A COMPARISON BETWEEN INCLINED STIRRUP AND U-STIRRUP BEAM-FINITE ELEMENT APPROACH

By

SWAPAN MATUBBER

MD. ALAMGIR HOSSAIN

MD. JOYNUL ABEDIN

MD. KAMRUL HASAN

MD. ALAUDDIN

SHAH MD. ARAFAT

A thesis submitted to the Department of Civil Engineering in partial fulfillment for the degree of Bachelor of Science in Civil Engineering

Department of Civil Engineering

Sonargaon University

147/I, Green Road, Dhaka-1215, Bangladesh

Section: 15A

Spring-2022

A COMPARISON BETWEEN INCLINED STIRRUP AND U-STIRRUP BEAM-FINITE ELEMENT APPROACH

By

SWAPAN MATUBBER (ID: BCE 1503006209)

MD. ALAMGIR HOSSAIN (ID: BCE 1803015084)

MD. JOYNUL ABEDIN (ID: BCE 1803015143)

MD. KAMRUL HASAN (ID: BCE 1803015166)

MD. ALAUDDIN (ID: BCE 1803015175)

SHAH MD. ARAFAT (ID: BCE 1602008194)

Supervisor
MD.FAZLA RABBI ANIK
LECTURER

A thesis submitted to the Department of Civil Engineering in partial fulfillment for the degree of Bachelor of Science in Civil Engineering

Department of Civil Engineering

Sonargaon University

147/I, Green Road, Dhaka-1215, Bangladesh

Section: 15-A

Semester -Spring-2022

BOARD OF EXAMINERS

The thesis titled “A COMPARISON BETWEEN INCLINED STIRRUP AND U-STIRRUP BEAM-FINITE ELEMENT” submitted by 1. SWAPAN MATUBBER, 2. MD. ALAMGIR HOSSAIN, 3. MD. JOYNUL ABEDIN, 4. MD. KAMRUL HASAN, 5. MD. ALAUDDIN, 6. SHAH MD. ARAFAT has been accepted as satisfactory in partial fulfillment of the requirement for the degree of Bachelor of Science in Civil Engineering on Date-of-Defense.

1. Md.Fazla Rabbi Anik Chairman
Lecturer,

Sonargaon University

……………………………………
2. Internal / External Member Member

………………………………………
3. Internal / External Member Member

DECLARATION

It is hereby declared that this thesis/project or any part of it has not been submitted elsewhere for the award of any degree or diploma.

STUDENT NAME	STUDENT ID.	SIGNATURE
MD. SWAPAN MATUBBER	(ID: BCE 1503006209)	
MD. JOYNUL ABEDIN	(ID: BCE 1803015143)	
MD. ALAMGIR HOSSAIN	(ID: BCE 1803015084)	
MD. KAMRUL HASAN	(ID: BCE 1803015166)	
MD. ALAUDDIN	(ID: BCE 1803015175)	
SHAH MD. ARAFAT	(ID: BCE 1602008194)	

Dedicated

to

“Our parents”

ACKNOWLEDGEMENTS

We are forever grateful to our Thesis Supervisor, Md. Fazla Rabbi Anik for his kind supervision and guidance throughout this important study and also with the continuous help from the beginning to the end of this study the work finally becomes positive to complete successfully.

We are grateful to our parents for their help & kind participation while preparing this Thesis book. This encouragement, helping hand, affection and sacrifice both directly and indirectly helped to reach on the today's position, without their love, appreciation and encouragement it was impossible to reach here. Acknowledgement of their contribution is too small as a reward.

The satisfaction and euphoria that accompany the successful completion of the task would be great but incomplete without the mention of the people who made it possible with their constant guidance and encouragement crowns all the efforts with success. In this context, we would like to thank all the other staff members, both teaching and non-teaching, which have extended their timely help and eased our task.

ABSTRACT

In this present study, a numerical analysis has been performed to compare the shear resisting capacities between conventional loop stirrup beams and inclined stirrup beams. Resisting the shear force is one of the key features of stirrups. A rarely used and uncommon inclined stirrup setup is analyzed in this study and new findings are compared with loop stirrup results. Two sets of beams (150x300x1960mm) are analyzed to determine the ultimate shear force resisting capacities, displacement, and stress resistance capacities. It was seen that the beam with an inclined stirrup showed ultimate shear resistance up to 187.24 KN compared to the beam with a loop stirrup setup which was 114.24 KN. Displacement was found in the inclined stirrup setup and the loop stirrup setup was 2.38 mm and 4.21 mm. Stress resistance and distribution pattern was also improved on inclined stirrup setup. Flexural and Shear Crack patterns of the beams at the ultimate load are also predicted from numerical analysis. Numerical study has been carried out by using Finite Element Software ABAQUS. Finally, these results were also compared with the theoretical formula available in the literature to justify the numerical analysis.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 General

In the field of structural analysis today, numerical analysis is using in a very large scale by simulating the result of a singular section or for a full model structure. Numerical analysis takes comparatively less time and is cost effective than experimental study. Moreover, finite element (FE) analysis can predict the experimental behavior and isolate the contributions of the individual elements of RCC Beam. Studies on RCC beam using FE analysis varying different structural and materials parameters of RCC beam are very limited.

For the analysis of RCC beam under a concentric load in 3D model and simulating the results with side-by-side comparison with experimental model, the most usable and realistic finite element model (FEM) analysis software is CAE simulation software. Through this, we can easily build up the 3D model of any desired section of the structure regarding the support condition and also can apply the service load such as concentric load in the section. Again, the effect of axial load such as earthquake load, wind load also can be simulated in FEM. Fire effect and impact load can be simulated in this platform with a statement of very high accuracy in this software. The most important application of this software is its visual representation. It represents the simulation result with a very easily understandable visualization with respect of color accuracy with value result. By this, it can be easily overviewed a full structure at a glance and find the most crucial and effective part/corner of the structure. This kind of simulation is very much useful in the respect of time. When a structure/section goes in service, after a certain time how will the structure/section will behave, it can be seen in the numerical analysis software. In this study, RCC beam in various formation are built up in FEM platform to analysis their behavior.

For analysis a section by FEM, it is divided into modules, where each module defines an aspect of the modeling process. For example, defining the geometry, defining material properties, and generating a mesh. As you move from module to module, each module contributes keywords, parameters, and data to form an input file that you submit to the finite element solver for analysis. For example, you use the Property module to define material and section properties and the Step module to choose an analysis procedure; the CAE postprocessor is called the Visualization module. From the visualization, we can find the value of any particular node of that model we want to know the deflection or displacement under the applied load. Systems.

In this study, FEM software is used to define the beam model and for their parametric studies further. A simply supported RCC beam is built up as model in the platform and concentric load was applied. The beam model differs in the respect of the concrete damage behavior, the steel ratio and the placement of stirrups in the beam.

1.2 Objective & Scope of the study

The main objectives addressed by this study are to-

1) To compare the model's ultimate strength capacity by varying of orientation of stirrups (closed, inclined, u-stirrup).

2) To compare the model's Stress capacity by varying of orientation of stirrups (closed, inclined, u-stirrup).

3) To predict the cracking patterns of the models.

1.3 Organization of the thesis

The report of the analysis is organized in this paper to represent and discuss the results and findings that come out from the studies performed by FEM of RCC beam in most convenient way with a comparison of higher strength established model.

Chapter 1 introduces the topic, in which an overall idea is presented before entering into the main studies and discussion.

Chapter 2 is literature Review, which represents the work performed so far in connection with it and is collected from various references. It also represents the strategy for moving forward with the success current issue.

Chapter 3 is all about the finite element modeling exclusively used in this study analysis and it also shows some figures associated with this study for proper presentation and understanding. It also describes the type of element selected, the shape of the mesh, the mechanical properties of the steel and concrete materials, and the boundary conditions implemented in the finite element model.

Chapter 4 is concerned with the verification of the model developed in Chapter 3 with reference to a theoretical study problem carried out by ACI Standard [2] and detailed description of the finite element model for the FEM beam is provided in Chapter 3, together with the nature of the test samples from the published literature.

Here numerical simulation results are provided to validate the development of finite element models under concentric loading conditions. Discussions are included on the comparison between the theoretical and numerical failure modes, peak stress, force versus time deformation curves, for different test groups.

It also presents the detailed parametric study conducted with the developed finite element model to cover the range of several geometric and material parameters on the behavior of FE beams.

Chapter 5 is about the findings of this parametric study are demonstrated and discussed in detail with a recommendation of further study scope and a conclusion.

CHAPTER 2

Literature Review

2.1 Introduction

Concrete frame structures are a very common type in modern buildings or may be the most common type internationally. As the name suggests, this type of building consists of a concrete frame or skeleton. The horizontal members of the frame are called beams and the vertical members are called columns. People walk on a flat concrete plane called a slab. Among them, the pillar is the most important because it is the main load element of the building. If the beam or floor in the building is damaged, it will only affect one floor, but damage to the column may damage the entire building.

Numerical Analysis is the study of algorithms that deals with the development and use of numerical methods for solving problems and it focuses on creating, analyzing, and implementing algorithms for solving the problems of continuous mathematics (Atkinson, 2007). In the field of structural analysis today, numerical analysis is using on a very large scale by simulating the result of a singular section or for a full model structure. Apart from that, it was seen from the previous research studies that Numerical analysis takes comparatively less time and is usually cost-effective than experimental study. The Finite Element Analysis (FEA) is one of the most used methods in Numerical Analysis. Perhaps it is known as the operation of simulating the behavior of a part under the provided prerequisites to assessing it by using the Finite Element Method (FEM). Moreover, Finite Element (FE) analysis can predict the experimental behavior and isolate the contributions of the individual elements of RCC Beam.

Reinforced Cement Concrete (RCC) Beam is a structural member, and it carries all types of vertical loads and resists it from bending. A RCC beam is a composite solid in which the upper part takes compression and the lower part take tension. Lower tensile strength and ductility of the concrete are counterbalanced by the addition of reinforcement which has great ductility and tensile strength. Moreover, Reinforced structures are majorly designed to survive tensile and shear stresses in specific regions of concrete that can be the reason of unwanted cracks and structural failures. In RCC structure, how a beam section will behave and how its strength will vary with the differentiation of its property is important. It is significantly a very time-consuming and costly procedure to cast beams under different types of loads such as concentric and eccentric loads with various materials properties. That's why the numerical modeling by finite element analysis (FEA) is encouraged nowadays.

A Beam fails in shear when the incoming shear stress crosses the permissible design limit of shear stress which a concrete can sustain. That's why now RC beam is treated with shear steel to resist the shear failure by adding ductility. A lot of studies were done to find the working principal of shear of the concrete. But still now cracking mechanisms of the concrete due to shear are not fully identified (Saravanakumar & Govindaraj, 2016). Meanwhile in flexural a member longitudinal strain is much smaller compared to the diagonal strain; that is why the shear cracks width is wider compared to flexural cracks (Adebar & Leeuwen, 1999) (Adebar, 2001). Diagonal cracks generated besides flexural zone in RC beam are the indication shear failure. Wider crack pattern is the differentiating point between shear cracks and flexural cracks and these cracks happened in the shear zone closer to the supports because shear force value is much higher near to the supports. The development of these shear cracks is so rapid and propagate without notice and cause sudden failure of the members. Shear reinforcement can lower this unexpected catastrophe and can also enhance the member ductility (Moayyad & M., 2013). Before the occurrence of diagonal crack, concrete was supposed to resist all the total shear alone. Redistribution of stress and debonding both occurs in concrete and shear steel after the diagonal crack formation. (Theodor, 1992) (Michael & Daniel, 1999) (Young-soo, et al., 1996). In RC beams the diagonal width of the cracks was influenced by stirrup ratio and bonding between concrete and stirrups (Witchukreangkrai, 2004) (Witchukreangkrai, 2006). It was observed that the bonding between stirrups and concrete was increased significantly by decreasing stirrups spacing. Vertical shear reinforced beams have greater shear cracks width than inclined shear reinforced beams (Zakaria, et al., 2009), (Zakaria, et al., 2011), (Hassan, 1985). Major shear crack load is caused by aggregate interlocking force so for resisting these shear forces a RC structure must have minimum shear reinforcement. (Songkram, 2013) (Sato, 2004). Inclined stirrups provided the most effective solution over conventional beam having high shear at distributed area (Saravanakumar & Govindaraj, 2016) (Suhaim,

2015) (Colajanni & n.d., n.d.). Inclined stirrups effectively enhanced the shear capacity of RC beam rather than loop stirrup beam (Zamri, et al., 2018).

In this study basically the strength behavior of RCC Beam will be discussed. When we say concrete in the construction industry, it actually refers to reinforced concrete. Its full name is Reinforced Cement Concrete or RCC. RCC is concrete that contains steel bars (called steel bars or steel bars). This combination works very well because the compressive strength of concrete is very high, it is easy to produce on site and is economical, and the tensile strength of steel is very strong. To make reinforced concrete, first create a mold called a template that will contain liquid concrete and give us the shape and shape we need. Then, the person looks at the drawings of the structural engineer and places them in the reinforcement and uses steel cables to tie them in place. Packaged steel is called a reinforcing cage because it has the shape of one. Once the steel is in place, it is time to begin preparing concrete by mixing cement, sand, stone chips of various sizes and water in a concrete mixer, and then pouring liquid concrete into the formwork right after mixing up to the level. Reach the level. Concrete hardens in a few hours, but it takes about a month to achieve maximum strength. Therefore, it usually remains at rest until this period. During this time, the concrete must harden by immersing under water to its surface, which is necessary for a normal chemical reaction within the concrete mixture.

Creating the exact “recipe” or ratio of each ingredient is a science in itself. This is called a concrete mix. A good mix design will start with the quality of the mix, then consideration of many factors and develop a detailed mix design. Field engineers typically order different types of mixtures for different purposes. For most construction applications, however, a standard mix is used. Therefore, the structure is actually a framework for connecting members, and each member is firmly connected. There are other types of joints, including articulated joints for steel structures, but the concrete frame structures have a 99.9% bending connection [3]. This frame becomes very strong, and must resist the various loads that act on a building during its life.

In a frame structure, beam section in a floor can be joined through multiple columns or can be joined as simply supported or also can be joined as T-beam. There are various types of loads are applied in a structure.

These include:

Static load: The downward force on a building due to the weight of the building itself, including structural elements, walls, and exterior walls.

Live load: The downward force on a building due to the expected weight of the passenger and his property (including furniture, books, etc.). These loads are usually specified in building codes, and structural engineers must design buildings to transport these or larger loads. These costs will vary with the use of space.

Dynamic Loads: these occur commonly in bridges and similar infrastructure due to the loads created by traffic, including braking and accelerating loads.

