FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE COLUMN FORCED DISPLACEMENT BEHAVIOUR

BY

MD. FORHAD AHMED BAPPY

&

MD. JAHID IQBAL BIN MOSTOFA

DEPARTMENT OF CIVIL ENGINEERING
DAFFODIL INTERNATIONAL UNIVERSITY

Course Title & Code: Thesis & CE-400

THE NAME OF THE THESIS:

FINITE ELEMENET ANALYSIS OF REINFORCED CONCRETE COLUMN FORCED DISPLACEMENT BEHAVIOUR.

A Thesis submitted to Department of Civil Engineering in partial fulfilment of the requirements for the Award of Degree of **Bachelors of Science in Civil Engineering.**

Submitted by:

Forhad Ahmed Bappy	ID No: 162-47-143
Md. Jahid Iqbal Bin Mostofa	ID No: 162-47-150

Supervisor:

Mr. Rayhan Md. Faysal

Assistant Professor

Department of Civil Engineering

Daffodil International University

Date: February, 2021

APPROVAL

The thesis title “Finite Element Analysis of reinforced concrete column forced displacement behaviour” submitted by Forhad Ahmed Bappy (ID: 162-47-143) & Jahid Iqbal Bin Mostofa (ID: 162-47-150) to the Department of Civil Engineering, Daffodil International University's work has been accepted as satisfactory for partial fulfilment of the requirements for the degree of B.Sc. in Civil Engineering, and its style and content presentation have been approved.

DECLARATION

It is stated that the work "Finite Element Analysis of reinforced concrete column forced displacement behaviour" reported in this thesis has been performed under the supervision of Mr. Rayhan Md. Faysal Assistant Professor, Department of Civil Engineering, Daffodil International University, Dhaka. The thesis contains no material previously published or written by another person, to the best of my knowledge and belief, except where due reference is made in the thesis itself.

Mr. Rayhan Md. Faysal

Assistant Professor

Department of Civil Engineering

Daffodil International University

Md. Forhad Ahmed Bappy
ID: 162-47-143

Md. Jahid Iqbal Bin Mostofa
ID: 162-47-150

ACKNOWLEDGEMENT

It gives us great pleasure to express our heartfelt gratitude to everyone who assisted us throughout the training and until the completion of this report.

We would like to express our gratitude to Allah, first and foremost, for bringing us here and giving us good health. Secondly, for all their support, love and prayers, we would like to thank our family.

It would not have been possible to write that thesis without the help and support of our supervisor, Mr. Rayhan Md. Faysal, Lecturer, Department of Civil Engineering, Daffodil International University, Dhaka, Bangladesh, for his valuable continuous guidance, helpful suggestions and constant encouragement for this going of the thesis.

Dr. Miah M. Hussainuzzaman, Associate Professor and Head, Department of Civil Engineering, Daffodil International University, Dhaka, is offered a very special debt of profound gratitude for his continuous encouragement and fellowship during this research.

At the same time, we recognize Daffodil International University and its staff for their academic and technical support. Thank you, because it has been such a good guide and gives us the opportunity to perform here. This report would be a reality, with their accompaniment. Once again, we are grateful for all of their help in successfully completing our report

DEDICATION

This thesis is dedicated to all of our parents and supervisor (FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE COLUMN FORCED DISPLACEMENT BEHAVIOUR).

ABSTRACT

The Finite Element Method (FEM) is a procedure for the numerical solution of the equations that govern the problems found in nature. Usually, the behaviour of nature can be described by equations expressed in differential or integral form. For this reason, the FEM is understood in mathematical circles as a numerical technique for solving partial differential or integral equations. The finite element method is a widely used method for numerically solving differential equations arising in engineering and mathematical modelling. Analysis of structures the FEM is a powerful method for computing the displacements, stresses and strains in a structure under a set of loads. We have done a column analysis using the finite element method. In the column, we have made 2 materials choices. The column stirrups are given. Here changed concrete compressive behaviour. We used axial load uniformly, this load is added to the free end section. And also used lateral load. We have found that using the finite element method adds force to the column and changes the properties of the concrete to control the displacement by placing the displacement at a certain distance.

Table of Contents

List of Figure

List of Table

CHAPTER-01

INTRODUCTION

1.1 GENERAL

The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained.

FEA utilizes numerical models to understand and measure the effects on a section or gathering of true conditions. These re-enactments, which are conducted by means of specific programming, allow experts to identify expected problems in a plan, including pressure zones and shaky areas.

A segment is the primary vertical part with or with-out minutes that supports compressive loads of the hub. Segments maintain vertical loads from the floors and rooftops and communicate these loads to the institutions. A segment's heap conveying limit relies on the materials used in its growth, its length, the state of its cross-area, and the limitations applied to its closures. The reformist breakdown of the outbound floors and a definitive all-out breakdown of the entire construction can be caused by the disappointment of a segment in a basic area.

Concrete is a heterogeneous substance brimming with micro-cracks even before adding a certain heap. After stacking, micro-cracks form, and the limitation of such micro-cracks into a major break leads to inevitable disappointment. It is becoming increasingly clear that such behaviour should be based on crack mechanics principles in numerical demonstrating. In a house, it is always necessary to construct RC segments, either because of helpless development, or because higher burdens are placed on the framework than those anticipated in the initial plan, or because of events such as pivotal burdens.

1.2 SIGNIFICANCE

The planner Imhotep used stone sections in ancient Egyptian design as early as 2600 BC, whose surface was cut to mirror the natural type of packaged reeds, similar to papyrus, lotus and palm. Also, facet chambers were normal in later Egyptian engineering. Their structure is thought to have originated in ancient reed-built sanctuaries. The segments were carved out of stone and embellished with cut and painted symbolic representations, messages, custom symbolism, and recurring themes. In the Great Hypostyle Hall of Karnack (around 1224 BC), where 134 segments are arranged in sixteen lines, Egyptian sections are widely present for certain segments reaching 24 meters in height.

The papyri structure sections are most likely the most common type. The origins of these segments can be traced back to the Fifth Dynasty. They are made of lotus stems that are put together into a pack embellished with groups: the capital grows out and then limits again like a blossom in the bud, rather than opening out into the state of a bellflower. A persistently repeating design of stipules can be found on the base, which tightens to form a half-circle like the stem of a lotus.

Early sections of stone were constructed, some of them out of a solitary piece of stone. Among the heaviest stones used in the design are solid sections. Other segments of stone are made of numerous stone, mortared or dry-fit segments together. Separated segments were cut with a middle opening or sadness in many old-style locations so they could be fixed together with stone or metal pins. In addition to a decrease in measurement along the tallness of the segment, the plan of most traditional sections includes (the incorporation of a slight outward bend in the sides) with the objective that the top is only 83 per cent of the base width. This reduction emulates the parallax effects that the eye hopes to see, and will generally make sections look taller and straighter than they are while that effect is added by entasis. Within a table, a column is a vertical group of values. It has multiple rows of values from a single field...

NOTE: While it's easy to confuse rows and columns, just keep in mind that columns are vertical, whereas rows are horizontal, like text rows. Here many types of columns.

1. Tied Column. This type of column is commonly construction from reinforced concrete.

2. Spiral Column. Spiral column is also construction from reinforced concrete.

3. Composite column.

4. Axially Loaded Column.

5. Column with Uniaxial Eccentric Loading.

6. Column with Biaxial Eccentric Loading.

7. Short Column.

8. Long Column.

To support the weight of the roof and/or the upper floors, a column is used. Many columns are used for decorative purposes nowadays. A column, when combined with load-bearing beams, can support a significant amount of weight.

Columns may be designed for lateral forces to resist. Because of the similar stress conditions, other compression members are commonly referred to as "columns." Beams or arches on which the upper parts of walls or ceilings rest are frequently used to support columns.

1.3 SCOPE OF THE STUDY

In ABAQUS, the study includes 3D finite element analyses representing RC columns. The investigation focuses on modelling axially loaded columns placed at the centre of the cross-sections of the columns. To model the columns, the finite element method was used.

ABAQUS is software for finite elements. Finite element analysis of different column types for different axial load and uniaxial moment position is being done in the present work. The study is based on the fact that the variation in stress and strain depends primarily on the position of the load on the cross-section of the column.

1.4 OBJECTIVES

The objective of Finite Element Analysis (FEA) is to simulate column displacement control using the Finite Element Method, a numerical technique. A computational instrument for performing engineering analysis is FEA as applied in engineering. It involves the use of mesh generation methods to divide a complex problem into small elements, as well as the use of FEM algorithm-coded software programs.

We used the finite element approach to perform a column analysis. We've picked two materials in the column. The stirrups for the column are given. We have altered concrete compressive behavior here. This load is applied to the free end portion and we used the axial load uniformly. And lateral load has also been used. We discovered that applying the finite element method to the column adds force and changes the properties of the concrete, allowing us to control the displacement by positioning it at a certain distance.

When a lateral load is applied, it induces column displacement. For the sake of contrast, we used a displacement of 250 mm in this model.

This amount cannot be measured normally. It is not feasible if we want to analyze periodic theory. We also developed a model for finite element analysis for this. All of the properties of concrete and steel were used in this project. Now, irrespective of the load, we'll control the displacement from here.

A concrete column's displacement is difficult to control normally. Since it is a material that is fragile. In several studies, the values of concrete properties such as compressive and tensile strength have been established here.

CHAPTER-02

LITERATURE REVIEW

2.1 INTRODUCTION

Abaqus FEA (formerly ABAQUS) is a software suite, initially released in 1978, for finite element analysis and computer-aided engineering. This software's name and logo are based on the abacus calculation tool. Five core software products make up the Abaqus product suite. The "Finite Element Analysis of Force Displacement Behaviour of Reinforced Concrete Column" was performed using the ABAQUS software.

Abaqus is a commercial finite element analysis software package. Abaqus/Standard, Abaqus/Explicit, and Abaqus/CAE are the three core products in the Abaqus product suite. Abaqus/Standard is a general-purpose solver that solves finite element analyses using a traditional implicit integration scheme. To resolve highly nonlinear transient dynamic and quasi-static analyses, Abaqus/Explicit uses an explicit integration scheme. For the analysis products, Abaqus/CAE provides integrated modelling (pre-processing) and visualization (postprocessing) environment.

In the automotive, aerospace, and industrial goods industries, Abaqus is used. Academic and research institutions prefer the product because of its extensive material modelling capabilities and ability to be customized. Abaqus also has a good collection of Multiphysics capabilities, such as coupled acoustic structural, piezoelectric, and structural-pore capabilities, which makes it appealing for production-level simulations involving multiple fields.

2.2 BACKGROUND STUDY

First of all, thanks to Amir Hossein Basravi for such a creative thesis. The thesis was published in December 2010. Here the effect of placing holes along reinforced concrete short braced columns in multi-story buildings was investigated using a finite element model. Using three-dimensional non-linear finite element analysis, ABAQUS modelled RC columns with different sizes and reinforcement with holes positioned at the centre of their cross-sections. Compared with the results obtained from laboratory testing of the same columns, the ultimate strengths of the columns obtained from the present study are compared with the design strengths recommended by the practice codes BS 8110 and ACI. It was highlighted that the load-carrying capacity of columns with holes has been reduced.

Another thesis that has been done at ABAQUS. At first, the specimens of the beam-column joint were subjected to reverse cyclic loading at the beam end. Based on the joint reinforcement description, the samples were divided into two groups. The first group member B. Venkatesan of three non-ductility specimens had joint detailing done according to IS456-2000, and the second group member R. Ilangovan of three ductility specimens had joint reinforcement done according to IS13920-1993, with the same axial load cases as the first group. The experimental findings are backed up by analytical studies using ABAQUS and finite element models. The findings show that the simulation of hysteresis is satisfactory for both un-strengthened and ferrocement-enhanced specimens. In addition, the strengthened beam-column joints exhibit better structural performance when ferrocement reinforcement is used than the un-strengthened specimens of about 31.56 percent and 38.98 for DD-T1 and DD-T2 respectively. The predictions of analytical shear strength were consistent with the test results reported in the literature, thus adding confidence in the validity of the models proposed.

