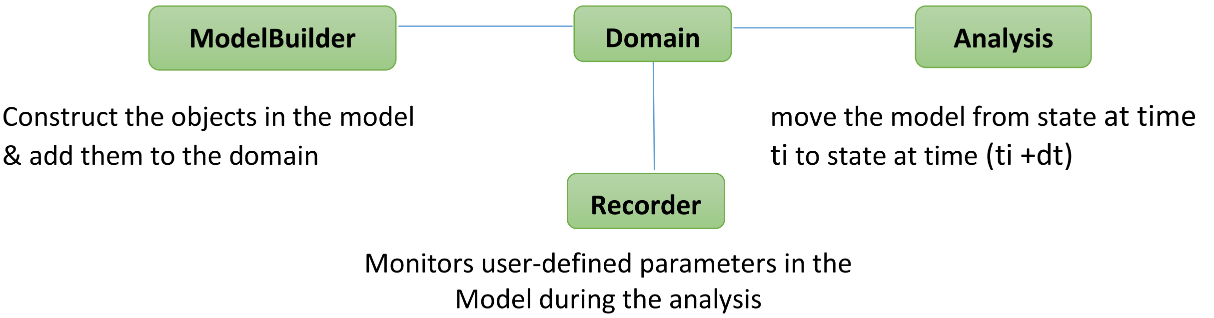


Fig.2.15 Main Abstraction in Opensees Framework

2.3.8 Nonlinear Analysis Algorithm

An incremental procedure is usually used to study nonlinear finite element problems. For nonlinear problems, however, the incremental procedure would lead to a build-up of error. An iterative procedure using a certain solution algorithm should be employed to correct the build-up errors. Therefore, the combination of the incremental approach and iterative procedure are used as the basis for most of the nonlinear finite element analyses. In the analysis procedure, the integrator determines the next predictive step during the analysis procedure, and specifies the tangent matrix and residual vector at any iteration. The algorithm determines the sequence of steps taken to solve the nonlinear equations during the iterative procedures. Hence, different algorithms are also described.

2.3.8.1 Load Control Iteration Scheme

To study the problem of a structure under monotonic loading, an incremental procedure can be used. The structural tangent stiffness matrix is related to the increments of loads to increments of displacements, which incorporates a tangential material constitutive matrix relating the increments of stresses to the increments of strains. Under load control, the total load is divided into small load increments. Each load step is applied in turn and iterations are performed until convergence is achieved at the structural level. Then the next load step is processed. In solving nonlinear equations, the commonly used solution algorithm is the full Newton-Raphson method. In each iteration, the stiffness matrix is iteratively refined until the convergence criterion is achieved. The stiffness matrix is computed from the last iterative solution during the iterative procedure until convergence is achieved. A modified Newton-Raphson procedure is also used in the solution algorithms. Different from the full Newton-Raphson method, the stiffness matrix from the last converged equilibrium was used during the iterative procedure until convergence is achieved. The Newton-Raphson method uses the initial stiffness matrix throughout the iterative procedure. When compared with the Newton-Raphson method and the modified Newton-Raphson method, the full Newton-Raphson method converges more rapidly, and the process will converge in less iteration and give smaller residual force at each iteration. However, it requires that the tangent stiffness matrix be evaluated at each iteration, which can be significant for large structures. In contrast to the full Newton-Raphson method, the initial stiffness matrix in the Newton-Raphson method is calculated at the beginning of the load step and the stiffness matrix remains the same throughout the procedure. A large number of iterations are required to achieve convergence. The modified Newton-Raphson method shows the balance between the computation and iteration numbers. Many algorithms, such as the KrylovNewton method, have been developed by improving the Newton-type methods with the acceleration technology. The KrylovNewton method is a modified Newton-Raphson method with “Krylovsubspace” acceleration, which greatly decreases the number of iterations in the solution.[11]

2.3.8.2 Displacement Control Iteration Scheme

Loads can be applied to a structure using either of two different methods: under load control or displacement control. To simulate the seismic behavior of a reinforced concrete structure subjected to reversed cyclic loading, the entire load–displacement curve, including the ascending branch, descending branch, and the hysteresis loops, can be obtained using the displacement control scheme. On the other hand, the displacement control method also has advantages over load control in the analysis procedure as described below.

1. Under load control it is impossible to indicate the behavior of the structure at a local limit such as the temporary drop of force due to the initial concrete cracking. More importantly, load control is incapable of producing the ultimate strength of the structure, or to trace the behavior of the structure in the post-peak region. Under load control the tangent stiffness matrix becomes nearly singular at the peak point of the load–displacement curve. The failure of the solution to

converge is not an indication that the structure has reached its collapse point, but rather a failure of the solution convergence. Under the displacement control, especially the displacement control with arc length scheme, it is possible to obtain the behavior of the structure beyond the crack point and the maximum point and to determine the entire response including ascending, descending, and cyclic branches. [14]

2. When there is no preference of load control or displacement control, the displacement control method shows faster convergence and is more stable than the load control method. This is observed in the nonlinear finite element analyses of reinforced concrete plane stress structures. Many researchers have proposed the displacement control scheme to overcome the limits of the load control method. Meanwhile, the arc length method has been developed to overcome the local and global limit points in the nonlinear analysis, which treated the load factor as a variable. The arc length method was originally proposed by Riks (1972) and was improved by Crisfield (1981). A displacement control with an arc length scheme originally proposed by Batoz and Dhatt (1979) is available [15]. For the Pushover analysis a displacement control strategy will be used. In displacement control we specify an incremental displacement that we would like to see at a nodal Dof and the strategy iterates to determine what the pseudotime (load factor if using a linear time series) is required to impose that incremental displacement. Nonlinear and nonlinear models do not always converge. The analysis is carried out inside a “while “loop as it can be seen in Opensees modeling at the appendix A1, A2, A3 and A4. The loop will either result in the model reaching its target displacement or it will fail to do so. At each step a single analysis step is performed. If the analysis step fails using standard Newton solution algorithm, another strategy using initial stiffness iterations will be attempted.

Integrator: The Integrator object is responsible for defining the contributions of the elements and Nodes to the system of equation and for updating the response quantities at the Nodes with the appropriate responses, given the solution to the system of equations.

Handler: The Handler object is responsible for ensuring that the single and multi-point constraints in the Domain are enforced.

Numberer: The Numberer object is responsible for mapping equation numbers in the system of equations to the degrees-of-freedom.

SystemOfEqn: The SystemOfEqn object encapsulates the system of equations and provides operations to solve the system.

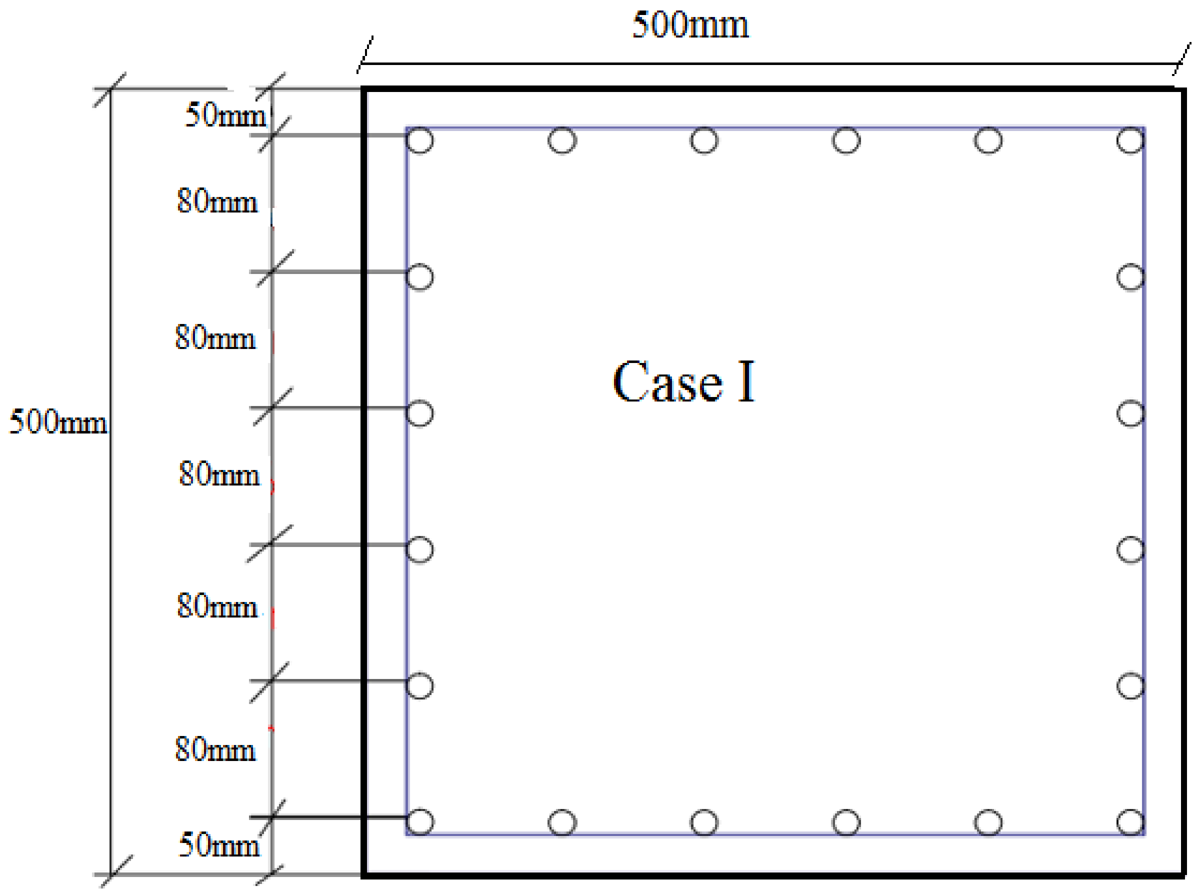
Equation Solvers: The following methods provide the solution of the linear system of equations Ku = P. Each solver is tailored to a specific matrix topology.

- ProfileSPD: - Direct profile solver for symmetric positive definite matrices
- BandGeneral:- Direct solver for banded unsymmetrical matrices
- BandSPD:-Direct solver for banded symmetric positive definite matrices
- SparseSPD:-Direct solver for unsymmetrical sparse matrices
- SparseSymmetric:-Direct solver for symmetric sparse matrices
- UmfPack General:-Direct UmfPack solver for unsymmetrical matrices
- FullGeneral:-Direct solver for unsymmetrical dense matrices
- ConjugateGradient:-Iterative solver using the preconditioned conjugate gradient method

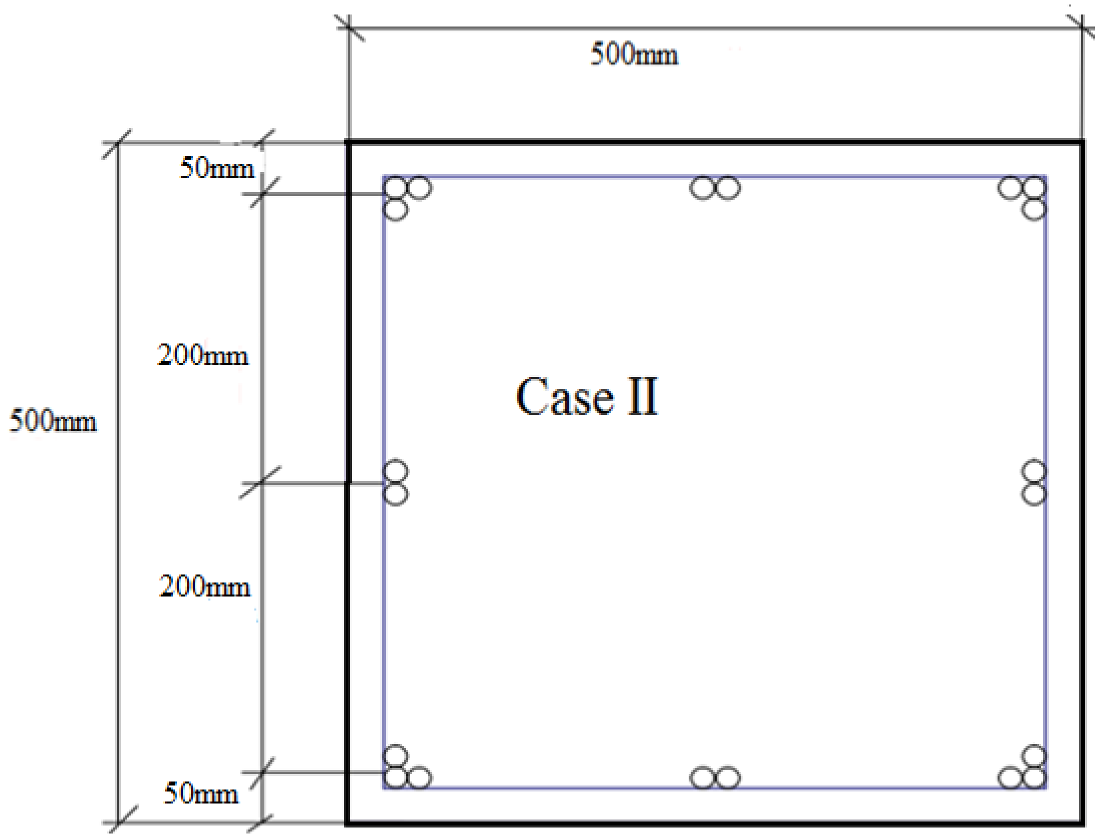
This paper investigates the effect of bundling reinforcing bars in column on non-linear modeling of two dimensional reinforced concrete structures using Opensees with static loading. Opensees is as its name indicates open software that can be modified based on the interest of analysts and a powerful finite element software for analysis of earthquake loading, and it is easy to use. Because it is the preferable software in researches as it can model both linear and nonlinear modeling of structures.

Chapter 3: Moment Capacity Determination

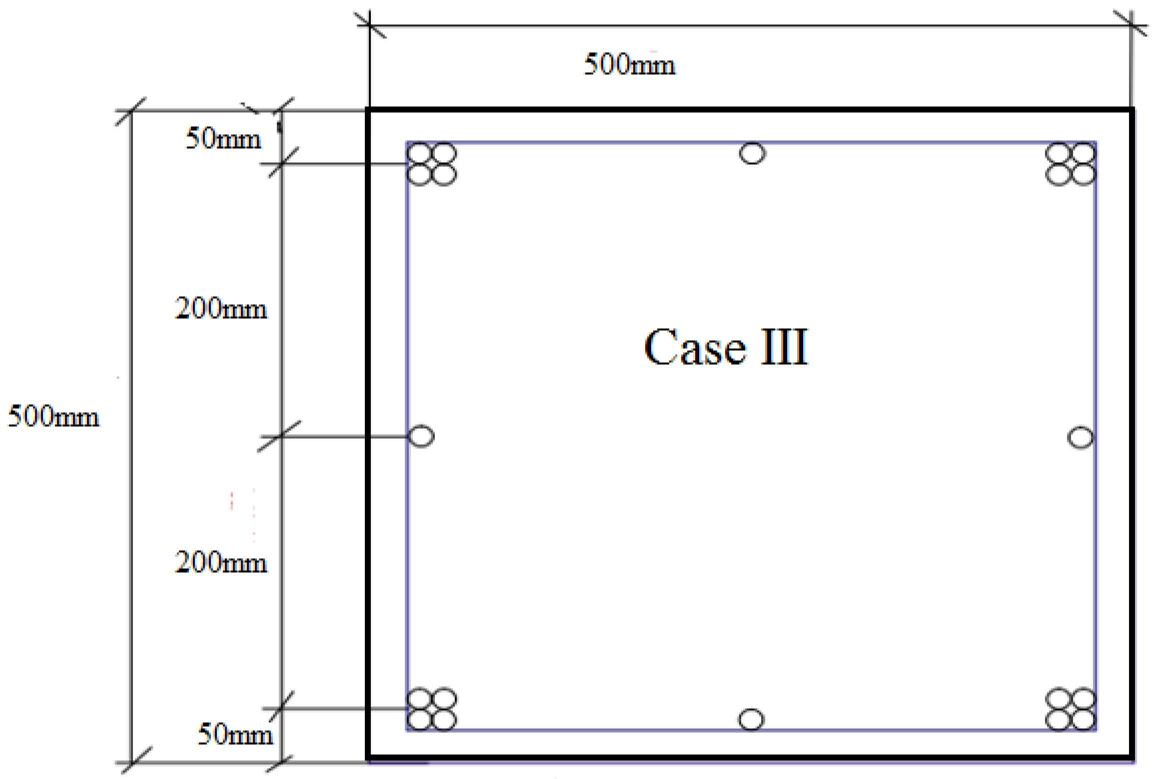
Almost all compression members in concrete structures are subjected to moments in addition to axial loads. These may be due to misalignment of the load on the column, or may result from the column resisting a portion of the unbalanced moments at the ends of the beams supported by the columns. The moment capacity determination of a column whose section is shown in the figures below using EBCS-EN 1992-1-1 code. Compare the interaction diagram for four columns each with the same material property, the same gross area A_g and the same total area of longitudinal steel A_{st}; the column differs in arrangements and size of bar. To obtain the same total areas of reinforcement in each bundle diameter of 4Ø16mm and 1Ø32mm were used. 4Ø16mm bars have an equivalent bar area of 804.247mm^2 which equates to an equivalent diameter of 1Ø32mm bar having 804.247mm^2, Fig 3.1 shows the arrangements of bars.