Wind Loads: This is a very important design factor, especially for tall buildings, or buildings with large surface area.

Earthquake Loads: in an earthquake, the ground vigorously shakes the building both horizontally and vertically, rather like a bucking horse shakes a rider in the sport of rodeo. This can cause the building to fall apart. The heavier the building, the greater the force on it.

In these loads' combination, in a structure at first, the load will be passed in the slab in frame, from slab it will be passed in the beam section on the floor and then column and from them to foundation and at last the load will be carried out into the soil beneath the foundation. In RCC structure, how a beam section will behave and how its strength will vary with the differentiation of its property is important. It is significantly very time consuming and costly procedure to cast beams under different types of loads such as concentric and eccentric loads with various materials properties. That's why the numerical modeling by finite element analysis (FEA) is encouraged now a days. In these model, RCC beam models are built up with same consideration of experimental procedure of laboratorial casting of beam. In these model different types of load combination can be applied in FEM platform. In this study, static concentric load was applied in the particular models for analyzing their moment and shear capacity with respect of their different material property.

Grade Beams and Floor beams are sometimes monolithically casted with slab or column of the floor. Beams are generally second load bearing element in a frame structure system. A typical frame structure and position of beams are given below-

Fig 2.1: Beam Sections in a frame Structure

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

A more detailed discussion on Reinforced Cement Concrete (RCC) materials and their use in beam sections, along with recent research advances in this field will follow.

2.2 RCC Beam Materials

Reinforced concrete (RC) (also known as reinforced cement concrete or RCC) is a composite material in which the lower tensile strength and ductility of the concrete are neutralized by the addition of reinforcing materials with greater tensile strength and ductility. As reinforcing material, steel bars are generally used that are passively buried before solidifying. Reinforcement schemes are generally designed to withstand tensile stresses in specific areas of concrete that can cause unacceptable cracks and structural damage. Modern reinforced concrete can contain a variety of reinforcements made of steel, polymer or other composite materials, whether or not used with steel bars. Reinforced concrete can also undergo permanent stress (compressed concrete, elastic concrete) to improve the performance of the final structure under the workload.

2.2.1 Concrete

Concrete is a building material consisting of cement, fine aggregate and coarse aggregates, as well as water that hardens over time. Portland cement is a type of cement commonly used to produce concrete. Concrete technology involves studying the properties of concrete and its practical application. In building construction, concrete is used to build foundations, columns, beams, slabs and other transport elements.

2.2.2 Steel Bars

Reinforced steel, referred to as steel or steel in galvanizing, is a steel or steel mesh used as a tensile tool in reinforced concrete and concrete structures. Reinforced masonry to reinforce and help to pull the concrete. Concrete is strong under compression but has weak tensile strength. Reinforcing steel significantly increases the tensile strength of the concrete and thus the structure. The surface of the steel bars is often deformed to facilitate better bonding with the concrete.

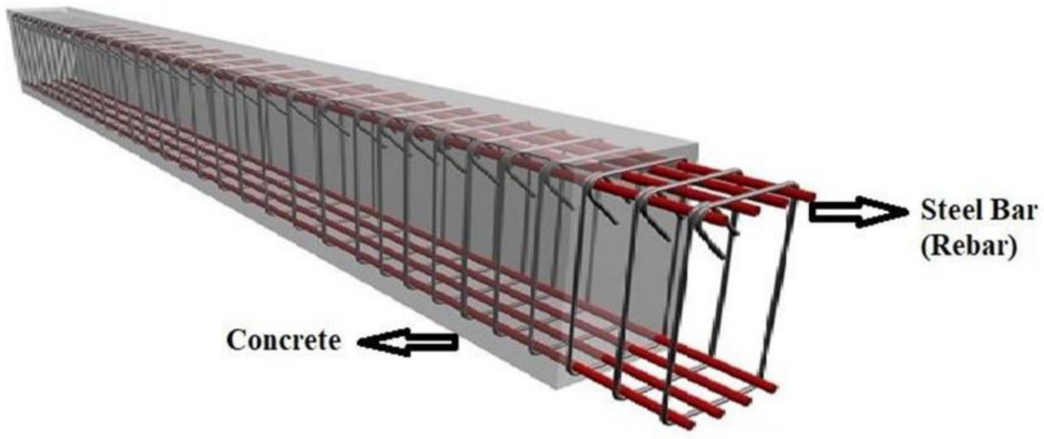


Fig 2.2: Reinforced Cement Concrete (RCC) Beam

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

2.3 Properties And Behavior of RCC Materials

Every chapter, except the Introduction and Conclusions chapters, may contain a summary. The summary section should summarize key points of the chapter and act as a bridge between the current chapter and the following chapter. The summary section can be 4-5 sentences long.

2.3.1 Tensile Behavior of Steel

Stress: The stress is the ratio of the force applied to the axial region, defined as “force per unit area”. There are some types of stress such as,
tensile stress – stress that tends to stretch or lengthen the material and acts normal to the stressed area
compressive stress – stress that tends to compress or shorten the material and acts normal to the stressed area
shearing stress – stress that tends to shear the material and acts in plane to the stressed area at right-angles to compressive or tensile stress

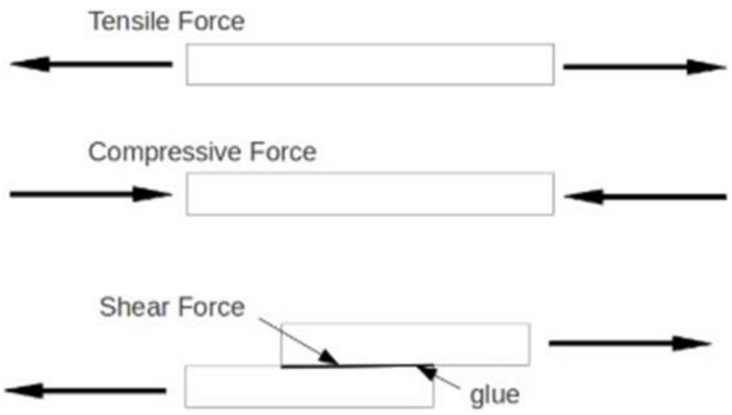


Fig 2.3: Types of stresses
(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Tensile strength is different for different varieties of steel. There are three types of tensile strength:

Yield Strength – The strength up to which a material can withstand loads without constant deformation.

Ultimate Tensile Strength – The maximum tensile stress encountered by a material without failure is called Ultimate tensile strength.

Fracture Strength – Tensile strength is the stress at which a material breaks or ruptures.

Strain is defined as “deformation of a solid due to stress”.

Normal strain – elongation or contraction of a line segment

Shear strain – change in angle between two-line segments originally perpendicular.

Normal strain and can be expressed as

$\varepsilon = \Delta l / lo = \sigma / E$ (i)

Where,

Δl = change of length (m, in) lo = initial length (m, in)

ε = strain – unit-less

E = Young’s modulus (Modulus of Elasticity) (Pa, (N/m2), psi (lbf/in2))

Young’s modulus can be used to predict the elongation or compression of an object when exposed to a force.

Poisson’s ratio is the ratio of relative contraction strain. The Stress vs Strain curve of Steel can be represented as-

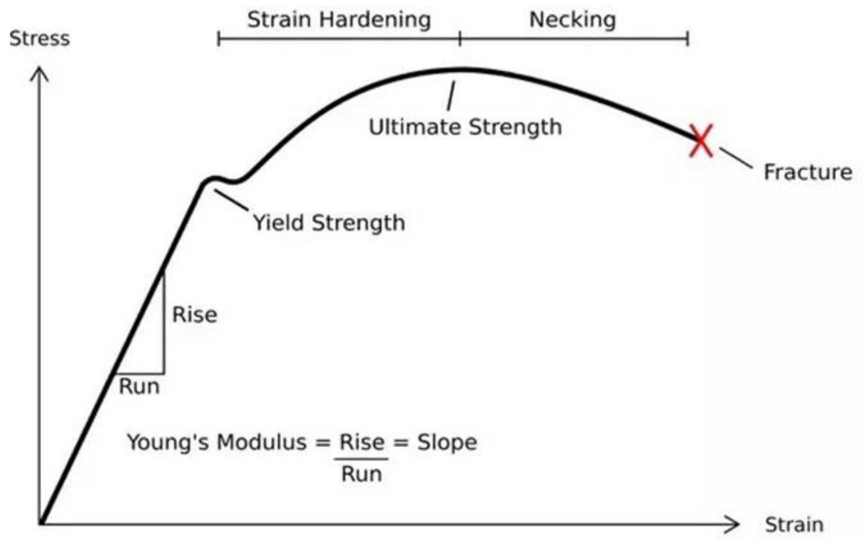


Fig 2.4- Stress vs Strain curve of Steel

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Steel bars have higher tensile strength due to its heavy ductility. The tensile strength of the structural steel is 400 Mpa, and the tensile strength of the carbon steel is 841 Mpa [4].

2.3.2 Compressive Behavior of Steel

Steel Materials are very strong in tension but comparatively weak in compression. The compressive strength of steel is not that much lower than its tensile strength. Maximum compressive strength of steel is 35-40 Mpa which is very lower than its tensile strength (250-400 Mpa). Stress vs Strain curve can be represented as

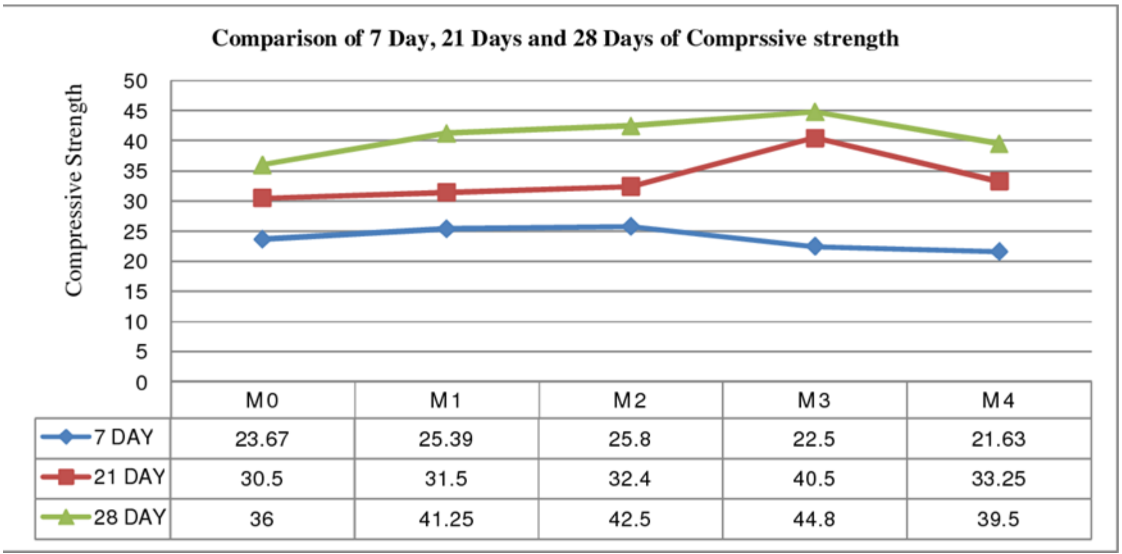


Fig 2.5- Comparison of 7 days and 28 days Compressive Strength with different % steel fiber

Fiber Description	Aspect Ratio	Compressive Strength (MPa)			Average Compressive Strength (MPa)
Steel fiber	25	40.89	41.78	41.30	41.33
	38	32.00	32.88	33.30	32.73
	50	28.44	27.11	29.30	28.29
GI fiber	25	40.00	39.56	39.60	39.70
	38	29.33	28.40	30.20	29.31
	50	26.20	26.66	26.70	26.46
Polypropylene fiber	800	36.89	37.33	37.80	37.33
	1600	24.00	24.40	24.00	24.13

Compressive Strength Test Results of SIFF with Steel Fiber, GI Fiber and Polypropylene Fiber for Different Aspect Ratio

Fig 2.6- Compressive Strength Test Results of SIFF with Steel Fiber, GI Fiber and Polypropylene Fiber for Different Aspect Ratio

2.3.3 Compressive Behavior of Concrete

Concrete mixtures can be designed to provide a wide range of mechanical properties and durability to meet structural design requirements. The compressive strength of concrete is the most common performance characteristic for engineers when designing structures. The compressive strength was measured by destroying a cylindrical concrete sample in a pressure tester. The compressive strength of the failure load is calculated by dividing the cross-sectional area by the load and expressed in pounds per square inch (psi) or megapascals (Mpa). For residential concrete, concrete compressive strength ($f'c$) ranges from 2500 psi (17 Mpa) to 4000 psi (28 Mpa), even higher in commercial buildings. Some applications use higher pressure, over 10,000 pounds (70 Mpa).

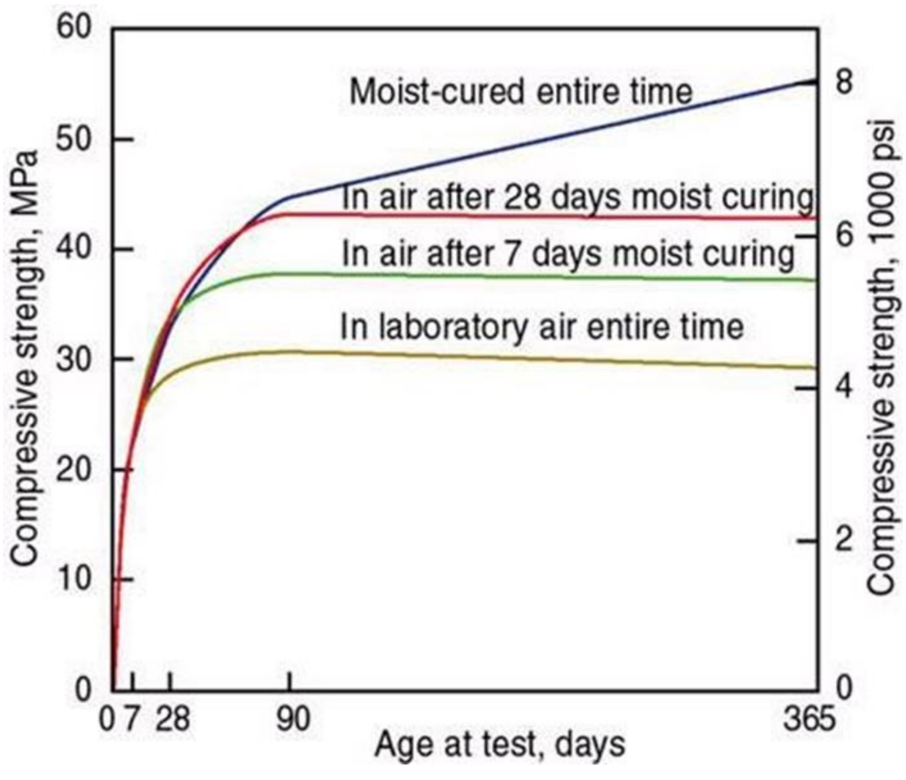


Fig 2.7- Maximum compressive strength of concrete

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Compressive strength results are used to ensure that the concrete mixture as delivered meets the requirements of the specified strength, $f'c$, in the job specification.