This is another thesis and I would thank Ahmed, F.S, Khan, M.A because they did a much better job. A structure must be safe against collapse and usable in order to serve its purpose. In order to keep the cracks within tolerable limits, serviceability requires that deflections be sufficiently small. Although the widely used ACI Code suggests a detailed method of deflection calculation, it frequently fails to accurately predict a structure's actual behaviour. Finite element analysis, using ABAQUS, was performed for this purpose. The goal of the study was to create a program that would analyse a column that was fixed at the base and measured lateral deflection. The concrete and reinforcement were modelled using the key point, node, meshing, and separate elements concepts. The study scope is limited to the lateral load applied only at the midpoint in the direction of X at the top height. The same operation can be carried out in the Y direction. The current application allows for the use of columns of varying sizes and heights. However, the work was done with only one-story height in mind to serve the purpose of a single story.

2.3 AXIAL LOAD

Axial loading is when a force is applied to a structure directly along with one of its axes. From: Finite Element Method Basic as Applied to Biomechanics of Injury, 2018.

Axial loading is a force that is applied along the axis line. When a force is applied to an object, the axial loads act along the object's axis. Alternatively, the axial force is seen as passing through a considered section's neutral axis, which is normal to the section's plane.

2.4 LATERAL LOAD

Lateral loads are live loads that are applied parallel to the ground; that is, they act on a structure as horizontal forces. For instance, they are unique to gravity loads that are vertical, downward forces. The following types are most common: wind load, seismic load.

2.5 CONCENTRED LOAD

The concentrated load is a load that, compared to the size of the loaded member, is applied to a relatively small area.

CHAPTER-03

MODELING & ANALYSIS

3.1 FEA of column frame:

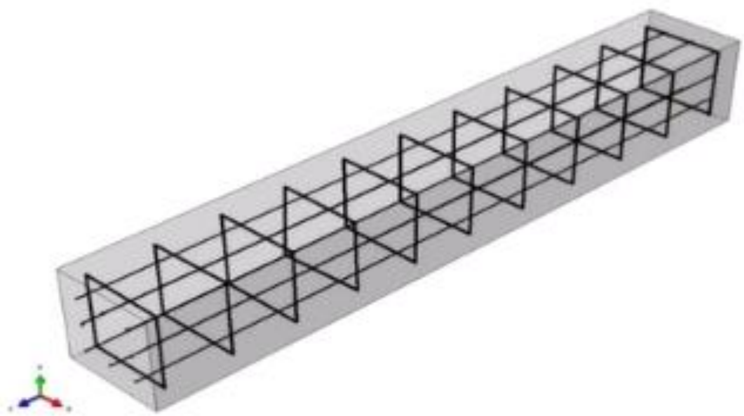

Figure 3. 1: FEA of column frame

3.2 Properties and Dimension:

Dimension: 2000*300*300(mm)

Boundary Conditions: One free, one fixed

Axial load: 3 N/mm^2

Concentrate Load: 5 N/mm^2

Lateral load: Displacement 250 mm

Main Bars Area: 254.47 mm Square

3.3 Points Coordinate of the main bars on the fixed end:

X	Y	Z
62.5	62.5	2000
62.5	150	2000
62.5	237.5	2000
150	237.5	2000
237.5	237.5	2000
237.5	150	2000
237.5	62.5	2000
150	62.5	2000

Table 3. 1: Point coordinate of the main bars on the fixed end.

Stirrups:

Area: 78.54 mm Square (240*240mm)

Point Coordinate for the first stirrup (30, 30, 1950)

3.4 Properties of steel and concrete:

Steel:

Density	P=7.8E-6kg/mm3
Elastic	E=200E+3 (MPa), u=0.3
Plastic	
Yield Stress (MPa)	Plastic Strain
280	0
370	0.1

Table 3. 2: Steel Property

Concrete:

Density			P=2.4E-6kg/mm3	
Elastic			E=17585 (MPa), u=0.17	
Plasticity				
Dilation Angle	Eccentricity	Fb0/fc0	K	Viscosity Parameter
30.5	0.1	1.16	0.666	0.001

Table 3. 3: Concrete Property

3.5 OUTLINE OF THE METHODOLOGY

A column is an essential part of any structure. It is like the legs that a structure stands on. It's made to withstand axial and lateral forces and safely transfer them to the ground's footings. Using a simple process outlined in this linked article, you can calculate the superimposed loads on a column in a structure manually.

The thesis dissertation was prompted by finite element analysis of reinforced concrete column force-displacement behaviour.

The concept has been completed with ABAQUS software and variations have been listed below;

- At first, we take a specific measurement of this column. (Modelling space is **3D**, FEA analysis in ABAQUS).
- Then we create 3 different parts concrete, bar, stirrups.
- We used two types of materials in this column steel & concrete. (This is mainly RC column design).
- This column’s approximate size is 600mm.
- Depth of the concrete section is 2000mm.
- Here bar size is 4000mm.
- Stirrups size is 480mm.
- Steel density is 7.8e-6.
- Concrete density is 2.4e-6.
- We used concrete damage plasticity – (Compressive behaviour & tensile behaviour).
- We take some data for Compressive & Tensile behaviours.
- Section of concrete Solid & Homogeneous.
- And section of bar Beam & Truss.
- Bar area is 254.74mm^2.
- This is a non-linear analysis.

- We used an embedded region.
- Coordinate the datum point X, Y, Z.
- Boundary condition is one free, one fixed.
- We used axial load uniformly (pressure), this load is added to the free end section. And also used lateral load.
- Magnitude is 3 N/m^2
- Mesh the concrete part.
- We used some data for step.
- For the force-displacement curve we used the reaction force in the fixed end and the displacement at the free end in the Y direction.
- We selected 2 nodes in the free end (X and Y-axis).
- We tried to draw the force-displacement curve of the column.

3.6 Model RC column Using ABAQUS:

Part Model:

The geometry of the individual components of your model is defined by parts and is, therefore, the building blocks of an Abaqus/CAE model. We can create parts of Abaqus/CAE that are native.

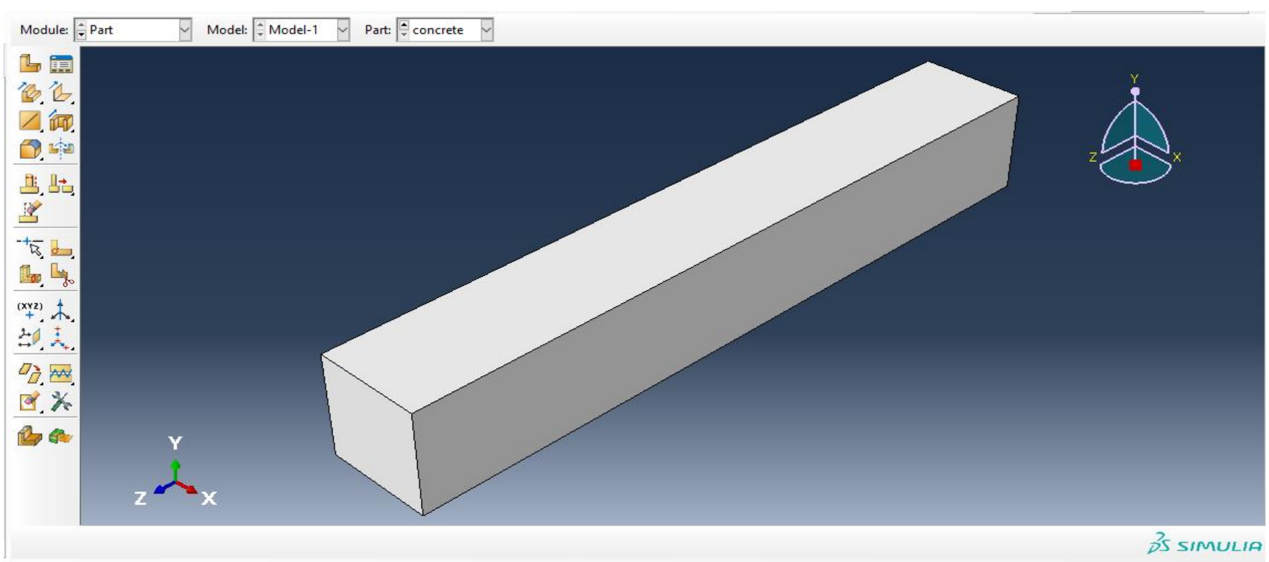

Figure 3. 2: Model of Parts

Property:

In Abaqus/Explicit, assigning contact properties to general contacts. Contact properties define the mechanical surface interaction models that govern the behaviour of surfaces when they come into contact and can be applied to specific regions within a larger contact domain.

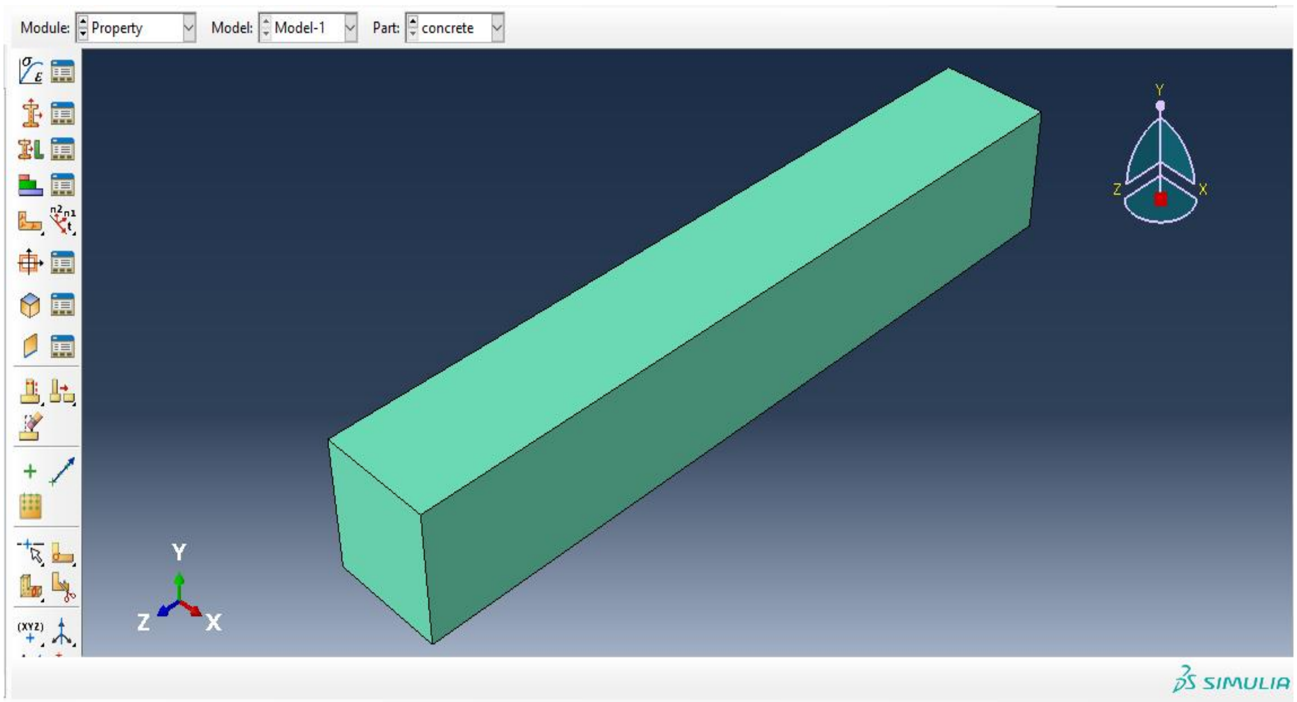

Figure 3. 3: Model of solid element

Assembly:

The assembly interface in Abaqus allows analysts to use an organizational scheme parallel to the physical assembly to create a finite element mesh. Part instances are the components that are assembled together in Abaqus.