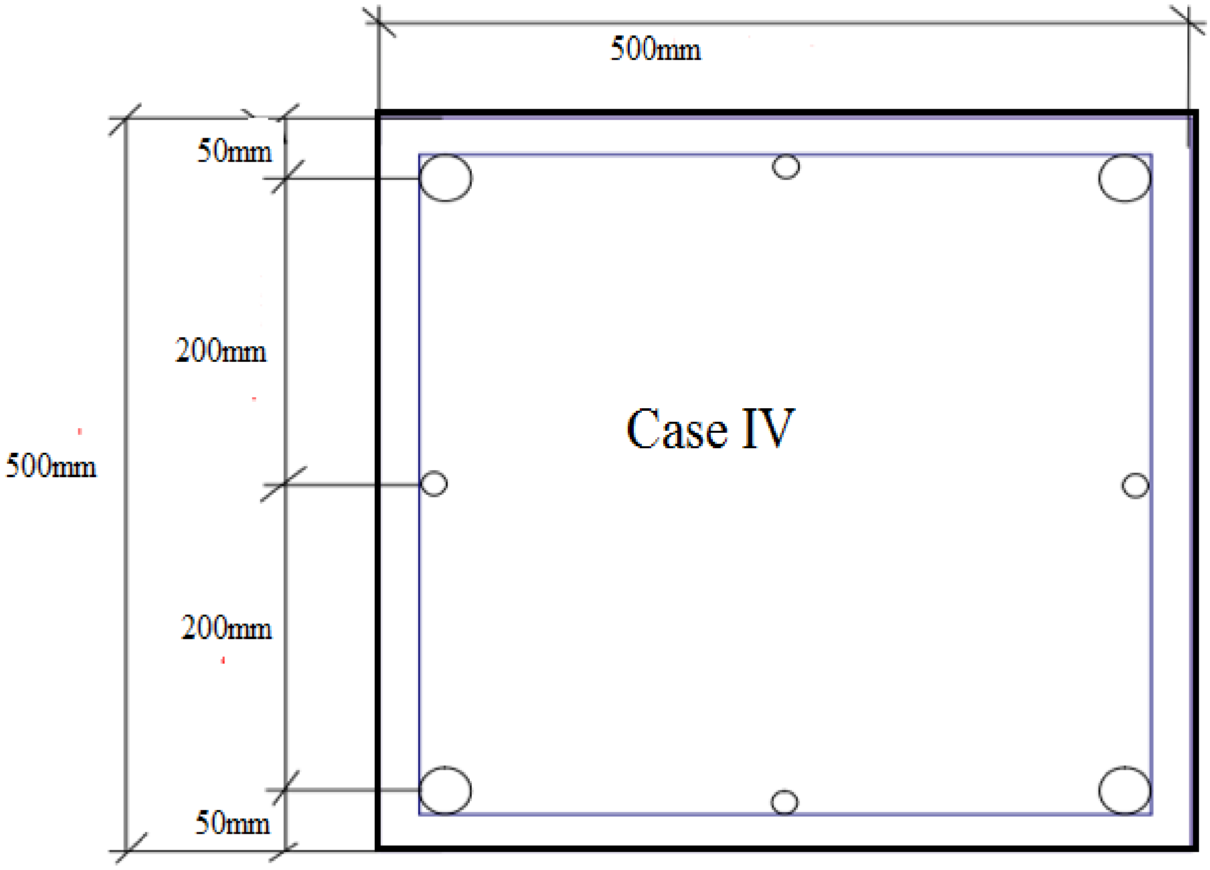
(a) *Column section with equally spaced longitudinal reinforcement (20Ø16)*



(b) *Column section with bundle bars place (3Ø16 at each corner & 2Ø16 at center)*



(c) *Column section with bundle bars place (4Ø16 at each corner & 1Ø16 at center)*



(d) Column section with equivalent bars (1Ø32 at each corner & 1Ø16 b/n corner bars)

Fig. 3.1. Arrangements of Reinforcing Bars for This Paper

3.1 Design data

$B\sec = 500mm$

$H\sec = 500mm$

$f_{ck} = 30N/mm^2$

$f_{yk} = 460N/mm^2$

$A_g = 250000mm^2$

$L = 3m$

$E_s = 200KN/mm^2$

$\gamma_c = 1.5$

$\gamma_s = 1.15$

$\varepsilon_{c2} = 0.002$

$\varepsilon_{cu2} = 0.00035$

3.2 Calculation method

Step 1: - find axial load carrying capacity of the concrete

$P_c = c_c * f_{ck} * B\sec * \bar{x}$ ……………………………………………………...Equation 3.1

$$c_c = \frac{\alpha_{cc}}{1.5}, \alpha_{cc} = 0.85$$

$$\bar{x} = 0.8 * x_u$$

Step 2: - find axial load carrying capacity of the steel

Strain in the steel e_{sci} for each row

$$\varepsilon_{sci} = 0.0035 * \frac{D_{sec} - d_i}{D_{sec}}, for - e_y \le e_{sci} \le e_y \text{ .... Equation 3.2}$$

Stress in the steel found by multiplying the strain value by modulus of elasticity of the steel

$$f_{si} = e_{sci} * E_s \text{ .... Equation 3.3}$$

Stress in the concrete

$$f_c = f_{cd}(1 - (1 - \frac{\varepsilon_c}{\varepsilon_{cu2}})^n), for \varepsilon_c < \varepsilon_{c2} \text{ .... Equation 3.4}$$

$$f_c = f_{cd}, for \varepsilon_{c2} < \varepsilon_c < \varepsilon_{cu2}$$

$$f_{cd} = \alpha_{cc} \frac{f_{ck}}{\gamma_c}$$

Axial load in the steel

$$P_{si} = \left(f_{si} - f_{ci} \right) * A_{si} \text{ .... Equation 3.5}$$

Step 3:- Total axial load p_u

$$P_u = P_c + \sum_{i=1}^{m} P_{si} \text{ .... Equation 3.6}$$

Step 4 :- Ultimate moment M_u

Ultimate moment carrying capacity of the concrete M_c

$$M_c = P_c \left(\frac{d}{2} - \frac{\bar{x}}{2} \right) \text{ .... Equation 3.7}$$

Ultimate moment carrying capacity of the steel M_s

$$M_s = \sum_{i=1}^{m} P_{si} \left(\frac{d}{2} - d_i \right) \text{ .... Equation 3.8}$$

Ultimate moment

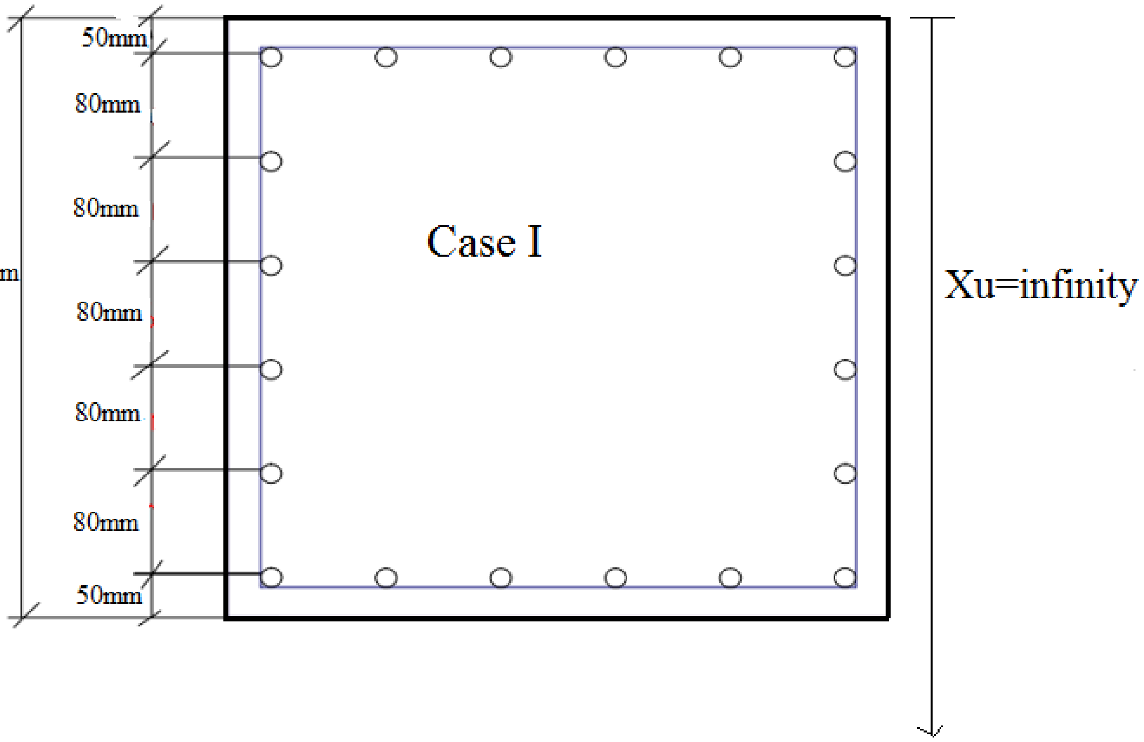
$$M_u = M_c + M_s \text{ .... Equation 3.9}$$

3.3 Worked Out Example

3.3.1. Determination of ultimate loads and ultimate moments using Equation 3.1 to Equation 3.9 and drawing interaction curve for Case I (the section with uniformly distributed longitudinal reinforcement. 20 bars of 16mm diameter reinforcement distributed uniformly along the four faces of the column as shown in Fig. 3.1(a))

1. Neutral axis depth greater than the depth of section $\boldsymbol{X_u > d}$

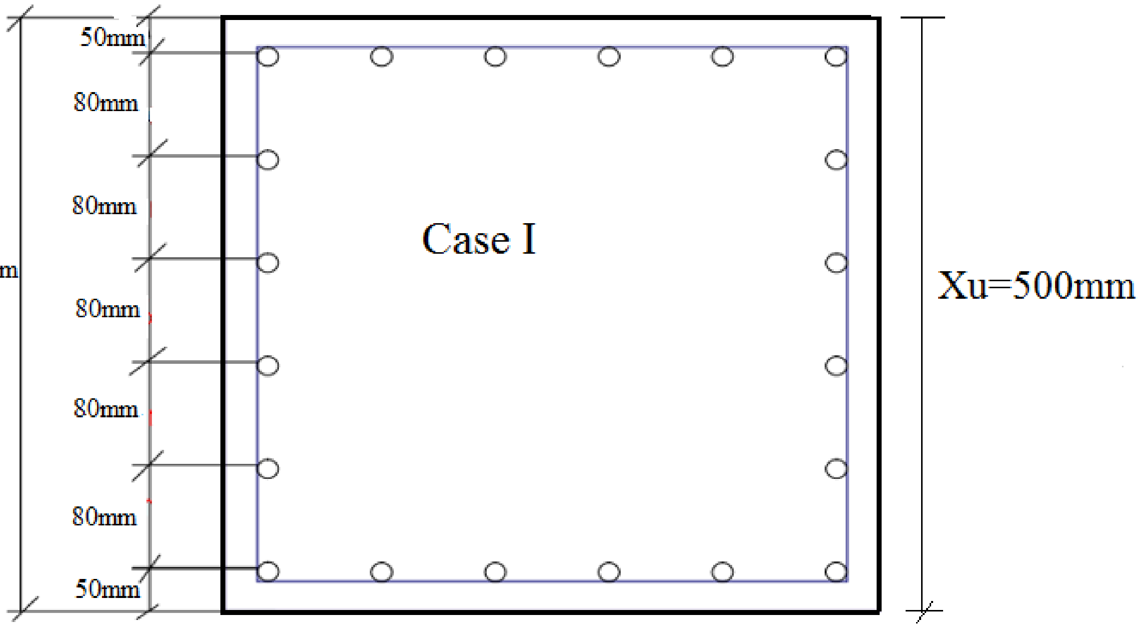
Xu at infinity distance



Ultimate load p_u = 5734.28KN Ultimate moment M_u = 0KNM

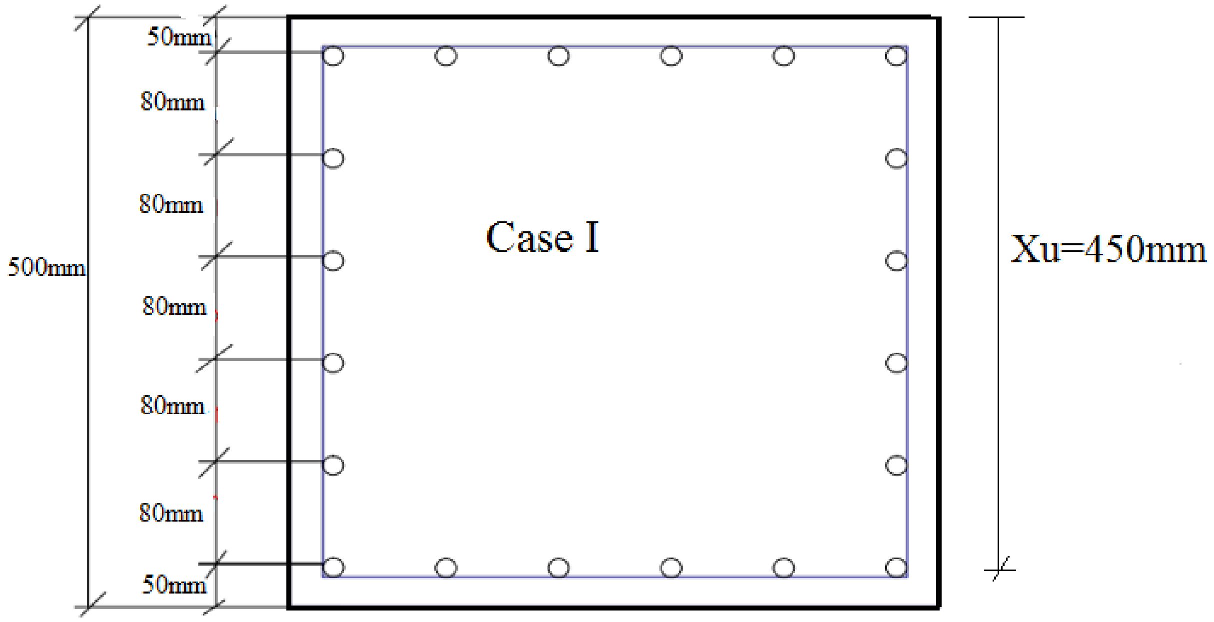
2. Neutral axis depth lies inside the section $\boldsymbol{X_u \leq d}$

When neutral axis $\boldsymbol{X_u}$ at level of 500mm



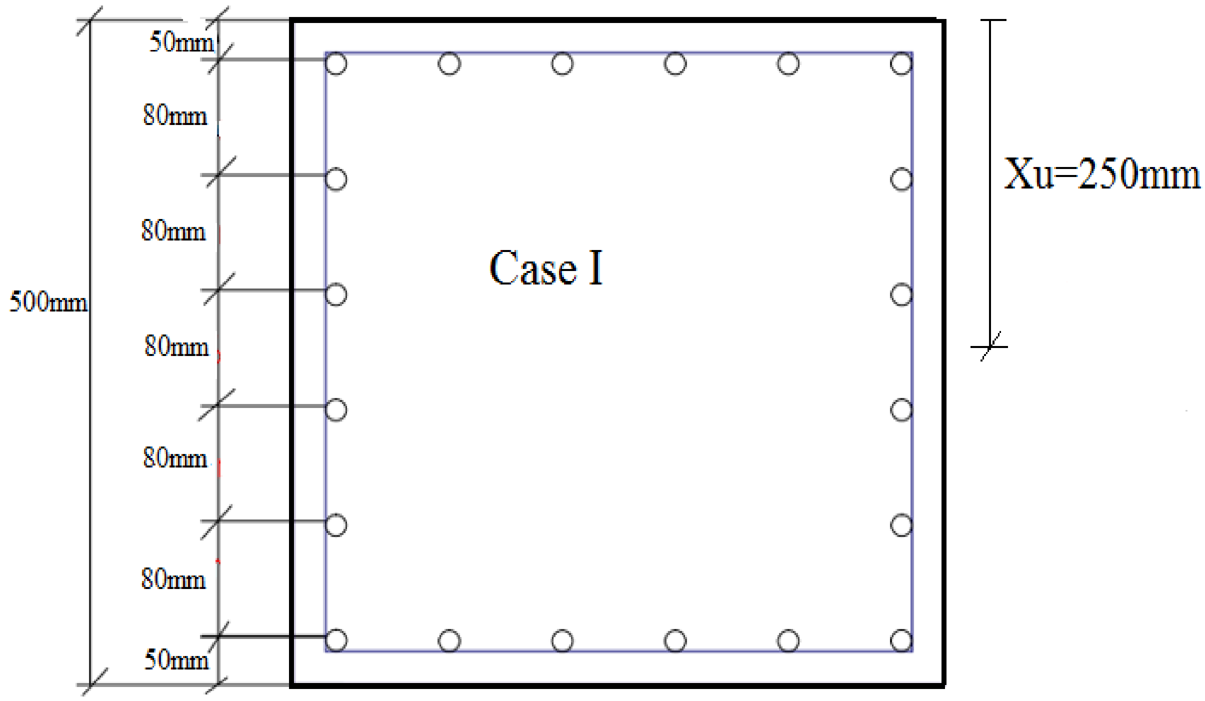
Ultimate load p_u = 4366.01 KN Ultimate moment M_u = 254.56 KNM