2.3.4 Tensile Behavior of Concrete

The tensile strength of concrete is fragile, but the compressive strength is relatively high. Σt (tensile strength) is approximately 1/10 of σc (compression force). In the water to cement ratio, the mortar is stronger than the corresponding concrete. Clay and cement aggregates are flexible and concrete is not. Maximum tensile strength fy is 4-6 Mpa for normally graded concrete, high strength concrete can carry up to 8-10 Mpa [7].

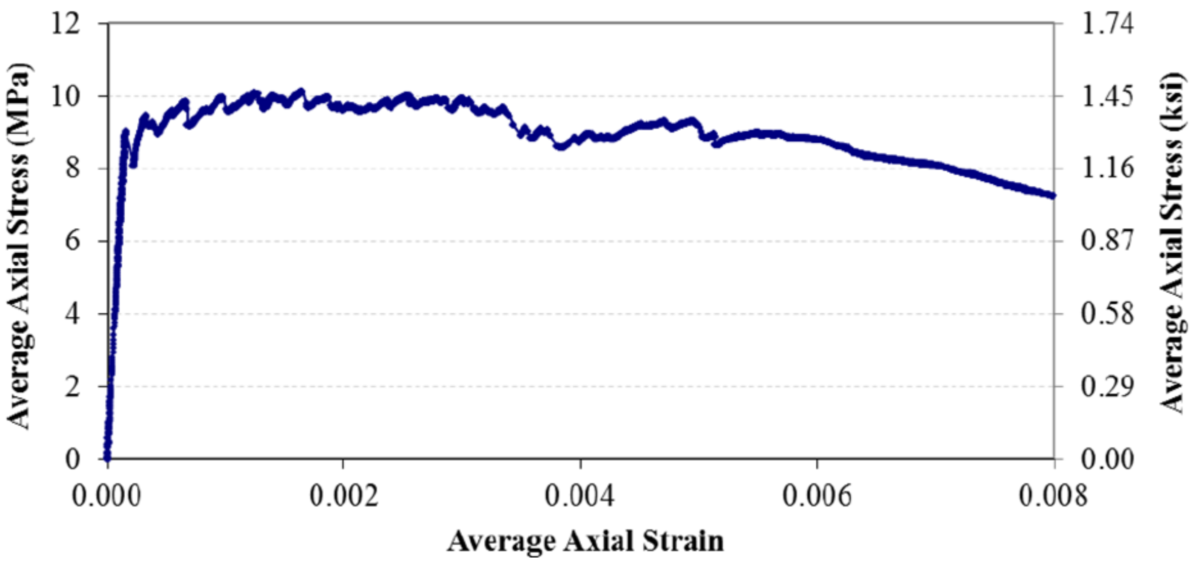


Fig 2.8- Maximum tensile strength fy

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

2.3.5 Mechanical Properties of Steel

Young's Modulus – or Young's Modulus alt. Modulus of Elasticity – is a measure of stiffness of an elastic material. It is used to describe the elastic properties of objects like wires, rods or columns when they are stretched or compressed.

Young's Modulus is defined as the "ratio of stress (force per unit area) along an axis to strain (ratio of deformation over initial length) along that axis"

It can be used to predict the elongation or compression of an object as long as the stress is less than the yield strength of the material. More about the definitions below the table.

Material	Tensile Modulus (Young's Modulus, Modulus of Elasticity) - E -		Ultimate Tensile Strength - σ_u -	Yield Strength - σ_y -
	(10^6 psi, Mpsi)	(GPa)	(10^6 Pa, MPa)	(10^6 Pa, MPa)
ABS plastics		1.4 - 3.1	40	
A53 Seamless and Welded Standard Steel Pipe - Grade A			331	207
A53 Seamless and Welded Standard Steel Pipe - Grade B			414	241
Concrete		17		
Steel, High Strength Alloy ASTM A-514			760	690
Steel, stainless AISI 302		180	860	502
Steel, Structural ASTM-A36	29	200	400	250

Fig 2.9: MOE, σu, σy of different materials

2.4 Failure Modes of RCC Beam

Basically, the reinforced concrete beams fail in two modes: flexural and shear. It is evident that in contrast with flexural failure mode which is a ductile one, the shear failure mode of concrete beam is sudden and with brittle nature. In addition, the time of occurrence is not easily predictable and shear failure does not present much warning before failure. Therefore, shear failure is more dangerous than flexural failure. As a result, the reinforced concrete beams should be designed in such a way that they can attain their total flexural capacity and under maximum loads, these beams must represent a ductile flexural failure mode.

There are also other types of failure can be happened in a beam without those two failures, such as [8],

1. FRP Rupture
2. Crushing of compressive concrete
3. Concrete cover separation
4. End interfacial debonding

Diagrams of various types of failure is given bellow-

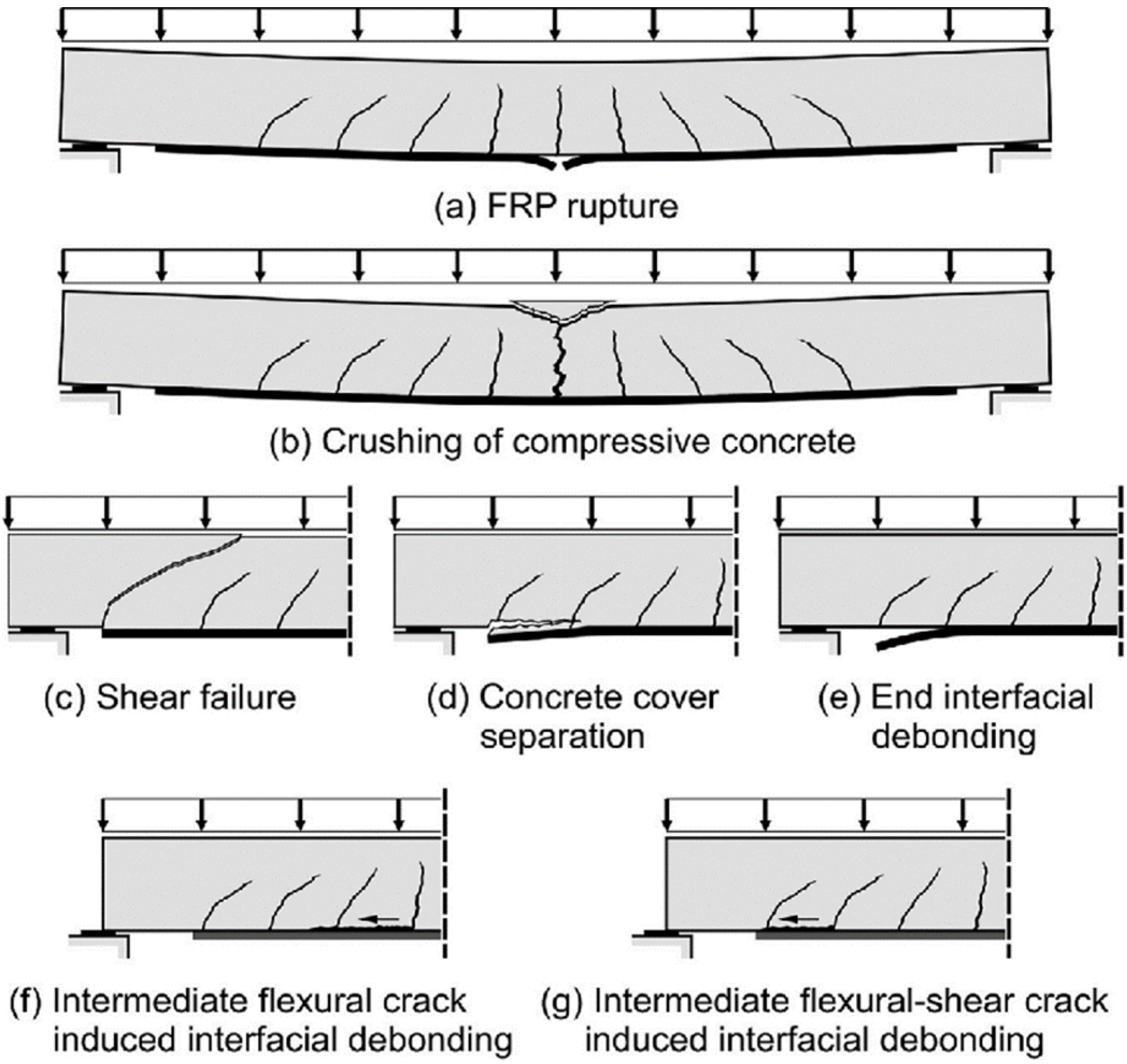


Fig 2.10- Various types of failure in beam

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Most of the concrete structures face problems in dealing with shear due to numerous reasons which include mistake in design computations, incorrect details of shear bar, and errors during construction and lack of their correction, changes in application of a structure with shift from lower service load to higher one, and reduction of shear bar due to corrosion under environmental conditions.

2.5 Concrete Damaged Plasticity

The plastic damage model of concrete is based on the assumption of scalar damage and is suitable for applications where concrete is subjected to arbitrary load conditions, including cyclic loading. This model takes into account the reduction in elastic stiffness caused by plastic stress during tensile and compression processes. This also explains the consequences of restoring stiffness under cyclic loading.

The concrete damaged plasticity model in FEM software:

1) Modeling provides universal modeling capabilities for all types of structures (beams, trusses, shells and concrete) made of concrete and other semi-brittle materials

2) It can be used with rebar to model concrete reinforcement

3) The concepts of isotropic tension and elasticity combined with compressive plasticity are used to represent the behavior of compressed plastic solids

4) It can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures

5) It can be defined to be sensitive to the rate of straining

The Drucker-Prager type potentials are commonly used to cover observed performance of concrete structures, compare among others. They characterize properly compressive behavior of the material though they fail to model its tensile properties. As an extension, the two-surface Drucker- Prager condition should be used to properly portray the compressive and tensile behavior of material.

Mechanical behavior

The model is a continuum, plasticity-based, damage model for concrete. It assumes that the main two failure mechanisms are tensile cracking and compressive crushing of the concrete material. The evolution of the yield (or failure) surface is controlled by two hardening variables, $\varepsilon{-}pl$
and $\varepsilon{-}pl$ linked to failure mechanisms under tension and compression loading, respectively. We refer to $\varepsilon{-}pl$ and $\varepsilon{-}pl$ as tensile and compressive equivalent plastic

strains, respectively. The following sections discuss the main assumptions about the mechanical behavior of concrete.

Uniaxial tension and compression stress behavior

The model assumes that the uniaxial tensile and compressive response of concrete is characterized by damaged plasticity, as shown in Figure 2.11

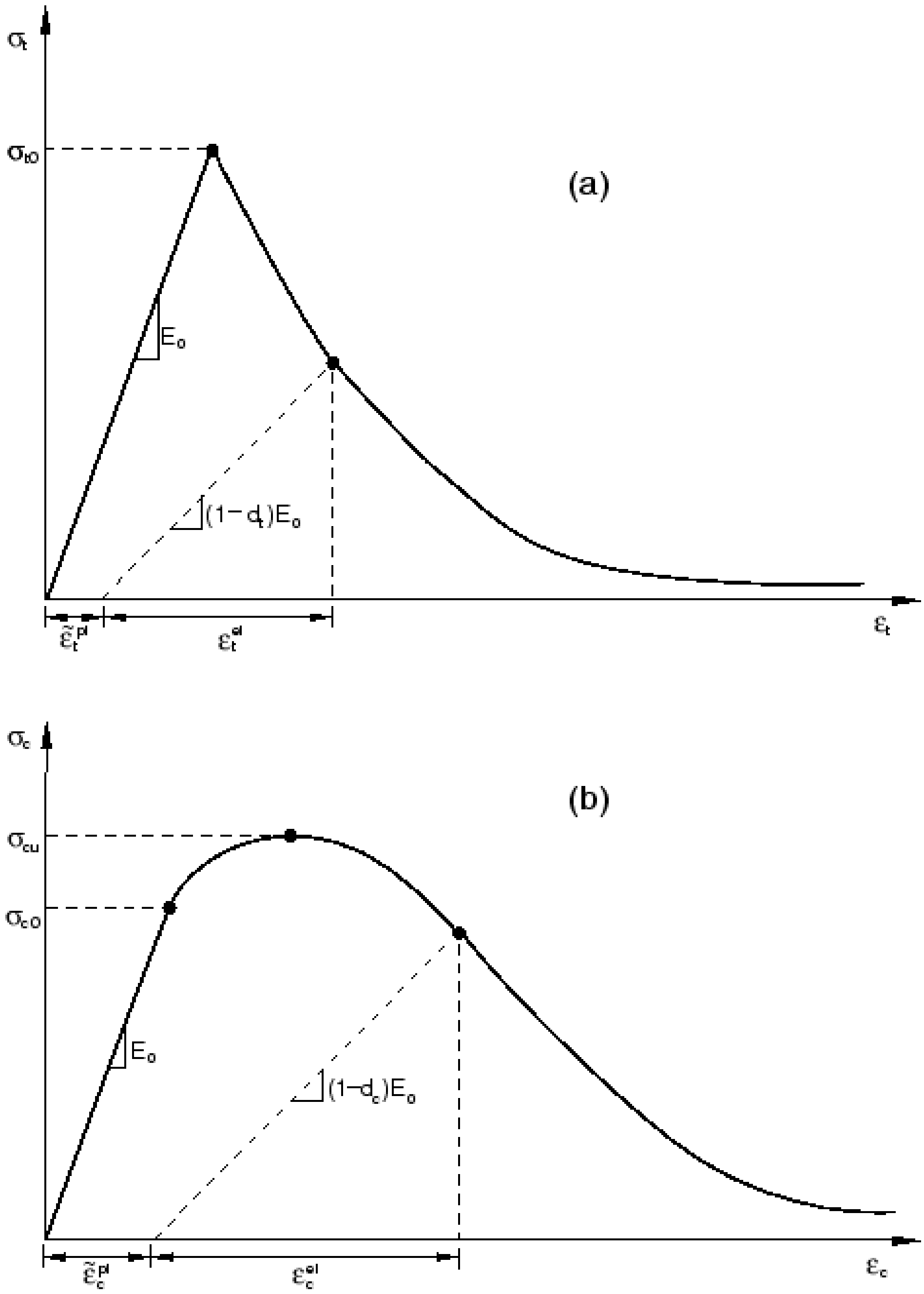


Figure 2.11. Response of concrete to uniaxial loading in tension (a) and compression (b).
(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Under uniaxial tension the stress-strain response follows a linear elastic relationship until the value of the failure stress, σt0, is reached. The failure stress corresponds to the onset of micro-cracking in the concrete material. Beyond the failure stress the formation of micro-cracks is represented macroscopically with a softening stress-strain response, which induces strain localization in the concrete structure. Under uniaxial compression the response is linear until the value of initial yield, σc0. In the plastic regime the response is typically characterized by stress hardening followed by strain softening beyond the ultimate stress, σcu. This representation, although somewhat simplified, captures the main features of the response of concrete .