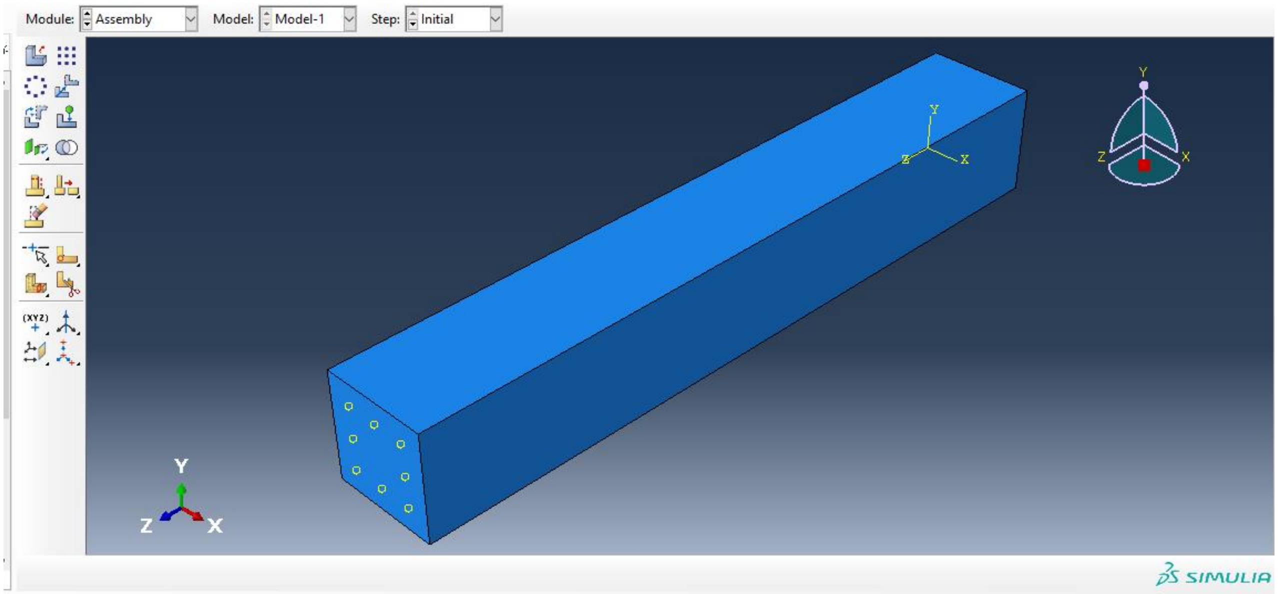

Figure 3. 4: Model of element assemble

Step:

A step is a thermal transient, a creeping hold, a dynamic transient, etc., any convenient phase of history. In its simplest form, a step can only be a static analysis of a load change from one magnitude to another in Abaqus/Standard. A description of each step that will appear in the data can be provided.

Interaction:

In Abaqus, general contact interactions are defined as an all-inclusive surface with all exterior faces, feature edges, and analytical rigid surfaces, edges based on beams and trusses, and Eulerian material boundaries.

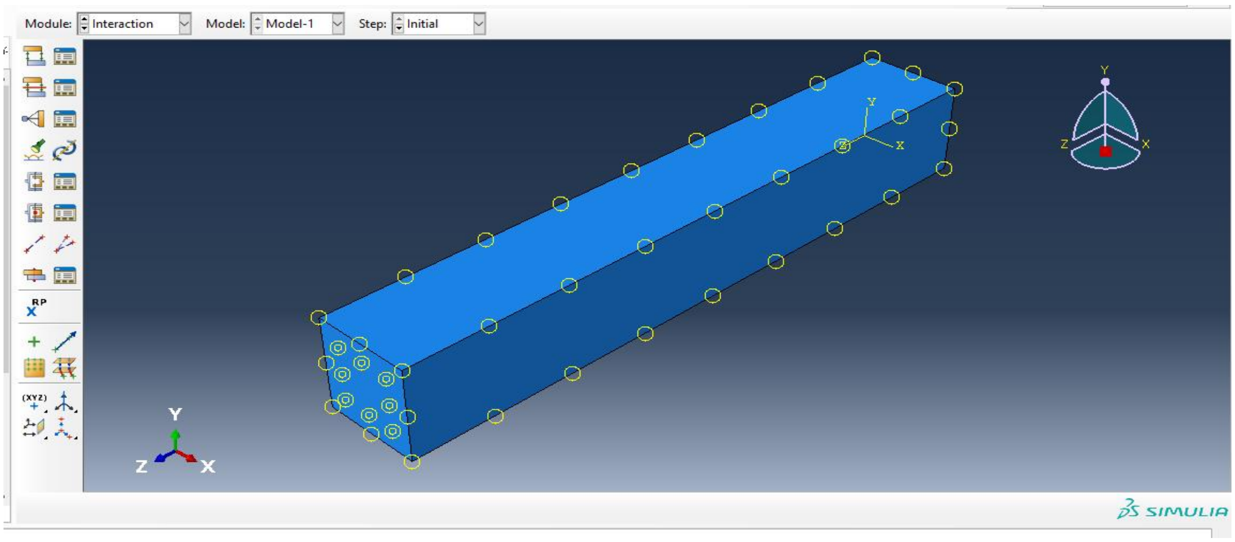

Figure 3. 5: Model of interaction

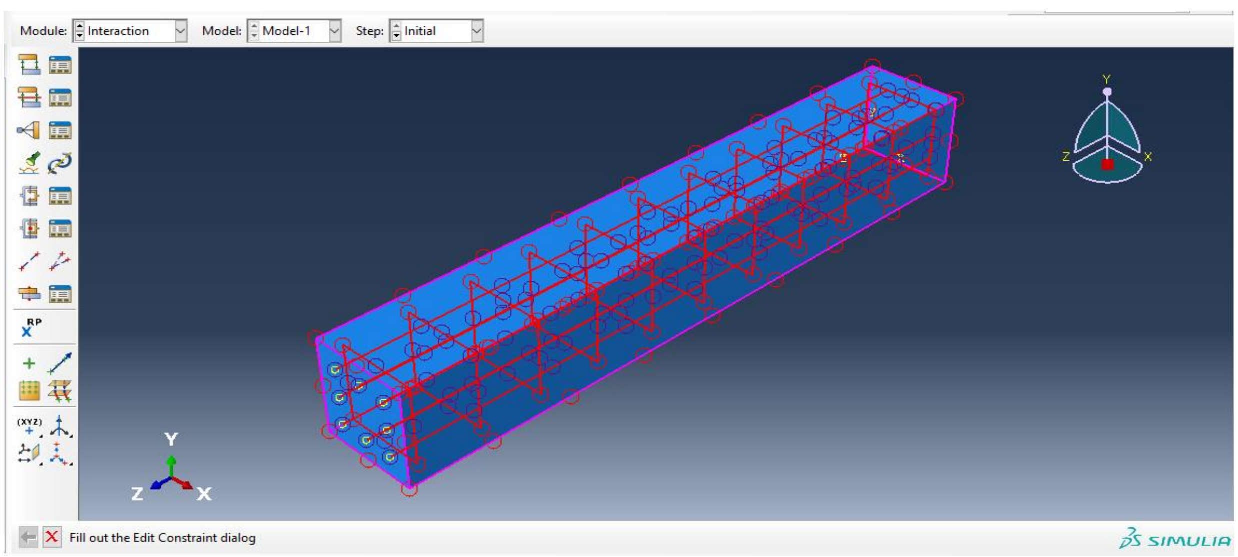

Figure 3. 6: Model of embedded region.

Load:

- Axial Load: A force-directed over the axial line is an axial load. The axial loads act along the object's axis if the object is loaded with force. σ is the normal stress here, F is the axial force, and A is the area of the cross-section.
- Lateral load: Lateral loads are live loads applied to the ground in parallel; that is, they are horizontal forces acting on a structure.
- Concentrated load: Concentrated load is a load applied compared to the size of the loaded member to a relatively small area.

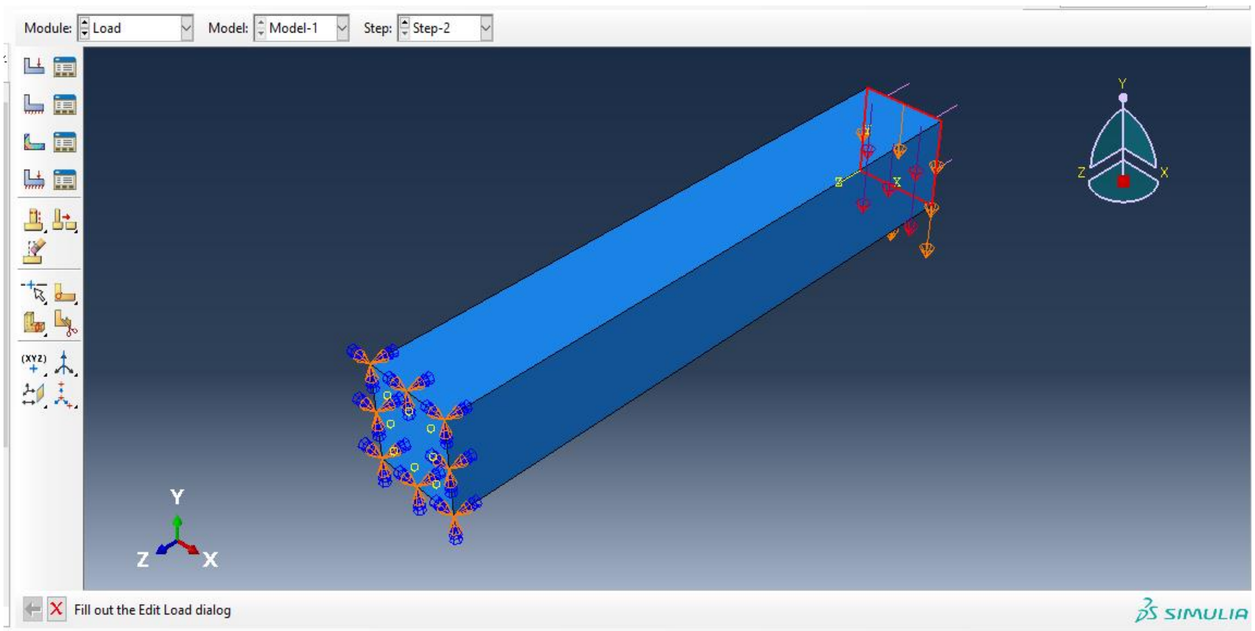

Figure 3. 7: Model of applied load.

Boundary Condition:

Boundary conditions can be defined as the total value of a variable when using the 'direct' format or, in a stress/displacement analysis, as the value of the velocity or acceleration of a variable. It is possible to define in a step as many boundary conditions as necessary.

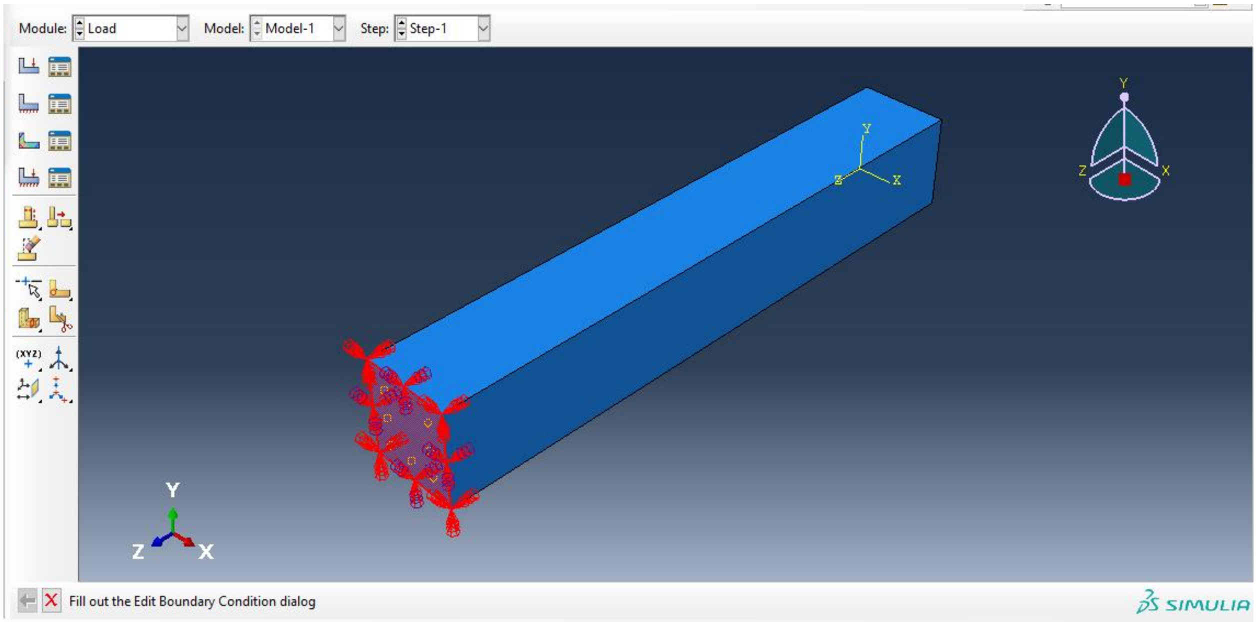

Figure 3. 8: Model of boundary condition (One fixed, one free).