When neutral axis depth is at level of 450mm



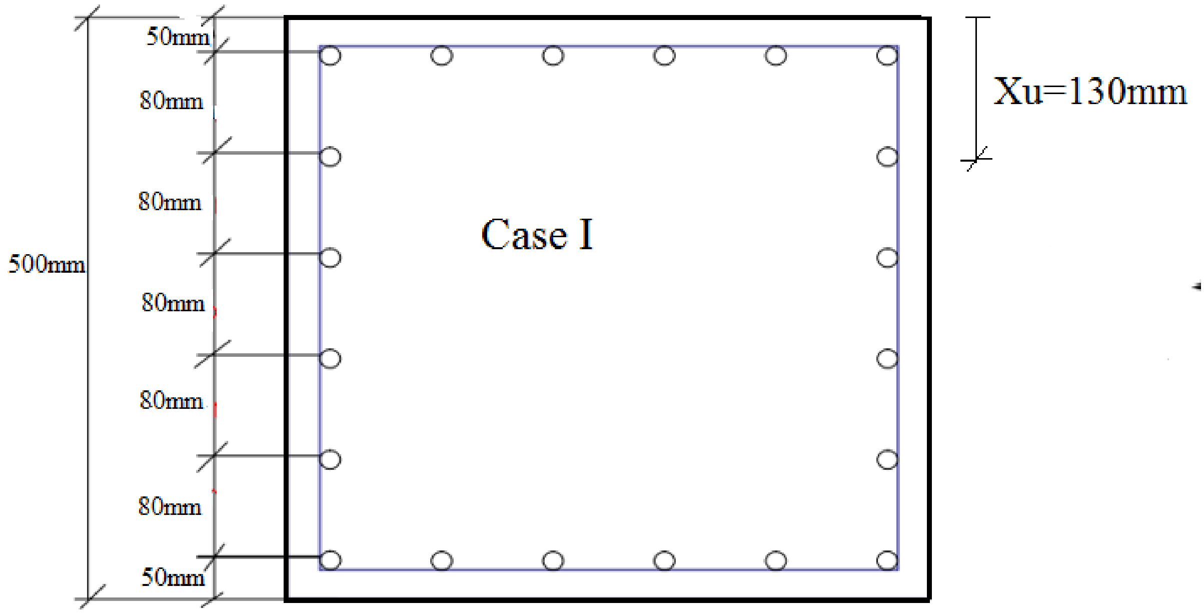
Ultimate load p_u = 3983.65 KN Ultimate moment M_u = 303.45 KNM

When neutral axis depth is at level of 250mm



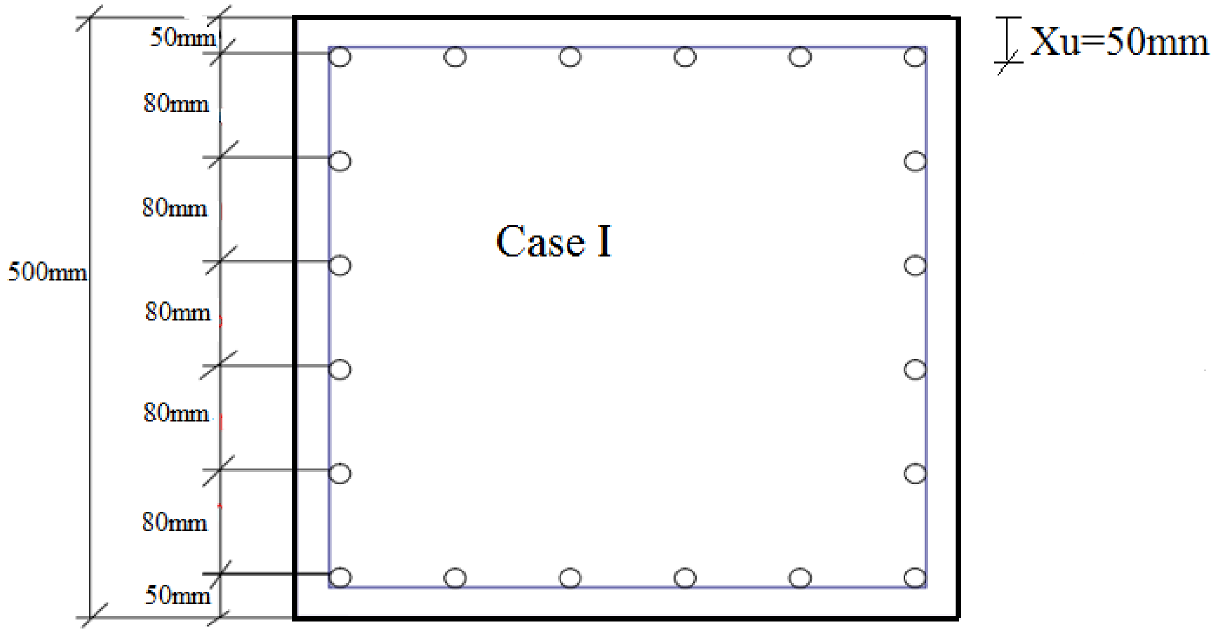
Ultimate load p_u = 1669.32 KN Ultimate moment M_u =468.96 KNM

When neutral axis X_u at level of 130mm



Ultimate load p_u = 481.09 KN Ultimate moment M_u =368.78 KNM

When neutral axis X_u at level of 50mm



Ultimate load p_u = -604.88 KN Ultimate moment M_u =176.84 KNM

When neutral axis X_u at level of 0mm

Ultimate load p_u = -1608.4 KN Ultimate moment M_u =0 KNM

Table 3.1. Values of Pu and Mu for uniformly distributed longitudinal reinforcement

Xu- Distance (mm)	X-axis Mu(KNM)	Y-axis Pu(KN)
Infinity	0	5734.28
500	254.56	4366.01
450	303.45	3983.65
250	468.96	1669.32
125	368.78	481.09
83.2	290.15	0.822
0	0	-1608.4

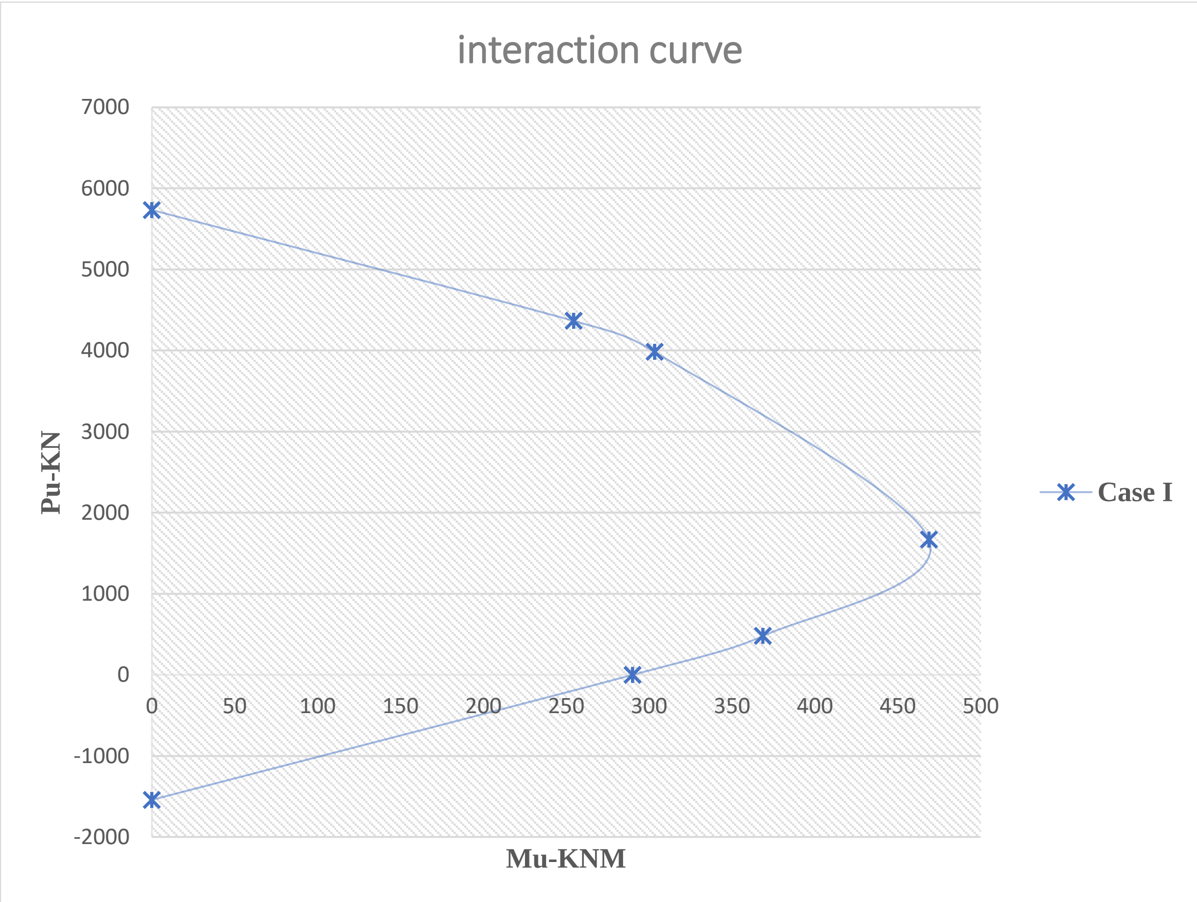


Fig 3.2 Interaction Curve for The Section with Uniformly Distributed Reinforcing Bars

3.3.2. Determination of ultimate loads and ultimate moments and drawing interaction curve for Case II (Three bar bundle of 16mm diameter at the corner of the section and two bar bundle of 16mm diameter at the center of the four faces of the section as shown in Fig 3.1(b))

Using similar procedure with that of having uniformly distributed longitudinal reinforcement section the calculated results Pu along Y-axis and Mu along X-axis for different rows of reinforcements are summarized below.

Table 3.2. Values of Pu and Mu for column section with bundle bars place (3Ø16 at each corner & 2Ø16 at center)

Xu-Distance (mm)	X-axis Mu(KNM)	Y-axis Pu(KN)
Infinity	0	5721.72
500	268.31	4326.3
450	327.83	3877.95
250	496.28	1665.82
198.5	225.61	-0.66
0	0	-1608.4

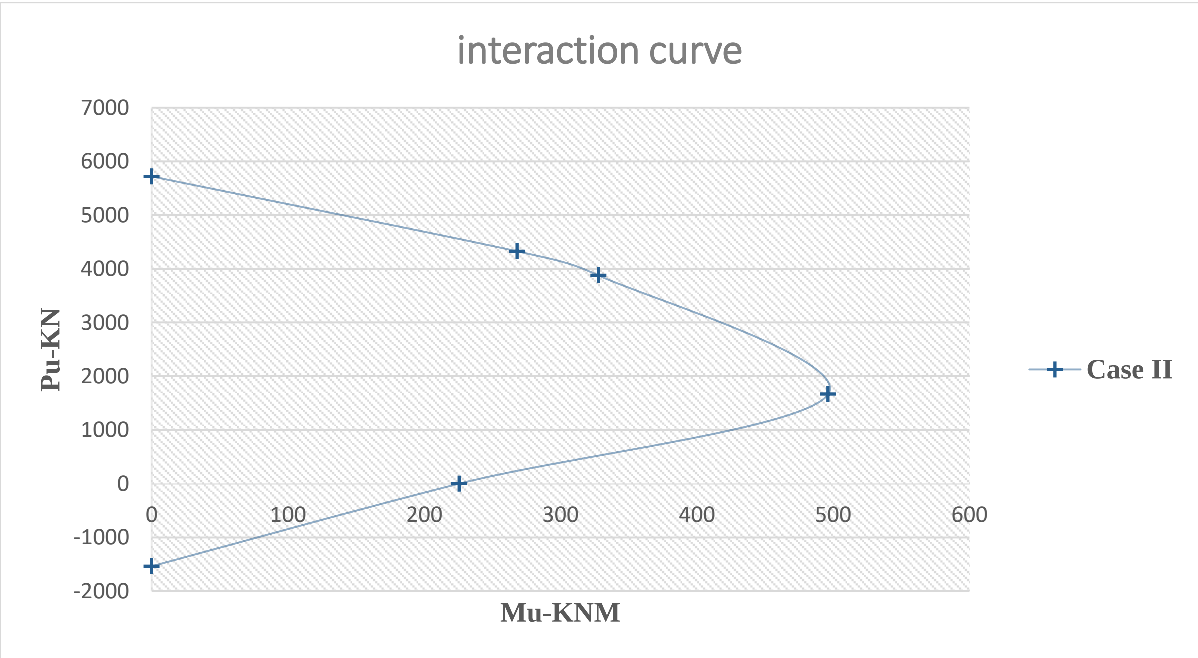


Fig 3.3 Interaction Curve for The Section with Bundle Bars Place (3Ø16 at each corner & 2Ø16 at center)

3.3.3 Case III (Four bar bundle of 16mm diameter at the corner of the section and one 16mm diameter bar is placed at the center of the four faces of the section as shown in Fig 3.1(c))

The calculated results Pu along Y-axis and Mu along X-axis for different rows of reinforcements are summarized below.

Table 3.3. Values of Pu and Mu for column section with bundle bars place (4Ø16 at each corner & 1Ø16 b/n corner bars)

Xu- Distance (mm)	X-axis Mu(KNM)	Y-axis Pu(KN)
Infinity	0	5721.72
500	281.02	4281.12
450	344.36	3820.86
250	527.08	1665.82
213.4	233.85	0.23
0	0	-1608.4

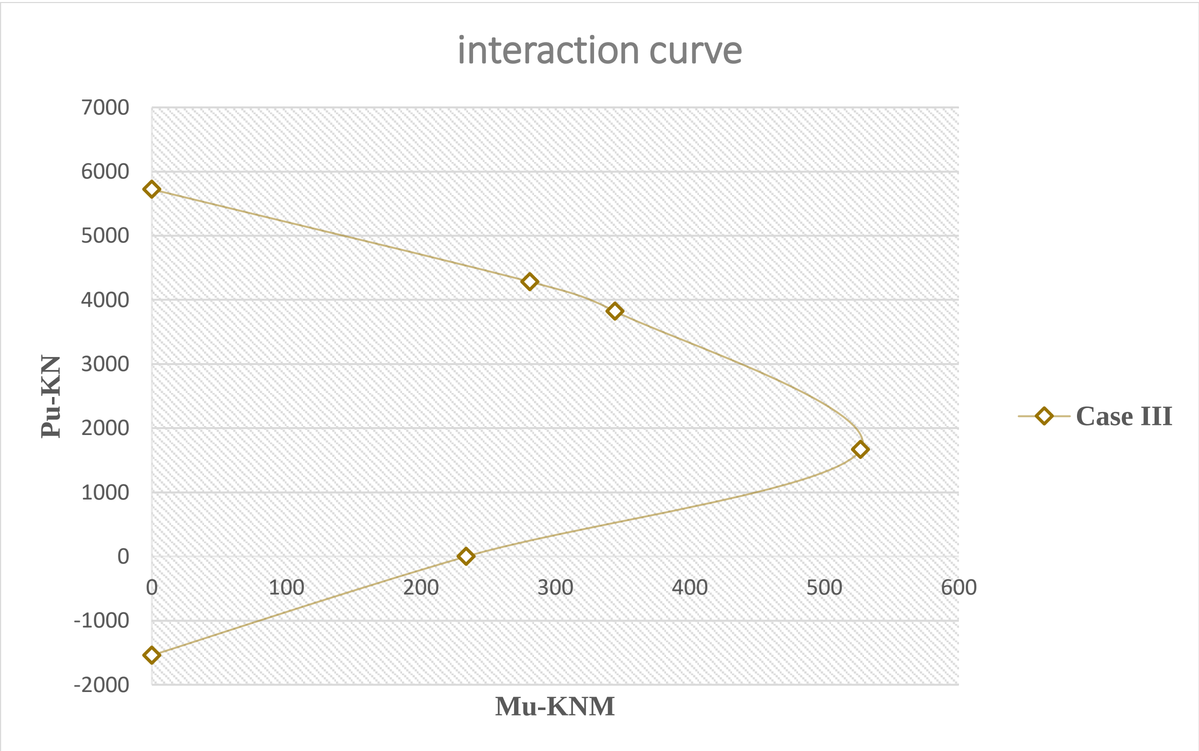


Fig 3.4 Interaction Curve for The Section with Bundle Bars Place (4Ø16 at each corner & 1Ø16 b/n corner bars)

3.3.4. Case IV (One 32mm diameter at the corner of the section and one 16mm diameter bar is placed at the center of the four faces of the section to check equivalent area as shown in Fig 3.1(d)). The calculated results Pu along Y-axis and Mu along X-axis for different rows of reinforcements are summarized below.

Table 3.4. Values of Pu and Mu for equivalent bar with bundled of 4Ø16 bars

Xu-Distance (mm)	X-axis Mu(KNM)	Y-axis Pu(KN)
Infinity	0	5721.72
500	281.02	4281.12
450	329.3	3864.35
250	496.28	1665.82
213.4	233.85	0.23
0	0	-1608.4

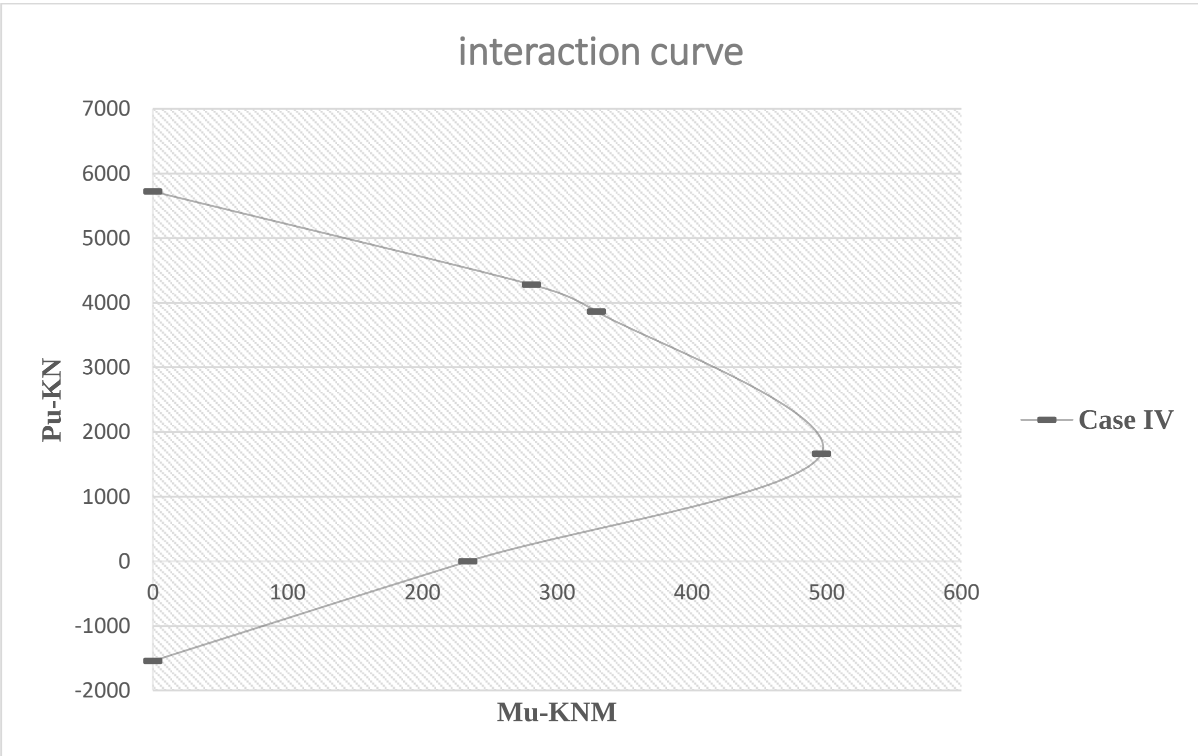


Fig 3.5 Interaction Curve for The Section with Equivalent Bars of 4Ø16 Bars

To verify the result using simple mechanics

Maximum compression capacity

Mu=0

$$p_u = (bh - A_{st}) f_{cd} + A_{st} f_{yd}$$
$$p_u = (500*500 - 4020)*17 + 4020*400)$$
$$p_u = 5789.66KN$$

Maximum tensile capacity

At this point assume all the reinforcement bars are yield

$$M_u = A_{st} f_{yd}$$
$$p_u = 4020*400$$
$$p_u = 1608KN$$

To verify the result using design chart provided in the code **uniaxial chart No 7**

$$V = \frac{N}{f_{cd} bh}$$
$$\mu = \frac{N}{f_{cd} bh^2}$$
$$\omega = \frac{A_{st} f_{yd}}{bh f_{cd}} = 0.378$$

The capacities of the curve obtained from analysis result for all case is safe by comparing ω used for analysis and ω obtained from the interaction curve. For all cases that was used in this paper the mechanical reinforcement ratio (ω=0.35) is within the curve obtained by (ω=0.378) which is provided mechanical reinforcement ratio.