It is assumed that the uniaxial stress-strain curves can be converted into stress versus plastic- strain curves. (This conversion is performed automatically by FEM from the user-provided stress versus "inelastic" strain data, as explained below.) Thus,

$\sigma t = \sigma t\ (\varepsilon_t^{-pl}, \varepsilon_t^{-pl}\ \theta, fi)$ ....(ii)

$\sigma c = \sigma c\ (, \varepsilon^{-pl}, \varepsilon^{-pl}\theta, fi)$ .... (iii)

2.5.1 Defining tension stiffening

The post failure behavior for direct straining is modeled with tension stiffening, which allows you to define the strain-softening behavior for cracked concrete. This behavior also allows for the effects of the reinforcement interaction with concrete to be simulated in a simple manner. Tension stiffening is required in the concrete damaged plasticity model. You can specify tension stiffening by means of a post failure stress-strain relation or by applying a fracture energy cracking criterion.

2.5.2 Post-failure stress-strain relation

In reinforced concrete the specification of post failure behavior generally means giving the post failure stress as a function of cracking strain, ε^{-ck}. The cracking strain is defined as the total strain minus the elastic strain corresponding to the undamaged material; that is, ε^{-ck} =

$\varepsilon_t - \varepsilon^{-el}_{0t}$ where, ε^{-el}_{0t}=σt/E0, as illustrated in Fig.2.12. To avoid potential numerical problems, Abaqus enforces a lower limit on the post failure stress equal to one-hundredth of the initial failure stress: σt≥σt0/100

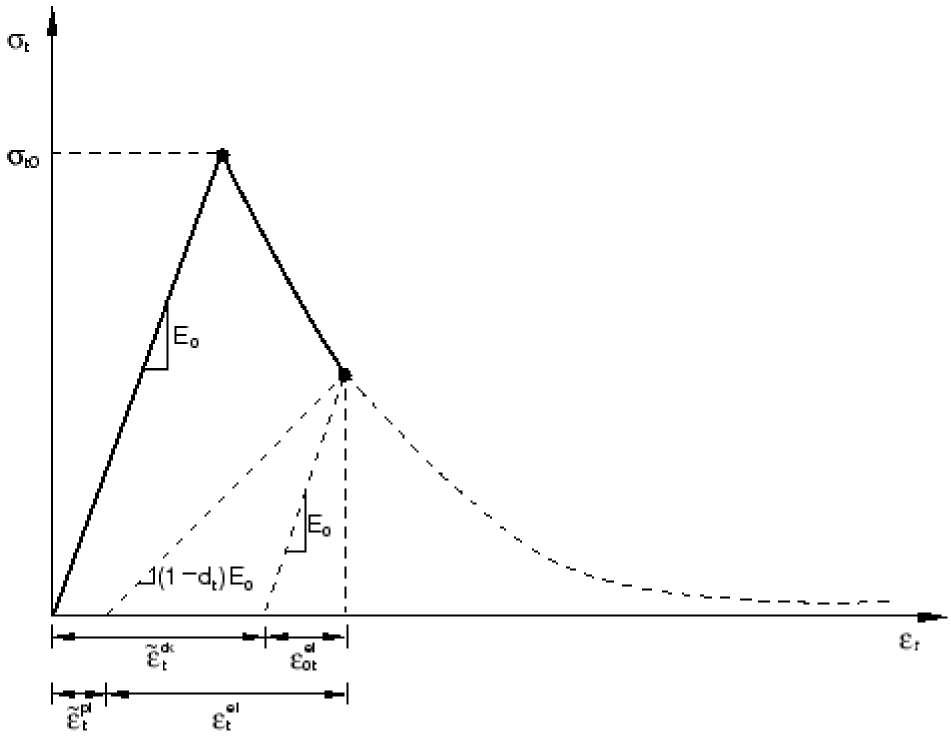


Figure 2.12. Illustration of the definition of the cracking strain ε^{-ck} used for the definition of tension stiffening data.
(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Tension stiffening data are given in terms of the cracking strain, $\varepsilon{-ck}$. When unloading data are available, the data are provided to Abaqus in terms of tensile damage curves, dt−$\varepsilon{-ck}$, as discussed below. Abaqus automatically converts the cracking strain values to plastic strain values using the relationship-

$$\tilde{\varepsilon}_t^{pl} = \tilde{\varepsilon}_t^{ck} - \frac{d_t}{(1-d_t)}\frac{\sigma_t}{E_0}. \quad \ldots\ldots\ldots\ldots\ldots\ldots \text{(iv)}$$

Abaqus will issue an error message if the calculated plastic strain values are negative and/or decreasing with increasing cracking strain, which typically indicates that the tensile damage curves are incorrect. In the absence of tensile damage $\varepsilon^{-pl} = \varepsilon^{-ck}$

In cases with little or no reinforcement, the specification of a post failure stress-strain relation introduces mesh sensitivity in the results, in the sense that the finite element predictions do not converge to a unique solution as the mesh is refined because mesh refinement leads to narrower crack bands. This problem typically occurs if cracking failure occurs only at localized regions in the structure and mesh refinement does not result in the formation of additional cracks. If cracking failure is distributed evenly (either due to the effect of rebar or due to the presence of stabilizing elastic material, as in the case of plate bending), mesh sensitivity is less of a concern.

In practical calculations for reinforced concrete, the mesh is usually such that each element contains rebars. The interaction between the rebars and the concrete tends to reduce the mesh sensitivity, provided that a reasonable amount of tension stiffening is introduced in the concrete model to simulate this interaction. This requires an estimate of the tension stiffening effect, which depends on such factors as the density of reinforcement, the quality of the bond between the rebar and the concrete, the relative size of the concrete aggregate compared to the rebar diameter, and the mesh. A reasonable starting point for relatively heavily reinforced concrete modeled with a fairly detailed mesh is to assume that the strain softening after failure reduces the stress linearly to zero at a total strain of about 10 times the strain at failure. The strain at failure in standard concretes is typically 10^{-4}, which suggests that tension stiffening that reduces the stress to zero at a total strain of about 10^{-3} is reasonable. This parameter should be calibrated to a particular case.

The choice of tension stiffening parameters is important since, generally, more tension stiffening makes it easier to obtain numerical solutions. Too little tension stiffening will cause the local cracking failure in the concrete to introduce temporarily unstable behavior in the overall response of the model. Few practical designs exhibit such behavior, so that the presence of this type of response in the analysis model usually indicates that the tension stiffening is unreasonably low.

2.5.3 Stiffness recovery

As discussed above, stiffness recovery is an important aspect of the mechanical response of concrete under cyclic loading. Abaqus allows direct user specification of the stiffness recovery factors Wt and Wc.

The experimental observation in most quasi-brittle materials, including concrete, is that the compressive stiffness is recovered upon crack closure as the load changes from tension to compression. On the other hand, the tensile stiffness is not recovered as the load changes from compression to tension once crushing micro-cracks have developed. This behavior, which corresponds to Wt=0 and Wc=1, is the default used by Abaqus. Figure 2.13 illustrates a uniaxial load cycle assuming the default behavior.

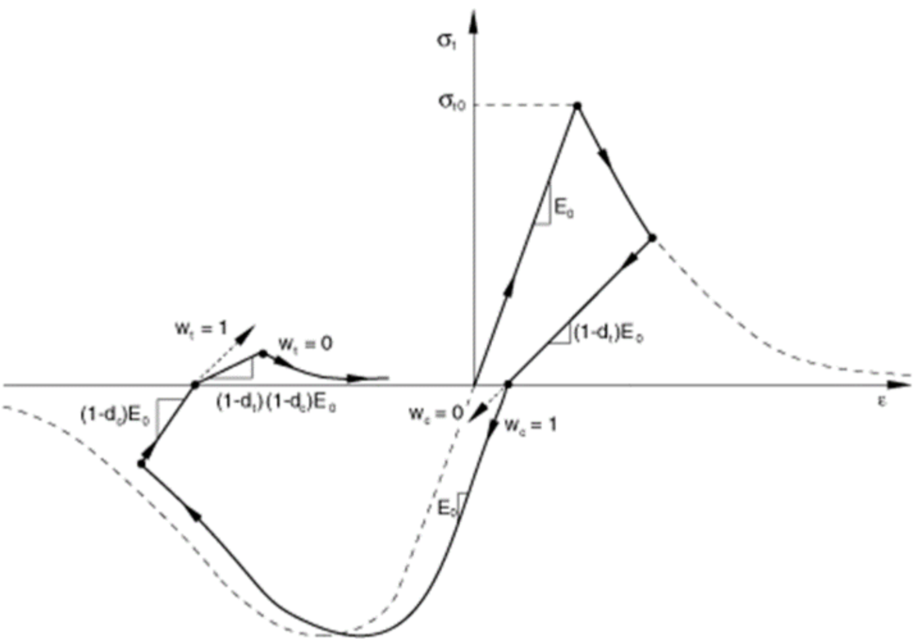


Figure 2.13. Uniaxial load cycle (tension-compression-tension) assuming default values for the stiffness recovery factors: Wt=0 and Wc=1

2.5.4 Yield function

The model makes use of the yield function of Lubliner et. Al. (1989), with the modifications proposed by Lee and Fenves (1998) to account for different evolution of strength under tension and compression. The evolution of the yield surface is controlled by the hardening variables, $\varepsilon - pl$ and $\tilde{\varepsilon} - pl$.

t c

In terms of effective stresses, the yield function takes the form

$$F = \frac{1}{1-\alpha}\left(\bar{q} - 3\alpha\bar{p} + \beta\left(\tilde{\varepsilon}^{pl}\right)\langle\hat{\bar{\sigma}}_{\max}\rangle - \gamma\langle-\hat{\bar{\sigma}}_{\max}\rangle\right) - \bar{\sigma}_c\left(\tilde{\varepsilon}_c^{pl}\right) = 0, \quad \text{.................. (V)}$$

with

$$\alpha = \frac{(\sigma_{b0}/\sigma_{c0})-1}{2(\sigma_{b0}/\sigma_{c0})-1};\ 0 \leq \alpha \leq 0.5, \quad \text{....................(Vi)}$$

$$\beta = \frac{\bar{\sigma}_c\left(\tilde{\varepsilon}_c^{pl}\right)}{\bar{\sigma}_t\left(\tilde{\varepsilon}_t^{pl}\right)}(1-\alpha) - (1+\alpha), \quad \text{....................(Vii)}$$

$$\gamma = \frac{3(1-K_c)}{2K_c-1}. \quad \text{.....................(Viii)}$$

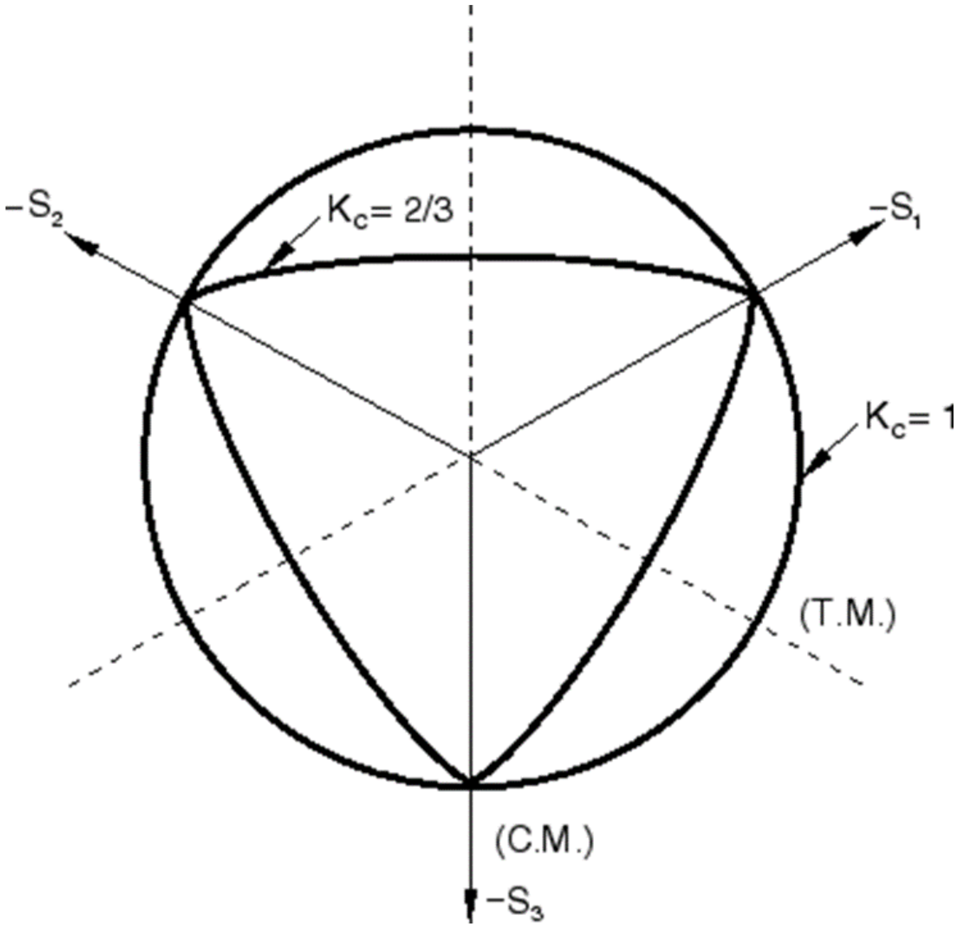


Figure 2.14. Yield surfaces in the deviatoric plane, corresponding to different values of Kc .
(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