Mesh:

The focus of Finite Element Analysis (FEA) is to simulate some physical phenomena using the Finite Element Method, a numerical technique.

In the ABAQUS model, units and meshing materials Better convergence is achieved during analysis, according to the force and deformation characteristics of the model. The concrete used is C3D8R, which has an eight-node reduced integral solid element. A two-node truss unit is used by Reinforcement (T3D2). Using a two-node truss unit, both the warp and weft fiber bundles of the fiber woven mesh are assumed to be ideal linear elastic materials (T3D2). Structured adaptive meshing performs model unit meshing. ABAQUS, the physical finite element model of the model and the mesh division, automatically generates the corresponding element mesh.

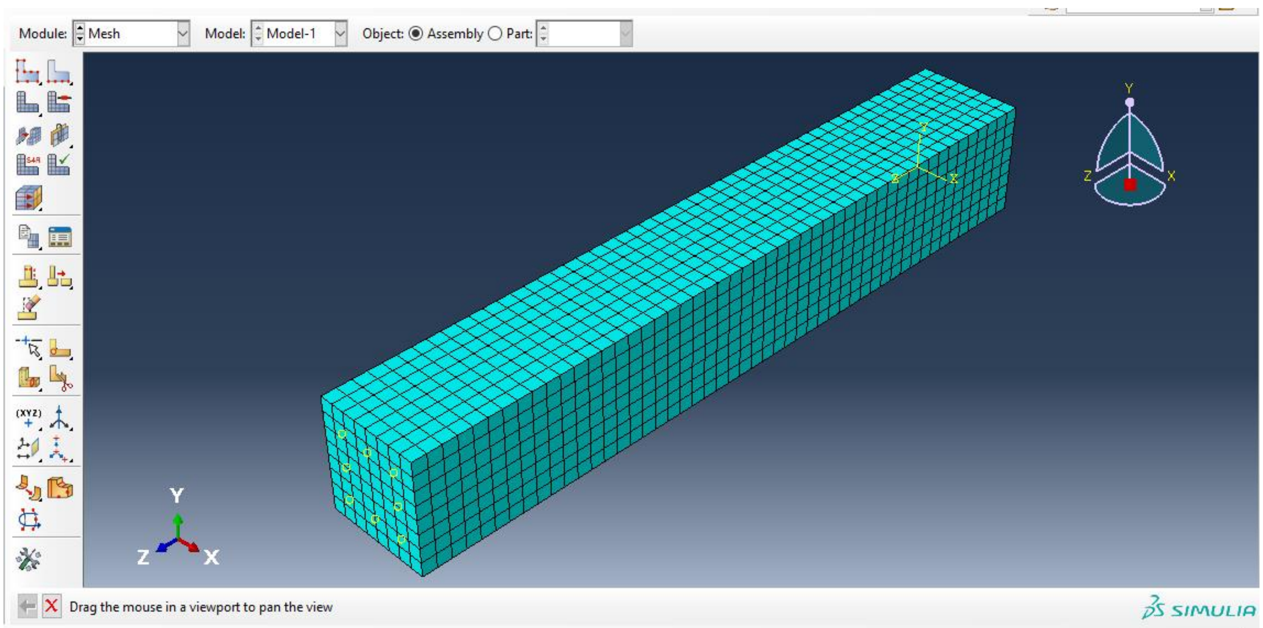

Figure 3. 9: RC column mesh in ABAQUS.

3.7 Two vary concrete proprieties compressive behaviour for develop simulation column.

Material Properties of concrete:

C-20			
Concrete compressive behavior		Concrete tensile behavior	
Yield stress	Inelastic strain	Yield stress	Cracking strain
10.2	0	1.16	0
12.8	7.74E-05	1.08	0.000036
15	0.000173585	0.975	0.000140293
16.8	0.000288679	0.905	0.000244566
18.2	0.000422642	0.84	0.000348534
19.2	0.000575472	0.78	0.000452218
19.8	0.00074717	0.724	0.000555638
20	0.000937736	0.672	0.000658814
19.8	0.00114717	0.624	0.000761762
19.2	0.001375472	0.58	0.000864499
18.2	0.001622642	0.538	0.00096704
16.6	0.001888679	0.5	0.001069399
15	0.002173585	0.464	0.00117159
12.8	0.002477358	0.431	0.001273623
10.2	0.0028	0.4	0.001375511
7.2	0.003141509		
3.8	0.003501887		

Table 3. 4: Concrete properties of C-20

Material Properties of concrete (Cont.):

C-30			
Concrete compressive behavior		Concrete tensile behavior	
Yield stress	Inelastic strain	Yield stress	Cracking strain
15.3	0	1.16	0
19.2	0.000048249	1.08	0.000036
22.5	0.000119844	0.975	0.000140293
25.2	0.000214786	0.905	0.000244566
27.3	0.000333074	0.84	0.000348534
28.8	0.000474708	0.78	0.000452218
29.7	0.000639689	0.724	0.000555638
30	0.000828016	0.672	0.000658814
29.7	0.001039689	0.624	0.000761762
28.8	0.001274708	0.58	0.000864499
27.3	0.001533074	0.538	0.00096704
25.2	0.001814786	0.5	0.001069399
22.5	0.002119844	0.464	0.00117159
19.2	0.002448249	0.431	0.001273623
15.3	0.0028	0.4	0.001375511
10.8	0.003175097		
5.7	0.003573541		

Table 3. 5: Concrete properties of C-30.

Material Properties of concrete (Cont.):

C-40			
Concrete compressive behavior		Concrete tensile behavior	
Yield stress	Inelastic strain	Yield stress	Cracking strain
20.4	0	1.16	0
25.6	2.67E-05	1.08	0.000036
30	0.00008	0.975	0.000140293
33.6	0.00016	0.905	0.000244566
36.4	0.000266667	0.84	0.000348534
38.4	0.0004	0.78	0.000452218
39.6	0.0004	0.724	0.000555638
40	0.00056	0.672	0.000658814
39.6	0.000746667	0.624	0.000761762
38.4	0.0012	0.58	0.000864499
36.4	0.001466667	0.538	0.00096704
33.6	0.00176	0.5	0.001069399
30	0.00208	0.464	0.00117159
25.6	0.002426667	0.431	0.001273623
20.4	0.0028	0.4	0.001375511
14.4	0.0032		
7.6	0.003626667		

Table 3. 6: Concrete properties of C-40

CHAPTER -04

RESULTS

4.1 RESULT

- We have shown here that displacement and load constant but reaction force is different. C40's reaction force is the highest. C30 reaction force is less than C40. And the C20 reaction force is less than the C40 and C30, respectively. We are able to calculate the exact displacement in this column model.
- And we keeping the same displacement and we need to provide higher reaction force for the high strength concrete.
- We know that the higher the strength of the concrete, the greater the force.
- We are able to control the exact displacement in this column model.
- We know, it takes more force to do the displacement due to more strength.
- And if we want to reduce the displacement due to lateral load and axial load then we have to use high strength material.
- Column displacement occurs when the lateral load is applied. We have taken 200 mm displacement in this model for the purpose of comparison.
- This quantity cannot be normally analysed. If we want to analyse in regular theory, it is not possible. For this, we have developed a model in finite element analysis. Here we used all the properties of concrete and steel. Now we will control the displacement from here regardless of the load.
- The displacement of a concrete column cannot be controlled normally. Because it is a brittle material. Here the values of concrete properties such as compressive and tensile strength have been developed through many experiments.

4.2 Compares of force displacement behaviour develop Abaqus model of the various RC Column:

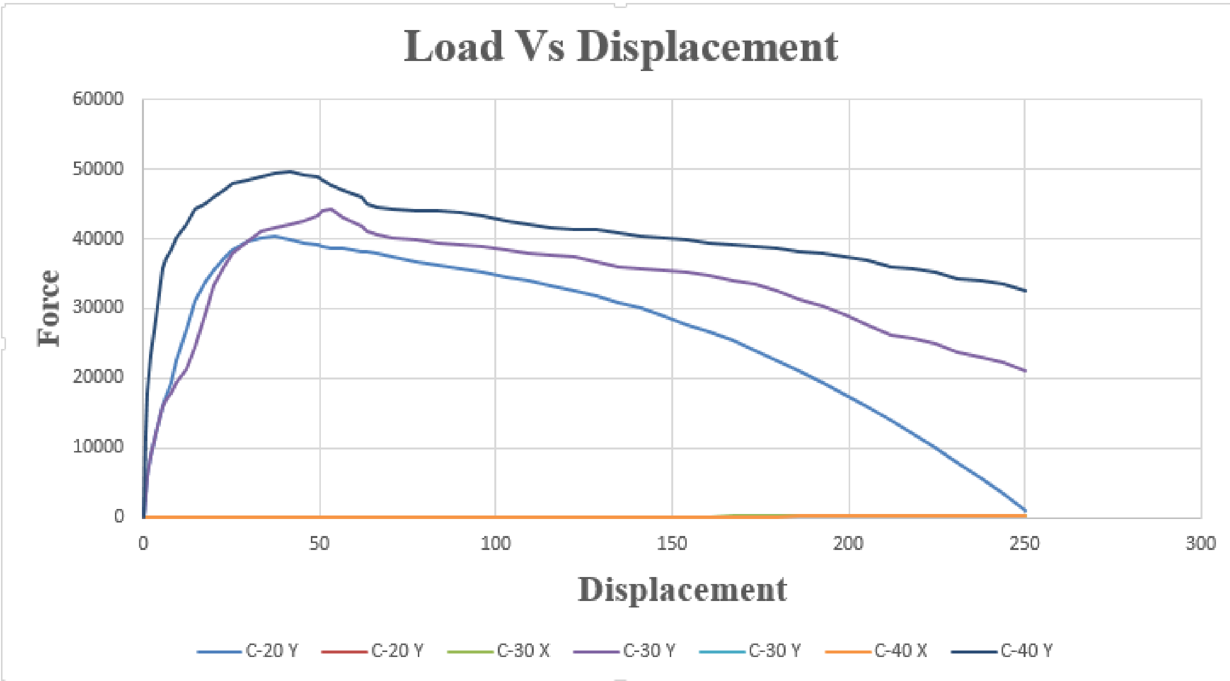

Figure 4. 1: Force displacement curve comparison chart.