Chapter 4: Program Development

4.1 Modeling steps

In this study, cantilever reinforced concrete column with different longitudinal bar arrangement is seen. The steps for the analysis of two dimensional columns for static loads is presented in this section. The step shown below is used to develop the program.

1. Enter dimension of the structure/2D
2. Build the node of the column
3. Enter boundary condition of the column/fixity
4. Build material properties of the column

    4.1. Enter nominal concrete compressive strength of the column

    4.2. Enter concrete elastic modulus

    4.3. Enter concrete shear modulus

    4.4. Enter compressive strength of confined concrete

    4.5. Enter strain at maximum stress for confined concrete

    4.6. Enter strain at ultimate stress for confined concrete

    4.7. Enter unconfined concrete strength of unconfined concrete

    4.8. Enter strain at maximum strength of unconfined concrete

    4.9 Enter ultimate stress for unconfined concrete

    4.10. Enter strain at ultimate stress for unconfined concrete

    4.11. Enter lambda (ratio between unloading slope and initial slope) for unconfined concrete

    4.12. Enter steel yield stress

    4.13. Enter tensile strength +tensions

    4.14. Enter tension softening stiffness

    4.15. Build core concrete (confined)

    4.16. Build cover concrete (unconfined)

    4.17. Build reinforcement material

5. Enter section geometry (width x depth)
6. Enter number of longitudinal reinforcement
7. Enter area of longitudinal reinforcement
8. Enter cover for reinforcement bar

9. Enter fiber section properties

9.1. Enter cover and core for both axes

10. Enter the core patch, four cover patches, and reinforcement layer respectively

11. Enter geometric transformation, integration points

12. Create nonlinear column/beam element (element dispbeamColumn)

13. Enter gravity loads at node 2

Enter constraints transformation, numberer plain, system BandGeneral, and test NormDispIncr, algorithm Newoton

Enter Integrator loadcontrol

Enter analysis static

Enter analyze

Enter loadConst-time 0.0

Enter lateral loads

14. Define recorder

14.1. Recorder node (displacement reaction forces at each node)

14.2. Recorder element (local,global,section forces,deformation,stress strain,stiffness)

Enter pushover analysis in loops

Print node

Print ele

Print sec

Display the analysis results

4.2 Program Development on Opensees

A 3m height cantilever reinforced concrete column for axial compression and lateral loads (static load) is studied. The dimension of the column is 0.5m by 0.5m, the cylindrical compressive strength of concrete is 30MPa and expected yield strength of steel deformed bars is 460MPa. The column is subjected to 100KN constant compressive load and 5KN gradually increasing static lateral load.

In two dimensional columns, the number of degrees of freedom (dof) per node is 3, namely dx, dy, and rz, and therefore, the total number of degrees of freedom is 3 times the number of nodes (**n**). However, every structure has supports with certain degrees of freedom constrained (otherwise the structure would be a free body capable of undergoing rigid-body motion). These constrained dof have zero displacements. As a result, the number of unknown degrees of freedom (**ndof**) is smaller than 3x**n**. The modeling for this paper using Opensees is prepared as follows in the following sequence:

Run the Opensees Application, the opened window is shown below

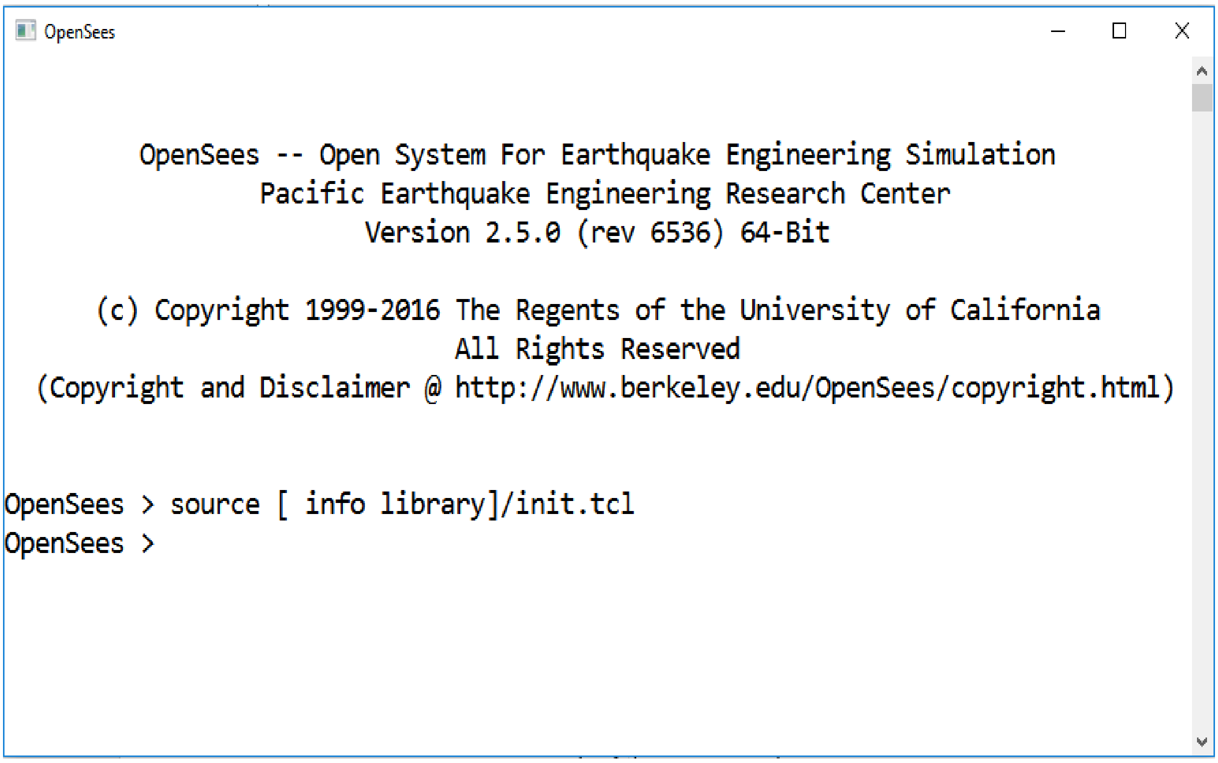


Fig. 4.1 Opensees Working Area

4.2.1 ModelBuilder- The model builder constructs as in any finite element analysis, the analyst's first step is to subdivide the body being studied into elements and nodes, to define loads acting on the elements and nodes, and to define constraints acting on the nodes. The ModelBuilder is the object in the program responsible for building the following objects in the model and adding them to the domain (Node, Mass, Material, Section, Element, LoadPattern TimeSeries, Transformation, Block, Constraint).

For two dimensional problems, 3 degrees of freedom at each node, two translations, and one rotations. Because Opensees does not use internal units; the user must keep track of the types of units being used.

4.2.2 Nodal Coordinates- Once the dimension of the problem is defined, it recommended that the user define the coordinates of the nodes, the mass associated with each node and DOF and the boundary conditions at the nodes. The nodal coordinates are defined using the node command. The numbers of parameters associated with this command are referenced to the model command. Nodal masses can be defined at the same time as the coordinates. In the two-dimensional problem considered here, only the z and y coordinates of each node need to be defined, and three mass parameters (two translations and one rotation).

4.2.3 Boundary Conditions-The boundary conditions are defined using the fix command. tag 0 represents an unconstrained (free end) degree of freedom; tag 1 represents a constrained (fixed end) degree of freedom.

4.2.4 Materials Once the nodes have been defined, the next step towards defining elements is the material (nDMaterial Command, uniaxialMaterial Command) definition. This step may not be necessary when using elastic element or sections, as the materials are defined with the element or section. There are two types of materials currently available in Opensees, uniaxial materials and nDmaterials. The different types of concrete and steel materials are among the uniaxial materials.

Concrete02 will be used for the structure under consideration of this paper, as the tensile strength of the concrete is of interest in the elastic range, and modeling linear tension softening is considered. The cover and core concrete will be modeled as different materials, using the same material type, but different stress and strain characteristics and different material tags. Steel02 will be used for the reinforcing steel. Because some material characteristics are dependent on others, it is recommended that the user define the material properties using variables. It is also a good idea to use variables for IDtags of materials, sections, elements, etc. This is done to ensure that the same IDtag is not used when defining the input.

Table 4.1. material properties for concrete and steel

confined concrete		**Unconfined concrete**		**Steel**	
Material properties	values	Material properties	values	Material properties	Values
Kfc	1.3				
Fc	30000	Fc	30000	Fy	400000
Ec	31000000	Ec	31000000	Es	200000000
fc1C	Kfc*fc	fc1U	fc	B	0.01
eps1C	2.*fc1C/Ec	eps1U	-0.0035	R0	20
fc2C	0.2*fc1C	fc2U	0.2*fc1U	cR1	0.925
eps2C	5*eps1C	eps2U	-0.001	cR2	0.15
ftC	-0.14*fc1C	ftU	-0.14*fc1U		
Lambda	0.1	Lambda	0.1		

4.2.5 Element Cross Section

some element types require that the element cross section be defined a-priori. This is done using the section command. The section is used to represent force-deformation (or resultant stress-strain) relationships at beam-column and plate sample points. While there are many types of sections available; the fiber sections will be used to define the cross section of the column. The fiber section has a general geometric configuration formed by sub regions of simpler, regular shapes (e.g. quadrilateral, circular and triangular regions.) called patches. In addition, individual or layer of reinforcement bars can be specified. The cross section of column core concrete cover and reinforcements are defined using fiber section command. In this work the fiber section was formed by quadrilateral region and reinforcement bars were arranged in layer for uniformly distributed bars and fiber command for bundled bars.

Fiber: - a single fiber can be defined, such as a single reinforcing bar. The coordinates, associated area, and material tag are prescribed with the fiber. (The coordinates are given with respect to the plane of the cross section; a coordinate transformation is later defined in the input using the transformation command)

Patch: - a patch defines an area that has a regular shape; quadrilateral or circular. A different material can be associated with each patch.

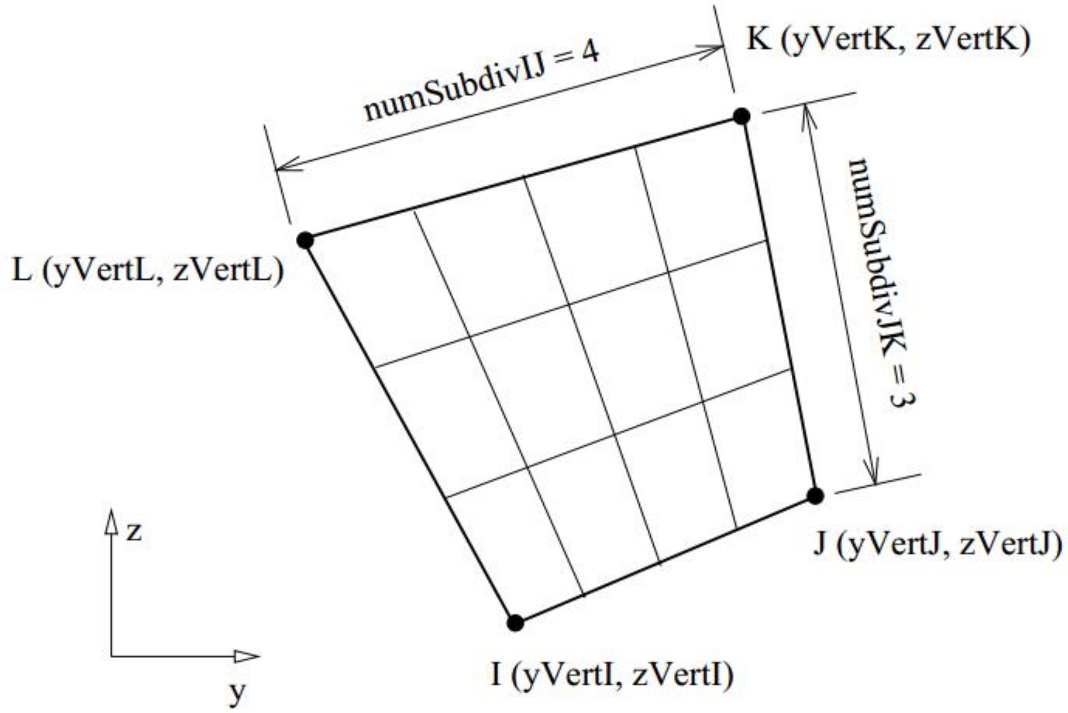


Fig 4.2 Quadratic Fiber Section

Layer: - (Straight Layer Command, Circular Layer Command) a layer defines a layer of reinforcement that has a regular shape: straight or circular. A different material can be associated with each layer.

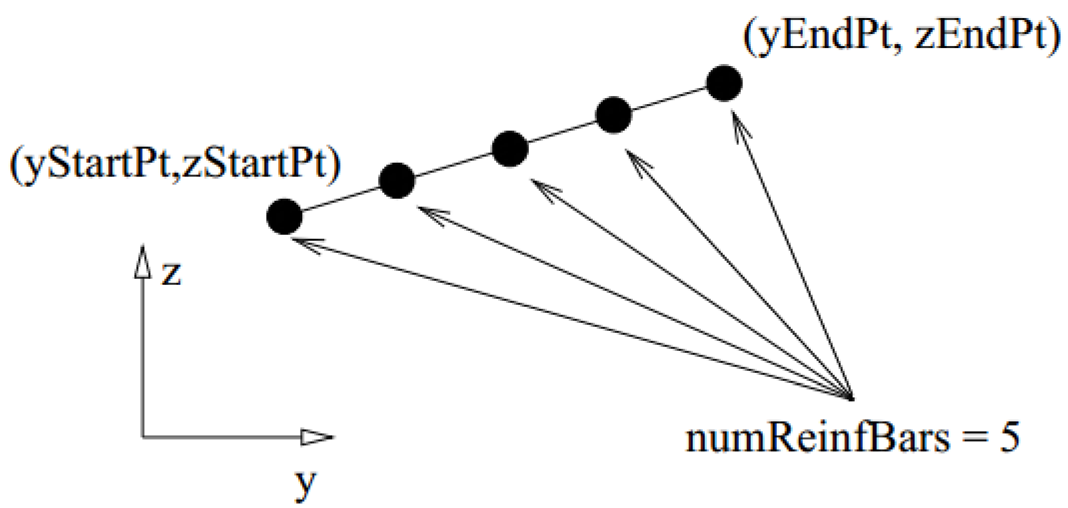


Fig 4.3 Straight Reinforcing Layer

4.2.6 Elements and Elements Connectivity

Once the element cross section has been defined, the geometric transformation is used to relate the local element, and section, coordinates to the global system coordinates.

4.2.7. Nonlinear Beam Column Element

This is used to construct a nonlinear BeamColumn element object, which is based on the non-iterative (or iterative) force formulation, and considers the spread of plasticity along the element. There are basically two types of Nonlinear BeamColumn Elements

- Force based elements
- Distributed inelasticity (nonlinear Beam-Column)
- Concentrated plasticity with elastic interior (beam with Hinges)
- Displacement based element
- Distributed inelasticity with linear curvature distribution (dispBeamColumn)

4.2.8. Loads and Analysis in Opensees

In Opensees loads is applied in a three-step process:

1. Loads must be defined in a load pattern
2. The analysis must be then defined and its features
3. The loads are then applied when the analysis is executed

Load definition: Loads are defined using the pattern command. Three types of patterns are currently available in Opensees: The pattern command is used to construct a LoadPattern and add it to the Domain. Each LoadPattern in OpenSees has a TimeSeries associated with it. In addition, it may contain ElementLoads, NodalLoads and SinglePointConstraints. Some of these SinglePoint constraints may be associated with GroundMotions. The command has the following form:

1. Plain Pattern: - this pattern is used to define the following:

1.1 Nodal loads, such as gravity loads and lateral loads (or load-controlled nodal displacements)

1.2. Single-point constraints, such as displacement control at a node (typically used for a constant displacement at a node)

1.3 Element loads, such as distributed gravity loads along the element (this is a new option, which still needs documentation). In this paper nodal loads are used

2. UniformExcitation Pattern: - this type of pattern imposes a user-defined acceleration record to all fixed nodes, in a specified direction.