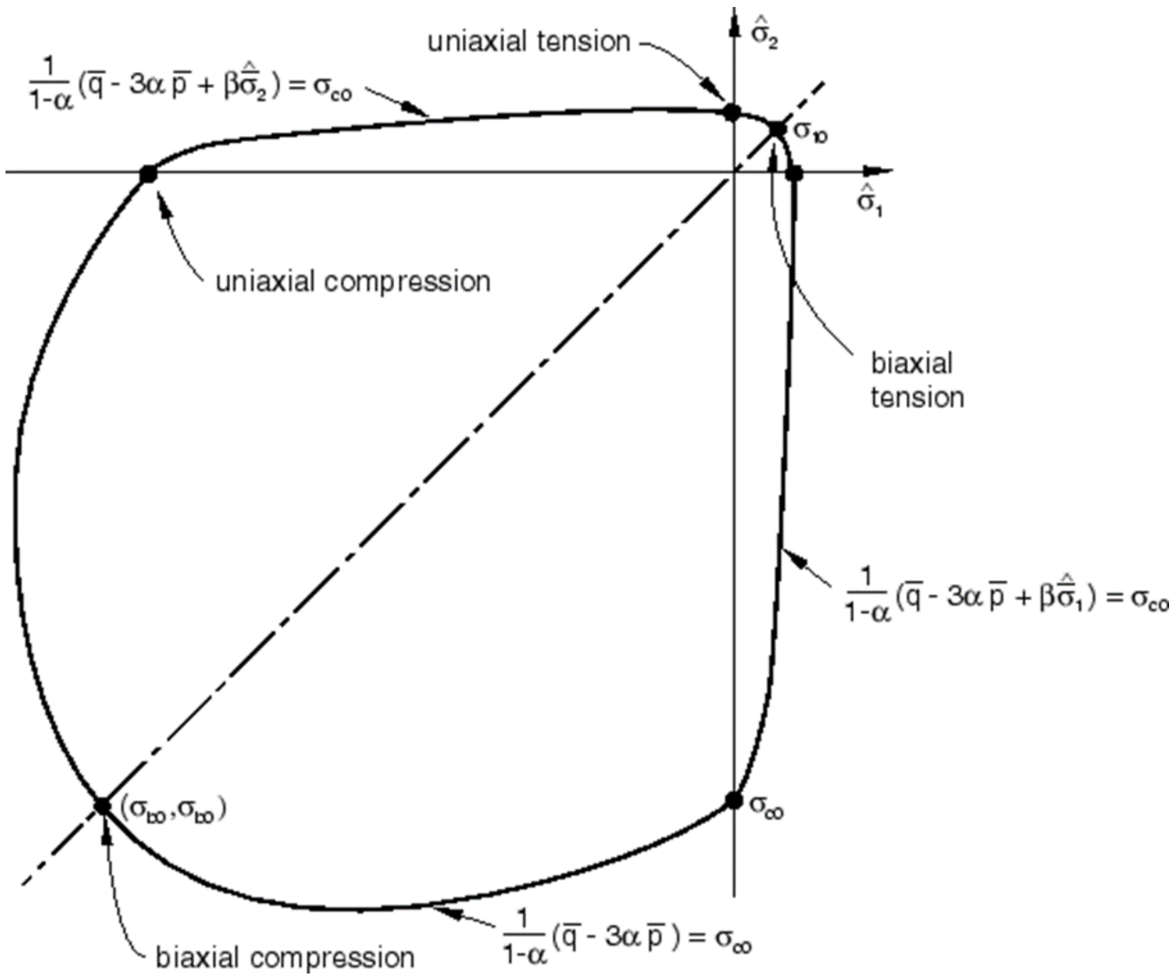


Figure 2.15. Yield surface in plane stress

In CDP Model of Abaqus platform, Reinforcement bars(rebar) are usually provided for reinforcing abacus in concrete structures. Rebars are one-dimensional bars that can be individually defined or integrated into a directional plane. Rebar is commonly used in metal-plastic models to describe the behavior of rebar materials and is attached to a standard type of grid used to model concrete. This modeling approach takes into account specific reinforcement-independent behavior. Effects such as slippage and the pin associated with the reinforcement-concrete interface are modeled by introducing “tensile strength” into the concrete simulation, simulating the transfer of load through the reinforcement through the reinforcement. Defining the rebar can be tedious in complex problems, but it is important that this be done accurately since it may cause an analysis to fail due to lack of reinforcement in key regions of a model.

CDP model can also be analyzed for cyclic load pattern, but in this study, CDP of a beam is only applied for static loading pattern to find out the behavior of tensile strength and compressive strength of RCC beam

2.6 Truss Element(T2D3) & Concrete C3D8 Model:

Truss element(T2D3)

Truss element(T2D3) are used in two and three dimensions to model slender, line-like structures that support loading only along the axis or the centerline of the element. No moments or forces perpendicular to the centerline are supported.

The two-dimensional truss elements can be used in axisymmetric models to represent components, such as bolts or connectors, where the strain is computed from the change in length in the r–z plane only. Two-dimensional trusses can also be used to define master surfaces for contact applications in Abaqus/Standard (see About contact interactions). In this case the direction of the master surface's outward normal is critical for proper detection of contact.

The 3-node truss element available in Abaqus/Standard is often useful for modeling curved reinforcing cables in structures, such as prestressed tendons in reinforced concrete or long slender pipelines used in the off-shore industry.

Eight-node brick element (C3D8)

The C3D8 element is a general-purpose linear brick element, fully integrated (2x2x2 integration points). The shape functions can be found in. The node numbering follows the convention of Figure 2.16 and the integration points are numbered according to Figure 2.17

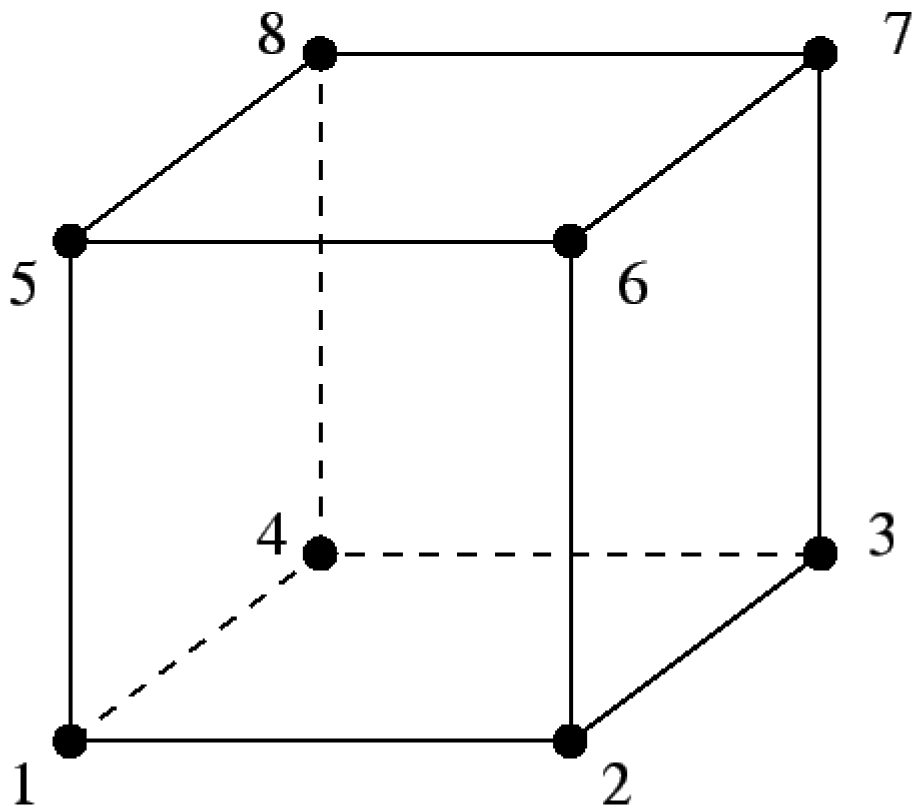


Figure 2.16-node brick element

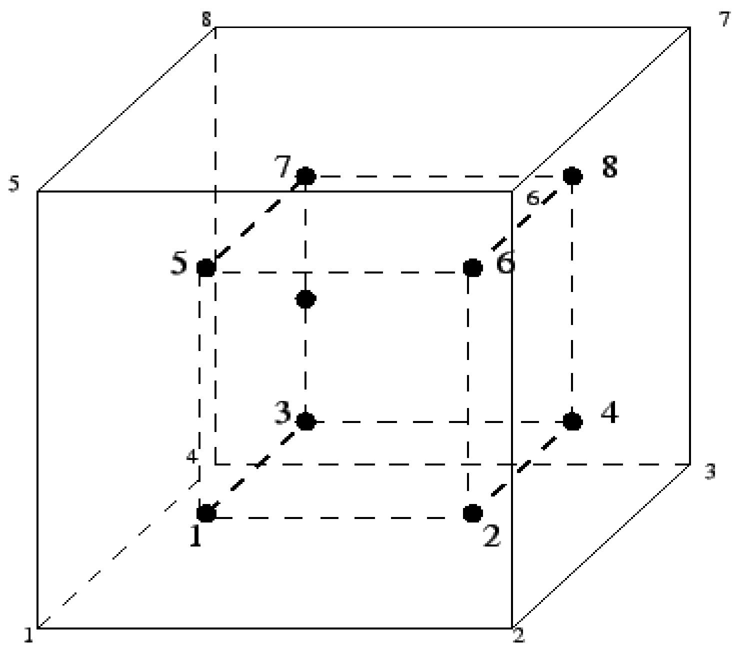


Figure 2.17: 2x2x2 integration point scheme in hexahedral elements.

(Source: https://images.app.goo.gl/SvMT6Tnt65wUnqa97)

Although the structure of the element is straightforward, it should not be used in the following situations:

- due to the full integration, the element will behave badly for isochoric material behavior,

  i.e. for high values of Poisson’s coefficient or plastic behavior.
- the element tends to be too stiff in bending, e.g., for slender beams or thin plates under bending.

2.7 Embedded Region Constrain

Constraints defined in the Interaction module define constraints on the analysis degrees of freedom, whereas constraints defined in the Assembly module define constraints only on the initial positions of instances. In the Interaction module you can constrain the degrees of freedom between regions of a model, and you can suppress and resume constraints to vary the analysis model.

In our study model, Embedded Constrain is used for fixing the host body under which the whole system works. An embedded region constraint allows you to embed a region of the model within a “host” region of the model or within the whole model.

The embedded element technique:

- is used to specify an element or a group of elements that lie embedded in a group of host elements whose response will be used to constrain the translational degrees of freedom and pore pressure degree of freedom of the embedded nodes (i.e., nodes of embedded elements);
- can be used in geometrically linear or nonlinear analysis;
- is not available for host elements with rotational degrees of freedom;
- can be used to model a set of rebar-reinforced membrane, shell, or surface elements that lie embedded in a set of three-dimensional solid (continuum) elements; a set of truss or beam elements that lie embedded in a set of solid elements; or a set of solid elements that lie embedded in another set of solid elements;
- will not constrain rotational degrees of freedom of the embedded

nodes when shell or beam elements are embedded in solid elements

Abaqus searches for the geometric relationships between nodes of the embedded elements and the host elements. If a node of an embedded element lies within a host element, the translational degrees of freedom and pore pressure degree of freedom at the node are eliminated and the node becomes an “embedded node.” The translational degrees of freedom and pore pressure degree of freedom of the embedded node are constrained to the interpolated values of the corresponding degrees of freedom of the host element.

In our model, embedded region constrain is applied for analyzing the specific nodal result of beam surface under compression and tension. Surface regions constrain is also applied for defining the support system of the beam.

2.8 Remarks

As far we discussed about the materials properties and the strength behavior the materials of RCC beam above, the modeling of beam for numerical analysis in Abaqus/CAE, all the parameters and the decisions are directly used from these literature review. In the model, in most of the case, exactly the same material properties and the mechanical properties are maintained due to gain the accuracy comparing to the experimental study. In this study, methodology of building beam model in the Abaqus/CAE were followed by the engineering knowledge of the concrete and steel properties. To fully simulate their behavior up to failure, numerical models which are capable of predicting the complexities of material nonlinearity, interaction between the steel and concrete surface are required. Therefore, an attempt has been made in current study to address these issues and thereby to develop a full-scale 3D finite element model for RCC strengthened beam under concentric loading. In the further study of this paper, it is showed that numerical model can adequately simulate the experimental results than it would be really helpful in the era of RCC structure strengthening.

CHAPTER 3
METHODOLOGY

3.1 Introduction

In this Chapter, a 3D finite model has been created to analyze the performance of Inclined stirrups over conventional stirrups on RC beams.

3.2 Geometric profile of the model

Total 2 beams had been analyzed. All the beams had the same dimensions of (150x300x1960) mm. Longitudinal and transverse profiles of the beams are shown in Fig 1 and Fig 2. Two 12 mm and two 8 mm diameter bars are used as longitudinal bars for modeling the beam. For loop stirrup beam (LSB) 8mm diameter bars are used as stirrups and placed perpendicularly to bottom bars with a spacing of 100 mm center to center. For inclined Stirrup beam (ISB), the same size bars were used for modeling the stirrups and placed them making an angle of 45 degree with the bottom reinforcement.

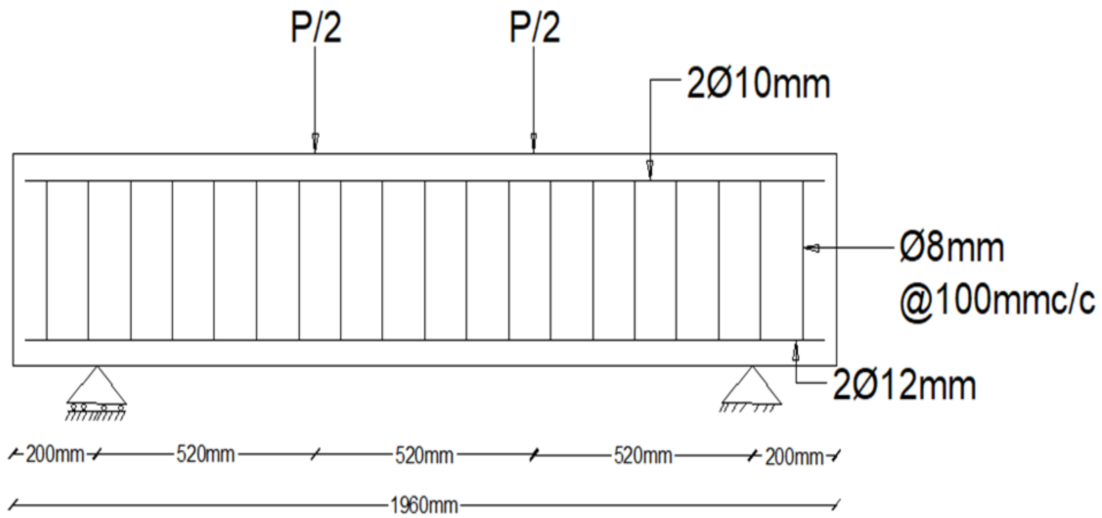


Fig 3.1: Longitudinal Profile of the loop stirrup beam (LSB).