❖ Displacement: X-axis:

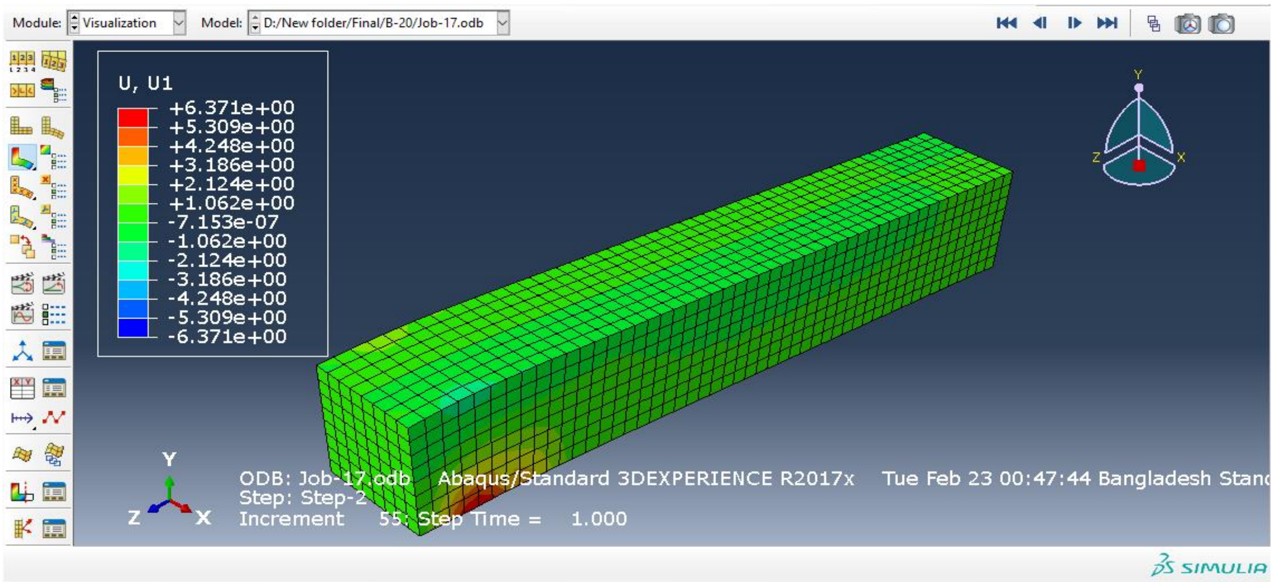

Figure 4. 2: Displacement of U (X direction)

❖ Displacement: Y-axis

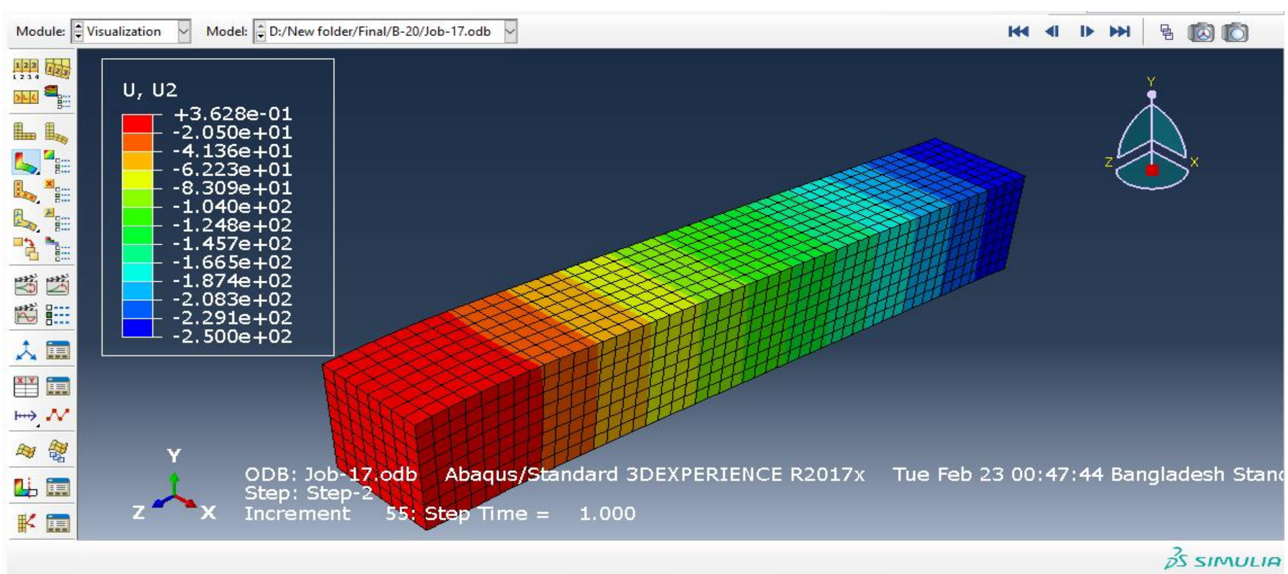

Figure 4. 3: Displacement of U (Y direction).

❖ Displacement: Z-Axis

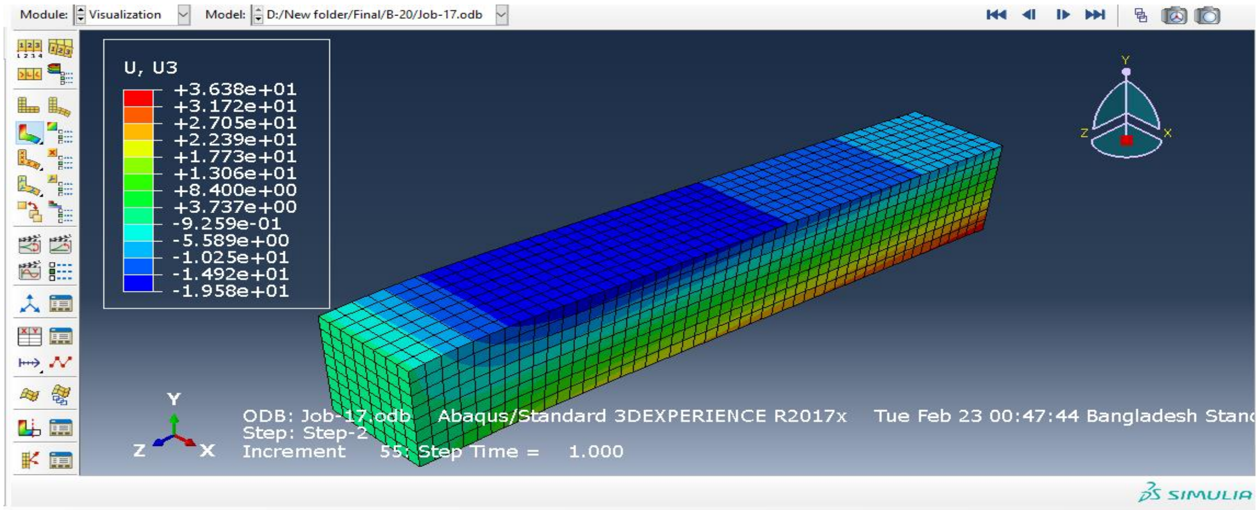

Figure 4. 4: Displacement of U (Z direction).

❖ Symbol of U3

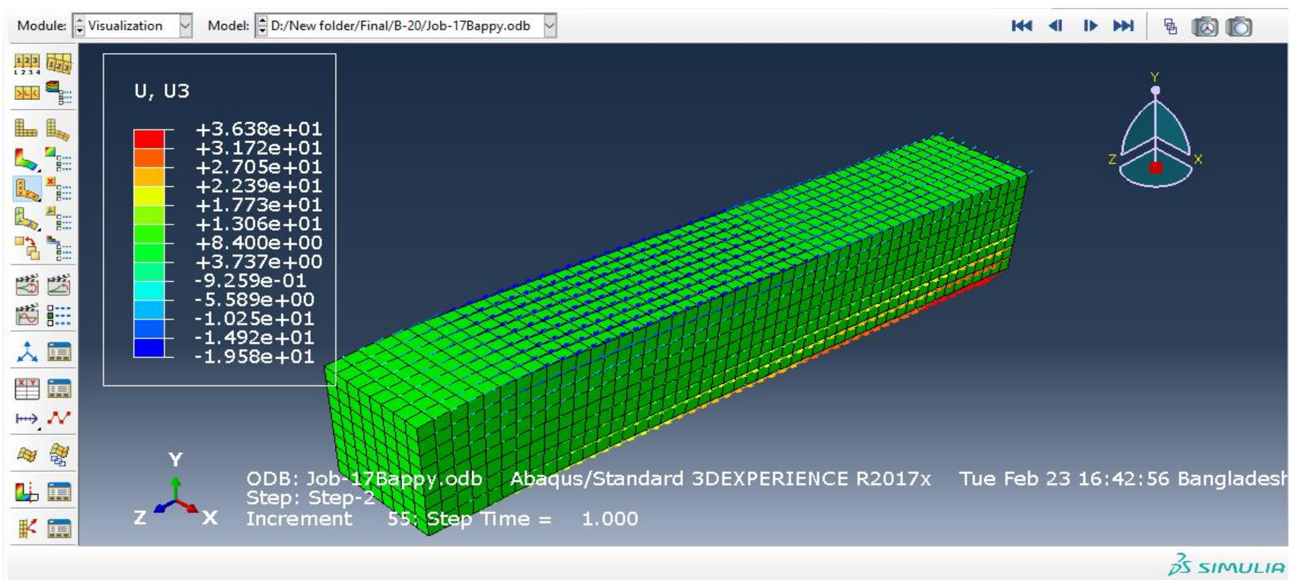

Figure 4. 5: Total stress of the column.

- **Symbol of U2**

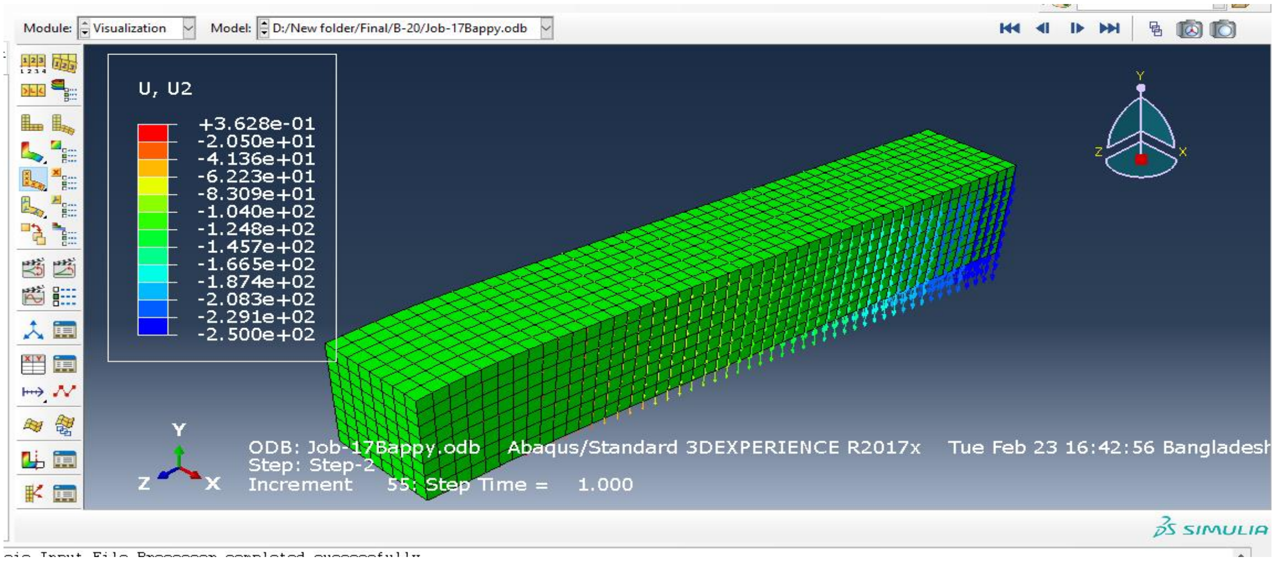

Figure 4. 6: Stress of the column (Y direction)

- **Symbol of U1**

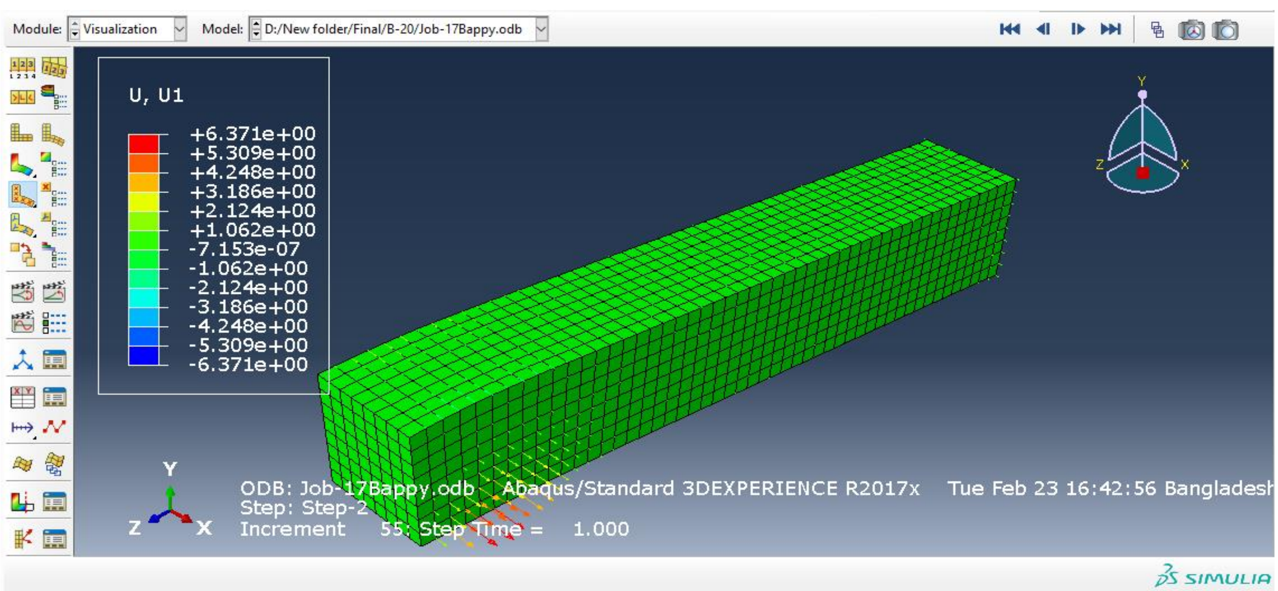

Figure 4. 7: Stress of the column (X direction)