3. Multi-Support Excitation Pattern:- The command to generate a multi-support excitation contains in **{ }** the commands to generate all the ground motions and the single-point constraints in the pattern. The command is as follows:

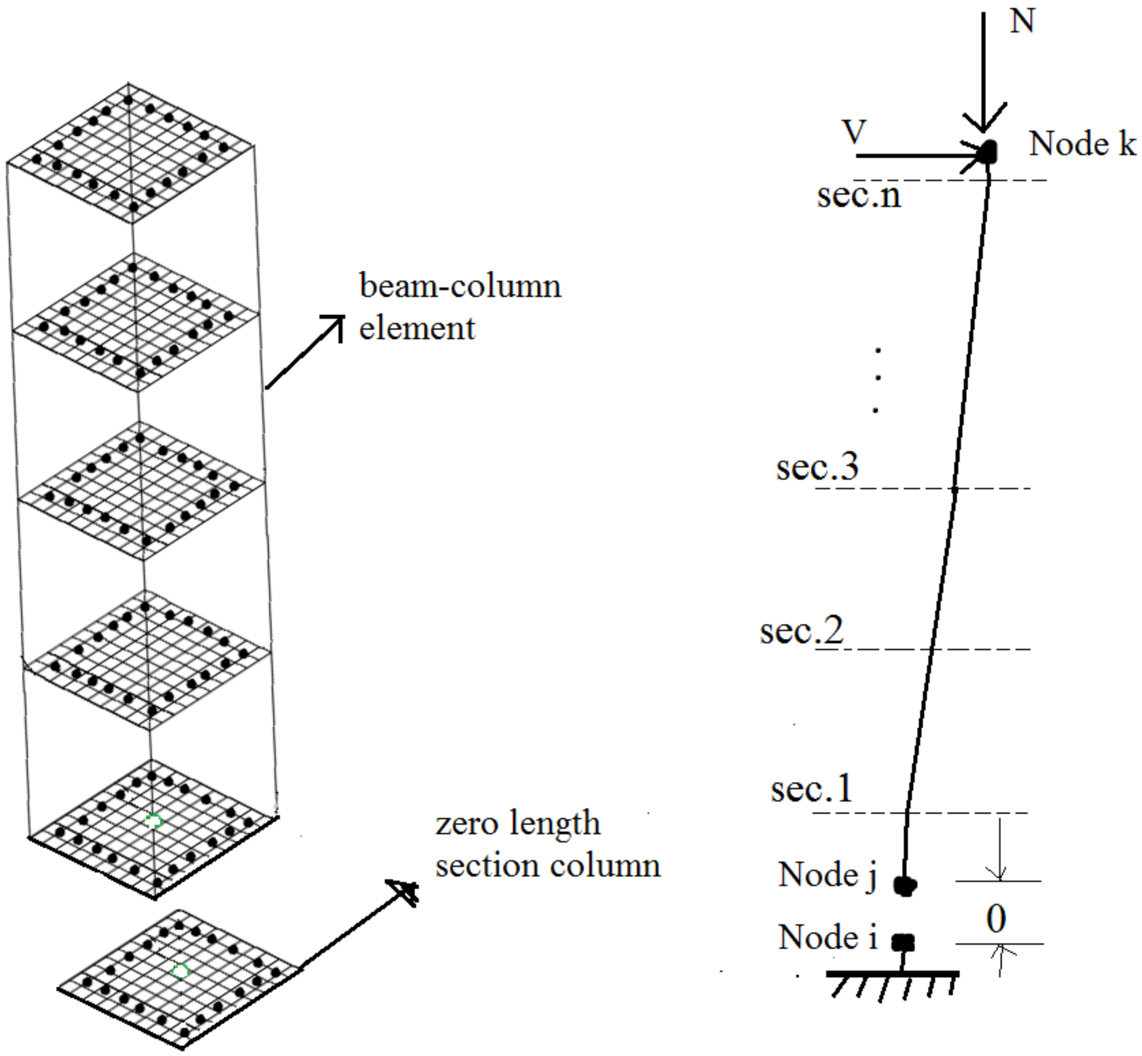


Fig 4.4 Adding a zero length section element to a beam column element

4.2.9 Analysis Object

The Analysis objects are responsible for performing the analysis. The analysis moves the model from state at time t_i to state at time $ti + dt$. This may vary from a simple static linear analysis to a transient non-linear analysis. In Opensees each Analysis object is composed of several component objects, which define the type of analysis how the analysis is performed. Analysis is a high-level class that is invoked on a Domain to advance the state. The object represents the mathematical abstractions for performing an analysis to solve the governing equations, as represented by the current state of the Domain. To accomplish this, the Analysis object is an aggregation of the following objects that are responsible for important aspects of an analysis.

system systemType? arg1? ... This command is used to construct the SystemOfEqn object, it encapsulates the system of equations and provides operations to solve the system. For this work a BandGeneralSOE linear system of equation object was used. As the name implies, this class

The type of solution algorithm created and the additional arguments required depends on the **algorithmType?** provided in the command. For this paper use **ModifiedNewton** algorithmType? to construct SolutionAlgorithm object, which uses the modified newton-raphson algorithm to solve the nonlinear residual equation.

analysisType?

• Static Analysis: - solves the KU=R problem, without the mass or damping matrices.

• Transient Analysis: - solves the time-dependent analysis. The time step in this type of analysis is constant. The time step in the output is also constant.

• Variable Transient Analysis: - performs the same analysis type as the Transient Analysis object. The time step, however, is variable. This method is used when there are convergence problems with the Transient Analysis object at a peak or when the time step is too small. For this paper use **static Analysis analysisType**? to construct analysis object.

4.2.10 Recorder Object

The recorder object monitors user-defined parameters in the model during the analysis.

What does a recorder do?

- Monitors the state of a domain component (node, element, etc.) during an analysis
- Writes this state to a file or to a database at selected intervals during the analysis
- There are also recorders for plotting and monitoring residuals
- Recorders can also be placed anywhere on a fiber section to measure fiber stresses and strains. When more than one material may occupy the location specified (such as a steel bar at the edge of the confined-concrete core), a preferred material can be specified. The location of the recorder is specified using the local coordinate system. If no fiber is located at that coordinate, a blank file will be output (very common error).

4.2.11 Pushover Analysis

Pushover analysis is a static, nonlinear procedure using simplified nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis used to determine the force-displacement relationship, or the capacity curve, for a structure or structural element. The analysis involves applying horizontal loads, in a prescribed pattern, to the structure incrementally, i.e. pushing the structure and plotting the total applied shear force and associated lateral displacement at each increment, until the structure or collapse condition. In this paper lateral loads are applied at node 2 with appropriate direction but, prior to it is an Opensees script called “loadConst -time 0.0” which means the Opensees program understand that gravity load is constant then additional lateral loads are applied. Finally, pushover analysis in a loop is made to run lateral load with increasing condition and after the prescribed iteration, the pushover analysis will be successful or fail.

4.2.12 Running the Script

In the output the results of the print commands are noticed in the file. A lot of warning messages will be seen with in the analysis script that the analysis step has failed and the alternative initial step iterations are being performed. Opensees Analysis Capabilities Linear equation solvers, time integration schemes, and solution algorithms are the core of the Opensees computational framework. The components of a solution strategy are interchangeable, allowing analysts to find sets suited to their particular problem.

To display the analysis results in Opensees window use Print command, Print node (to display node Displacements, unbalanced Load and reaction), Print ele (to display End Forces (End1 and End2)). In Opensees modeling, if the final analysis shows that it is completed successfully, the program is said to be run successfully, “Pushover analysis completed SUCCESSFULLY”.

4.3 Output Format

The format of the output is typically dependent on the element and/or section type. In general, however, the output follows the order of the degrees of freedom. Here are some cases:

element global Force 2D, 3dof: FY FX MZ

local Force 2D, 3dof: Fy Fx Mz

section force Fy Fx Mz

deformation axial-strain curvature

StressStrain stress-strain

The element deformations, section forces, stress strain of section at the cross section level can be recorded at any integration point. For example, in this paper column deformation, section forces, stress strain of the section are recorded at the first integration point.

4.4 Model Verification

In this section a verification of the model by using SAP2000 is presented. The chosen column is modeled in SAP2000 using material type concrete for concrete and rebar for reinforcement steel. The section for column rectangular element with 500mm x 500mm dimensions created by section designer.

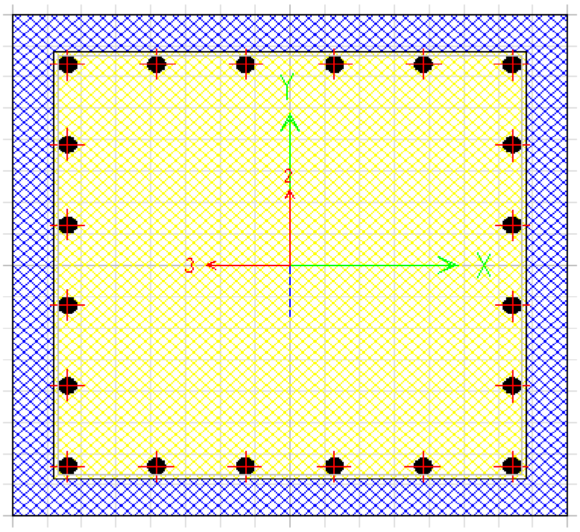
Table 4.2. concrete model parameter

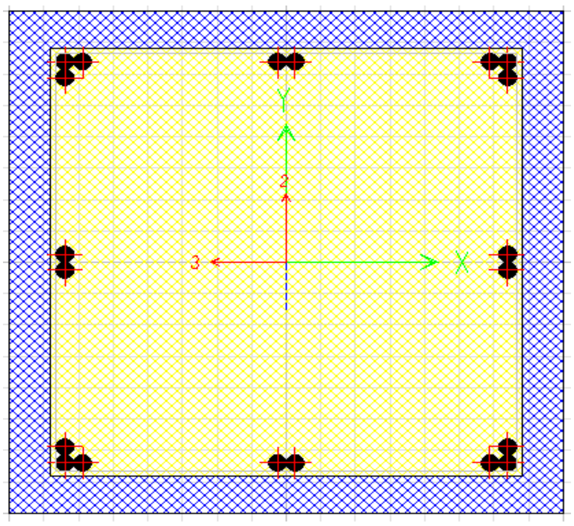
Parameters	Values	Unit
Mass Density	2.5	Kg/M^3
Compression Strength	30	Mpa

Table 4.3. reinforcement model parameter

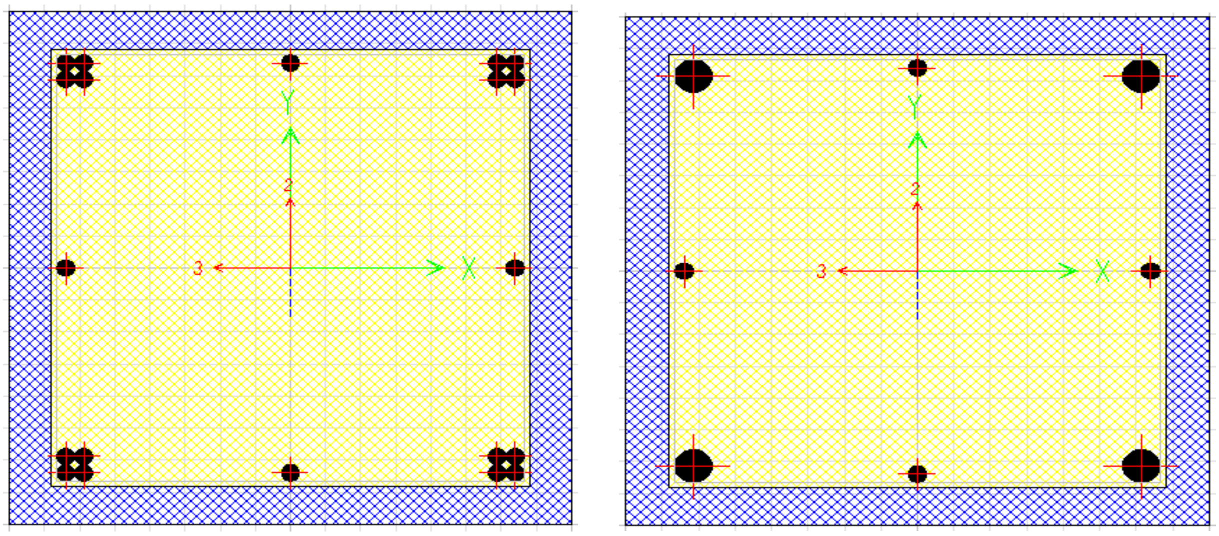
Parameters	Values	Unit
Mass Density	7.85	Kg/M^3
Modules of Elasticity	200	Gpa
Poison's Ratio	0.3	
Yield Stress	400	Mpa

Column was fixed from the bottom in all direction and free from the top.





a) Column sections with equally spaced bars b) Column sections with bundled bars



c) Column sections with bundled bars d) Column sections with equivalent bars

Fig. 4.5 Column Sections Created by Section Designer

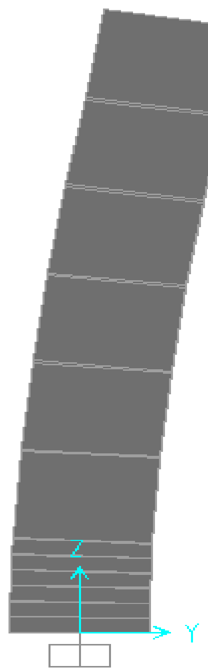


Fig. 4.6 Deformed Shape of the Column

From the analysis result the pushover curve for all cases was similar. Deflection of the column at maximum load of 157.28KN is 59.69mm as shown in Fig. 4.7 which was in a very good agreement compared to 64.2mm (case IV), 51mm (case II) and 53.9mm (case I) from Opensees result. The shape of the deflection curve for both SAP2000 and Opensees were nearly identical.

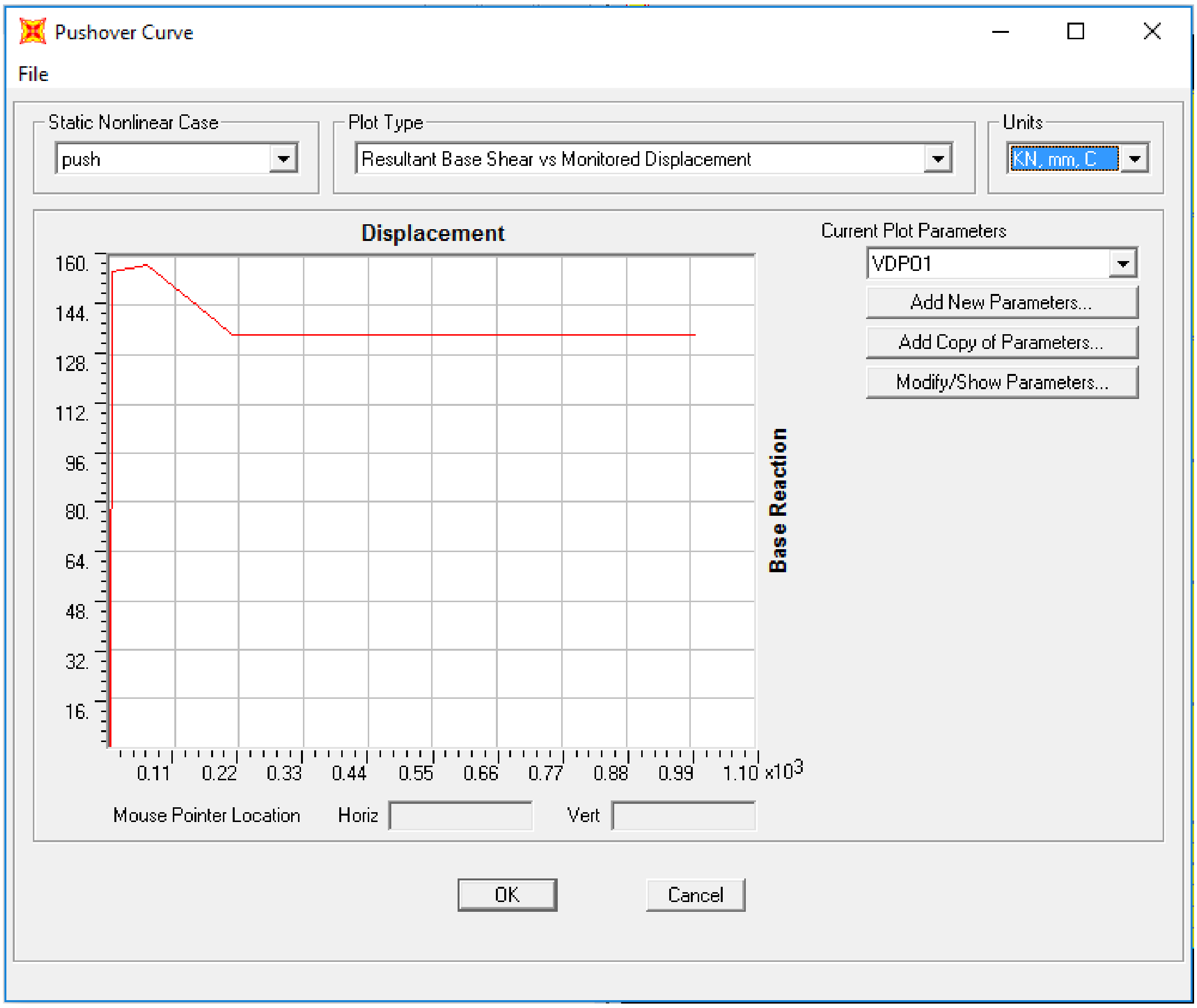


Fig. 4.7 Static Pushover Curve

Chapter 5: Results and Discussions

In this thesis, moment and static lateral load resisting capacity of rectangular reinforced concrete column with bundle bars has been simulated numerically. The following arrangements of reinforcing bars are targeted and discussed in this research: the first case was column with uniformly disturbed longitudinal bars along the four faces 20Ø16 (conventional one), the second case, column with 2Ø16 bars bundled between the corner bars on four faces and 3Ø16 bars bundled at the corners, the third case, column with 1Ø16 bar between the corner bars on four faces and 4Ø16 bars bundled at the corners, and the fourth case for this paper was column with equivalent bars 1Ø32 with the 4Ø16 bars bundle. For all cases, axial load vs. moment capacity using interaction curve for different strain profile and also lateral deformation capacity by applying static lateral load was determined. The developed program produced the output in the external file. The result constitutes all the necessary analysis outputs such as nodal displacement, reaction forces, local and global forces, section forces, and deformation. Stress-strain relationship of either rebar, confined concrete or unconfined concrete is also manipulated. Fig. 5.1 shows the intended output recording location; stress-strain relationship for core concrete was recorded at the center of section one and for reinforcement bars it was recorded at the corner of positive y-z axis in section one.