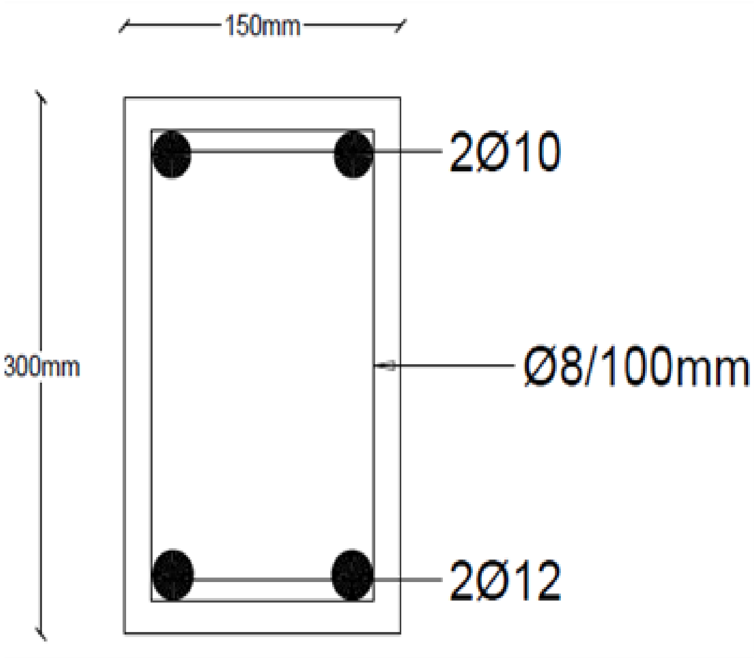


Fig 3.2: Transverse Profile of the loop stirrup beam (LSB).

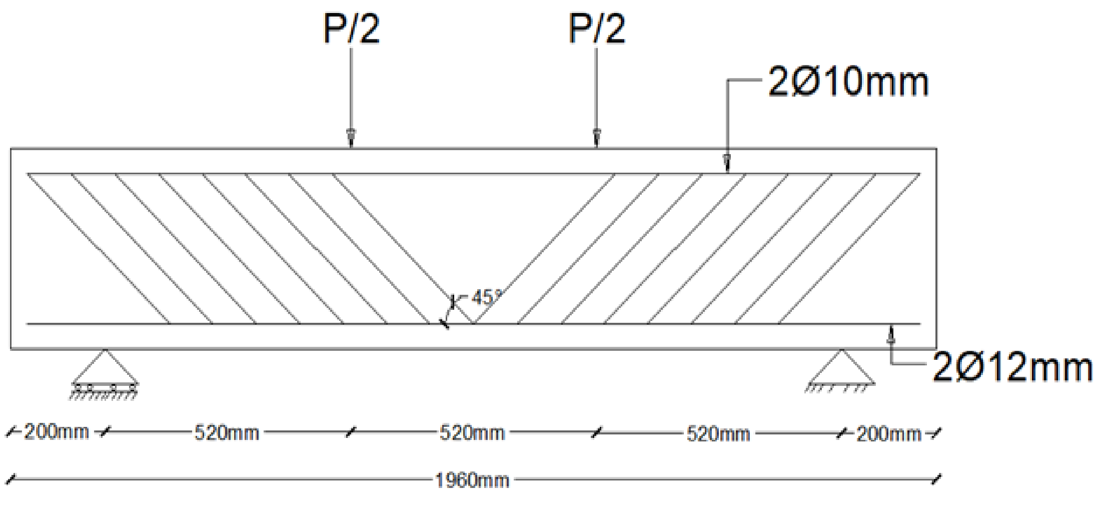


Fig 3.3: Longitudinal Profile of the inclined stirrup beam (ISB).

3.3 Material property of concrete used in numerical analysis

It was assumed that the concrete is homogeneous and initially isotropic. Concrete crushes under compression and cracks under tension. These are the two chief failure mechanisms of concrete. The Yield Stress-Cracking strain correlation in compression for concrete used in ABAQUS is characterized in Fig 3.4. Poisson's ratio of 0.20 was used for concrete.

Table 3.1: Material property of concrete used in numerical analysis.

Modulus Of elasticity, E (MPa)	26000
Poisson's ratio	0.20
Compressive strength, f'_c (MPa)	30
Tensile strength(f_t), MPa	1.81

Table 3.2: Concrete damage plasticity data used in numerical analysis.

Dilation Angle	31
Eccentricity	0.1
fb0/fc0	1.16
k	0.67
Viscosity parameter	0.00001

For tension, the stress-strain property trails a linear elastic connection till the value of the failure stress is reached. Stress-Strain properties of concrete in tension used in ABAQUS as shown in Fig 3.5 (Hafezolghorani, 2017). The failure stress relates to the onset of micro-cracking in the concrete material. Concrete damage plasticity (CDP) data are used in ABAQUS modeling to capture the cracking patterns of the concrete under compression and tension.

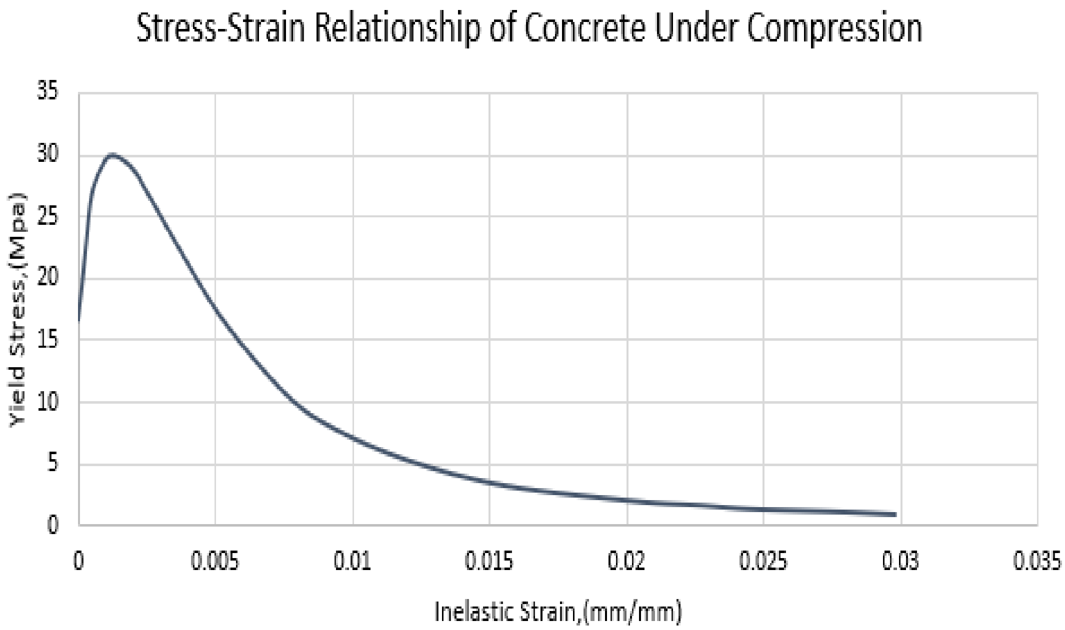


Fig 3.4: Stress-strain relationship for concrete under uniaxial compression used in FE analysis.

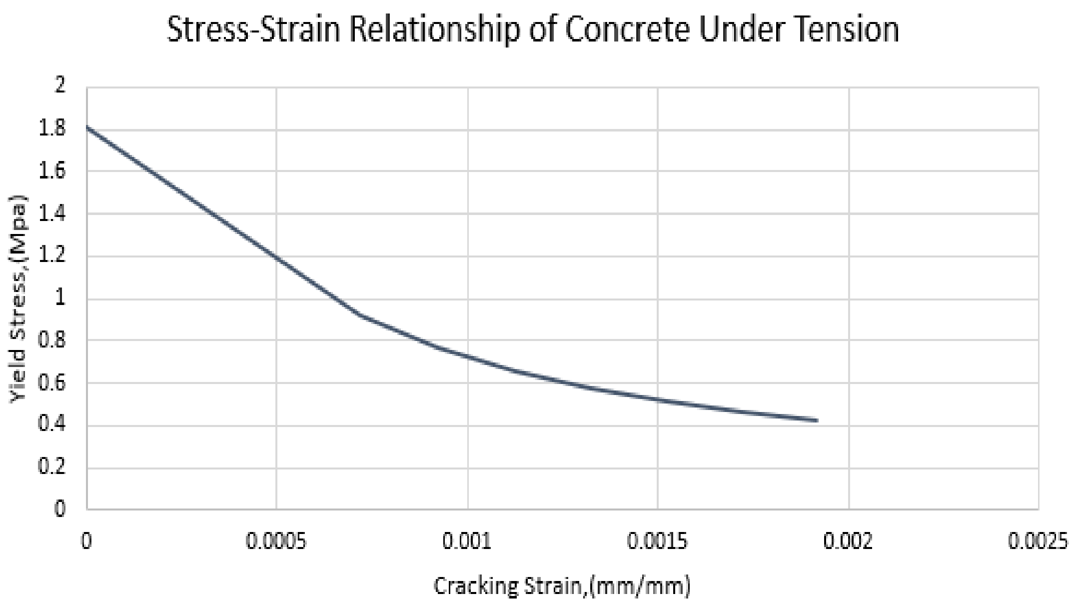


Fig 3.5: Stress-strain relationship for concrete under uniaxial compression used in FE analysis.

3.4 Material property of steel used in numerical analysis

Steel is supposed to be a perfectly elastic-plastic material that is similar in both tension and compression behavior. Poisson's ratio was used 0.3 as a property of the steel reinforcement in this study.

Table 3.3: Properties of reinforcing steel bars and stirrups.

Elastic modulus E (GPa)	209
Nominal Diameter (mm)	12 and 10
Yield Stress (N/mm2)	507
Poisson's ratio	0.3
Density (tone/mm^3)	7.8e-8

3.5 Model Development with ABAQUS

For Concrete beam modeling a 3D linear brick element(C3D8) with reduced integration and hourglass control was used. For general-purpose linear brick elements, the C3D8 element is used which has 2x2x2 integration points. The node numbering follows the way of Figure 6 and the integration points are numbered according to Figure 7. The solid element (C3D8) has eight nodes. All of the nodes have three degrees of freedom which is translations in the nodal x, y, and z directions (Mohamed, et al., 2019). Cracking in three orthogonal directions, Plastic deformation and crushing are the significant property of the element. (Systèmes, 2013).

The embedded steel bars were modeled using linear two-node truss elements(T3D2) showed in Fig 8. Each of the nodes has three degrees of freedom. Truss elements (T3D2) are capable of modeling slender, line-like structures that support loading only

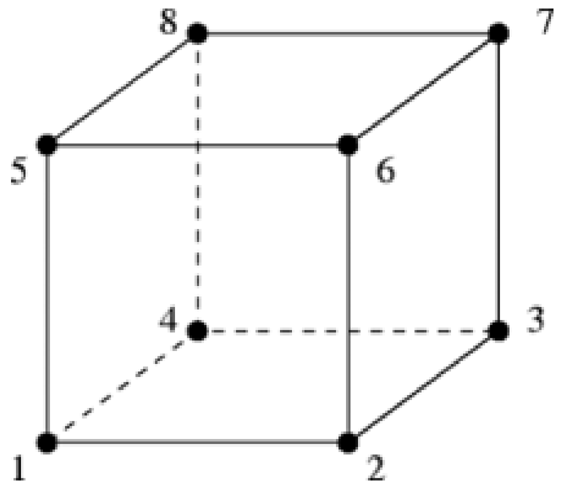


Fig 3.6: Nodes of the brick element.

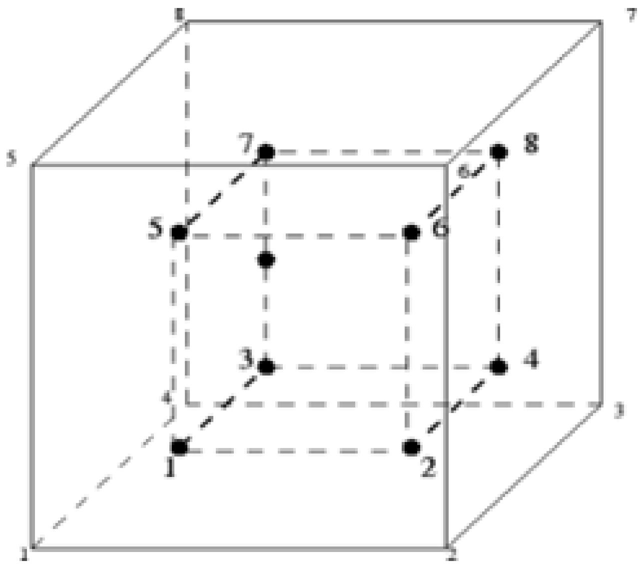


Fig 3.7: 2x2x2 integration point scheme in hexahedral elements.

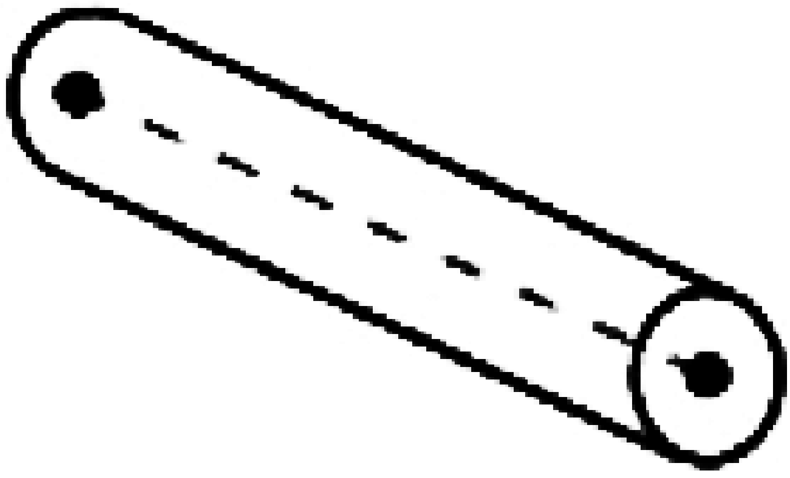


Fig 3.8: T3D2 truss element.

Necessary partitions are made on the 3D beam to facilitate load application and meshing. Embedded region constraint is used to make the bonding between the beam and reinforcement.