❖ Symbol of PE

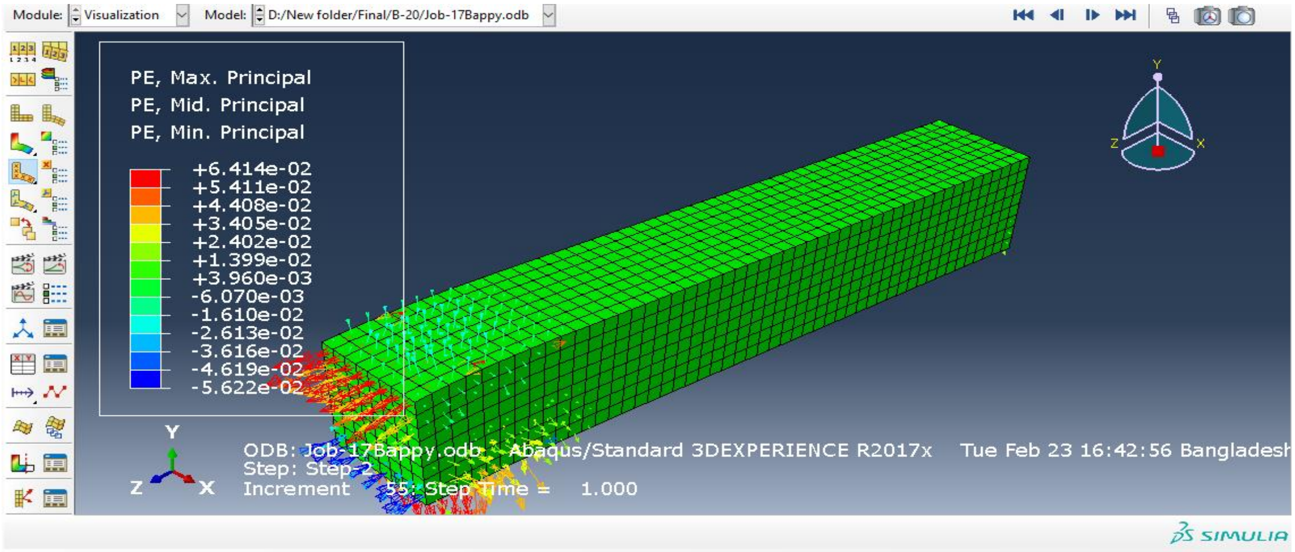

Figure 4. 8: Maximum, Medium & Minimum principal plastic stress of the column

❖ Symbol of Shear

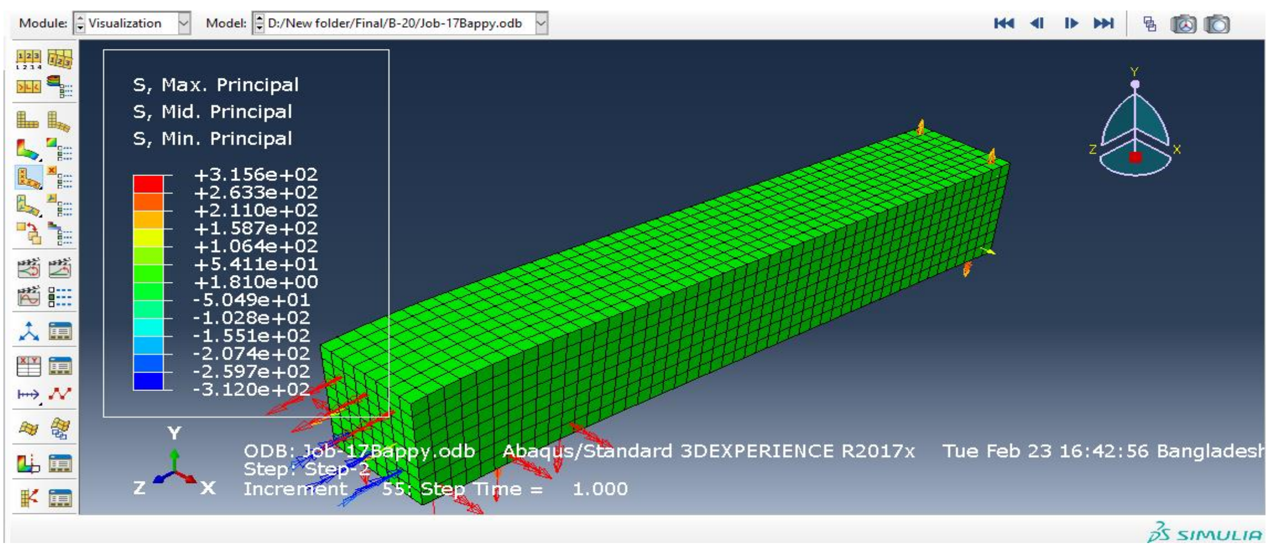

Figure 4. 9: Maximum, Medium & Minimum principal shear stress of the column

❖ Primary Ac Yield

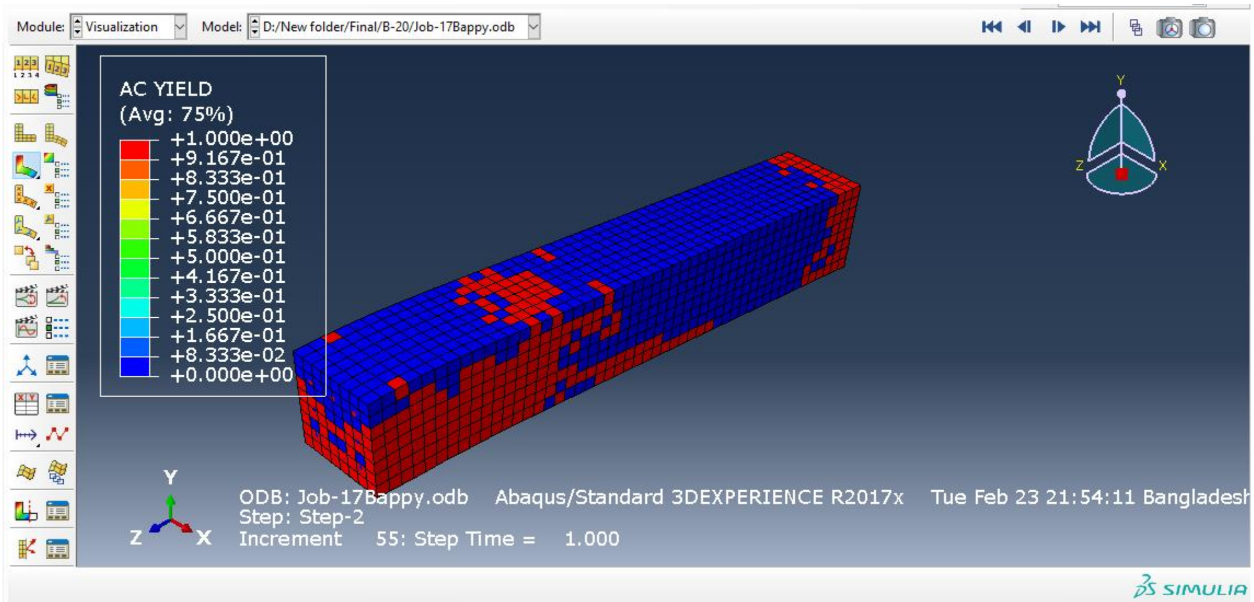

Figure 4. 10: Model of primary Ac Yield of the column

❖ Rebar Frame Displacement U3

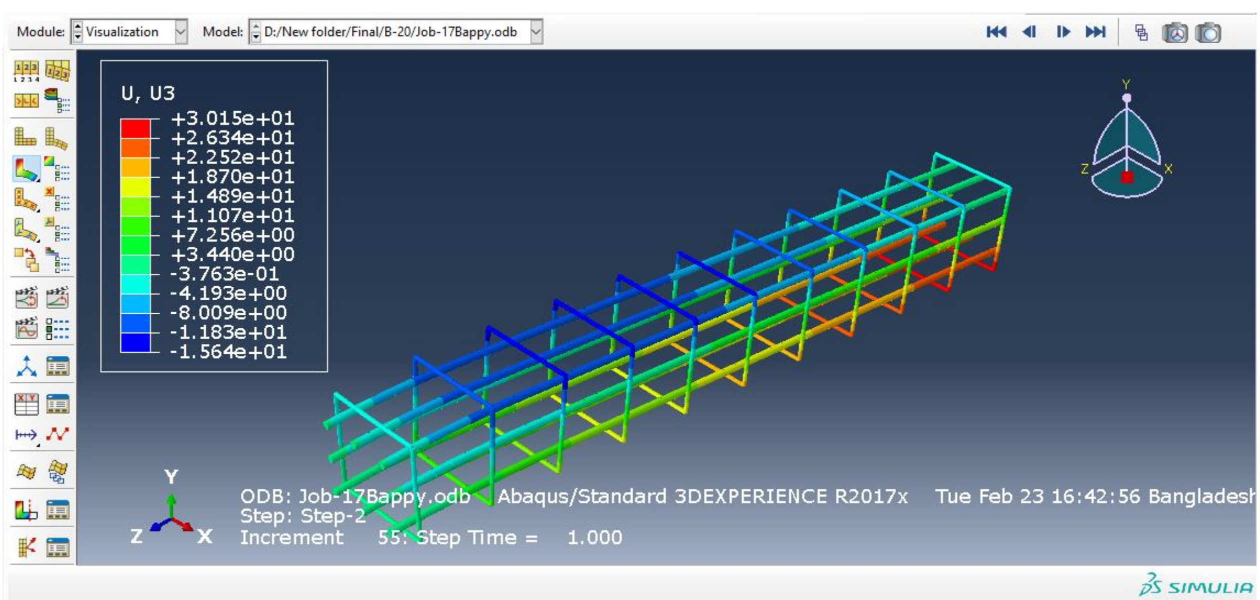

Figure 4. 11: Reinforcement displacement (Z direction).