Opensees program has “put” command that assure whether the modeling is correct or wrong, so “put” command was used and pushover analysis run successfully. The accuracy of the solution may depend on the number of iteration, integration points, fiber sections, etc. However, as we increase the above mentioned conditions, super computer may be needed to solve the problem and it may be time consuming. The Opensees scripts developed for cantilevered column is attached in Appendix –A.

5.1 Moment capacity

Fig. 5.3 and Table 5.1 shows that the moment capacity of a reinforced concrete column is increased when the longitudinal reinforcing bars becomes bundled. Table 5.1 shows that column with four bars bundle have largest capacity, its capacity increases 25% of the customary one; Column with larger bar diameter (equivalent bars with bundled bars) and column with three bars bundle have higher capacity, its capacity increases 15% and 7.6% respectively of column with uniformly distributed longitudinal bars.

5.2 Lateral load resisting capacity

It is found that the unbalanced load vs nodal displacement characteristics of a column for different modeling is shown in Fig. 5.2. The numerical value of a column with bundled bars have less displacement with higher force. On the other hand, column with uniformly distributed bars and column with larger diameter bars which is equivalent to bundled bars relatively displaced more with small force.

Table 5.1 shows that column with four bars bundle have largest lateral load capacity, its capacity increases 25% of the customary one; Column with larger bar diameter (equivalent bars with bundled bars) and column with three bars bundle have higher lateral load capacity, its capacity increases 15% and 6% respectively of column with uniformly distributed longitudinal bars. From the figures column with bundled bars displaced less with higher section and unbalanced load.

In this paper the properties of column with bundle bars is studied in terms of shear vs nodal displacement and moment capacity.

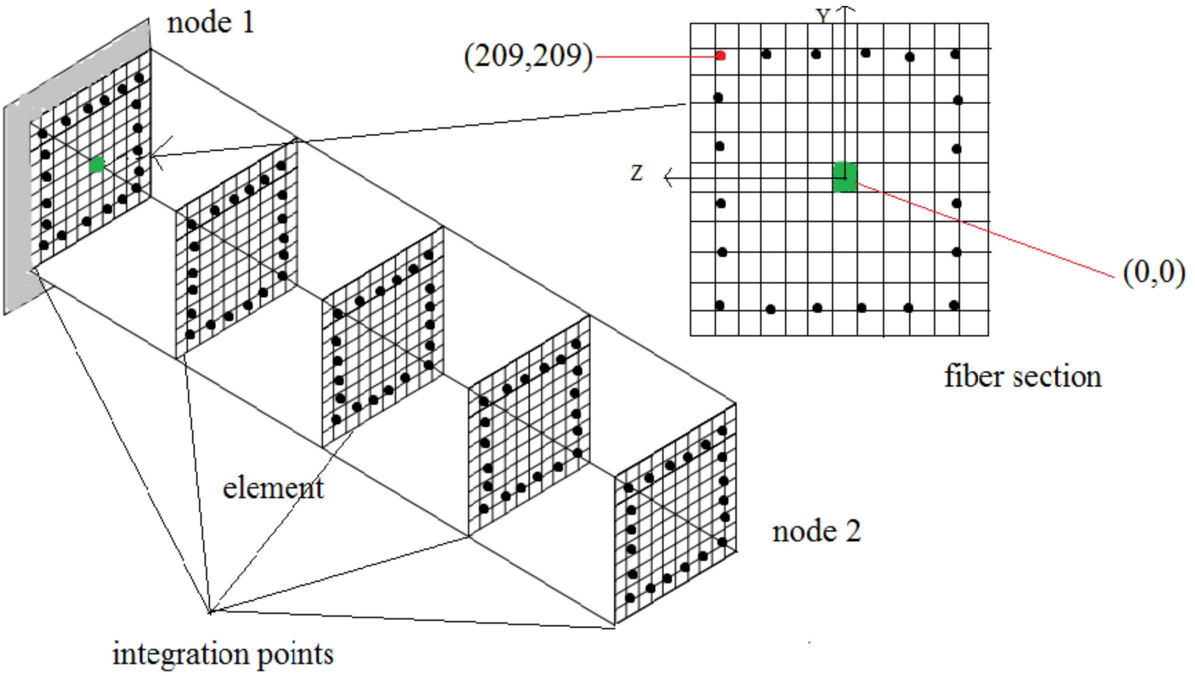


Fig. 5.1 Element and Fiber Section

Sample example

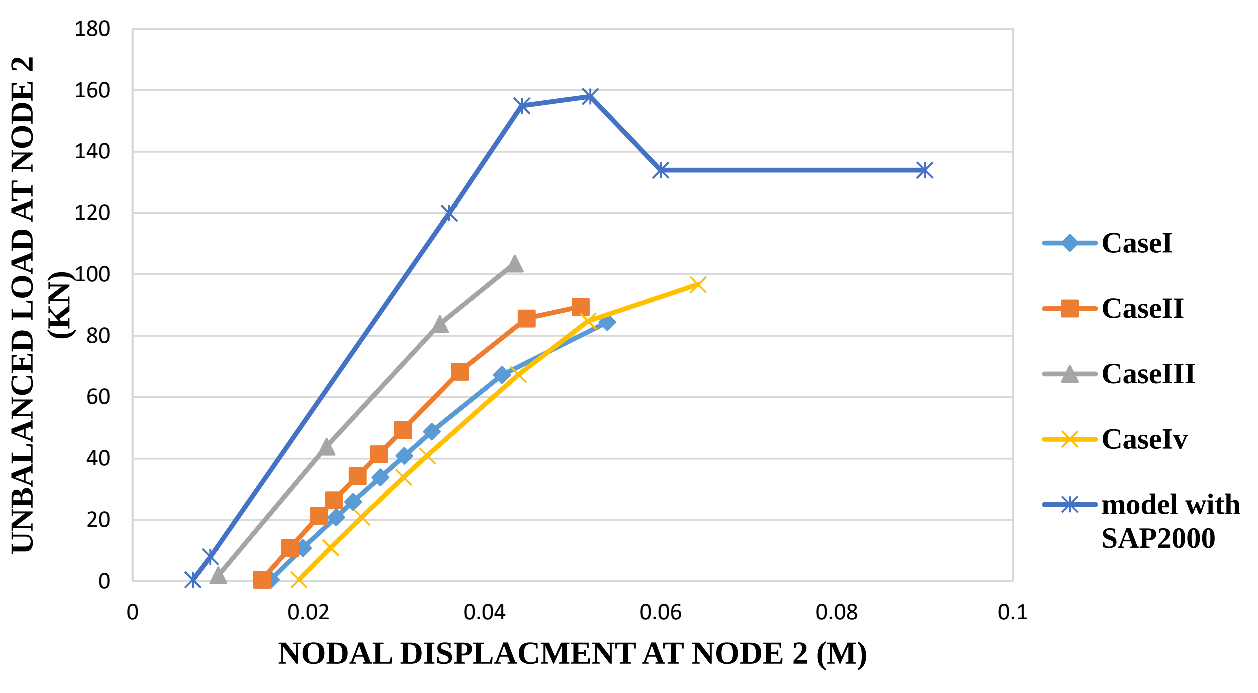


Fig. 5.2 Unbalanced Load Vs. Nodal Displacement at Free End for The Sections

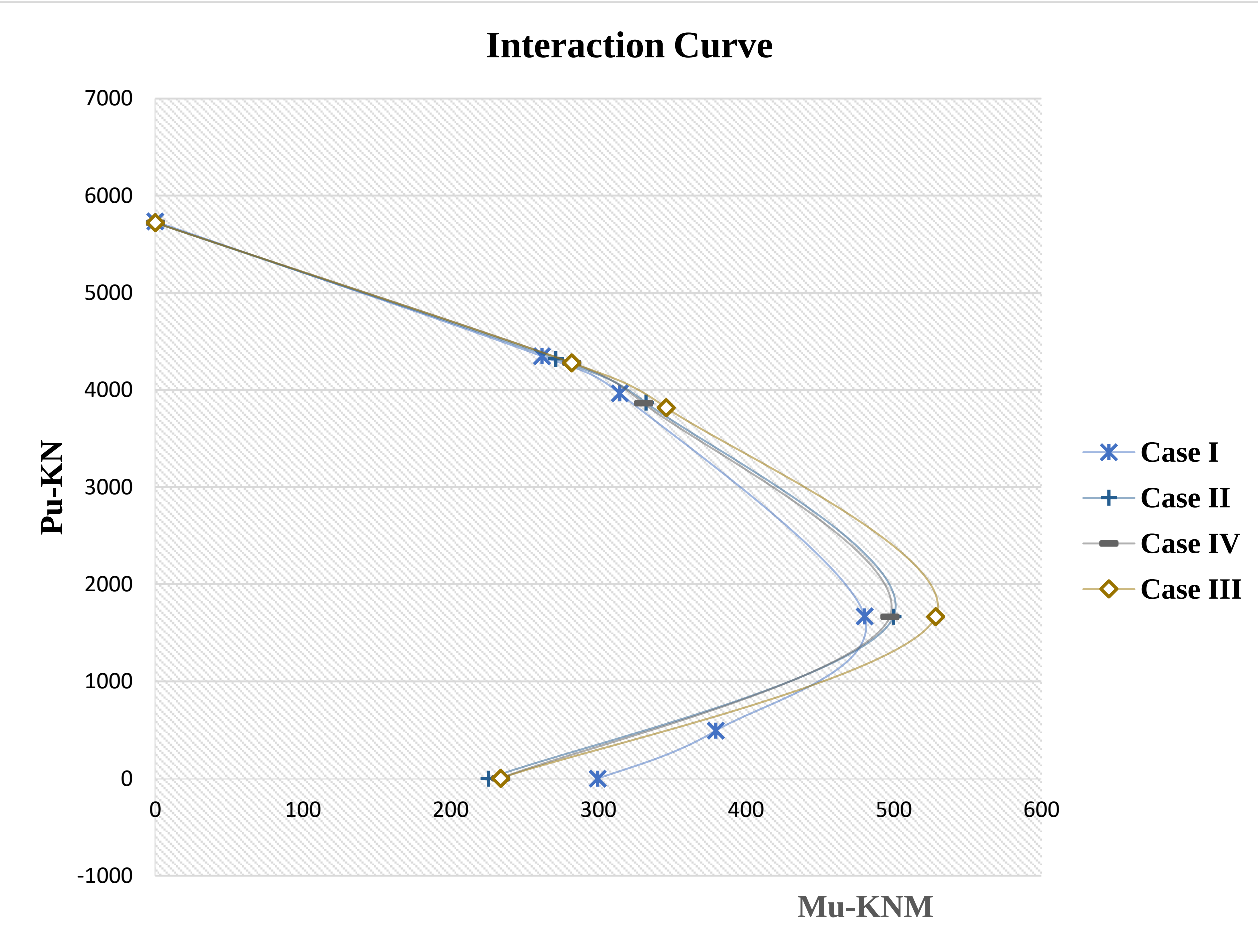


Fig. 5.3 Interaction Curve for The Section with Four Different Bar Arrangements

Table 5.1 Global forces for the sections at failure

		P	V	M	ΔV	ΔM	% increment of V	% increment of M
Case I	End 1	98.2	84.48	253.95	0	0	0	0
	End 2	-98.2	-84.48	-0.49	0	0	0	0
Case II	End 1	78.5	89.41	273.2	4.93	19.25	5.84	7.58
	End 2	-78.5	-89.41	-4.97	-4.93	-4.48	5.84	0
Case III	End 1	100	106	318	21.52	64.05	25.47	25.22
	End 2	-100	-106	0	-21.52	0.49	25.47	0
Case IV	End 1	100	97.5	292.5	13.02	38.55	15.41	15.18
	End 2	-100	-97.5	0	-13.02	0.49	15.41	0

Chapter 6: Conclusion and Recommendations

6.1. Conclusions

The result obtained in the research leads to the following conclusions.

1. Columns with bundled longitudinal reinforcement exhibit higher resisting moment compared to columns with uniformly distributed longitudinal reinforcement irrespective of the location of neutral axis

2. Column with bundled bars gives higher stress and strain values at every row of reinforcements compared to the conventional column with uniformly distributed reinforcement.

3. Lateral load resisting capacity of a reinforced concrete column is increased when the longitudinal reinforcing bars becomes bundled.

4. According to the results of the numerical simulation conducted in this study, the global forces (internal forcers or action) of the column modeled with bundled bars are greater than column with uniformly distributed bars.

5. It indicates that a column with bundled longitudinal bars with less diameter is safe and economical for the construction of column.

6.2. Recommendations

Based on the achievements and work of current research, it is recommended to study several improvements that may be targeted for future work.

1. It is recommended that, the designers must know about the capacity of column with bundled bars and develop the trend of design column with bundle bars for safety and economy.
2. The developed program is advisable to be used for nonlinear modeling of reinforced concrete column with different longitudinal bar arrangement. But, the modeling can be modified and used for similar structure.
3. For highly earthquake prone regions should be used column with uniformly distributed bars to ensure ductility.
4. Other parametric studies like different bar size in the bundle, spacing between each bundle of the longitudinal bars should be conducted to study their effect on a column subjected to lateral load.
5. It is advisable to use experimental studies to prove the analysis result.

References

[1] Ethiopian building code standard, “structural use of concrete”, (EBCS 2-1995), ministry of work and urban development, Addis Ababa, Ethiopia, 1995.

[2] American concrete institute committee 318,” building code requirement for structural concrete and commentary” ACI 318-11, American concrete institute, Detroit, MI, 2011.

[3] Daniel Bruce Grant, B.S. “Bond and Development of Bundled Reinforcing Steel” The University of Texas at Austin, August 1994.

[4] B.G Naresh et al. “study on the behavior of rectangular column with unequally spaced longitudinal reinforcement” International journal of Engineering research and Technology, vol.4 july-2015.

[5] Muhammad N.S Hadi “Axial and flexural performance of square reinforced columns wrapped with CFRP under eccentric loading” university of Wollongong, 2012.

[6] Mustafa, Sema “Determination of moment capacity for rectangular reinforced concrete column” published in Teknik Dergi Vol.20, no 1 january 2009.

[7] Maurizio et al. “flexural behavior of reinforced columns externally wrapped with FRP sheets” Dica university of palerno, Italy.

[8] Anneli Dahlgren and Louise Svensson “Guidelines and Rules for Detailing of Reinforcement in Concrete Structures” Chalmers university of technology, Sweden 2013.

[9] Ke Du, Jingjiang Sun & Weixiao Xu Evaluation of Section and Fiber Integration Points in Fiber Model. Institute of Engineering Mechanics, China Earthquake Administration, Harbin 150080, China

[10] Vojko kilar and peter fajfar Simplified Push-Over Analysis of Building Structures. University of Ljubjana, Faculty of Civil Engineering and Geodesy, Nstttute of Structural and Earthquake Engineering Jamova 2 61000 Ljubljana, Slovania

[11] Silvia Mazzoni, F. M. (2007). OpenSees Command Language Manual. San Diego: UC.

[12] Brent Welch (January 13,1995) Practical Programing in Tcl and Tk.

[13] Berkeley, California, U.S.A., 2000, Open System for Earthquake Engineering Simulation (OpenSees).

[14] Armin Gharakhanloo (June 2014) Distributed and Concentrated Inelasticity Beam-Column Elements used in Earthquake Engineering. master's thesis at the Norwegian University of Science and Technology (NTNU).

[15] Apostolos K.Kanstantinidis " Earth Quake resistant buildings from reinforced concrete, Volume A the Art of construction and the Detailing."

[16] Gregory G. Deierlein et al. Nonlinear Structural Analysis for Seismic Design, NIST GCR 10-917-5 NEHRP Seismic Design Technical Brief No. 4

Appendices

A. OpenSees scripts

A.1 cantilever column with uniformly distributed longitudinal reinforcing bars

```
# units KN, m, sec;
# start of model generation
model BasicBuilder -ndm 2 -ndf 3   # create modelBuilder with 2 dimension and 3 DOF/node
set dataDir Data
# Create nodes; tag x y
node 1 0.0 0.0
node 2 0.0 3.0
# tag Dx Dy Rz
fix 1 1 1 1    # fix support at base of column
# MATERIAL parameters -------------------------------------------------------------
set IDconcCore 1; # material ID tag -- confined core concrete
set IDconcCover 2; # material ID tag -- unconfined cover concrete
set IDreinf 3; # material ID tag -- reinforcement
# nominal concrete compressive strength
set fc 30000; # CONCRETE Compressive Strength, (+Tension, -Compression)
set Ec 31000000; # Concrete Elastic Modulus;set GC [expr $Ec/2.4]
# confined concrete
set Kfc 1.3; # ratio of confined to unconfined concrete strength
set fc1C [expr $Kfc*$fc]; # CONFINED concrete (mander model), maximum stress
set eps1C [expr 2. *$fc1C/$Ec]; # strain at maximum stress
set fc2C [expr 0.2*$fc1C]; # ultimate stress
set eps2C [expr 5*$eps1C]; # strain at ultimate stress
# unconfined concrete
set fc1U $fc; # UNCONFINED concrete (todeschini parabolic model), maximum stress
set eps1U -0.0035; # strain at maximum strength of unconfined concrete
set fc2U [expr 0.2*$fc1U]; # ultimate stress
set eps2U -0.001; # strain at ultimate stress
set lambda 0.1; # ratio between unloading slope at $eps2 and initial slope $Ec
# tensile-strength properties
set ftC [expr -0.14*$fc1C]; # tensile strength +tension
```

```
set ftU [expr -0.14*$fc1U]; # tensile strength +tension
set Ets [expr $ftU/0.002]; # tension softening stiffness
set Fy 400000; # STEEL yield stress;
set Es 200000000; # modulus of steel
set b 0.01; # strain-hardening ratio
set R0 20; # control the transition from elastic to plastic branches
set cR1 0.925; # control the transition from elastic to plastic branches
set cR2 0.15; # control the transition from elastic to plastic branches
# build core concrete (confined)
uniaxialMaterial Concrete02 $IDconcCore $fc1C $eps1C $fc2C $eps2C $lambda $ftC $Ets;
# build cover concrete (unconfined)
uniaxialMaterial Concrete02 $IDconcCover $fc1U $eps1U $fc2U $eps2U $lambda $ftU $Ets;
# build reinforcement
uniaxialMaterial Steel02 $IDreinf $Fy $Es $b $R0 $cR1 $cR2;
# section GEOMETRY -------------------------------------------------------------
set HSec 0.5; # Column Depth
set BSec 0.5; # Column Width
set coverSec 0.025; # Column cover to reinforcing steel NA.
set numBarsSec1 6; # number of longitudinal-reinforcement bars in steel layer. (Symmetric
set numBarsSec2 4; # top & bot);
set barAreaSec [expr 0.25*3.14*0.000256];
# area of longitudinal-reinforcement bars(diam.=16mm)
set SecTag 1; # set tag for symmetric section
# FIBER SECTION properties -------------------------------------------------------------
# column section:
set coverY [expr $HSec/2.0]; # The distance from the section z-axis to the edge of the cover concrete -- outer edge of cover concrete
set coverZ [expr $BSec/2.0]; # The distance from the section y-axis to the edge of the cover concrete -- outer edge of cover concrete
set coreY [expr $coverY-$coverSec]; # The distance from the section z-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set coreZ [expr $coverZ-$coverSec]; # The distance from the section y-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set nfCoreY 30; # number of fibers for concrete in y-direction -- core concrete
set nfCoreZ 30; # number of fibers for concrete in z-direction
set nfCoverY 30; # number of fibers for concrete in y-direction -- cover concrete
```

```
set nfCoverZ 30; # number of fibers for concrete in z-direction
set nfCoverY1 1; # number of fibers for concrete in y-direction
set nfCoverZ1 1; # number of fibers for concrete in z-direction
set Cd1 0.016; # distance from center of bars to outer edge of core
set Cd2 0.0836; #clear distance between bars
# Define the fiber section (define the core patch, four cover patchs and Reinforcement layer respectively)
section Fiber $SecTag {
patch quadr $IDconcCore $nfCoreZ $nfCoreY -$coreY $coreZ -$coreY -$coreZ $coreY -$coreZ $coreY $coreZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 $coverY -$coverZ -$coreY $coreZ $coreY $coreZ $coverY $coverZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 -$coreY -$coreZ -$coverY -$coverZ $coverY -$coverZ $coreY -$coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 -$coverY $coverZ -$coverY -$coverZ -$coreY -$coreZ -$coreY $coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 $coreY $coreZ $coreY -$coreZ $coverY -$coverZ $coverY $coverZ
layer straight $IDreinf $numBarsSec1 $barAreaSec [expr $coreY-$Cd1] [expr $coreZ-$Cd1] [expr $coreY-$Cd1] [expr -$coreZ+$Cd1]
layer straight $IDreinf $numBarsSec1 $barAreaSec [expr -$coreY+$Cd1] [expr $coreZ-$Cd1] [expr -$coreY+$Cd1] [expr -$coreZ+$Cd1]
layer straight $IDreinf $numBarsSec2 $barAreaSec [expr $coreY-$Cd1-$Cd2] [expr $coreZ-$Cd1] [expr -$coreY+$Cd1+$Cd2] [expr $coreZ-$Cd1]
layer straight $IDreinf $numBarsSec2 $barAreaSec [expr $coreY-$Cd1-$Cd2] [expr -$coreZ+$Cd1] [expr -$coreY+$Cd1+$Cd2] [expr -$coreZ+$Cd1]
}
# number of column integration points (sections)
# ID tag for column transformation, defining element normal
set IDcolTransf 1
#geomTransf Linear $transfTag $vecxzX $vecxzY $vecxzZ <-jntOffset $dXi $dYi $dZi $dXj $dYj $dZj>
geomTransf Linear $IDcolTransf
set colSec 1
set np 5
# Create the non-linear column elements' connectivity
# element dispBeamColumn $eleTag $iNode $jNode $np $secTag $transfTag
```

```
element dispBeamColumn 1 1 2 $np $colSec 1
 # Set axial load
 pattern Plain 1 Constant {
 load 2 0.0 -100 0.0
}
  initialize
 integrator LoadControl 0.1;
 system BandGeneral
 test NormDispIncr 1.0e-5 1000 5
 numberer Plain
 constraints Plain
 algorithm ModifiedNewton -initial
 analysis Static
 # perform the gravity load analysis,
 analyze [expr 1]
 # Set the gravity loads to be constant & reset the time in the domain
 loadConst -time 0.0
 remove recorders
 # set lateral load
 pattern Plain 2 Linear {
load 2 5 0.0 0;
}
 system BandGeneral
 constraints Transformation
 numberer Plain
 # Create a recorder to monitor nodal displacement and element forces
recorder Node -file nodeTop.out -node 2 -dof 1 disp
 recorder Node -file nodeDisp.out -time -node 2 -dof 1 2 3 disp
 recorder Node -file Rxnnode.out -time -node 1 -dof 1 2 3 reaction
 recorder Element -file ele1global.out -time -ele 1 globalForce
 recorder Element -file el1local.out -time -ele 1 localForce
 recorder Element -file elesX.out -time -ele 1 globalForce
 recorder display node.out "Nodal Displacement" 10 10 400 400 -columns 1 2
 # recorder for element1 section1 steel stress/strain and section force-def.
 recorder Element -file ele1sec1Force.out -time -ele 1 section 1 force
 recorder Element -file ele1sec1Defo.out -time -ele 1 section 1 deformation
```

```
recorder Element -file ele1sec1Stiff.out -time -ele 1 section 1 stiffness
#recorder Element -file ele1sec1StressStrain.out –time -ele 1 section 1 fiber $y $z<$matID> stressStrain
recorder Element -file ele1sec1StressStrain3.out -time -ele 1 section 1 fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] 3 stressStrain
recorder Element -file ele1sec1StressStrain2.out -time -ele 1 section 1 fiber 200 200 2 stressStrain
recorder Element -file ele1sec1StressStrain1.out -time -ele 1 section 1 fiber 0 0 1 stressStrain
recorder Element -file Sect_ForceandDeformation.out -ele 1 section 1 forceAndDeformation
recorder plot ele1sec1StressStrain.out s-e 0.20 0.20 400 400 -columns 1 2
test NormDispIncr 1.0e-5 1000
algorithm ModifiedNewton -initial
analysis Static
set maxU 0.1; # Max displacement
set ok 0
set currentDisp 0.0; # perform the analysis;
while {$ok == 0 && $currentDisp < $maxU} {set ok [analyze 1]}; # if the analysis fails try initial tangent iteration;
if {$ok! = 0} {puts "regular newton failed ... let’s try an initail stiffness for this step";
test NormDispIncr 1.0e-5 1000; algorithm KrylovNewton -initial; set ok [analyze 1]; if {$ok ==0} {puts "that worked ... back to regular newton"}; test NormDispIncr 1.0e-12 1000; algorithm Newton}
if {$ok == 0} {puts "Pushover analysis completed SUCCESSFULLY";} else {puts "Pushover analysis FAILED";}
```

A.2 cantilever column with three bar bundles at corner and two bar bundles at center

```
# units KN, m, sec;
# start of model generation
# create modelBuilder with 2 dimension and 3 DOF/node
model BasicBuilder -ndm 2 -ndf 3
set dataDir Data
# Create nodes; tag x y
node 1 0.0 0.0
node 2 0.0 3.0
# fix support at base of column; tag Dx Dy Rz
fix 1 1 1 1
# MATERIAL parameters -------------------------------------------------------------------
set IDconcCore 1; # material ID tag -- confined core concrete
set IDconcCover 2; # material ID tag -- unconfined cover concrete
set IDreinf 3; # material ID tag -- reinforcement
# nominal concrete compressive strength
set fc 30000; # CONCRETE Compressive Strength, (+Tension, -Compression)
set Ec 31000000; # Concrete Elastic Modulus;set GC [expr $Ec/2.4]
# confined concrete
set Kfc 1.3; # ratio of confined to unconfined concrete strength
set fc1C [expr $Kfc*$fc]; # CONFINED concrete (mander model), maximum stress
set eps1C [expr 2. *$fc1C/$Ec]; # strain at maximum stress
set fc2C [expr 0.2*$fc1C]; # ultimate stress
set eps2C [expr 5*$eps1C]; # strain at ultimate stress
# unconfined concrete
set fc1U $fc; # UNCONFINED concrete (todeschini parabolic model), maximum stress
set eps1U -0.0035; # strain at maximum strength of unconfined concrete
set fc2U [expr 0.2*$fc1U]; # ultimate stress
set eps2U -0.001; # strain at ultimate stress
set lambda 0.1; # ratio between unloading slope at $eps2 and initial slope $Ec
# tensile-strength properties
set ftC [expr -0.14*$fc1C]; # tensile strength +tension
set ftU [expr -0.14*$fc1U]; # tensile strength +tension
```

```
set Ets [expr $ftU/0.002]; # tension softening stiffness
set Fy 400000; # STEEL yield stress;
set Es 200000000; # modulus of steel
set b 0.01; # strain-hardening ratio
set R0 20; # control the transition from elastic to plastic branches
set cR1 0.925; # control the transition from elastic to plastic branches
set cR2 0.15; # control the transition from elastic to plastic branches
# build core concrete (confined)
uniaxialMaterial Concrete02 $IDconcCore $fc1C $eps1C $fc2C $eps2C $lambda $ftC $Ets;
# build cover concrete (unconfined)
uniaxialMaterial Concrete02 $IDconcCover $fc1U $eps1U $fc2U $eps2U $lambda $ftU $Ets;
# build reinforcement
uniaxialMaterial Steel02 $IDreinf $Fy $Es $b $R0 $cR1 $cR2;
# section GEOMETRY -------------------------------------------------------------
set HSec 0.5; # Column Depth
set BSec 0.5; # Column Width
set coverSec 0.025; # Column cover to reinforcing steel NA.
set numBarsSec1 2; # number of longitudinal-reinforcement bars in steel layer. (Symmetric top & bot);
set barAreaSec [expr 0.25*3.14*0.000256];
# area of longitudinal-reinforcement bars(diam.=16mm)
set SecTag 1; # set tag for symmetric section
# FIBER SECTION properties -------------------------------------------------------------
# column section:
set coverY [expr $HSec/2.0]; # The distance from the section z-axis to the edge of the cover concrete -- outer edge of cover concrete
set coverZ [expr $BSec/2.0]; # The distance from the section y-axis to the edge of the cover concrete -- outer edge of cover concrete
set coreY [expr $coverY-$coverSec]; # The distance from the section z-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set coreZ [expr $coverZ-$coverSec]; # The distance from the section y-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set nfCoreY 30; # number of fibers for concrete in y-direction -- core concrete
set nfCoreZ 30; # number of fibers for concrete in z-direction
set nfCoverY 30; # number of fibers for concrete in y-direction -- cover concrete
set nfCoverZ 30; # number of fibers for concrete in z-direction
set nfCoverY1 1; # number of fibers for concrete in y-direction
```

```
set nfCoverZ1 1; # number of fibers for concrete in z-direction
set Cd1 0.016; # distance from center of bar to outer edge of core for both axis
set Cd2 0.201; # distance from center of bar to center of core
# Define the fiber section (define the core patch, four cover patchs and Reinforcement layer respectively)
section Fiber $SecTag {
patch quadr $IDconcCore $nfCoreZ $nfCoreY -$coreY $coreZ -$coreY -$coreZ $coreY -$coreZ
$coreY $coreZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 $coverY -$coverZ -$coreY $coreZ $coreY $coreZ
$coverY $coverZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 -$coreY -$coreZ -$coverY -$coverZ $coverY -
$coverZ $coreY -$coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 -$coverY $coverZ -$coverY -$coverZ -$coreY -
$coreZ -$coreY $coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 $coreY $coreZ $coreY -$coreZ $coverY -$coverZ
$coverY $coverZ
fiber [expr $coreY-$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr -$coreZ+2*$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr $coreZ-$Cd2-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr -$coreZ+$Cd2+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-$Cd2-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+$Cd2+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd2-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd2+$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd2+$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
}
```

```
# number of column integration points (sections)
# ID tag for column transformation, defining element normal
set IDcolTransf 1
#geomTransf Linear $transfTag $vecxzX $vecxzY $vecxzZ <-jntOffset $dXi $dYi $dZi $dXj $dYj
$dZj>
geomTransf Linear $IDcolTransf
set colSec 1
set np 5
# Create the non-linear column elements' connectivity
# element dispBeamColumn $eleTag $iNode $jNode $np $secTag $transfTag
element dispBeamColumn 1 1 2 $np $colSec 1
 # Set axial load
 pattern Plain 1 Constant {
 load 2 0.0 -100 0.0
}
  initialize
 integrator LoadControl 0.1
 system BandGeneral
 test NormDispIncr 1.0e-5 1000 5
 numberer Plain
 constraints Plain
 algorithm ModifiedNewton -initial
 analysis Static
 # perform the gravity load analysis,
 analyze [expr 1]
 # Set the gravity loads to be constant & reset the time in the domain
 loadConst -time 0.0
 remove recorders
 # set lateral load
 pattern Plain 2 Linear {
load 2 5 0.0 0;
}
 system BandGeneral
 constraints Transformation
 numberer Plain
# Create a recorder to monitor nodal displacement and element forces
```

```
recorder Node -file nodeTop.out -node 2 -dof 1 disp
 recorder Node -file nodeDisp.out -time -node 2 -dof 1 2 3 disp
 recorder Node -file Rxnnode.out -time -node 1 -dof 1 2 3 reaction
 recorder Element -file ele1global.out -time -ele 1 globalForce
 recorder Element -file el1local.out -time -ele 1 localForce
 recorder Element -file elesX.out -time -ele 1 globalForce
 recorder display node.out "Nodal Displacement" 10 10 400 400 -columns 1 2
 # recorder for element1 section1 steel stress/strain and section force-def.
 recorder Element -file ele1sec1Force.out -time -ele 1 section 1 force
 recorder Element -file ele1sec1Defo.out -time -ele 1 section 1 deformation
 recorder Element -file ele1sec1Stiff.out -time -ele 1 section 1 stiffness
 #recorder Element -file ele1sec1StressStrain.out –time -ele 1 section 1 fiber $y $z<$matID> stressStrain
 recorder Element -file ele1sec1StressStrain3.out -time -ele 1 section 1 fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] 3 stressStrain
recorder Element -file ele1sec1StressStrain2.out -time -ele 1 section 1 fiber 200 200 2 stressStrain
recorder Element -file ele1sec1StressStrain1.out -time -ele 1 section 1 fiber 0 0 1 stressStrain
recorder Element -file Sect_ForceandDeformation.out -ele 1 section 1 forceAndDeformation
 recorder plot ele1sec1StressStrain.out s-e 0.20 0.20 400 400 -columns 1 2
DisplayModel2D deformedShape 5
recorder display DisplayModel2D 10 10 500 500
 test NormDispIncr 1.0e-5 1000
 algorithm ModifiedNewton -initial
 analysis Static
set maxU 0.1; # Max displacement
set ok 0
set currentDisp 0.0; # perform the analysis;
while {$ok == 0 && $currentDisp < $maxU} {set ok [analyze 1]}; # if the analysis fails try initial tangent iteration;
if {$ok! = 0} {puts "regular newton failed ... let’s try an initail stiffness for this step";
test NormDispIncr 1.0e-5 1000; algorithm KrylovNewton -initial; set ok [analyze 1]; if {$ok ==0} {puts "that worked ... back to regular newton"}; test NormDispIncr 1.0e-12 1000; algorithm Newton}
if {$ok == 0} {puts "Pushover analysis completed SUCCESSFULLY";} else {puts "Pushover analysis FAILED";}
```