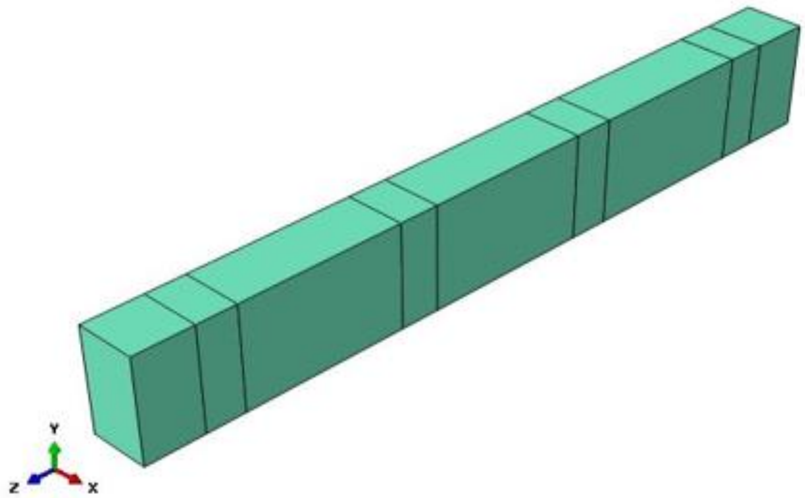


Fig 3.9: 3D-Solid beam developed for FE analysis.

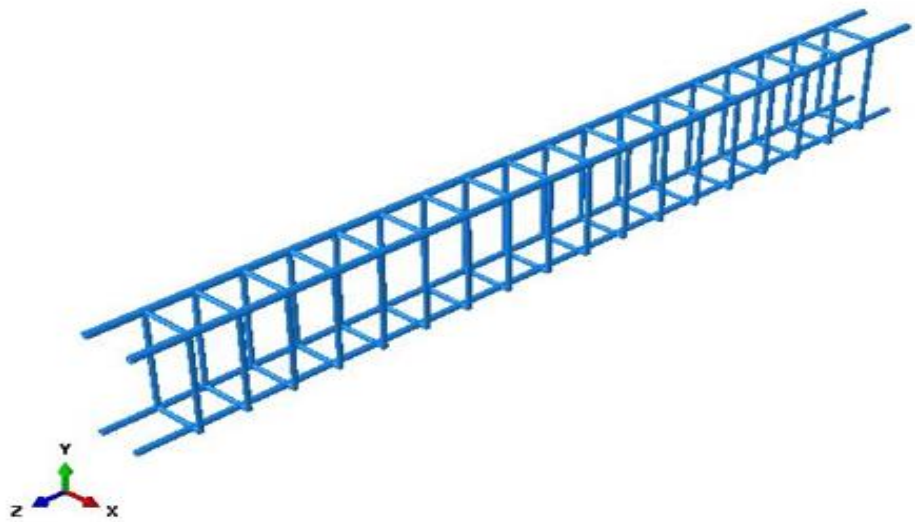


Fig 3.10: Reinforcement and Loop Stirrups 3D modeling.

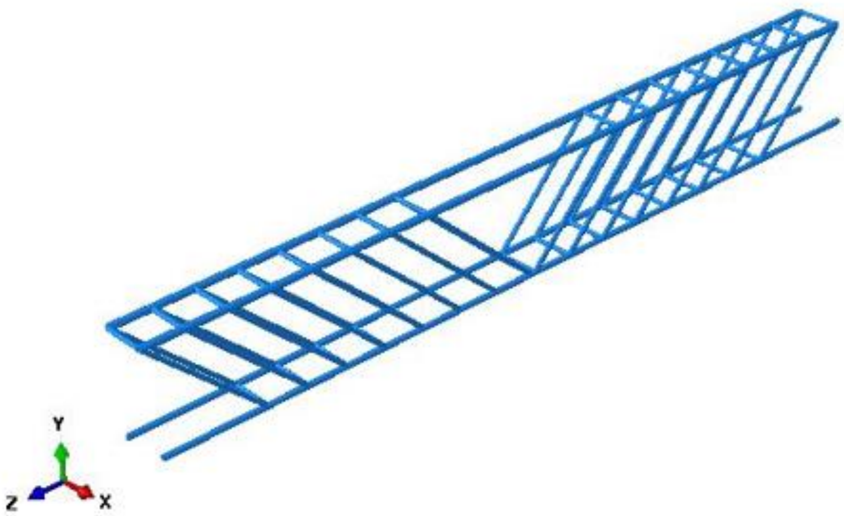


Fig 3.11: Reinforcement and Inclined Stirrups 3D modeling.

3.6 Mesh sensitivity and model validation

For generating nodes and elements meshing is required. Mesh is created by nodes defining and used them to define the elements. Meshing is important for accruing the most logical results from a finite element analysis. After assembling and assigning the properties, a mesh sensitivity test had been done for finding the optimum mesh.

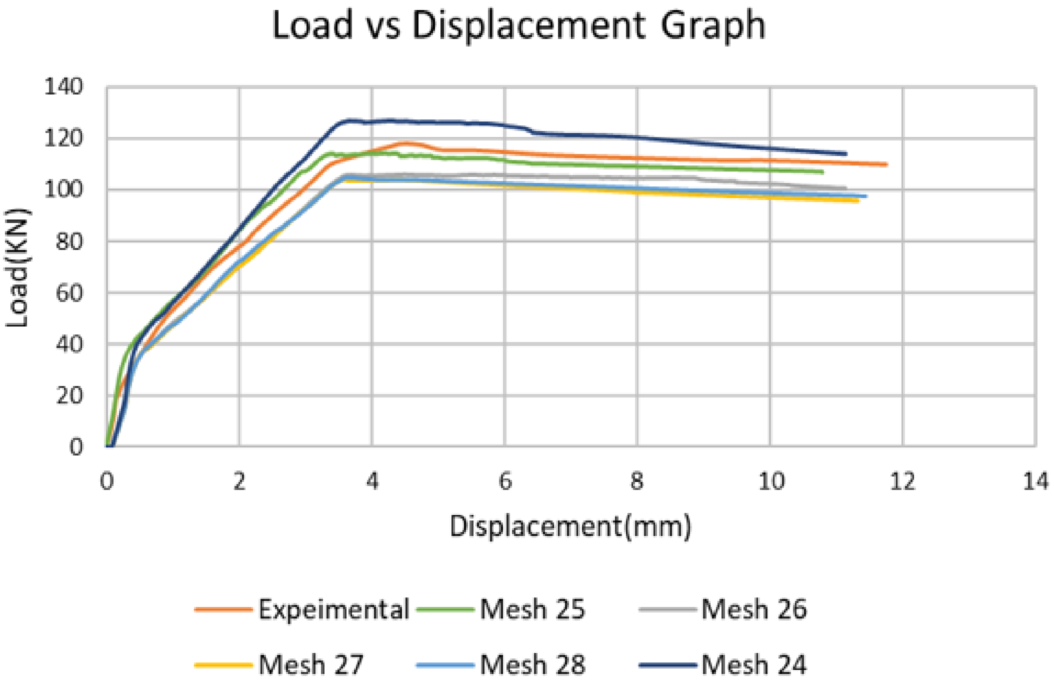


Fig 3.12: Mesh sensitivity test results of the numerical models.

From the load vs Displacement graph, it was seen that the FE model almost followed the experimental graph. Which indicates the accuracy of the FE model.

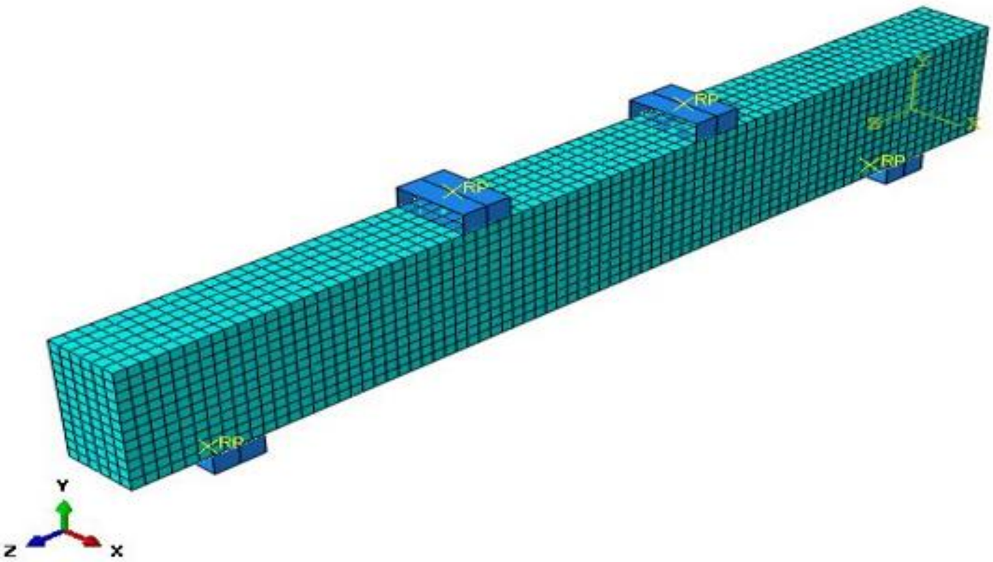


Fig 3.13: Final meshed condition of the beam.

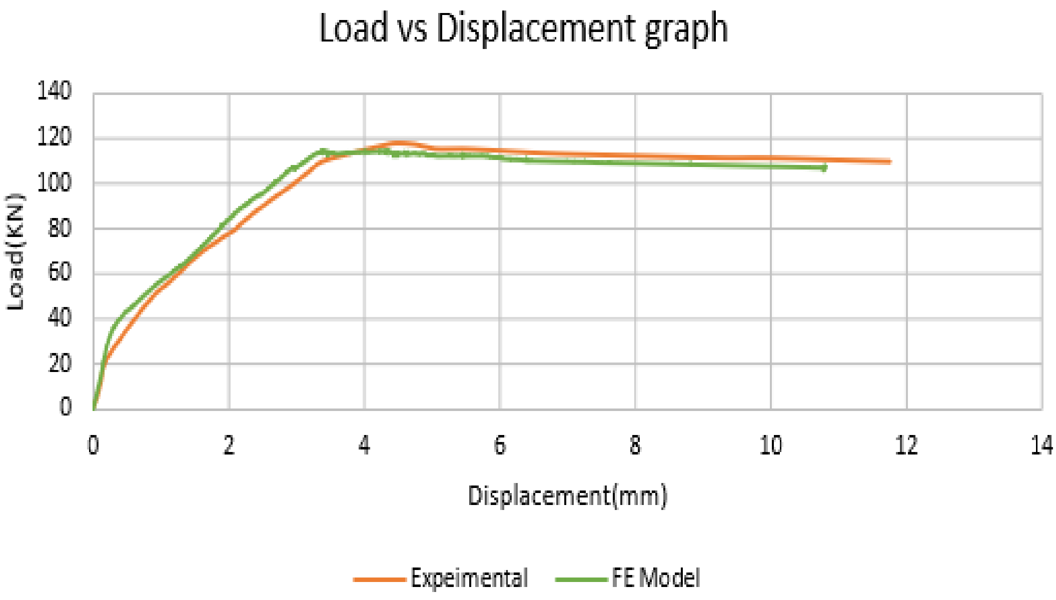


Fig 3.14: load-displacement graph of FE model and experiment work.

The analysis had been carried out under displacement control. Two load-cell had been placed on the top of the beam and a constant displacement of 12mm had been applied at a rate of 1mm/min. Two hinge support had been placed under the beam at a distance of 200mm from the two ends. A full Newton-raphson nonlinear analysis method has been directed for each of the models.

CHAPTER 4
RESULTS AND DISCUSSION

4.1 Introduction

This chapter presents the findings of the research. The discussion on the results can be presented in this chapter also.

4.2 Load-Displacement behavior of the beam

Fig 15 illustrates the contrast between the loop and inclined stirrup beams in terms of the load-displacement relationship. From the figure, it is seen that the LSB showed a linear displacement till the start of flexural cracks. The nonlinearity of the load-deflection curve indicates the start of the flexural cracks. During the post-cracking stage, the displacement amplified at a bigger ratio till the yielding of tension reinforcement took place. The maximum load carried by the LSB was 114.24 KN and the ISB was 187.24 KN. Displacement obtained for the maximum load of the LSB was 4.21 mm and for the ISB was 2.38 mm. The difference in deflection values for both beams may be due to the stiffness of the beam. Generally, stiffness is an initial slope of the load-deflection curve. All beams were displayed similar load-displacement performance up to the elastic limit. Then, for the effect of Stirrups, the stiffness of the beams amplified and it showed changed behavior. The behavior of RC beam with inclined stirrups was found more ductile than RC beam with loop stirrups. Thus, RC beam with inclined stirrups had experienced smaller deflection compared with the RC beams with loop shear reinforcement.

Table 4.1: Comparison of ultimate load and displacement of the loop and inclined stirrup beam.

Specimen	Ultimate load (KN)	Load capacity increased (%)	Displacement at ultimate load (mm)	Displacement decrease %
Loop stirrup beam	114.24		4.21	
Inclined stirrup beam	187.24	63.90	2.38	43.47

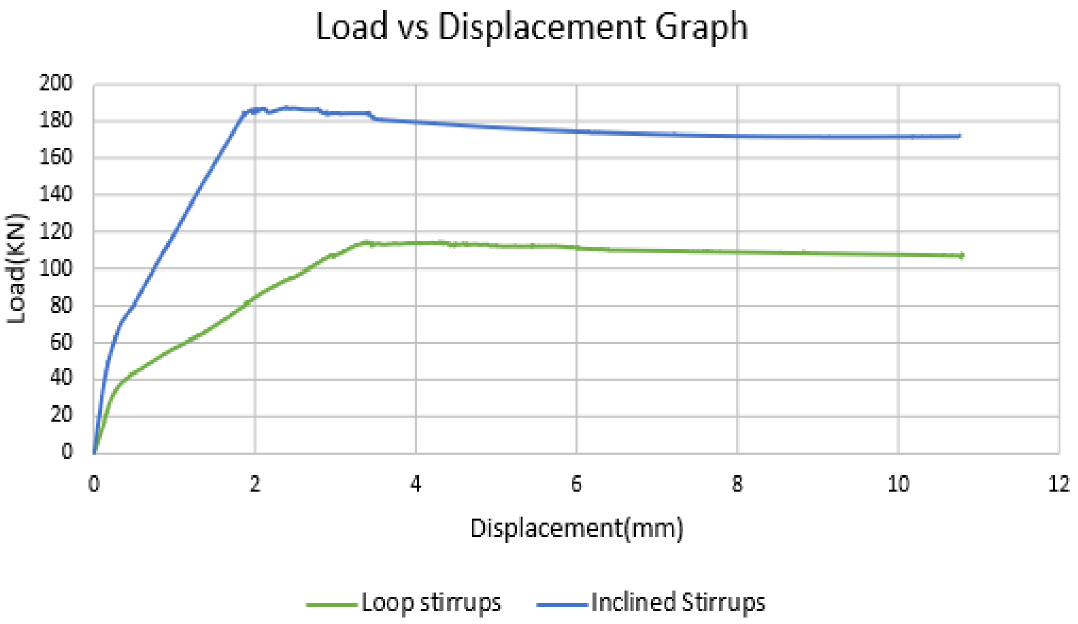


Fig 4.1: Load-deflection behavior of Loop and Inclined stirrup beams.