❖ Rebar Frame Displacement magnitude

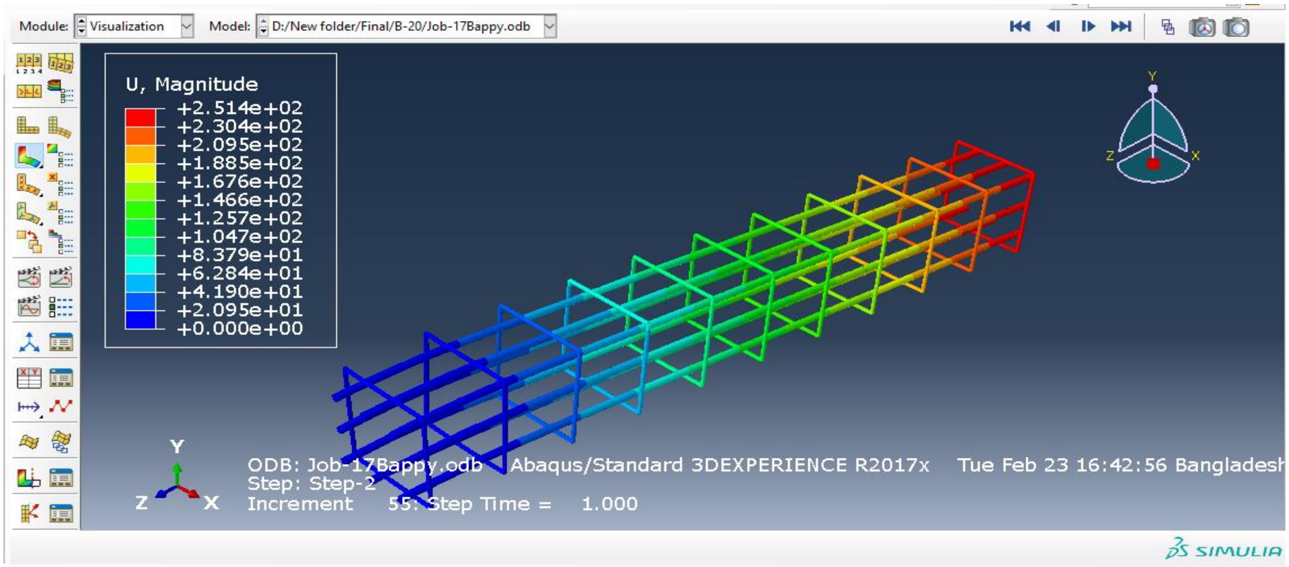

Figure 4. 12: Reinforcement magnitude displacement

❖ Rebar Frame Displacement U1

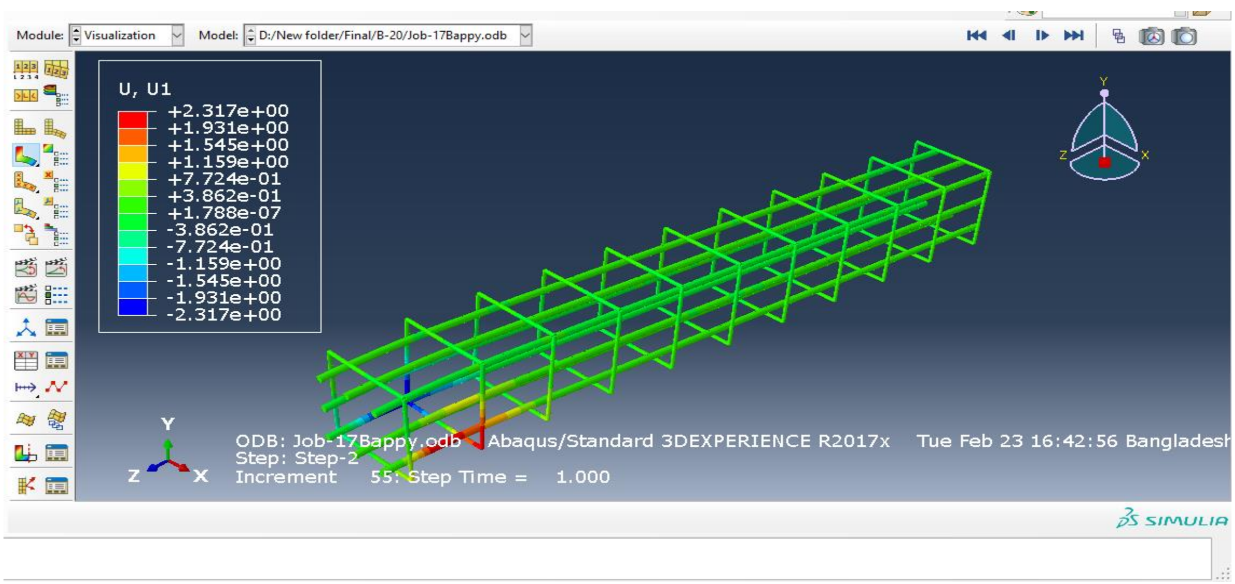

Figure 4. 13: Reinforcement displacement (X direction)

❖ Rebar Frame Displacement U2

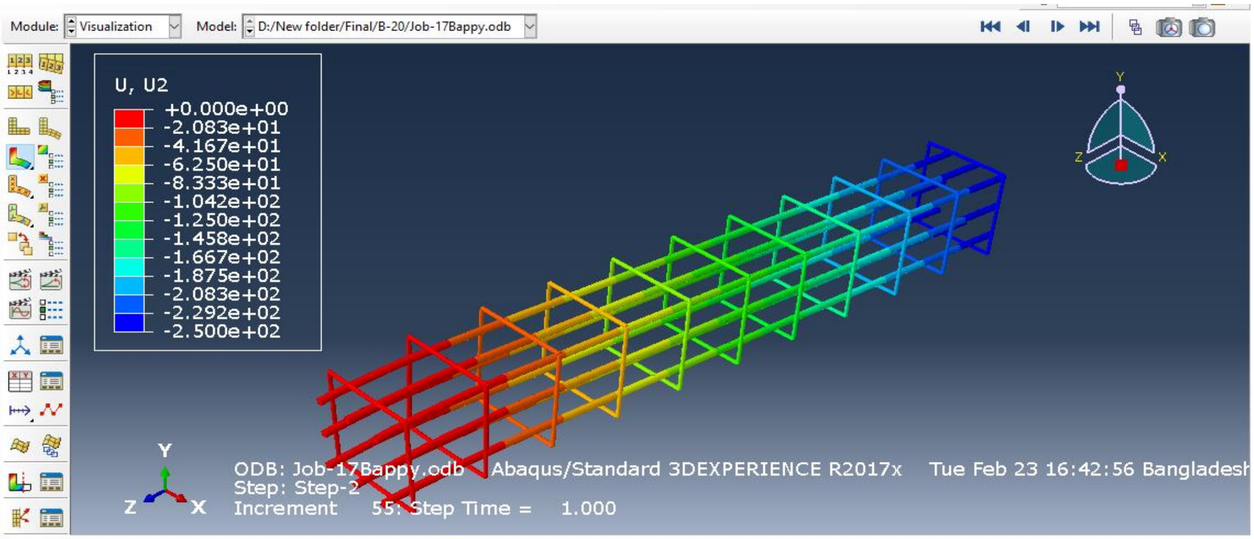

Figure 4. 14: Reinforcement displacement (Y direction)