A.3 cantilever column with four bar bundles at corner and one bar at center of faces

```
# units; KN, m, sec;
# start of model generation
# create modelBuilder with 2 dimension and 3 DOF/node
model BasicBuilder -ndm 2 -ndf 3
set dataDir Data
# Create nodes; tag x y
node 1 0.0 0.0
node 2 0.0 3.0
# fix support at base of column; tag Dx Dy Rz
fix 1 1 1 1
# MATERIAL parameters -------------------------------------------------------------------
set IDconcCore 1; # material ID tag -- confined core concrete
set IDconcCover 2; # material ID tag -- unconfined cover concrete
set IDreinf 3; # material ID tag -- reinforcement
# nominal concrete compressive strength
set fc 30000; # CONCRETE Compressive Strength, (+Tension, -Compression)
set Ec 31000000; # Concrete Elastic Modulus;set GC [expr $Ec/2.4]
# confined concrete
set Kfc 1.3; # ratio of confined to unconfined concrete strength
set fc1C [expr $Kfc*$fc]; # CONFINED concrete (mander model), maximum stress
set eps1C [expr 2. *$fc1C/$Ec]; # strain at maximum stress
set fc2C [expr 0.2*$fc1C]; # ultimate stress
set eps2C [expr 5*$eps1C]; # strain at ultimate stress
# unconfined concrete
set fc1U $fc; # UNCONFINED concrete (todeschini parabolic model), maximum stress
set eps1U -0.0035; # strain at maximum strength of unconfined concrete
set fc2U [expr 0.2*$fc1U]; # ultimate stress
set eps2U -0.001; # strain at ultimate stress
set lambda 0.1; # ratio between unloading slope at $eps2 and initial slope $Ec
# tensile-strength properties
set ftC [expr -0.14*$fc1C]; # tensile strength +tension
set ftU [expr -0.14*$fc1U]; # tensile strength +tension
```

set Ets [expr $ftU/0.002]; # tension softening stiffness
set Fy 400000; # STEEL yield stress;
set Es 200000000; # modulus of steel
set b 0.01; # strain-hardening ratio
set R0 20; # control the transition from elastic to plastic branches
set cR1 0.925; # control the transition from elastic to plastic branches
set cR2 0.15; # control the transition from elastic to plastic branches
build core concrete (confined)
uniaxialMaterial Concrete02 $IDconcCore $fc1C $eps1C $fc2C $eps2C $lambda $ftC $Ets; # build cover concrete (unconfined)
uniaxialMaterial Concrete02 $IDconcCover $fc1U $eps1U $fc2U $eps2U $lambda $ftU $Ets; # build reinforcement
uniaxialMaterial Steel02 $IDreinf $Fy $Es $b $R0 $cR1 $cR2;
section GEOMETRY ---
set HSec 0.5; # Column Depth
set BSec 0.5; # Column Width
set coverSec 0.025; # Column cover to reinforcing steel NA.
set numBarsSec1 2; # number of longitudinal-reinforcement bars in steel layer. (Symmetric top & bot);
set barAreaSec [expr 0.25*3.14*0.000256];
area of longitudinal-reinforcement bars(diam.=16mm)
set SecTag 1; # set tag for symmetric section
FIBER SECTION properties ---
column section:
set coverY [expr $HSec/2.0]; # The distance from the section z-axis to the edge of the cover concrete -- outer edge of cover concrete
set coverZ [expr $BSec/2.0]; # The distance from the section y-axis to the edge of the cover concrete -- outer edge of cover concrete
set coreY [expr $coverY-$coverSec]; # The distance from the section z-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set coreZ [expr $coverZ-$coverSec]; # The distance from the section y-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set nfCoreY 30; # number of fibers for concrete in y-direction -- core concrete
set nfCoreZ 30; # number of fibers for concrete in z-direction
set nfCoverY 30; # number of fibers for concrete in y-direction -- cover concrete
set nfCoverZ 30; # number of fibers for concrete in z-direction
set nfCoverY1 1; # number of fibers for concrete in y-direction

```
set nfCoverZ1 1; # number of fibers for concrete in z-direction
set Cd1 0.016; # distance from center of bar to outer edge of core for both axis
set Cd2 0.209; # distance from center of bar to center of core
# Define the fiber section (define the core patch, four cover patchs and Reinforcement layer respectively)
section Fiber $SecTag {
patch quadr $IDconcCore $nfCoreZ $nfCoreY -$coreY $coreZ -$coreY -$coreZ $coreY -$coreZ $coreY $coreZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 $coverY -$coverZ -$coreY $coreZ $coreY $coreZ $coverY $coverZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 -$coreY -$coreZ -$coverY -$coverZ $coverY -$coverZ $coreY -$coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 -$coverY $coverZ -$coverY -$coverZ -$coreY -$coreZ -$coreY $coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 $coreY $coreZ $coreY -$coreZ $coverY -$coverZ $coverY $coverZ
fiber [expr $coreY-$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr -2*$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-2*$Cd1] [expr -$coreZ+2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr -$coreZ+2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+2*$Cd1] [expr $coreZ-2*$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-0.201-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec $IDreinf
fiber [expr $coreY-$Cd1] [expr $coreZ-0.201-$Cd1] $barAreaSec $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-0.201-$Cd1] $barAreaSec $IDreinf
}
```

```
# number of column integration points (sections)
# ID tag for column transformation, defining element normal
set IDcolTransf 1
#geomTransf Linear $transfTag $vecxzX $vecxzY $vecxzZ <-jntOffset $dXi $dYi $dZi $dXj $dYj
$dZj>
geomTransf Linear $IDcolTransf
set colSec 1
set np 5
# Create the non-linear column elements' connectivity
# element dispBeamColumn $eleTag $iNode $jNode $np $secTag $transfTag
element dispBeamColumn 1 1 2 $np $colSec 1
# Set axial load
 pattern Plain 1 Constant {
 load 2 0.0 -100 0.0
}
  initialize
 integrator LoadControl 0.1
 system BandGeneral
 test NormDispIncr 1.0e-5 1000 5
 numberer Plain
 constraints Plain
 algorithm ModifiedNewton -initial
 analysis Static
 # perform the gravity load analysis,
 analyze [expr 1]
 # Set the gravity loads to be constant & reset the time in the domain
 loadConst -time 0.0
 remove recorders
 # set lateral load
 pattern Plain 2 Linear {
load 2 5 0.0 0;
}
 system BandGeneral
 constraints Transformation
 numberer Plain
# Create a recorder to monitor nodal displacement and element forces
```

```
recorder Node -file nodeTop.out -node 2 -dof 1 disp
 recorder Node -file nodeDisp.out -time -node 2 -dof 1 2 3 disp
 recorder Node -file Rxnnode.out -time -node 1 -dof 1 2 3 reaction
 recorder Element -file ele1global.out -time -ele 1 globalForce
 recorder Element -file el1local.out -time -ele 1 localForce
 recorder Element -file elesX.out -time -ele 1 globalForce
 recorder display node.out "Nodal Displacement" 10 10 400 400 -columns 1 2
 # recorder for element1 section1 steel stress/strain and section force-def.
 recorder Element -file ele1sec1Force.out -time -ele 1 section 1 force
 recorder Element -file ele1sec1Defo.out -time -ele 1 section 1 deformation
 recorder Element -file ele1sec1Stiff.out -time -ele 1 section 1 stiffness
 #recorder Element -file ele1sec1StressStrain.out –time -ele 1 section 1 fiber $y $z<$matID> stressStrain
 recorder Element -file ele1sec1StressStrain3.out -time -ele 1 section 1 fiber [expr $coreY-$Cd2-$Cd1]
[expr $coreZ-$Cd1] 3 stressStrain
recorder Element -file ele1sec1StressStrain2.out -time -ele 1 section 1 fiber 200 200 2 stressStrain
recorder Element -file ele1sec1StressStrain1.out -time -ele 1 section 1 fiber 0 0 1 stressStrain
recorder Element -file Sect_ForceandDeformation.out -ele 1 section 1 forceAndDeformation
recorder plot ele1sec1StressStrain.out s-e 0.20 0.20 400 400 -columns 1 2
test NormDispIncr 1.0e-5 1000
algorithm ModifiedNewton -initial
analysis Static
set maxU 0.1; # Max displacement
set ok 0
set currentDisp 0.0; # perform the analysis;
while {$ok == 0 && $currentDisp < $maxU} {set ok [analyze 1]}; # if the analysis fails try initial
tangent iteration;
if {$ok! = 0} {puts "regular newton failed ... let’s try an initail stiffness for this step";
test NormDispIncr 1.0e-5 1000; algorithm KrylovNewton -initial; set ok [analyze 1]; if {$ok ==0} {puts
"that worked ... back to regular newton"}; test NormDispIncr 1.0e-12 1000; algorithm Newton}
if {$ok == 0} {puts "Pushover analysis completed SUCCESSFULLY";} else {puts "Pushover analysis
FAILED";}
```

A.4 cantilever column with equivalent bar

```
# units; KN, m, sec;
# start of model generation
model BasicBuilder -ndm 2 -ndf 3 # create modelBuilder with 2 dimension and 3 DOF/node
set dataDir Data
# Create nodes; tag x y
node 1 0.0 0.0
node 2 0.0 3.0
# fix support at base of column; tag Dx Dy Rz
fix 1 1 1 1
# MATERIAL parameters ------------------------------------------------------------------
set IDconcCore 1; # material ID tag -- confined core concrete
set IDconcCover 2; # material ID tag -- unconfined cover concrete
set IDreinf 3; # material ID tag -- reinforcement
# nominal concrete compressive strength
set fc 30000; # CONCRETE Compressive Strength, (+Tension, -Compression)
set Ec 31000000; # Concrete Elastic Modulus;set GC [expr $Ec/2.4]
# confined concrete
set Kfc 1.3; # ratio of confined to unconfined concrete strength
set fc1C [expr $Kfc*$fc]; # CONFINED concrete (mander model), maximum stress
set eps1C [expr 2. *$fc1C/$Ec]; # strain at maximum stress
set fc2C [expr 0.2*$fc1C]; # ultimate stress
set eps2C [expr 5*$eps1C]; # strain at ultimate stress
# unconfined concrete
set fc1U $fc; # UNCONFINED concrete (todeschini parabolic model), maximum stress
set eps1U -0.0035; # strain at maximum strength of unconfined concrete
set fc2U [expr 0.2*$fc1U]; # ultimate stress
set eps2U -0.001; # strain at ultimate stress
set lambda 0.1; # ratio between unloading slope at $eps2 and initial slope $Ec
# tensile-strength properties
set ftC [expr -0.14*$fc1C]; # tensile strength +tension
set ftU [expr -0.14*$fc1U]; # tensile strength +tension
set Ets [expr $ftU/0.002]; # tension softening stiffness
```

```
set Fy 400000; # STEEL yield stress;
set Es 200000000; # modulus of steel
set b 0.01; # strain-hardening ratio
set R0 20; # control the transition from elastic to plastic branches
set cR1 0.925; # control the transition from elastic to plastic branches
set cR2 0.15; # control the transition from elastic to plastic branches
# build core concrete (confined)
uniaxialMaterial Concrete02 $IDconcCore $fc1C $eps1C $fc2C $eps2C $lambda $ftC $Ets;
# build cover concrete (unconfined)
uniaxialMaterial Concrete02 $IDconcCover $fc1U $eps1U $fc2U $eps2U $lambda $ftU $Ets;
# build reinforcement
uniaxialMaterial Steel02 $IDreinf $Fy $Es $b $R0 $cR1 $cR2;
# section GEOMETRY -------------------------------------------------------------
set HSec 0.5; # Column Depth
set BSec 0.5; # Column Width
set coverSec 0.025; # Column cover to reinforcing steel NA.
set numBarsSec1 2; # number of longitudinal-reinforcement bars in steel layer. (Symmetric top & bot);
set barAreaSec1 [expr 0.25*3.14*0.001024]; # area of longitudinal-reinforcement bars(diam.=32mm)
set barAreaSec2 [expr 0.25*3.14*0.000256]; # area of longitudinal-reinforcement bars(diam.=16mm)
set SecTag 1; # set tag for symmetric section
# FIBER SECTION properties -------------------------------------------------------------
# column section:
set coverY [expr $HSec/2.0]; # The distance from the section z-axis to the edge of the cover concrete -- outer edge of cover concrete
set coverZ [expr $BSec/2.0]; # The distance from the section y-axis to the edge of the cover concrete -- outer edge of cover concrete
set coreY [expr $coverY-$coverSec]; # The distance from the section z-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set coreZ [expr $coverZ-$coverSec]; # The distance from the section y-axis to the edge of the core concrete -- edge of the core concrete/inner edge of cover concrete
set nfCoreY 30; # number of fibers for concrete in y-direction -- core concrete
set nfCoreZ 30; # number of fibers for concrete in z-direction
set nfCoverY 30; # number of fibers for concrete in y-direction -- cover concrete
set nfCoverZ 30; # number of fibers for concrete in z-direction
set nfCoverY1 1; # number of fibers for concrete in y-direction
set nfCoverZ1 1; # number of fibers for concrete in z-direction
```

```
set Cd1 0.024; # distance from center of bar to outer edge of core for both axis
set Cd2 0.193; # distance from center of bar to center of core
# Define the fiber section (define the core patch, four cover patchs and Reinforcement layer respectively)
section Fiber $SecTag {
patch quadr $IDconcCore $nfCoreZ $nfCoreY -$coreY $coreZ -$coreY -$coreZ $coreY -$coreZ $coreY $coreZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 $coverY -$coverZ -$coreY $coreZ $coreY $coreZ $coverY $coverZ
patch quadr $IDconcCover $nfCoverY $nfCoverZ1 -$coreY -$coreZ -$coverY -$coverZ $coverY -$coverZ $coreY -$coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 -$coverY $coverZ -$coverY -$coverZ -$coreY -$coreZ -$coreY $coreZ
patch quadr $IDconcCover $nfCoverZ $nfCoverY1 $coreY $coreZ $coreY -$coreZ $coverY -$coverZ $coverY $coverZ
layer straight $IDreinf $numBarsSec1 $barAreaSec1 $coreY $coreZ $coreY -$coreZ
layer straight $IDreinf $numBarsSec1 $barAreaSec1 -$coreY $coreZ -$coreY -$coreZ
fiber [expr $coreY-$Cd1] [expr $coreZ-$Cd1] $barAreaSec1 $IDreinf
fiber [expr $coreY-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec1 $IDreinf
fiber [expr -$coreY+$Cd1] [expr $coreZ-$Cd1] $barAreaSec1 $IDreinf
fiber [expr -$coreY+$Cd1] [expr -$coreZ+$Cd1] $barAreaSec1 $IDreinf
fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] $barAreaSec2 $IDreinf
fiber [expr $coreY-$Cd2-$Cd1] [expr -$coreZ+$Cd1] $barAreaSec2 $IDreinf
fiber [expr $coreY-$Cd1] [expr $coreZ-$Cd2-$Cd1] $barAreaSec2 $IDreinf
fiber [expr -$coreY+$Cd2] [expr $coreZ-$Cd2-$Cd1] $barAreaSec2 $IDreinf
}
# number of column integration points (sections)
# ID tag for column transformation, defining element normal
set IDcolTransf 1
#geomTransf Linear $transfTag $vecxzX $vecxzY $vecxzZ <-jntOffset $dXi $dYi $dZi $dXj $dYj $dZj>
geomTransf Linear $IDcolTransf
set colSec 1
set np 5
# Create the non-linear column elements' connectivity
# element dispBeamColumn $eleTag $iNode $jNode $np $secTag $transfTag
element dispBeamColumn 1 1 2 $np $colSec 1
```

```
# Set axial load
 pattern Plain 1 Constant {
 load 2 0.0 -100 0.0
}
  initialize
 integrator LoadControl 0.1
 system BandGeneral
 test NormDispIncr 1.0e-5 1000 5
 numberer Plain
 constraints Plain
 algorithm ModifiedNewton -initial
 analysis Static
 # perform the gravity load analysis,
 analyze [expr 1]
 # Set the gravity loads to be constant & reset the time in the domain
 loadConst -time 0.0
 remove recorders
 # set lateral load
 pattern Plain 2 Linear {
load 2 5 0.0 0;
}
 system BandGeneral
 constraints Transformation
 numberer Plain
# Create a recorder to monitor nodal displacement and element forces
recorder Node -file nodeTop.out -node 2 -dof 1 disp
 recorder Node -file nodeDisp.out -time -node 2 -dof 1 2 3 disp
 recorder Node -file Rxnnode.out -time -node 1 -dof 1 2 3 reaction
 recorder Element -file ele1global.out -time -ele 1 globalForce
 recorder Element -file el1local.out -time -ele 1 localForce
 recorder Element -file elesX.out -time -ele 1 globalForce
 recorder display node.out "Nodal Displacement" 10 10 400 400 -columns 1 2
 # recorder for element1 section1 steel stress/strain and section force-def.
 recorder Element -file ele1sec1Force.out -time -ele 1 section 1 force
 recorder Element -file ele1sec1Defo.out -time -ele 1 section 1 deformation
 recorder Element -file ele1sec1Stiff.out -time -ele 1 section 1 stiffness
```

```
#recorder Element -file ele1sec1StressStrain.out –time -ele 1 section 1 fiber $y $z<$matID> stressStrain
recorder Element -file ele1sec1StressStrain3.out -time -ele 1 section 1 fiber [expr $coreY-$Cd2-$Cd1] [expr $coreZ-$Cd1] 3 stressStrain
recorder Element -file ele1sec1StressStrain2.out -time -ele 1 section 1 fiber 200 200 2 stressStrain
recorder Element -file ele1sec1StressStrain1.out -time -ele 1 section 1 fiber 0 0 1 stressStrain
recorder Element -file Sect_ForceandDeformation.out -ele 1 section 1 forceAndDeformation
recorder plot ele1sec1StressStrain.out s-e 0.20 0.20 400 400 -columns 1 2
test NormDispIncr 1.0e-5 1000
algorithm ModifiedNewton -initial
analysis Static
set maxU 0.1; # Max displacement
set ok 0
set currentDisp 0.0; # perform the analysis;
while {$ok == 0 && $currentDisp < $maxU} {set ok [analyze 1]}; # if the analysis fails try initial tangent iteration;
if {$ok! = 0} {puts "regular newton failed ... let’s try an initail stiffness for this step";
test NormDispIncr 1.0e-5 1000; algorithm KrylovNewton -initial; set ok [analyze 1]; if {$ok ==0} {puts "that worked ... back to regular newton"}; test NormDispIncr 1.0e-12 1000; algorithm Newton}
if {$ok == 0} {puts "Pushover analysis completed SUCCESSFULLY";} else {puts "Pushover analysis FAILED";}
```