4.3 Ultimate Shear Force Capacities of Beam

Table 3.1 From that, it was observed that the inclined shear reinforcement improves the load-carrying capacity of the beam. it showed 63.90% higher ultimate strength than the loop shear reinforced beam. The shear force of the beams was projected as per ACI 318-11 to compare with FE results.

The shear force of the concrete beam was calculated through this equation,

$V''c = 0.17 \lambda \sqrt{fc'} bwd''$ ……………………………..(4.1)

Where Vc = Shear strength of concrete without reinforcement,

f'c= Ultimate compressive strength of the concrete

λ=1.0 normal weight concrete

bw and d are the section dimensions.

Ultimate shear strength of the concrete including shear reinforcement was calculated by,

$$Vu=(\emptyset Avfyd)/s+\emptyset Vc \ldots\ldots\ldots\ldots\ldots\ldots\ldots.(4.2)$$

Where Vu= Ultimate shear strength

Av= area of shear reinforcement

fy= yield stress of the shear reinforcement.

S= Spicing between two shear reinforcement.

And Ultimate shear strength of the concrete including inclined shear reinforcement was calculated by,

$$Vu=(\emptyset Avfy(Sin\theta+cos\theta)d)/s+\emptyset Vc \ldots\ldots\ldots\ldots\ldots\ldots(3.3)$$

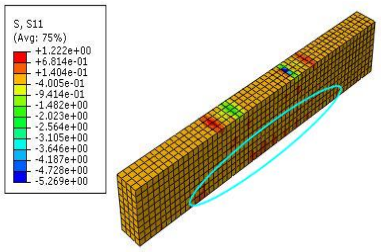
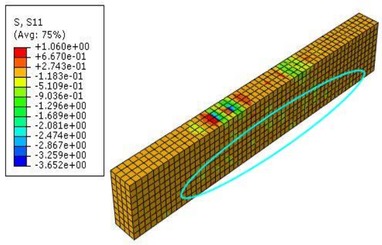
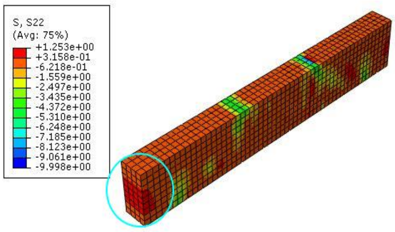
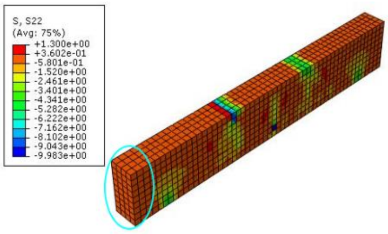
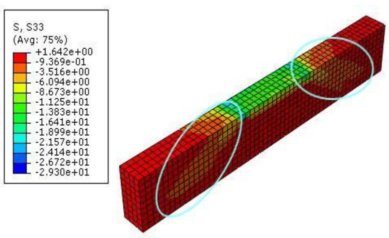
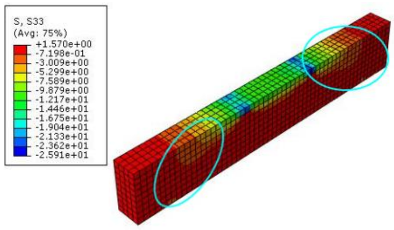
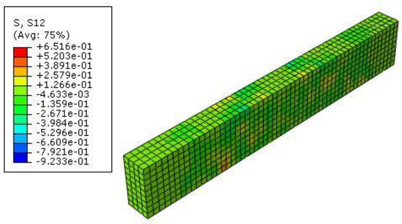
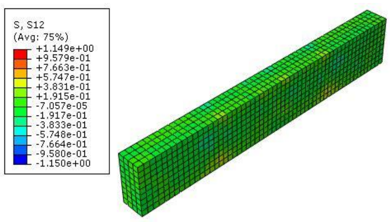
Where θ is angle between horizontal reinforcement and shear reinforcement.

Table 4.2: Comparison between analytical and FE results

Specimen	FE Shear Result	Analytical Shear Result	$\frac{FE(cal)}{Analytical(cal)}$
Loop shear rein. Beam	114.24	130.47	0.88
Inclined shear rein. Beam	187.24	178.84	1.04

4.4 Shear Stress Capacities of Beam

Stresses generated on the beams can be also found from ABAQUS output results. In ABAQUS X, Y, Z directions are denoted by 1,2,3 accordingly. So S11, S22, S33indicates principal stress in X, Y, Z direction and S12 indicates shear stress of the beam.

Loop Stirrup Beam (LSB)	Inclined Stirrup Beam(ISB)	Description
		The principal stress in X direction is equally distributed in the lower portion of the model “ISB”. Whereas, in model “LSB” there were some nodes that are experiencing much more stress than the neighboring nodes.
		In model “LSB” a certain amount of principal stress in Y direction occurs in the mid-portion at the end of the beam. But this compressive stress was absent in model “ISB”.
		Other than producing much less principal stress in Z direction, model “ISB” helped to distribute stress more accurately than that of model “LSB”.
		Having the same area, Model “ISB” resisted more shear stress of about 1.152 MPa than of Model “LSB” having the value of about 0.652 MPa, which concluded that Model “ISB” carries more Shear force than Model “LSB” before failing.

4.5 Crack Patterns

The ABAQUS program has the ability of recording crack patterns. Once the principal tensile stress goes beyond the ultimate tensile strength of the concrete a block changes it's color from blue to red to show the sign of cracking or crushing. The cracking sign propagates perpendicularly to the way of the principal stress. ABAQUS program shows blocks at locations of cracking or crushing in concrete elements. It shows the appearance of flexural cracks, diagonal shear cracks, and compression cracks. Concrete damage plasticity model was used in this analysis to capture the crack pattern. Loop stirrup beam showed flexural cracks and shear cracks. Pure Flexural cracks propagated perpendicular at the middle portion of the beam because of two-point loading; no shear force occurred at the middle part of the beam. shear cracks were propagated near the end of the support which is the shear zone of the beam. From the crack pattern of the loop stirrup beam, it is seen that flexural and shear cracks propagated through the weak zone of the concrete, and no proper load distribution occurred. But in inclined stirrup beam crushing of the concrete occurred at the bottom of the beam because the load was distributed all through the bottom face of the beam. Which improve the weak zone of the concrete and helped to increase the load-carrying capacities of the beam. Although inclined stirrups resist the diagonal cracks to propagate.

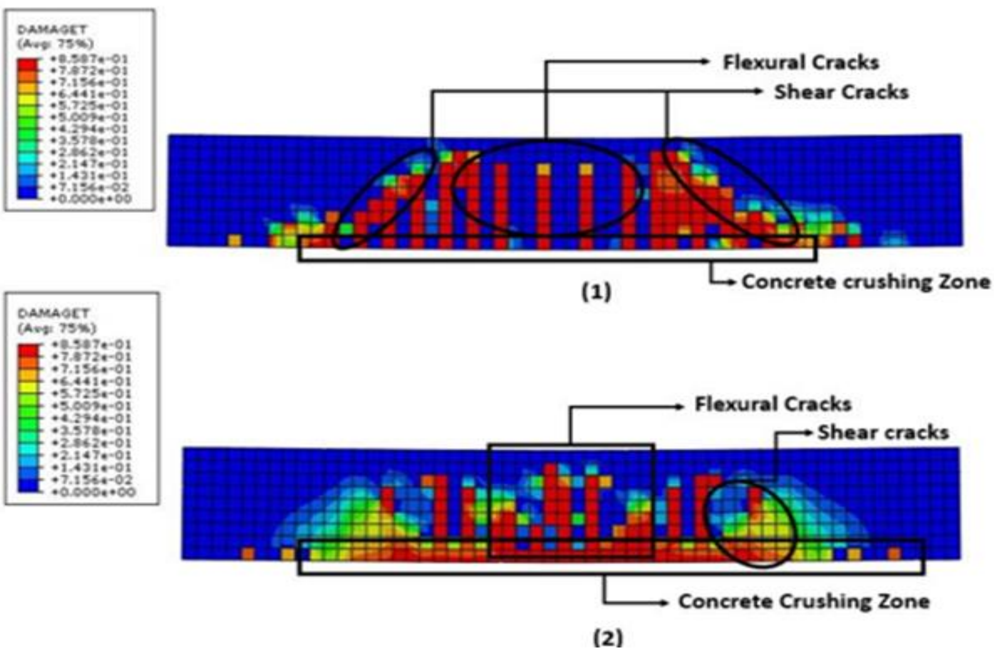


Fig 4.2: (1) (2) Crack propagation simulation of LSB and ISB

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 Conclusion

The following conclusion can be made from the above study:

1. From the analysis conducted above it is seen that the Inclined stirrups setup increased the shear capacities of the beam up to 63.90 % compared to the loop stirrups beam.
2. The beam with inclined stirrups shows less displacement rather than the beam with loop stirrups. Displacement reduced about 43.47% on inclined stirrups beam compared to loop stirrups beam due to increase of stiffness.
3. Shear stress resistance capacities increased significantly on inclined stirrups beam rather than loop stirrups beam.
4. The predicted theoretical results justify the numerical results.
5. From the Crack pattern analysis, it is seen that the inclined stirrup crushed the concrete at the bottom compared to the loop stirrup beam which increases the load-carrying capacities of the inclined stirrup beam.

5.2 Recommendation

1. In this study, only the Abaqus 6.14 software is used for the analysis. The software was a trial version; original software will be gives the more accurate results.
2. If the analysis results compare with the actual hand calculation data, then more reliable results will be found. It should be done in the future work.
3. Inclined stirrups are also provided generally at 45^{o} for resisting diagonal tension.
4. The results obtained from numerical analysis (FEM) using ABAQUS are in good agreement with experimental results.
5. It allows replacing experimental tests that are costly and time taking.

REFERENCES

Theodor, K., 1992. Minimum shear reinforcement based on interface shear transfer. *ACI Structural Journal,* pp. 99-105.

Adebar, P., 2001. Diagonal cracking and diagonal crack control in structural concrete. *Design and Construction Practices to Mitigate Cracking,ACI,* pp. 85-116.

Adebar, . P. & Leeuwen, J. V., 1999. Side-face reinforcement for flexural and diagonal cracking in large concrete beams. *ACI Structural Journal,* 96((5)), pp. 693-704.

Atkinson, K. E., 2007. Numerical analysis. *Scholarpedia.*

Colajanni, p. e. a. & n.d., n.d. Shear Capacity in Concrete Beams Reinforced by Stirrups with Two Different Inclinations. *Engineering Structures,* 81((1)), pp. 444-453.

Hafezolghorani, M. e. a., 2017. Simplified Damage Plasticity Model for Concrete. *Structural Engineering International Nr.*

Hassan, e. a., 1985. Fatigue test of reinforced concrete beams with various types of shear reinforcement. *Transaction of Japan Concrete Institute,* Volume 7, pp. 277-284. .

Michael, C. & Daniel, K., 1999. How safe are our large, lightly reinforced concrete beams,slabs, and footings. *ACI Structural Journal,* pp. 482-490.

Moayyad, A.-N. & M., 2013. Shear failure investigation of reinforced concrete beams with swimmer bars. *Journal of Civil Engineering and Construction Technology, ,* 4((2)), p. 56–74.

Mohamed, O., Khattab, R. & Okasha, N., 2019. Numerical Analysis of RC Slab with opening strengthened with CFRP Laminates. *s.n.* .

Saravanakumar, p. & Govindaraj, A., 2016. Influence Of Vertical And Inclined Shear Reinforcement On Shear Cracking Behavior In Reinforced Concrete Beams. *International Journal of Civil Engineering and Technology (IJCIET),* 7(6), pp. 602-610.

Sato, e. a., 2004. Diagonal tensile failure mechanism of reinforced concrete beams. *Journal of Advanced Concrete Technology,* 2((3)), pp. 327-341.

Songkram, 2013.

Suhaim, A., 2015. Inclined Link as Shear Reinforcement in reinforced Concrete Beam. *Universiti Teknologi Malaysia.*

Systèmes, D., 2013. ABAQUS 6.13 analysis user’s guide. *Dassault Systèmes.*

Witchukreangkrai, . e. a., 2004. Control of diagonal cracking in partially prestressed concrete beams. *Proceedings of Japan Concrete Institute,* 26((2)), pp. 727-732. .

Witchukreangkrai, e. a., 2006. Evaluation of shear crack width in partially prestressed concrete members. *Proceedings of Japan Concrete Institute,* 28((2)), pp. 823-828.

Young-soo, Y., William, C. & Denis, N., 1996. Minimum shear reinforcement in normal,medium, and high-strength concrete beams.. *ACI Structural Journal,* p. 576–84.

Zakaria, M., Ueda, T. & Wu, Z., 2011. Evaluating and proposing prediction models of shear crack width in concrete beams. *Journal of Japan Society of Civil Engineers Ser E2 (Materials and Concrete Structures),* 67((2)), pp. 245-263.

Zakaria, M., Ueda, T., Wu, Z. & Meng, L., 2009. Experimental Investigation on Shear Cracking Behavior in Reinforced Concrete.

Zamri, N. F., Mohamed, . R. N., Khalid, N. A. & Chiat, K. Y., 2018. The Effects Of Inclined Shear Reinforcement In Reinforced Concrete Beam. *Malaysian Journal of Civil Engineering,* 30((1)), pp. 85-96.

https://images.app.goo.gl/SvMT6Tnt65wUnqa97

https://images.app.goo.gl/SvMT6Tnt65wUnqa99

https://images.app.goo.gl/SvMT6Tnt65wUnqa77

https://images.app.goo.gl/SvMT6Tnt65wUnqa67

https://images.app.goo.gl/SvMT6Tnt65wVnqa97

https://images.app.goo.gl/SvMT6Tnt75wUnqa89