❖ Rebar Shear Stress Pressure

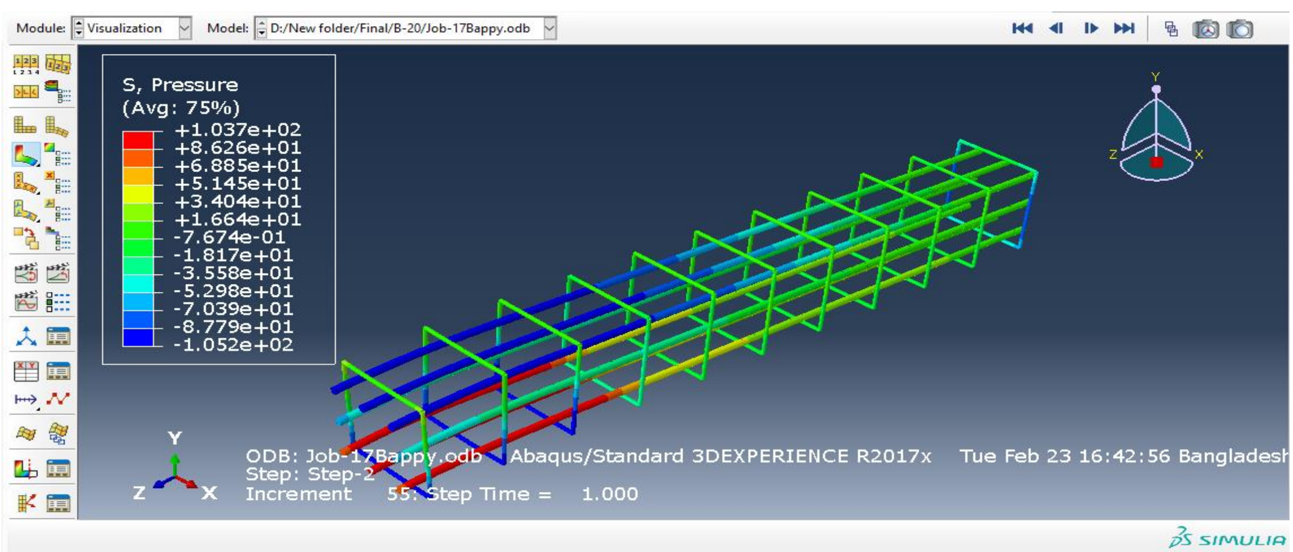

Figure 4. 15: Reinforcement Shear Stress pressure

❖ **Rebar Shear Stress Principal**

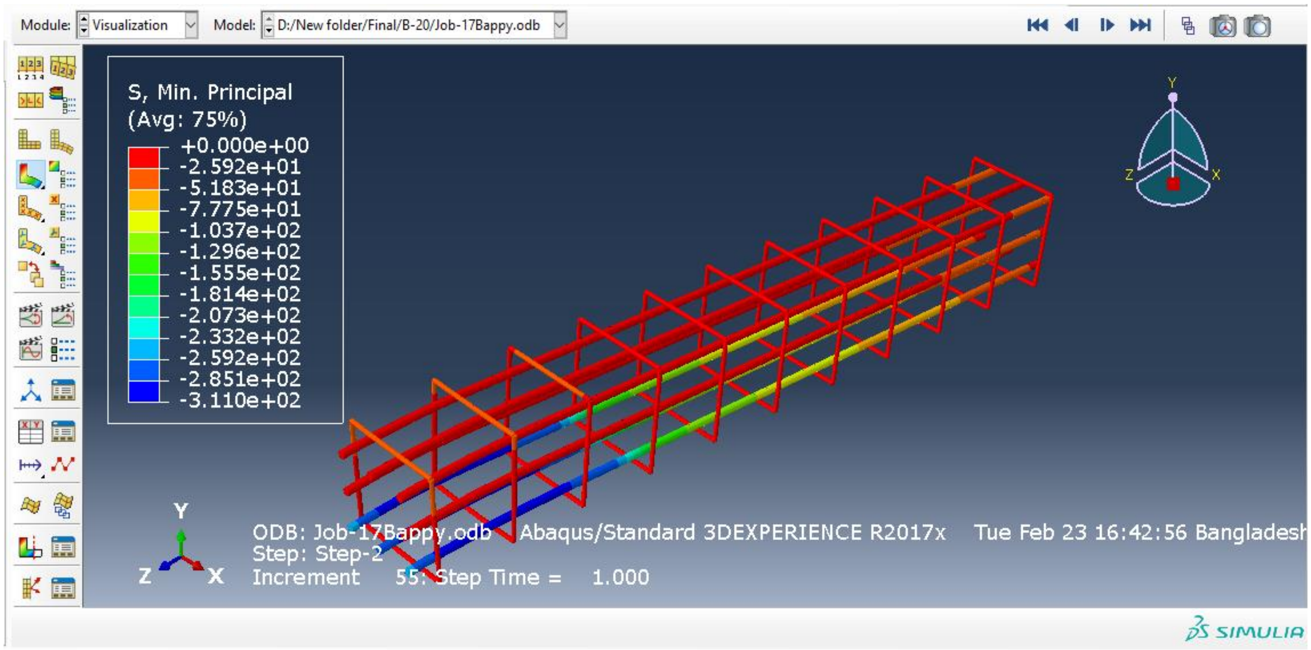

Figure 4. 16: Reinforcement Shear stress Minimum principal

❖ **Rebar Shear Stress Principal**

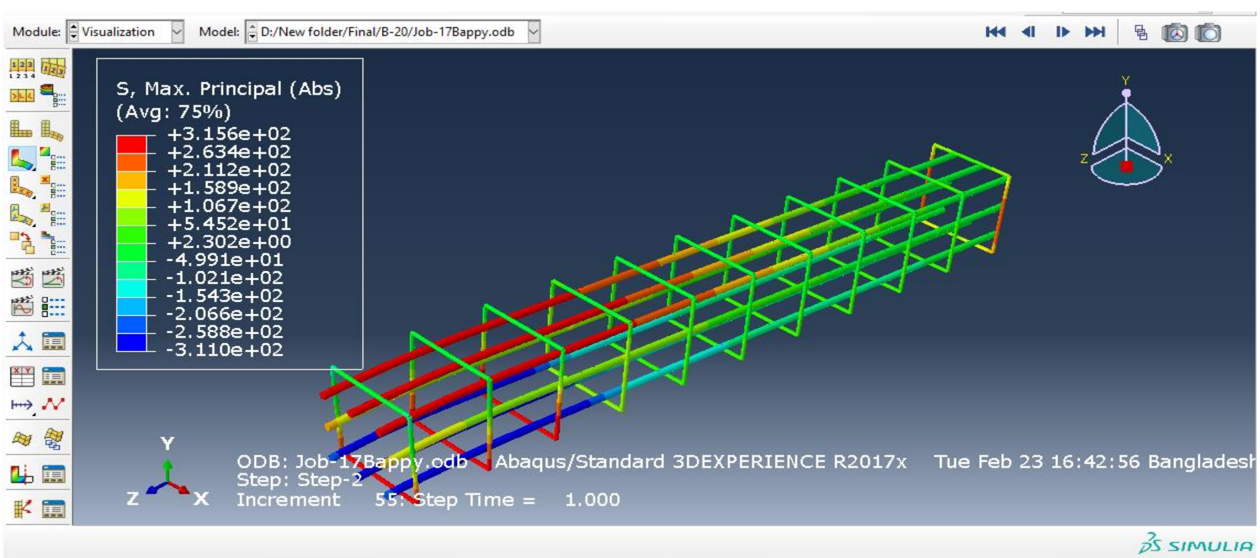

Figure 4. 17: Reinforcement Shear Stress Maximum Principal

❖ **Rebar Shear Stress Principal**

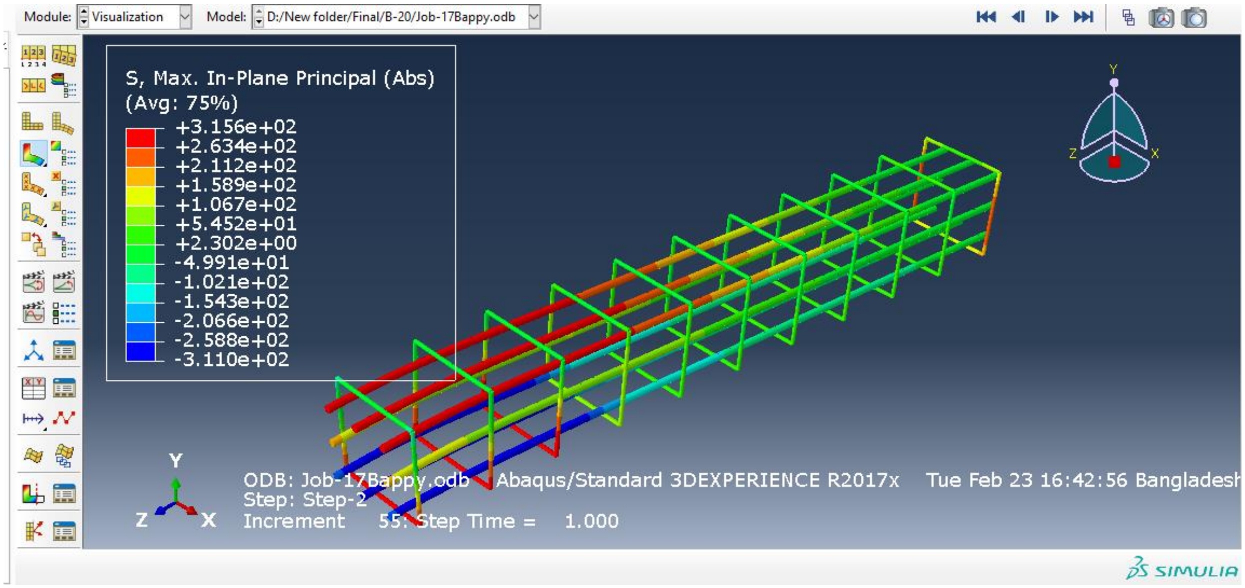

Figure 4. 18: Shear Stress Max, In-plane Principal

❖ **Rebar Shear stress Mises**

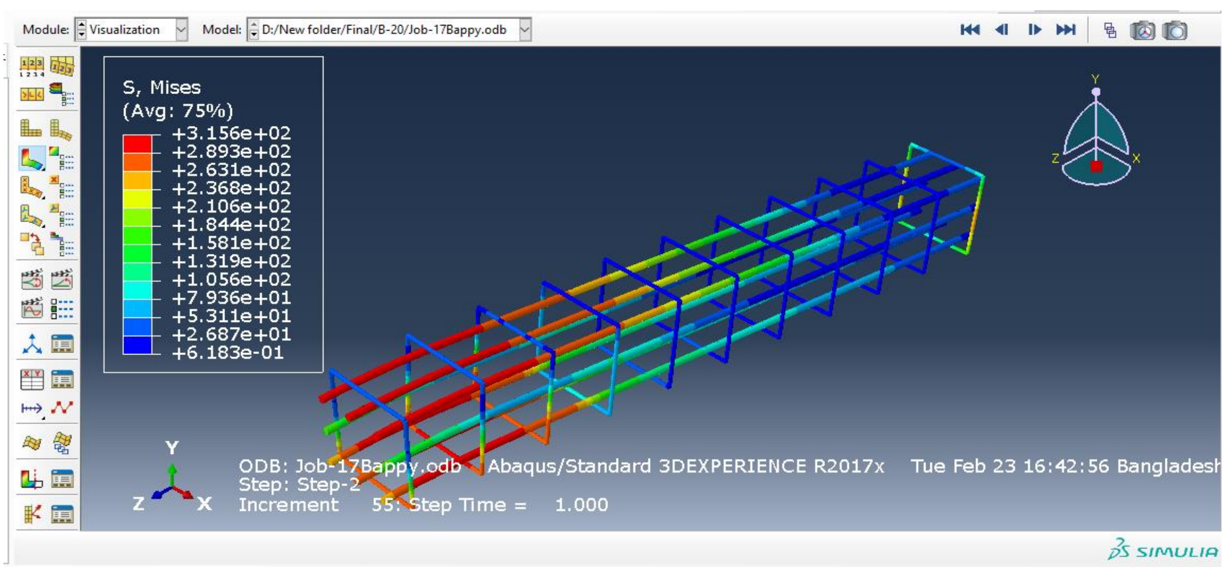

Figure 4. 19: Shear Stress Mises, In-plane Principal

❖ Rebar Shear stress AC Yield

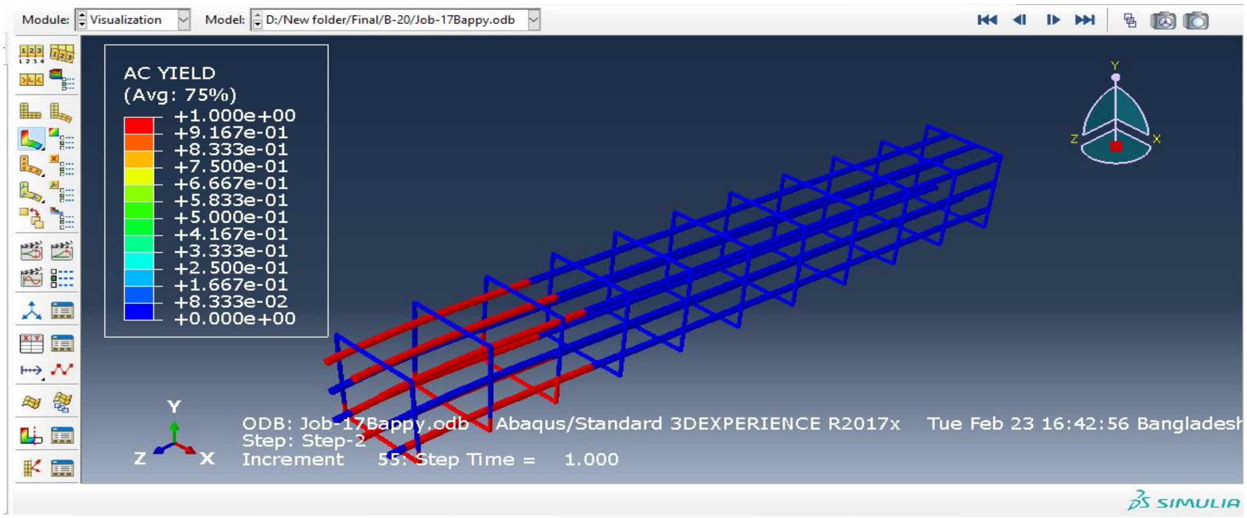

Figure 4. 20: Reinforcement shear stress Ac yield

❖ Rebar PE Magnitude

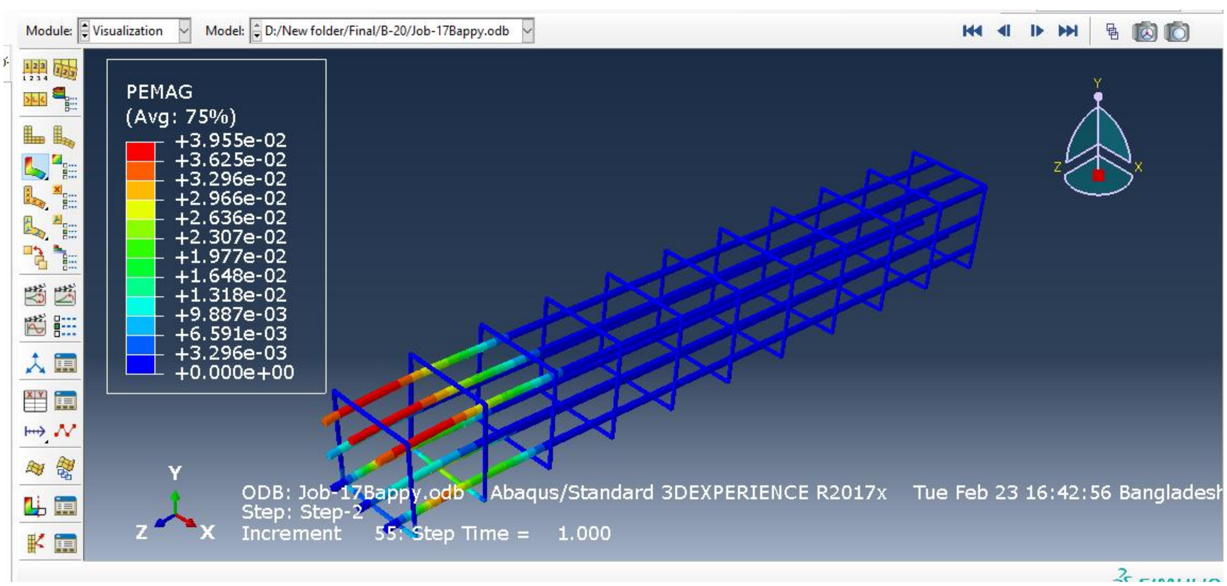

Figure 4. 21: Reinforcement deformed plastic stress

CHAPTER -5

CONCLUSION AND RECOMMENDASION

5.1 Conclusions

We used the finite element approach to do a column analysis. We have 2 material choices made in the column. The stirrups in the column are given. We have altered concrete compression behavior here. We used the axial load uniformly, adding this load to the free end portion. And lateral load was used also. We discovered that applying the finite element method to the column adds force and changes the properties of the concrete, allowing us to monitor the displacement by setting the displacement at a certain distance.

As Conclusion, outline structure displaying and investigation utilizing ABAQUS programming showed that the displacement of the column occurs when the lateral load is applied. For comparison purposes, we have used 250 mm displacement in this model.

We used all the concrete and steel characteristics here. We showed three different curves where we were able to control the displacement regardless of the load.

5.2 Recommendation:

We need this thing for further research people can also consider this,

- Steel properties can be change.
- Support condition can be change.
- Can be used force control model.
- Concrete tensile behavior can be change
- Concrete properties percent’s can higher or lower.
- External support such as FRP or Various concrete.

References: