

INFLUENCE OF LOW VELOCITY IMPACT ON STEEL CONCRETE COMPOSITE BRIDGE DECK

A
PROJECT REPORT

Submitted in partial fulfillment of the requirements for the award of the degree

of

BACHELOR OF TECHNOLOGY

IN

CIVIL ENGINEERING

Under the supervision

of

Dr. Tanmay Gupta

(Assistant professor)

by

**PEMA ZANGMO (161630)
PARMILA GURUNG (161695)**

to



JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY

WAKNAGHAT, SOLAN – 173234

HIMACHAL PRADESH, INDIA

June-2020

STUDENT'S DECLARATION

We hereby declare that the work presented in the Project report entitled “**INFLUENCE OF LOW VELOCITY IMPACT ON STEEL CONCRETE COMPOSITE BRIDGE DECK**” submitted for partial fulfillment of the requirements for the degree of Bachelor of Technology in Civil Engineering at **Jaypee University of Information Technology, Waknaghat** is an authentic record of my work carried out under the supervision of **Dr. Tanmay Gupta**. This work has not been submitted elsewhere for the reward of any other degree/diploma. We are fully responsible for the contents of our project report.



Pema Zangmo
161630
Department of
Civil Engineering
June 5th, 2020



Parmila Gurung
161695
Department of
Civil Engineering
June 5th, 2020

CERTIFICATE

This is to certify that the work which is being presented in the project report titled **“INFLUENCE OF LOW VELOCITY IMPACT ON STEEL CONCRETE COMPOSITE BRIDGE DECK”** in partial fulfillment of the requirements for the award of the degree of Bachelor of Technology in Civil Engineering submitted to the Department of Civil Engineering, **Jaypee University of Information Technology, Wagnaghat** is an authentic record of work carried out by **Pema Zangmo (161630) and Parmila Gurung (161695)**, during a period from July, 2019 to June, 2020 under the supervision of **Dr. Tanmay Gupta** Department of Civil Engineering, Jaypee University of Information Technology, Wagnaghat.

The above statement made is correct to the best of our knowledge.

Date: June 5th, 2020



Dr. Tanmay Gupta
Assistant Professor
Department of Civil Engineering
JUIT, Wagnaghat

Dr. Ashok Kumar Gupta
Professor and Head of Department
Department of Civil Engineering
JUIT, Wagnaghat

ACKNOWLEDGEMENT

This report presents the work and the results that have been achieved for the project entitled “Influence of Low Velocity Impact on Steel-Concrete Composite Bridge Deck” under the supervision of guide Dr. Tanmay Gupta.

First of all, we would like to take an opportunity to extend our heartfelt gratitude to our Project guide Dr. Tanmay Gupta for his kind support and for entertaining all our doubts despite his busy schedule. For guiding us through well often when we were stuck and also for providing us some essential materials, we would like to express our gratitude to Dr. Pankaj Kumar. We would also like to thank Mr. Rajesh Kumar Sahu and Mr. Pradeep for providing us technical support while working on Abaqus software for modeling. Without all of their support, we would not have made any progress with the project report and reached the final display of report as it is now.

Date: June 5th, 2020

ABSTRACT

Steel-concrete composite is the most commonly preferred combination of materials for construction which have applications in bridges, multi-storey buildings and factories. Bridge structures are often subjected to varying magnitudes of impact loads when vehicles cross across the deck pavement or when there is collision between vehicles and the components of bridge. Such loads overtime may lead to the generation of defects in the overall bridge structures. Thus, based on the intensity and magnitude of impact loads, the corresponding effect have tendency to compromise the overall structural response and may as well lessen the service life of bridge structure. Noticeably, in the last couple of decades ,the application of steel-concrete composites have escalated in bridge structures due to the enhanced property of being more stiffer and stronger than when steel and concrete are used individually. The use of steel-concrete composites helps in acquiring the advantageous property of strong compressive strength from concrete and strong tensile strength from steel section. Hence, in bridge construction, the deck pavement are preferred to be constructed using steel-concrete composites for resisting impact loads imposed by vehicular impacts, ship impacts and air craft impacts. Impact loads refers to concentrated loads which falls in the range slow velocity impact about 10^1 S^{-1} . To consider the application of steel-concrete composite in bridge deck, it has to be ensured that the various parameters of the material are checked for its effectiveness. In this study, the components of composite will be analyzed individually.

Keywords: Steel-concrete composite; bridge decks; impact loading; compressive stress/strain; tensile stress/strain; ABAQUS

TABLE OF CONTENTS

Contents	Page number
Student's declaration	i
Certificate	ii
Acknowledgement	iii
Abstract	iv
List of figures	vii
List of Symbols	ix
List of Abbreviation	x
CHAPTER 1: INTRODUCTION	
1.1 General	1
1.2 Brief history	1
1.2.1 Ancient era	1
1.2.2 Origins of composites	1
1.3 Generations of composites	2
1.3.1 The first generation	2
1.3.2 The second generation	2
1.3.3 The third generation	2
1.3.4 The fourth generation	3
1.3.5 Mid-2000s	3
1.4 Classifications of composites	3
1.4.1 Based on Matrix Materials	3
1.4.2 Based on Size and Shape of Dispersed phase	4
1.5 Types of connections in composites	5
1.5.1 Adhesives	5
1.5.2 Welding	5
1.5.3 Mechanical connections	5
1.6 Scope of Thesis	5
1.7 Organization of Thesis	6
1.7.1 Chapter 1	6
1.7.2 Chapter 2	6
1.7.3 Chapter 3	6
1.7.4 Chapter 4	7
1.7.5 Chapter 5	7
CHAPTER 2: LITERATURE REVIEW	
2.1 Introduction	8
2.2 Previous studies	8
2.3 Summary of Literature review	10
2.4 Research Gap	11
2.5 Research Objective	11
CHAPTER 3: METHODOLOGY	
3.1 Introduction	12
3.2 ABAQUS Software	12
3.2.1 Model Geometry	12
3.2.2 Model Processing	13
3.3 Structural Steel Modelling	13
3.4 Concrete Cube Modelling	17

3.5 Steel Beam Modelling	20
3.6 Concrete Slab Modelling	22
3.7 Stud Connector Modelling	24
3.8 Impactor Modelling	25
3.9 Interaction Model	27
CHAPTER 4: RESULTS AND DISCUSSIONS	
4.1 Introduction	30
4.2 Tensile Coupon Test	30
4.2.1 Visualization Module	32
4.3 Compression Test On Concrete Cube	34
4.3.1 Visualization Module	35
4.4 Low Impact Velocity on Simply Supported Composite Beam	36
CHAPTER 5: CONCLUSION	37
REFERENCES	
PLAGIARISM VERIFICATION CERTIFICATE	

LIST OF FIGURES

Fig number	Title	Page number
1.1	Classification of Composites	3
3.1(a)	Geometry and Dimension of Steel Coupon	13
3.1(b)	Section A-A	13
3.2	Assigning steps to steel model	15
3.3	Application of boundary condition and pressure loading to steel coupon	16
3.4	Creating concrete cube in ABAQUS/Explicit	17
3.5	Assigning Steps to concrete cube	18
3.6	Applying Boundary Condition and Pressure Load to concrete cube	19
3.7	Steel beam girder in ABAQUS/Explicit	20
3.8	Meshing of steel beam girder modal in ABAQUS/Explicit	21
3.9	Creating a part of concrete slab in ABAQUS/Explicit	22
3.10	Meshing of the concrete slab in ABAQUS/Explicit	23
3.11	Creating a part of shear stud in ABAQUS/Explicit	24
3.12	Meshing of the shear stud model	25
3.13	Impactor model without mesh	26
3.14	Meshed model of Impactor	27
3.15	Picture showing stud connectors on top flange of steel beam	28
3.16	Picture showing the stud connected steel-concrete composite girder	28
3.17	Picture showing the boundary conditions of various components	29
3.18	Figure of meshed model of impactor on composite surface	29
4.1	Showing the Visualization of Stress distribution in coupon steel at S, Mises	30
4.2	Showing the Visualization of Stress Distribution in coupon steel at S22	31
4.3(a)	Comparison between analytical result and reference result of Stress vs. Strain	32

4.3(b)	Comparison between analytical result and reference result of force vs. Displacement	33
4.4	Showing the visualization of stress distribution on concrete cube at mises	34
4.5(a)	Comparison between ABAQUS generated and reference paper curve of stress vs. strain	35
4.5(b)	Comparison between ABAQUS generated and reference paper curve of load vs. deformation	36

LIST OF SYMBOLS

E	Modulus of elasticity
f_y	Yield strength of steel
ϵ_y	Yield strain of steel
f_u	Ultimate strength of steel
ϵ_u	Ultimate strain of steel
ϵ_L	Failure strain of steel
ρ	Density of concrete (kg/mm^3)
ψ	Angle of Dilation
e	Eccentricity
σ_{b0}/σ_{c0}	Ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress
K_c	Coefficient of deviatoric cross-section's shape
μ	Viscosity Parameter
δ	Deformation

LIST OF ABBREVIATIONS

FEM	Finite Element Method
AISC	American Standard
BS	British Standard
FEA	Finite Element Analysis
NLFEA	Non-linear finite element analysis
EC	British Standard (euro code)
IS	Indian Standard

CHAPTER 1

INTRODUCTION

1.1 General

A composite is a material which consist of combination of two or more different material with different chemical and physical properties which, when combined together gives stronger and improved properties than by individual materials. The use of steel concrete composite provides stiffer and stronger properties as compared to steel beam and concrete slab alone.

Relatively, the performance of such composite member predominantly depends on connection between steel beam and concrete slab [1, 2]. The very purpose of the connection at steel-concrete interface is to resist the applied load as well as to transfer the stresses from one component to the other.

The degree of interaction between the steel and concrete interface is of paramount significance as the efficiency of composite is highly dependent on it. Further, the degree of interaction depends on shear stiffness of the composite connection. There is a direct relation between the degree of interaction and degree of shear connection [2, 7]

1.2 Brief History

1.2.1 Ancient era

Around 3400 B.C, the ancient Mesopotamians were the first to use composite material when they joined wood strips at different angles to make plywood in Iraq. After that in 1200 A.D, composite bows was evolved by Mongols which were made up of combination of wood, bamboo, bone, cattle tendons, horn and silk with pine resin[1, 2].

About 1500 B.C, Egyptians and Mesopotamians builders and artisans utilized straw as reinforce in mud bricks and boats.

From 1870s to 1890s, the uprising of the chemicals changed the initiation of composites. New synthetic resins were advanced with the operation of polymerization. In the beginning of 1900s, the evolution of chemicals had lead to the initiation of plastics and the reinforcement was required for the hardness and durability [3, 5].

1.2.2 Origins of Composites

The three main reasons for the speedy development and application of composite materials which started in the 1940s are as follows [3, 4]:-

- For the military purpose like airplanes, helicopters and rockets with affective strength and

light weight materials

- Due to the growth and expansions of polymer industries, the evolution of the light-weight polymer took place
- An exceptionally high strength materials like glass fibers were evolved

1.3 Generations of Composites

There are four generations of composites as follows [3, 4]:-

1.3.1 The first generation

The first generation of composite took place during 1940s

- With the aim to achieve high strength and light-weight material for the components of helicopter rotors, wings and nosecones, the engineers came up with the idea of development of fibers to get well built materials.
- In the beginning of 1932, Owens started to manufacture glass fibers, by 1935 they tied up with Corning for the mass production of fibers
- The copyright for epoxy resins in 1938 to P. Castan in Switzerland
- The use of GRFP took place in the field of marine industries as fiberglass boats were used in the beginning of 1940s in replacement of conventional materials.
- In 1943, the tooling operation was moved forward to develop GFRP components by Republic Aviation corporation

1.3.2 The second generation

The second generation of composites took place during 1960s

- Due to the project of Soviet Sputnik lead to the evolution of metal matrix composites as a solution to the requirement of lighter and well built properties than GFRP
- In 1964, Stephanie Kwolek developed a polymer which was resistance to melting. Based on this, DuPont came up with the development of fiber in 1971
- In 1960s, due to the applications of different kinds of fibers and matrices has lead to the evolution of composites [3,4]

1.3.3 The third generation

The third generation took place during 1970s and 1980s

- By 1970s, the importunity for the fibers in space and military purpose decreased. Thus, those composites made up of costly fibers were transferred to civil application [3,4]

1.3.4 The fourth generation

The fourth generation took place during 1990s

- There was initiation of composites in both academic and industrial research fields in much little amount. Thus, leading to the evolution of hybrids and nano composites [3,4]

1.3.5 Mid-2000s

The advancement of the 787 Dreamliner approved composites for greater strength and rigid. The continuous progress of the development of finished technology had lead to expansion of use of automotive appliances and consumer product industries [5]

1.4 Classification of composites

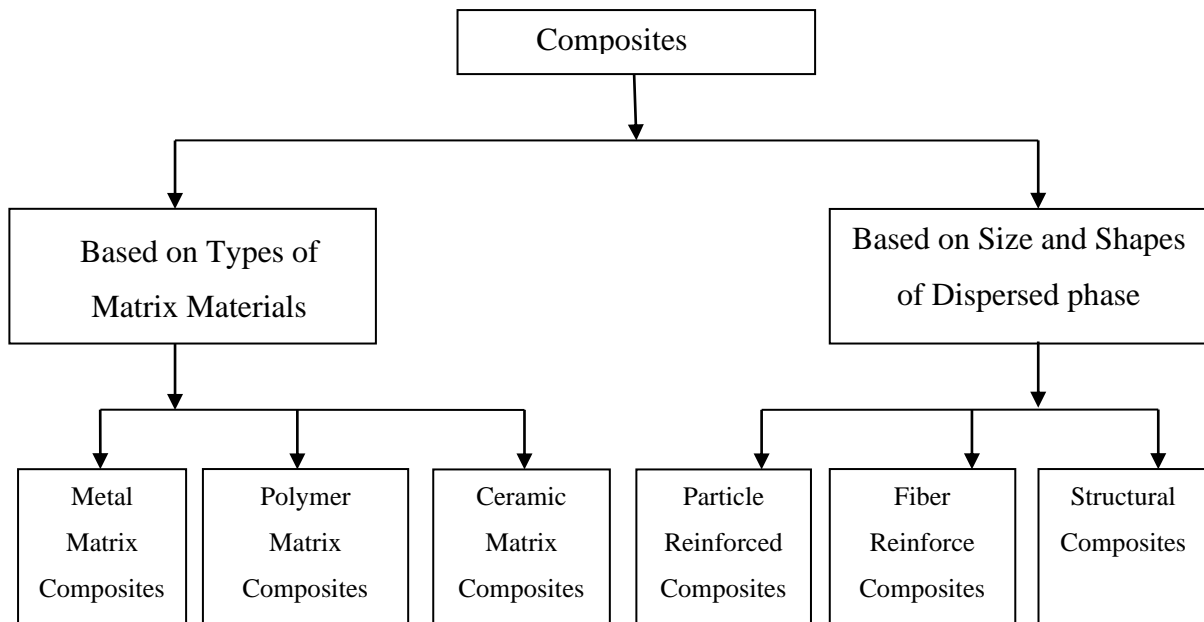


Fig 1.1 Classifications of Composites ^[1]

1.4.1 Based on Matrix Materials

• Metal Matrix Composites:

A metal matrix composites (MMC) consists of at least two constituents, one part of the constituent being metal and the other constituent being a different metal such as ceramic or organic compound [1].

Matrix is the stuff which protects and transfers load between the fibers and has better shear properties. It is also a ductile metal with low density

Example: Polyester, Vinyl Ester and Epoxy [1].

- **Polymer Matrix Composites:**

A polymer matrix composite consists of different kinds of fibers which may be short or continuous and are held together by an organic polymer matrix (resin).

Example: Fibre reinforced polymer [1]

- **Ceramic Matrix Composites:**

Ceramic matrix composites are composed of ceramic fibers which are embedded in a ceramic matrix to form a ceramic fibre reinforced ceramic material. The common fibers used are carbon and aluminum oxide and matrix materials are mostly silica carbide, silica nitride and aluminum oxide [1]

Example: Carbon fibre in silicon carbide matrix.

1.4.2 Based on Size and Shape of Dispersed phase

- **Particle reinforced Composites:**

These are widely used composites due to its cheap cost and being easily accessible.

Example: Concrete, where cement is used as the matrix and the sand or gravel is used as the dispersed phase [1]

- **Fiber reinforced Composites:**

Fibre reinforced composites is a building material which consists of more than one constituent materials with different physical properties of its own [1].

1 It consists of three components as given below:

2. Fibers as dispersed phase
3. Matrix as the continuous phase
4. Fine Interface region

- **Structural Composites**

Structural composites are those composites whose properties depend on the properties of the constituents and on the geometrical design of different of different structural elements [1].

There are two classes of composites which are widely used:

1. Laminates
2. Sandwich panels

1.5 Types of connections in composites

Different types of connection methods are [2]:

- Adhesives
- Welding
- Mechanical connections

1.5.1 Adhesives

Adhesives are those substances which can be applied to any surface of the two items in order to bind them together and to withstand their separation.

1.5.2 Welding

Welding involves a process of fabrication or sculptural methods in order to join various materials like metals or thermoplastics. The welding process is different from low temperature based methods such as soldering and brazing.

However, composites cannot be welded with the welding methods of metals. Therefore, the polymer matrixes are being melted by using a laser, induction or ultrasonic welding B [2, 9].

1.5.3 Mechanical connections

They are used to develop the composite action between steel beams and concrete. It is provided in order to resist longitudinal shear and it is referred to as the “shear connection”.

Mechanical connections is also a type of connection which utilizes traditional fasteners such as bolts, rivets, screws and pins and it is currently most preferred connection method between composites as well as between composites and other materials like concrete, wood and steel[2, 8].

1.6 Scope of Thesis

Nonlinear behaviour of simply supported and stud connected steel-concrete composite deck under low impact velocity is studied numerically. The main focus of the study has been to develop and validate a three-dimensional finite element model. Results obtained are compared with that of the reference paper for validation. The model of composite deck is developed using ABAQUS software. For modelling concrete, nonlinear damage plasticity model is considered. Appropriate interface elements along with constraints are used to portray interaction amongst the steel beam, concrete slab and shear studs. Along with the interaction

between the steel and concrete, proper value of friction coefficient has been given. The validation of model generated for steel coupon test and concrete cube has been done by comparing the stress vs. strain and load vs. deformation curve generated from Abaqus with that of the reference paper graph. The two curves generated from Abaqus corroborated well with the curves of the reference paper.

Upon going through available research papers, it can be noted that the behaviour of steel-concrete composite elements connected with mechanical connectors under static and cyclic loading has been examined in detail. However, the behaviour of steel-concrete composite elements under extreme loading which includes dynamic ,impact and blast loading have not been examined. This study proves an attempt towards examination of steel-concrete composite beam element under low impact velocity.

1.7 Organization of Thesis

The composition of the thesis is divided into five main chapters. Every chapter starts with an introduction to provide an overview of the contents of that exact chapter. The results obtained and the conclusions drawn from the study are summarized at the end of the chapter.

1.7.1 Chapter 1

Highlights general ideas about composite which includes brief history, its origin, classification and types of connections used. A brief explanation of each of those topics along with flowchart has been presented in this chapter.

1.7.2 Chapter 2

Provides an inclusive review on the available research work whichever was accessible and that which relates to the areas of interest in the current thesis. This includes the essential findings and results based on different experimental and analytical works. It also includes the relevant design rules and guidelines as per some international codes and standards. The chapter commences with an introduction of the chapter and concludes with a summary.

1.7.3 Chapter 3

Presents the finite element analysis developed to simulate the impact response of stud connected steel -concrete composite deck. A three-dimensional finite element model of stud connected steel-concrete composite deck is developed using the software ABAQUS. The

finite element method consists of three main parts which includes pre-processing, solution and post-processing. A brief explanation of each part has been presented in this chapter along with the explanation of how each model has been developed. Based on the data of reference paper, a model of steel coupon, concrete cube and model of simply supported composite deck has been presented showing in detail the material property, analysis step, interaction property, boundary conditions and mesh sizes.

1.7.4 Chapter 4

Presents and explains the results obtained from running the simulation of models in finite element analysis software ABAQUS. The results of steel coupon and concrete cube have been presented through stress-strain curve and load-deformation curve by comparing it the curves of reference paper. This chapter also presents the snapshot of S, Mises and U, Magnitude result obtained from running the analysis in the software.

1.7.5 Chapter 5

Presents a brief summary of the conclusions noted from the results and discussions. This chapter also provides the general overview drawn from working on the current thesis. The models of steel coupon and concrete cube have effectively shown its result in comparison with that of reference paper. However, the result of impact response of stud connected steel-concrete composite couldn't be shown properly due to not having computational facility in the current situation.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This section presents an overview of already existing studies on the analysis and properties of the use of steel concrete composites in construction. It contains guidelines, procedures and the latest as well as already completed studies regarding steel concrete composites in construction from different researchers.

2.2 Previous Studies

Domingo J. carreira and Kuang-Han Chu (1985) [6]:

In this research paper “Stress-Strain Relationship for Plain Concrete in Compression”, the researchers have worked on proposing an equation based on experimental data in order to represent the relationship of stress-strain in the form of serpentine curve for plain concrete in compression. The parameter required for generating a relationship of a general stress-strain depends on the testing conditions as the parameters are not inherent properties of concrete.

Carreira and Chu (1986) [7]:

In this research paper “Stress-Strain Relationship for Reinforced Concrete in Tension”, the researchers have proposed a stress-strain relationship which shows the behavior of reinforced concrete in tension including the combined effects of cracking and slippage at cracks alongside the reinforcement.

It is concluded that the behavior of reinforced concrete in tension is dependent on number of factors such as restraining action of reinforcement on the crack distribution, generation of external and internal cracks which are formed around and along the bars, strength of concrete and shrinkage effect.

Ho et al (2019) [8]:

In this research paper “Modelling tensile tests on high strength S690 steel materials undergoing large deformations”, the researchers have performed tensile test by taking two steel coupons of S690 and S275 each to generate a true stress-strain curve showing large deformations up to a fracture.

They have used three transformation rules which are Power Law Method, Linear Law Method and Instantaneous Area Method to obtain accurate true stress-strain curve after data analysis. After experimental and numerical investigation they have concluded that the instantaneous area method is effective in generating accurate true stress-strain curve as compared to power

Law method and linear law method for predicting load extension curve.

They have also concluded that it is important to conduct detail experimental and numerical investigations of steel having same grades but manufactured differently according to delivery conditions as it is considered that the various steel material having same grades with different delivery conditions shows different properties of deformation.

Chaudhary et al (2017) [9]:

In this research paper “Behaviour of adhesive bonded and mechanically connected steel-concrete composites under impact loading”, the researchers have done the study on steel concrete composite connections with impact loading, considering two types of connections namely adhesively bonded connection and mechanically connected. Two test were carried out which were push out test and drop weight impact test to determine the shear capacity of the composite specimen.

According to experimental work, the adhesively bonded connection required more number of blows to generate cracks as compared to mechanically stud connected composite specimen. They have concluded that composite specimen which were mechanically connected were more ductile when compared to adhesively bonded composite specimens.

Zheng et al (2009) [10]:

In this research paper “Finite element investigation of the structural behaviour of deck slabs in composite bridges”, the researchers have carried out the study on the variation of structural behaviour of deck slabs of steel concrete composite bridge when subjected to static patch loads. They have also investigated the component which plays a vital role in identifying structural behavior of deck slabs of concrete bridge that is the compressive member action (CMA).

In this study, they have used ABAQUS 6.5 software to develop a non-linear 3D finite element analysis model. They have validated and calibrated the proposed FEM models from the experimental datas of one-span composite bridge structure. Non-linear finite element analysis (NLFEA) were used to obtain some structural parameters like stress-strain distribution along the thickness of deck slab, which is difficult to acquire using other methods of analysis.

Kumar et al (2018) [11]:

In this research paper “Effect of bond layer thickness on behaviour of steel-concrete composite connections”, the researchers have worked on the experimental study of bonded connection behaviour. For this, they have carried out direct shear test with twenty five casted specimens in order to accomplish the ideal thickness of adhesive layer.

The verification of the results were done through experimental studies with the help of finite element analysis. The critical observation of the study was that the failure pattern of the bonded connection varies with the variation of thickness layer of adhesive. They have concluded that bonded connections are more effective than mechanical shear stud connections when compared in terms of its load-slip behaviour.

Da-Silva et al (2010) [12]:

In this research paper “Composite (steel-concrete) highway bridge fatigue assessment”, the researchers have presented a methodology for the analysis purpose which helped to assess the fatigue of the highway bridges comprising of steel and composite with regard to service life.

They have concluded that the loading intensity on the composite bridge is dependent on the position of vehicles on the bridge deck. As per their conclusion, the vehicles moving on the lateral lane of the bridge experiences higher stress values than those obtained with the vehicles moving on the central lane of the bridge.

Lam et al (2005) [13]:

In this research paper “Behaviour of headed stud shear connectors in composite beam”, the researchers have proposed a numerical model with the use of finite element method to simulate the push-out test of headed stud shear connectors in solid reinforced concrete slab. They have validated the model as per the results obtained from the experimental test and also compared to the data provided in the code (BS5950, EC4 and AISC).

They have carried out some parametric studies for the model created to determine the variation in concrete strength and shear stud diameter.

Kaviti et al (2019) [14]:

In this research paper “Modeling and simulation on composite laminates subjected to low velocity impact”, the researchers have studied the elastic behaviour of fiber-reinforced plastics by carrying out the numerical analysis of dynamic low velocity impact using ABAQUS software.

The researchers have also used an 18 ply finite element model and a rigid impactor to indicate the low velocity impact

2.3 Summary of literature review

This chapter concludes that, there is not much research done based on our topic up until now. From the available papers, it is noted that, nowadays the steel-concrete composites are predominantly preferred over the bridges constructed using other materials because of the

enhanced properties of compressive and tensile strength acquired from the steel and concrete when combined. On considering the overall performance of composite structure, it is equally important to analyze the connection at the interface of steel-concrete since the resistance of applied load depends on the connection. And it also helps in transferring the stress from one component to another component. Therefore, stronger and stiffer bridges of steel-concrete composites compared to the bridges constructed using steel and concrete alone can be obtained.

2.4 Research Gap

- The composite behaviour of steel-concrete composite elements connected with mechanical connectors under static and cyclic loading has been examined in detail.
- The behaviour of steel-concrete composite elements under extreme loading has not been examined. The wide application of composite elements in bridge infrastructure show investigator curiosity towards its behaviour in extreme loading as dynamic, impact and blast loading. This study proves an attempt towards examination of steel-concrete composite beam element behaviour in slow velocity impact.

2.5 Research Objective

- Numerical Simulation of structural steel and Reinforced Cement Concrete behaviour under static loading
- Design of single span steel-concrete composite beam connected with head stud connector
- Analysis of steel-concrete composite beam behaviour under slow velocity impact loading

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this project, we are working on software analysis to study the influence of low impact velocity on the steel-concrete composite bridge deck. The steel concrete composite exhibits stiffer and stronger properties than individual properties of steel and concrete. The steel are very strong in tension while concrete has very strong compressive strength, on combining them hence, results in excessively efficient and lightweight component.

For steel and concrete composite, a software analysis is carried out to determine the parameters of stress and strain curve for steel and concrete each. Therefore, two models of steel and concrete are carried out in Abaqus for the validation by taking the required properties for both the steel and concrete from the research paper.

3.2 ABAQUS Software

ABAQUS is the finite element software containing various analyses modules which are, ABAQUS standard for general nonlinear solid mechanics, ABAQUS Explicit for dynamic problems and ABAQUS Aqua for fluid mechanics, CAE and viewer for pre-processing and post-processing. Abaqus CAE is a graphical interface which enables to insert all the model details, run analysis and outlook the results. The successful model for finite element analysis can be generated only if all the details of the model are inserted in the software. Therefore, for the validation of the model, correct considerations of all the steps are required before running the analysis [22]

In ABAQUS Software the model work comprise of two factors: model geometry and model history.

3.2.1 Model Geometry

The model geometry has various sections of the model which includes the material properties of the individual sections and how they are gathered together. A mesh of nodes and elements represents particular part with particular node of specified location in a Cartesian coordinate system and, the number of these nodes represents each element.[22]

Material behaviour of the model is generated by the material definitions, inserting the required information such as nonlinear properties and elastic material of the model. For steel model, the ABAQUS allows to input Young’s modulus, Poisson’s ratio and yield stress. For concrete cube model, the material property is generated by smeared crack approach.[22]

3.2.2 Model Processing

The model processing consists of steps, loads, boundary conditions and output requests. The model requires a minimum of two steps in which the first step is being generated automatically for the boundary condition and, the second step is for the generation of loadings and outputs.

Different types of loadings like point load on nodes and distributed loads on elements can be generated. Boundary condition is formed by preventing multiple degrees of freedom at a node, and it counts on the elements used. The boundary condition is generated in order to do opposing action towards the enforced load on the structure.

In ABAQUS, the generation of results has a standard record of outputs if nothing has been defined and it consists of stress, strain and displacement properties of the structure. Then the results obtained are put to the output database for Viewer.[22]

3.3 Structural Steel modelling

For the validation of steel model, the tensile coupon is done test by taking structural steel of S275 as per BS EN ISO 6892-1:2009 in Abaqus/Explicit.

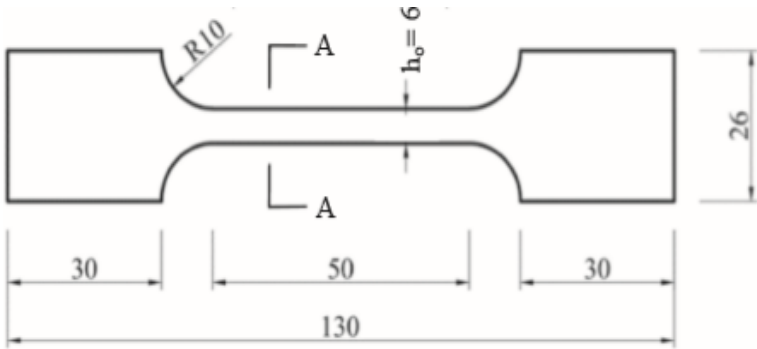


Fig 3.1 (a): Geometry and dimension of steel coupon [12]

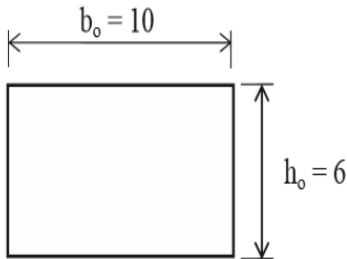


Fig 3.1 (b): Section A-A [12]

3.3.1 Member Sizes

The dimensions of coupon steel are taken from the research paper and carried out the model work as per the dimensions which were given in millimeters (mm). For the analysis of coupon steel, nominal dimensions of 10 mm wide and 6 mm deep, and the maximum available length of the coupon steel of 130 mm have been used.[12]

3.3.2 Element Used

For the modelling of the coupon steel, the element used was C3D8R which is a three-dimensional eight node element. C3D8R was preferred due to its ability to enhance the amount of convergence. [12]

3.3.3 Material Properties

Under the material options the material properties of coupon steel are enhanced and the data are being extracted from the research paper. Elastic property of the coupon steel with Young's modulus of 215000 N/mm^2 and Poisson's Ratio of 0.28 are taken. From the results obtained from the test as per the research paper, the plastic property is given with Yield strength (f_y) of 340 N/mm^2 , Yield strain (ϵ_y) of 0.0017 mm/mm, Ultimate strength (f_u) of 528 N/mm^2 , Ultimate strain (ϵ_u) of 0.2028 mm/mm and Failure strain (ϵ_L) of 0.4442 mm/mm. [12]

3.3.4 Analysis steps

For the generation of steps analysis, general static is used as it is effective for the nonlinear behaviour of the steel model. A default value of an initial time increment of 0.1, minimum time increment of 1×10^{-9} and the maximum time increment of 1 are used.

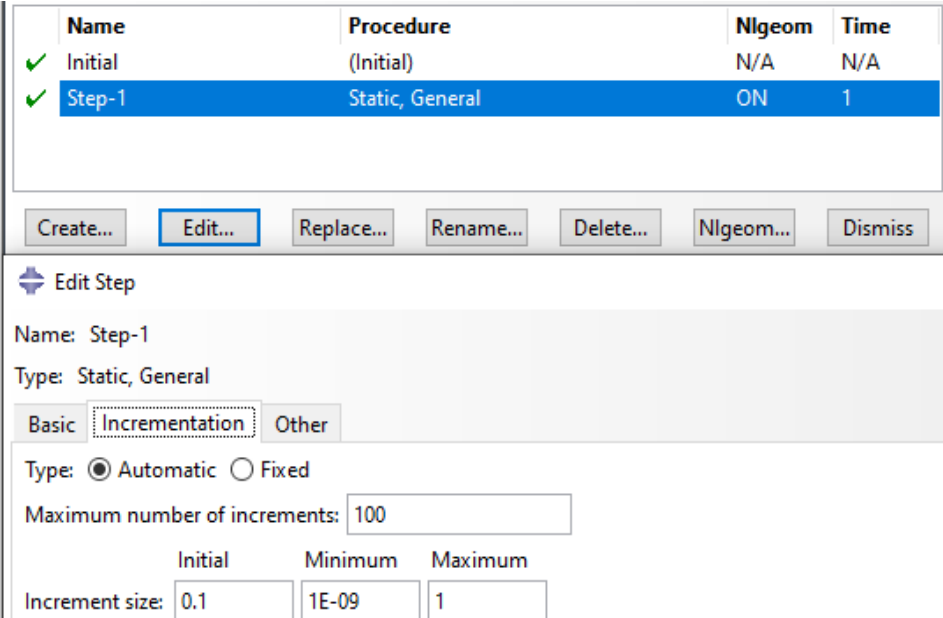


Fig 3.2: Assigning steps to steel model

3.3.5 Boundary Conditions

The boundary conditions for the steel coupon are given from the boundary options in ABAQUS. For the steel coupon model, the boundary condition is crucial for the enhancement of appropriate results of the model. At the bottom surface of the model, encastre was assigned in which all the degrees of freedom have been restricted to zero.

3.3.6 Loading Application

For the tensile test of steel coupon modelling, only one type of loading has been applied. At the top surface of the steel coupon model, a uniform distribution of pressure load of magnitude -700N/mm^2 is applied.

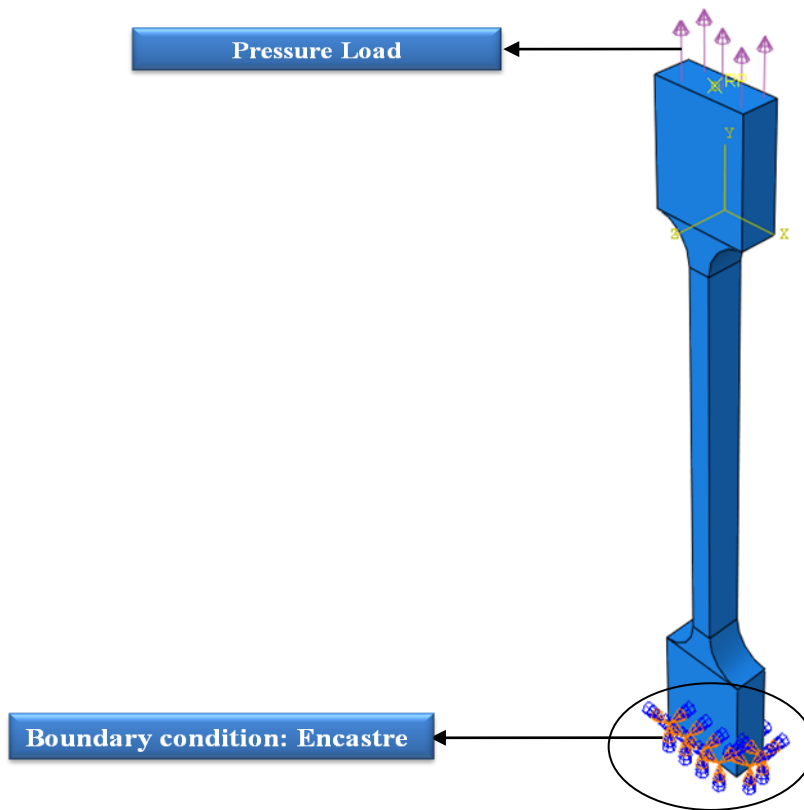


Fig 3.3: Application of boundary condition and pressure load to steel coupon

3.3.7 Meshing of the modal

An exceptional mesh is a principal problem during finite element analysis. It should be acceptable for a better feature where the data is required. For the steel coupon, the element was meshed by utilizing the seeds. The mesh must not be too fine as it consumes a large amount of time for the results. A default mesh size was taken for the enhancement of the accurate result that is a global mesh size of 2

3.4 Concrete Cube Modelling

For the validation of concrete model, the compression test on concrete cube in Abaqus/Explicit. Concrete damaged plasticity model was used. For concrete cube, M35 Grade of Concrete as per IS: 456-2000 and properties required are taken from the research paper.[9, 10]

3.4.1 Member Sizes

For the compression test, a concrete cube of dimension 150mm × 150mm × 150mm are taken and modelled in ABAQUS.

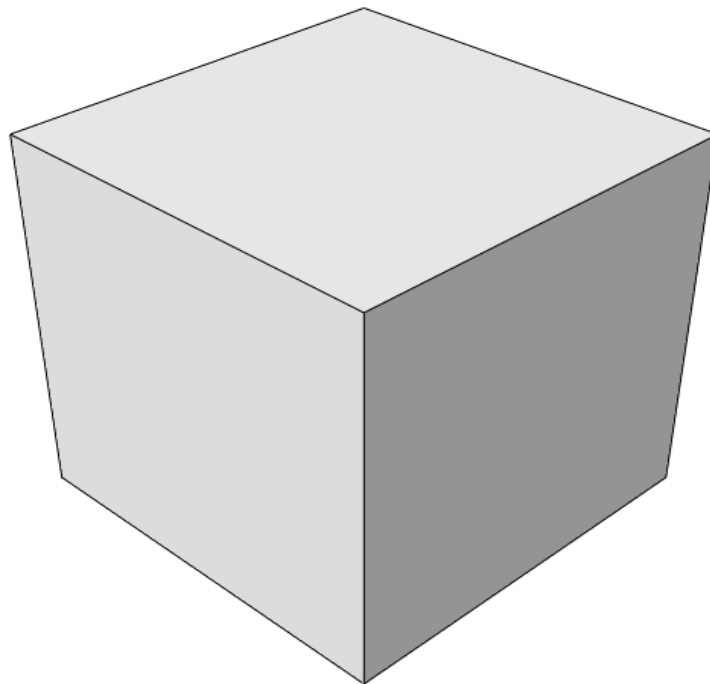


Fig 3.4: Creating concrete cube in ABAQUS/EXPLICIT [9, 10]

3.4.2 Element Used

For modelling of concrete cube model, the solid element C3D8R which is a three dimensional eight node element is used.

3.4.3 Material Properties

From the material option, mechanical properties of the concrete model have been enhanced. Elastic property of the concrete with Yong's modulus 29580.39892 N/mm² and Poison's Ratio: 0.2 are used. Modulus of elasticity of concrete is calculated according to IS: 456 as $5000\sqrt{f_c}$, N/mm², where the f_c is the compressive strength of concrete. The concrete of density 2.4E-09 Kg/mm³ and the concrete damage plasticity is used for the study of tensile cracking and possible crushing failure of the concrete in a brittle manner the principal failure

Under the CDP model, the assumed value for the dilation angle (ψ) is taken as 36° , which helps to inspect the volumetric change of concrete in the process of plastic deformation. For the flow potential eccentricity (e), the default value of 0.1 was taken and it elucidate the material has nearly identical dilation angle over a large range of enclosed pressure stress.

The default value was taken for both the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress (σ_{b0}/σ_{c0}) and Coefficient of deviatoric cross-section's shape (K_c) as 1.16 and 0.6667 respectively. For the viscosity Parameter (μ) the default value of 0 have been assigned as it is neglected in ABAQUS/Explicit. [9, 10]

3.4.4 Analysis Steps

For the generation of steps analysis, general static is used as it is effective for the nonlinear behaviour of the concrete cube model. An initial time increment of 0.1, minimum time increment of 1×10^{-5} and the maximum time increment of 1 are assigned.

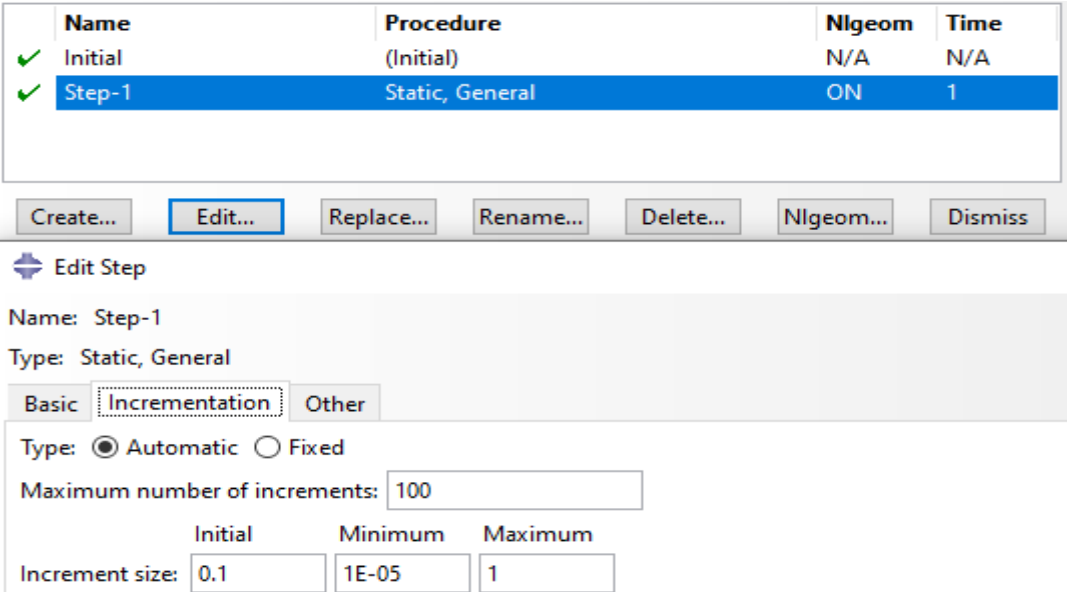


Fig 3.5: Assigning steps to concrete cube

3.4.5 Boundary Condition

For the compression test of concrete cube, boundary condition assigned to concrete cube was encastre at the bottom surface of the concrete cube model in order to restrict the movement of all the degrees of freedom.

3.4.6 Loading Application

For the simulation of concrete cube model only one type of loading which is pressure load, downward in direction with a magnitude of 100 N/mm^2 are provided and with uniform distribution along the surface.

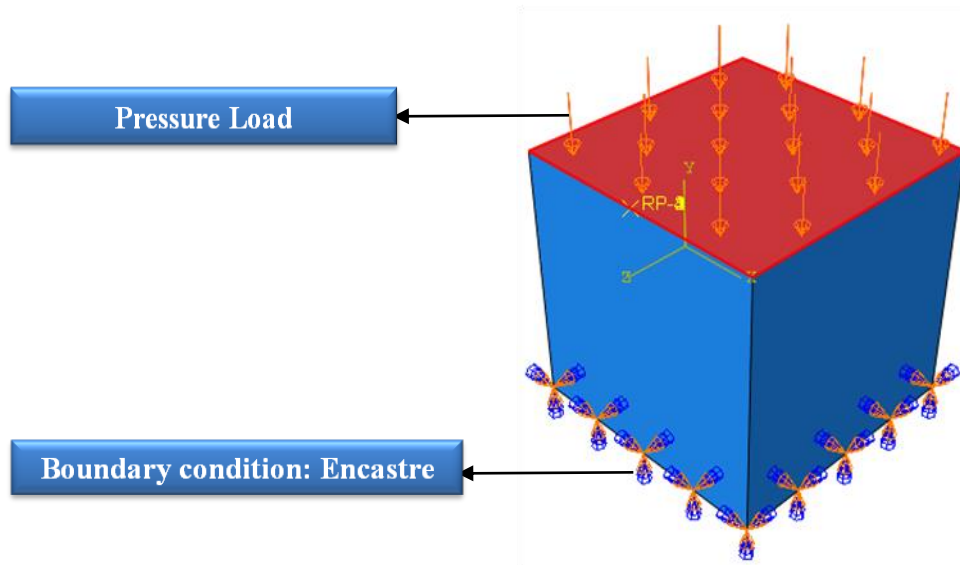


Fig 3.6: Applying Boundary condition and pressure load to concrete cube

3.4.7 Meshing of the modal

After the boundary has been assigned to the rigid bodies, the modal is meshed to simulate for a reasonable results. Therefore, the meshed modal of concrete cube with the default global size of 1 mesh size is provided.

3.5 Steel beam Modelling (Girder)

Simply supported stud connected steel-concrete composite girder is modeled using ABAQUS 6.13. First of all, steel I-girder beam was modeled followed by concrete deck modelling and then headed stud connector to enhance the interaction between the two components. After defining material property and assigning individual sections to them, the different components were assembled to act as a composite. All components which includes concrete slab, steel beam, support plates, reinforcement bars and headed shear studs are meshed by using global or sweep features. [17]

3.5.1 Member Sizes

The dimensions of the component are taken according to a reference paper [17] which includes steel beam having total depth of 306.6 mm, width of flanges as 165.7 mm, thickness of flanges as 11.8 mm and thickness of web as 6.7 mm.

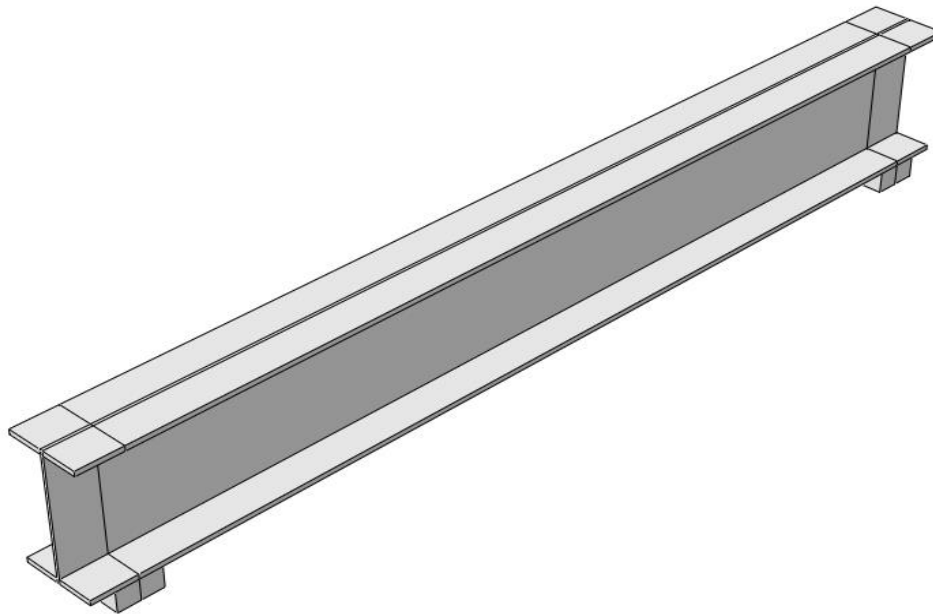


Fig 3.7: Steel beam girder in ABAQUS/Explicit

3.5.2 Element Used

For modelling of steel beam model, the solid element C3D8R which is a three dimensional eight node element is used

3.5.3 Material Properties

For the steel beam model material properties are generated from material option. An elastic modulus of 210000N/mm^2 and Poisson's ratio of 0.3. The density of 7850 kg/m^3 along with ultimate tensile strength of 510 N/mm^2 and nominal yield strength of 255 N/mm^2 has been provided. [17]

3.5.4 Analysis Steps

For the generation of steps analysis, general static is used as it is effective for the nonlinear behaviour of the steel beam girder model. An initial time increment of 0.1, minimum time increment of 1×10^{-5} and the maximum time increment of 1 are assigned.

3.5.5 Boundary Condition

The boundary condition was assigned at the support plate of the steel beam. A hinge and a roller condition are assigned on the two plates respectively.

3.5.6 Loading Application

As it's a composite model design, the loads are assigned on the composite section and not on the individual component

3.5.7 Meshing of the modal

For the steel beam girder modal, structured mesh is used with hexahedral elements. Global seeds with approximate global size of 25 mesh are used.

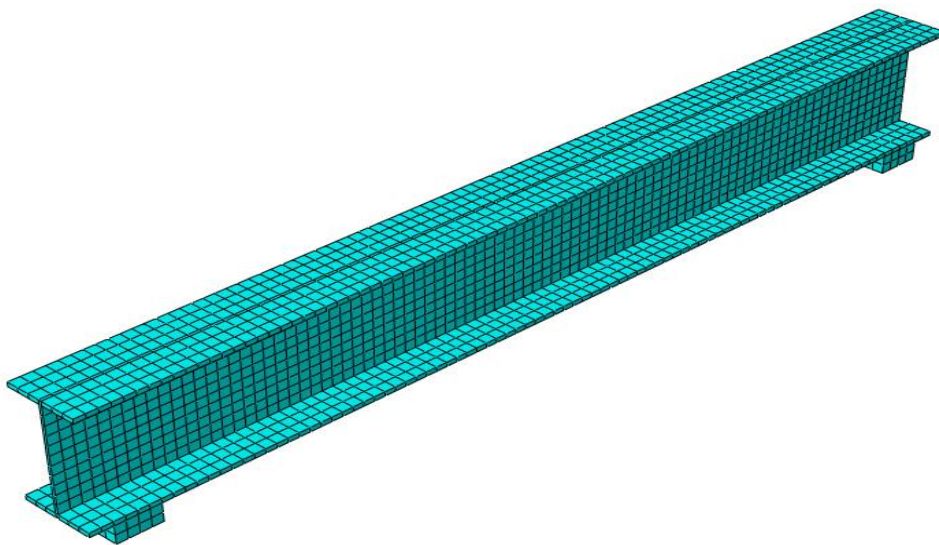


Fig 3.8: Meshing of Steel beam girder modal in ABAQUS/Explicit

3.6 Concrete Slab Modelling

After modelling the steel beam, concrete slab specimen is modelled using FE analysis by determining the required properties and accurate method of interaction between steel beam and the concrete slab. The concrete slab is placed on top of steel girder and the stud connector which is welded on the top flange of the steel beam helps in transferring the shear forces across the steel-concrete interface. [24]

3.6.1 Member Sizes

The concrete slab is modelled based on the dimensions taken from reference paper [17]. For the composite action concrete slab having width 600 mm and thickness 150 mm is considered.

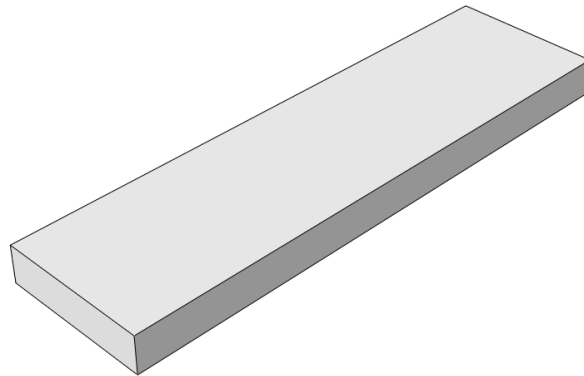


Fig 3.9: Creating a part of concrete slab in ABAQUS/Explicit

3.6.2 Element Used

In order to simulate concrete slab a three dimensional eight node element which is indicated by C3D8R is considered.

3.6.3 Material Properties

Elastic property of the concrete with Yong's modulus 30000 N/mm^2 and Poison's Ratio: 0.2 are used. The concrete of density 2500 Kg/mm^3 and the concrete damage plasticity is used for the study of tensile cracking and possible crushing failure of the concrete slab.

For the CDP model, the assumed value for the dilation angle (ψ) is taken as 35° , which helps to inspect the volumetric change of concrete in the process of plastic deformation. For the flow potential eccentricity (e), the default value of 0.1 was taken and it elucidate the material has nearly identical dilation angle over a large range of enclosed pressure stress.

The default value was taken for both the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress (σ_{b0}/σ_{c0}) and Coefficient of deviatoric cross-section's shape (K_c) as 1.16 and 0.6667 respectively. For the viscosity Parameter (μ) the value of 0.007895 have been assigned as it is neglected in ABAQUS/Explicit. [17]

3.6.4 Analysis Steps

Same as that of steel beam is considered as we they are modelled as a composite ultimately. An initial time increment of 0.1, minimum time increment of 1×10^{-5} and the maximum time increment of 1 are assigned.

3.6.5 Boundary Condition

The Boundary conditions are placed to denote a simply supported composite structure. The four edges of the top surface of the composite deck slab are provided pinned boundary condition in order to study response of the structure specimen under the effect of the loading. The pinned boundary condition basically allows rotation and restricts translation.

3.6.6 Loading Application

Equivalent pressure load with magnitude of 6000N/mm² has been provided on the top surface of the deck slab in order to generate the FE analysis of the composite under particular loading.

3.6.7 Meshing of the Modal

For the appropriate simulation of the model, the structured mesh using hexahedral elements is considered. Approximate global size of 25 is used for the proper confinement of the concrete slab model.

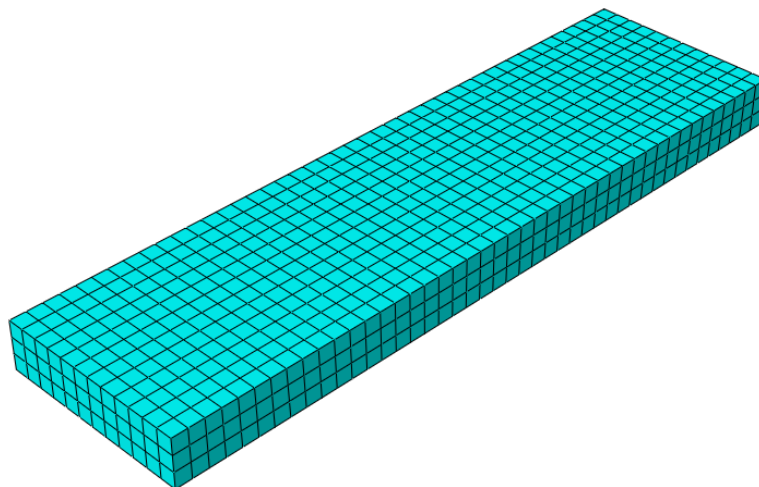


Fig 3.10: Meshing of the concrete slab in ABAQUS/Explicit

3.7 Stud Connector Modelling

The stud connectors are the principal component for carrying out the shear forces at the interface between the steel beam and concrete slab. It is the key member for the enhancement of the composite actions and is essential for protecting the composites from the vertical disconnection. [22]

3.7.1 Member Sizes

Shear stud connectors which plays a very vital role in modelling of shear interaction between concrete and steel components. The geometry dimensions of the stud are considered from reference paper. The stud that we modelled has diameter of shank as 20 mm, diameter of head as 30 mm, total length of head as 100 mm and the thickness of head as 10 mm. [17]

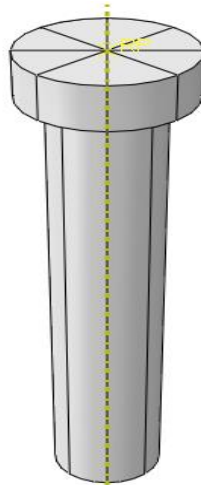


Fig 3.11: Creating a part of shear stud in ABAQUS/Explicit

3.7.2 Elements Used

Since for all the members of the composites the elements used were C3D8R, for the shear stud connector modelling the element type as C3D8R is considered..

3.7.3 Material Properties

The material properties of the shear stud specimen are given as per the reference paper with yield strength of 680N/mm^2 and ultimate strength of 900N/mm^2 . [17]

3.7.4 Analysis Steps

The analysis step which is considered for steel beam and concrete slab is considered for stud connector as well.

3.7.5 Meshing of Model

Hexahedral shaped mesh is generated for the shear stud with approximate global size of 2.

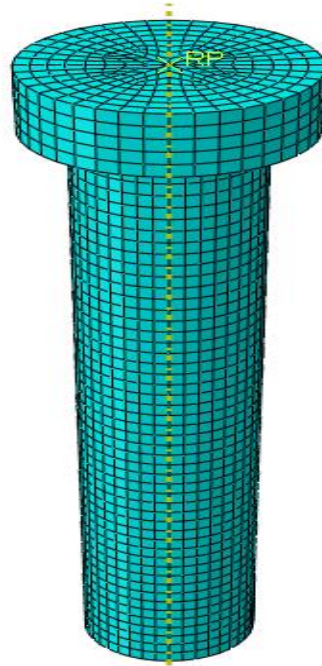


Fig 3.12: Meshing of the shear stud model

3.8 Impactor Modelling to Simulate Impact Phenomenon on composite beam

After completion of modeling the components of steel-concrete composite girder, we modelled impactor to induce low impact velocity on the composite. There are three ways possible to model the impactor. One way is to model the impactor by assuming it as a rigid body without assigning any property. It means that the impactor is provided with infinite rigidity or simply it's fully rigid. Another method is to model impactor by assuming it as a rigid body by assigning property .It means that the impactor is with real rigidity. And the last way is to consider the impactor as a deformable body with consideration of property assignment. In this paper, first method in which impactor is assumed as a rigid body with no property assigned is used.

3.8.1 Member Sizes

In this study, impactor is modelled for the simulation of t impact phenomenon on the composite. The impactor has been modelled as a lumped mass in a spherical shape and having a radius of 150 mm. The geometry dimensions considered are presented in the picture given below.

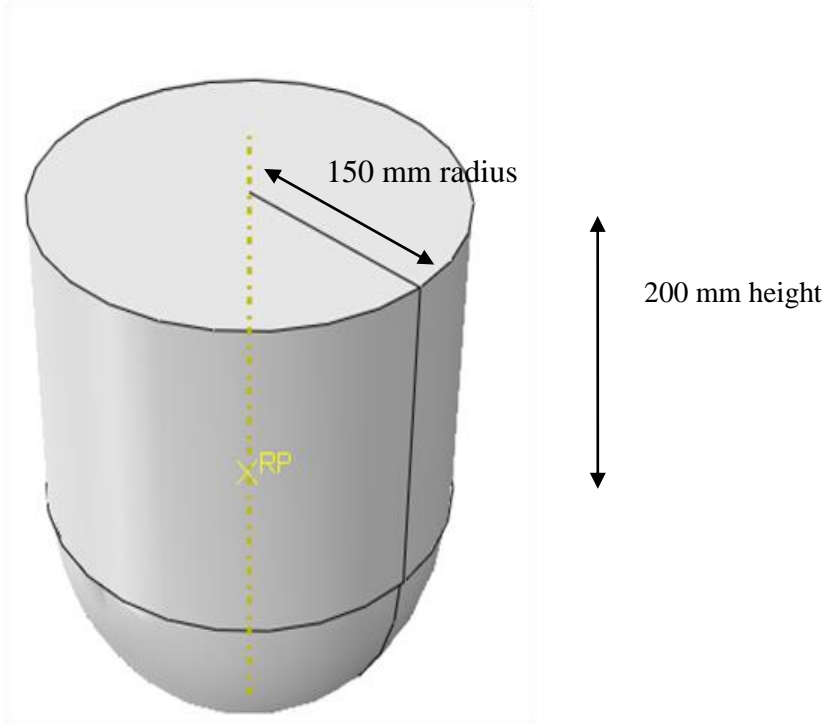


Fig. 3.13: Impactor model without mesh

3.8.2 Boundary Condition

For the impactor, displacement/rotation boundary condition is provided at initial step by selecting the reference point of the component. Similarly, velocity/angular velocity boundary condition is provided at step-1 by once again selecting the reference point of the component.

3.8.3 Meshing of the Model

The impactor is modeled with meshed using sweep technique and quad-dominated mesh is generated in the component. Global seeds having approximate global size of 10 are used.

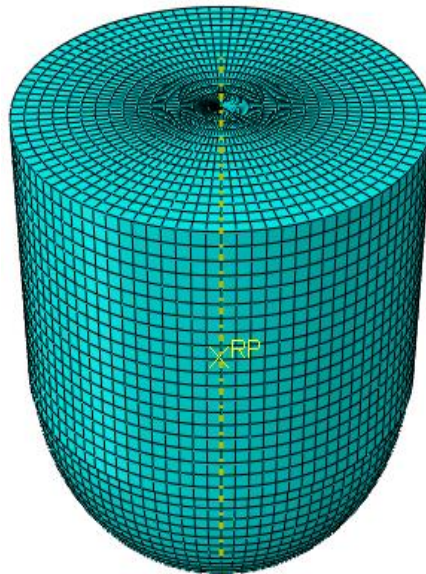


Fig.3.14: Meshed model of Impactor

3.9 Interaction Model

The welded regions of stud connector with the top flange of steel beam are modelled using tie constraints without considering any separation at weld locations. The reinforcements in both horizontal and transverse direction are provided with embedded region option. In order to represent the interaction between concrete slab and the top flange of steel beam, the surface of top flange of I-shaped steel beam is modelled as master surface and the bottom face of the concrete slab as the slave surface.

Likewise, the shank surface of concrete is modelled as master surface and the concrete surface around the shear stud is modelled as slave surface. The mechanical connection between the concrete surfaces of deck and the stud is modelled using frictional formulation in tangential direction. For tangential behaviour considered, penalty method is used by taking coefficient of friction as 0.4. [17]

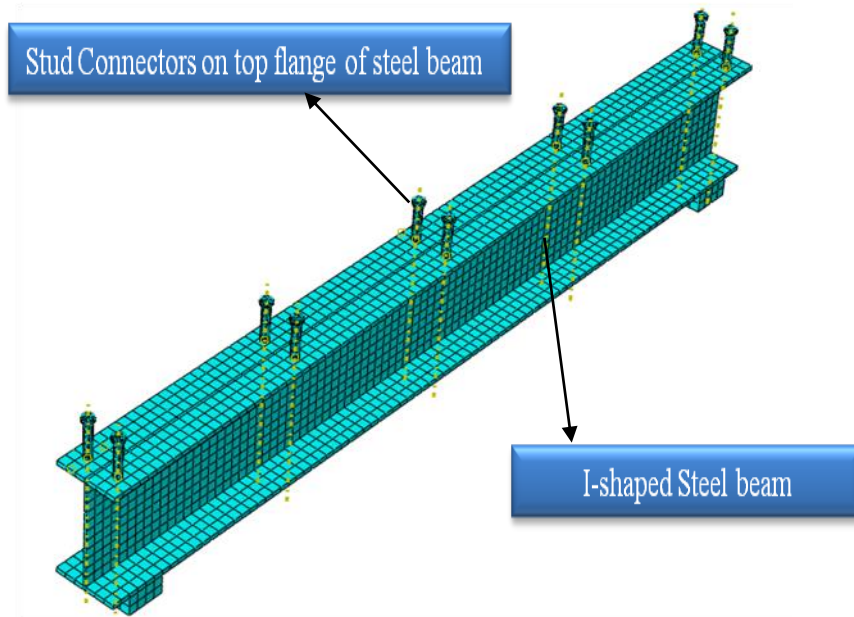


Fig 3.15: Picture showing stud connectors on top flange of steel beam

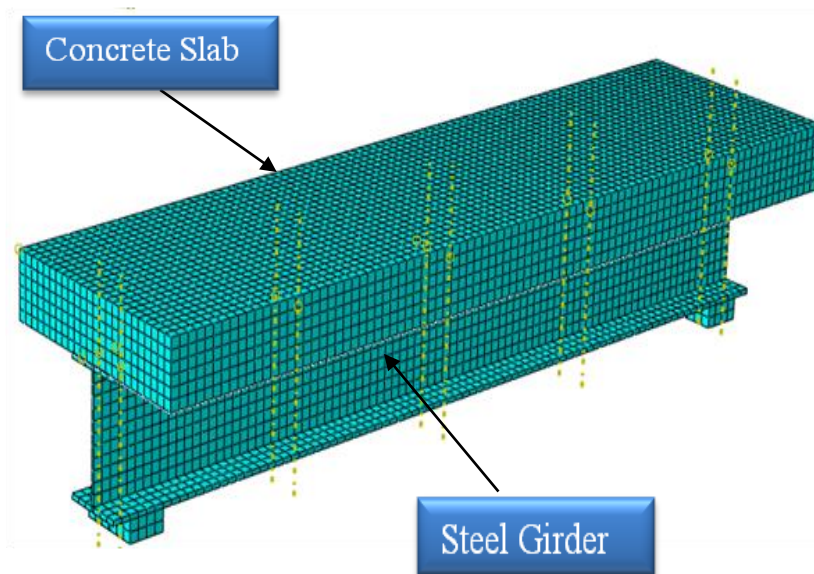


Fig 3.16: Picture showing the stud connected steel-concrete composite girder

3.9.1 Boundary Condition

Boundary conditions for the assembled components have been set up to represent a simply supported composite structure. The necessary boundary conditions considered for the model has been showcased in the figure provided below.

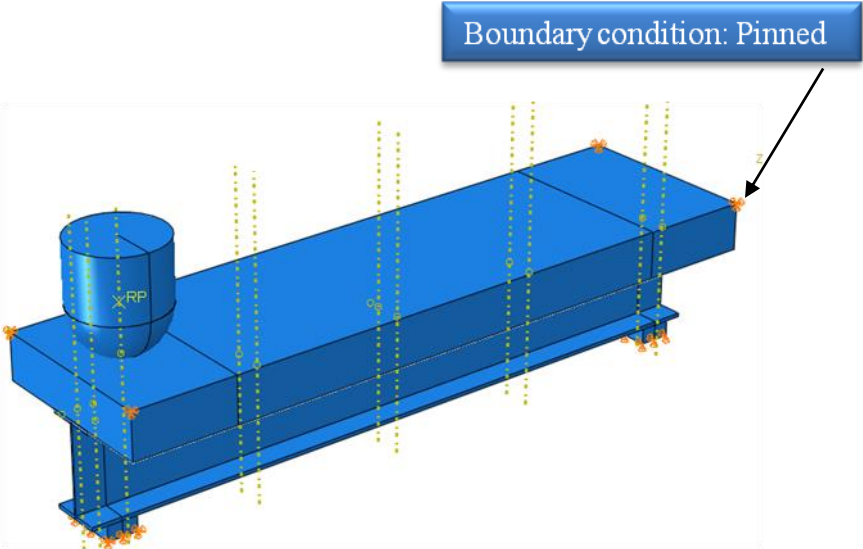


Fig.3.17: Picture showing the boundary conditions of various components

3.9.2 Impact Analysis

The impactor is considered as a rigid body for the analysis. The impactor is placed on the impacting surface of the deck slab with 400mm × 600mm.

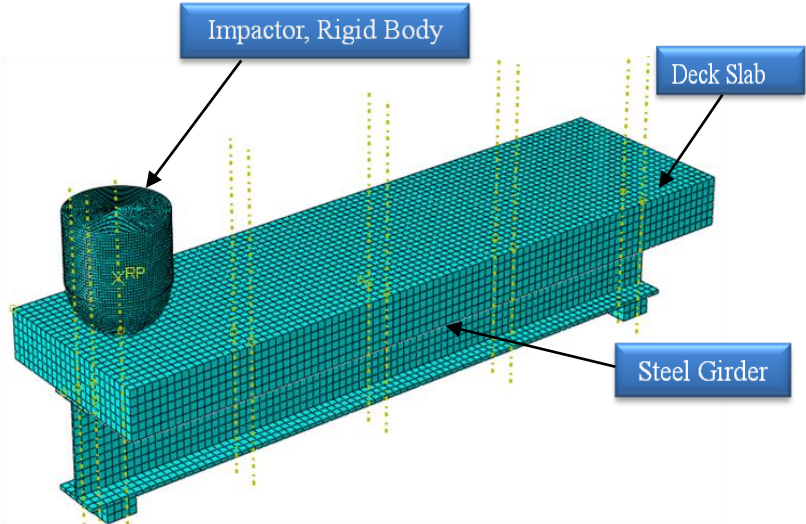


Fig.3.18: Figure of meshed model of impactor on composite surface

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

This section provides and describes the results acquired from the finite element analysis. The steel-concrete composite bridge deck with low impact velocity is modeled using ABAQUS Software in which Abaqus/Explicit is used. From the ABAQUS we can obtain the results of stress, strain and deflection as it analyse the nonlinear behaviour of the model.

The outcome of the stress from the analysis is a key to determine the ability of the parts to resist the load given. The salient feature of acquiring the plastic strain from the analysis is that it can discover whether the steel model is overloaded or over yielded. While, the outturn of the deflection can help to inspect the serviceability needed.

4.2 Tensile Coupon Test

In this section it displays the results acquired from the FE analysis of tensile coupon steel in ABAQUS/Explicit.

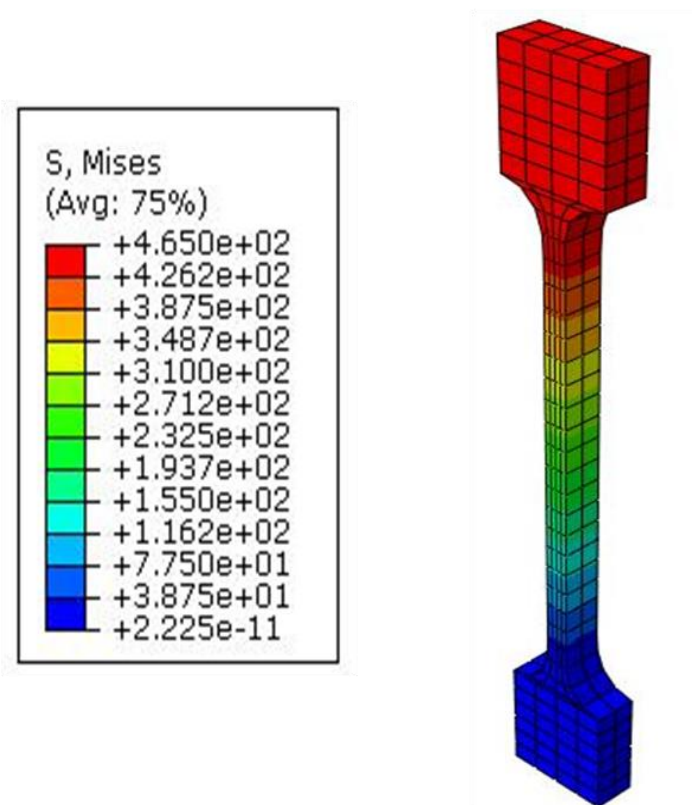


Fig 4.1: Showing the visualization of stress distribution in coupon steel at misses

Under this portion the outline of stresses and strains in the members of the model, and the displacement of the overall coupon steel model is enhanced. Therefore, the visualization output of the coupon steel model has been shown in fig 4.1.

The fig 4.1 clearly shows the details of the highest tensile stress accumulated at each element in the top surface of the coupon steel model in Von misses. The tension is determined by the red, yellow and orange region. The greatest stresses are in the regions where the direct loading is expected to be applied.

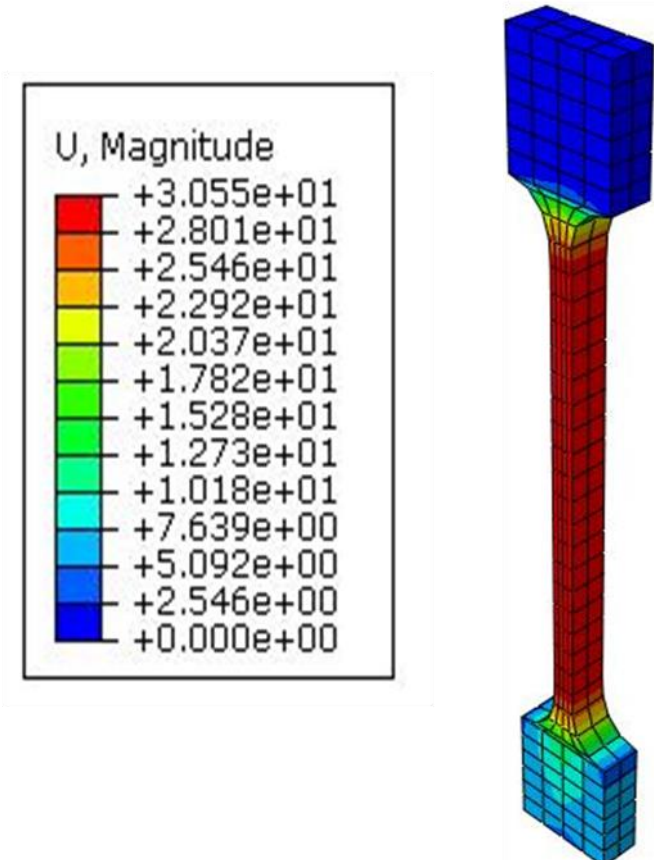


Fig 4.2: Showing the visualization of stress distribution in coupon steel at S22

The fig 4.2 exhibits the distribution of stress in the steel coupon model. From the stress placement as shown in fig 4.1, it can explained that the large amount of stress are being found in the middle areas designated with red color while the blue color specify the least amount of stress at the supports. The red color also shows that the failure has occurred at the middle region of the specimen at S22 which is global y direction.

4.2.1 Visualization Module

After the completion of generating the job in the analysis, the results in the visualization module are obtained.

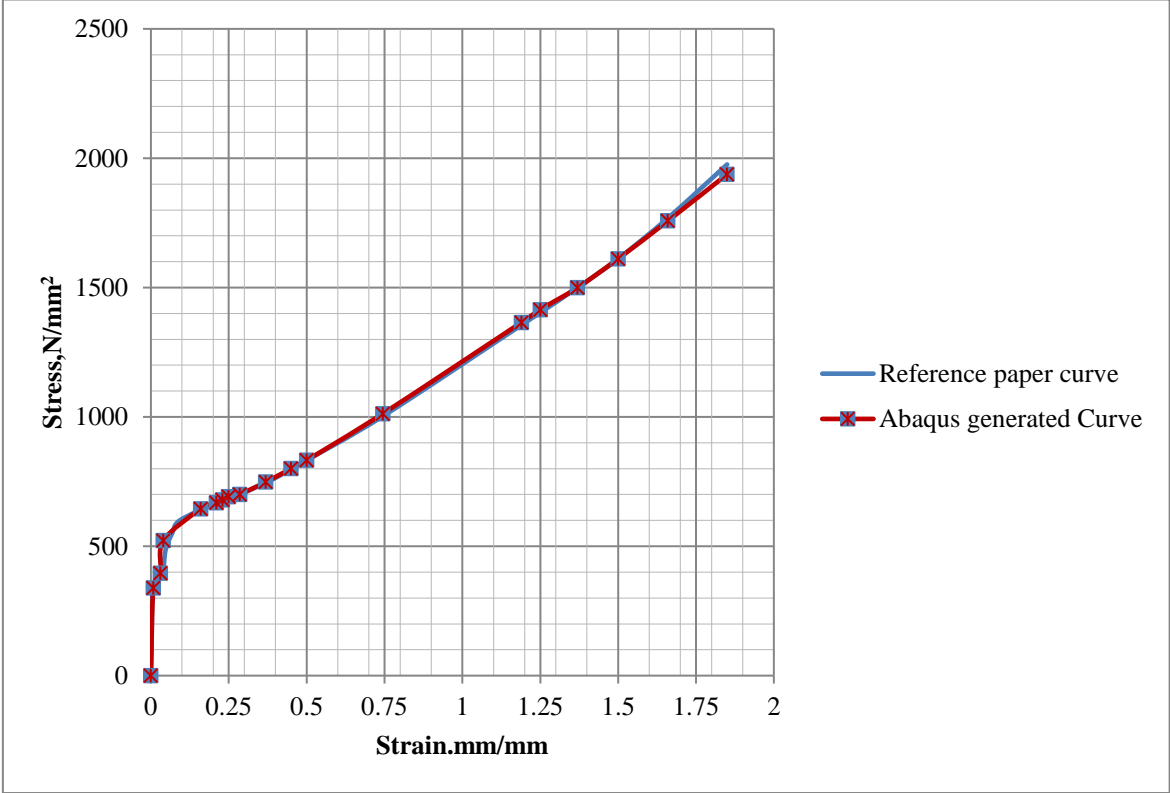


Fig4.3 (a): Comparison between analytical result and reference result of stress vs. strain

The fig 4.3(a) presents the stress vs strain curve simulated by the FE analysis of the structural steel coupon of S275. It can be observed that the maximum stress is acquired at about 1900N/mm² at 1.89 strain.

In this graph between stress vs strain, the strain rises and the curve proceeds to the right leading to a remarkable higher values of ultimate stress. Thereby, it shows a true curve of stress-strain for both the ABAQUS generated and reference paper results.

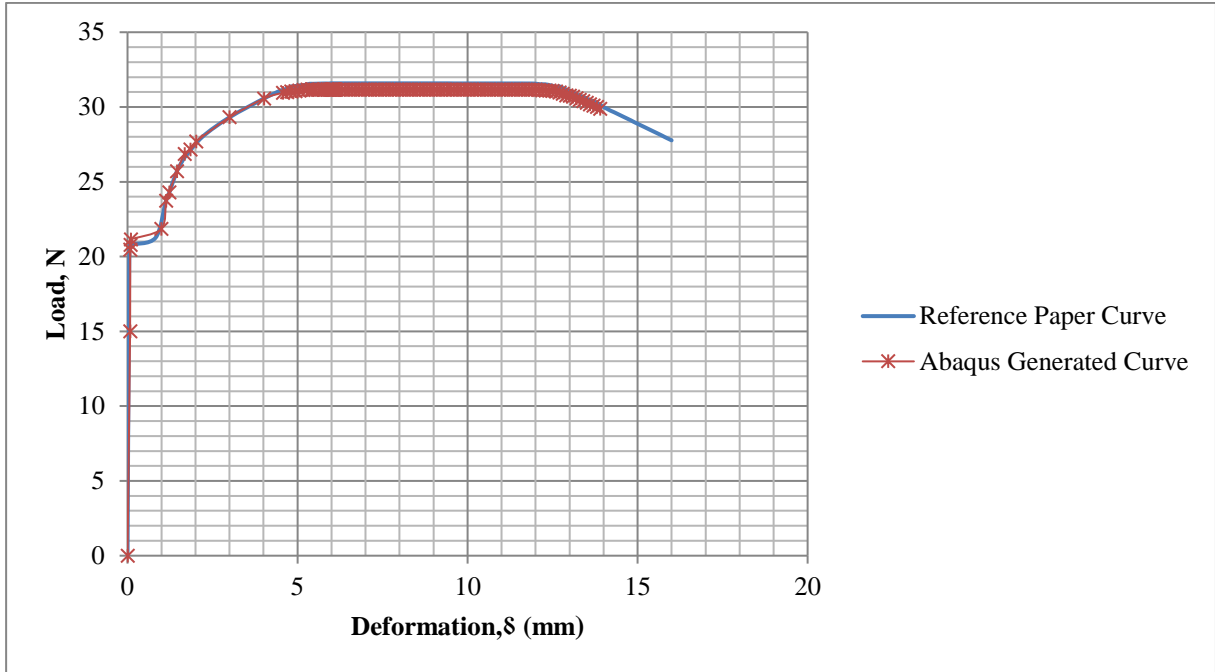


Fig 4.3 (b): Comparison between analytical result and reference result of force vs. Displacement

Fig 4.3(b) represents the load-displacement curve of structural steel S275 coupon. Primarily the load-displacement curve consists of the four regions which are elastic deformation, hardening, and necking and localized necking regions.

As the load increases about 21N beyond, the behaviour of the structure steel specimen changes. At about 31N it is observed that the ultimate strength has been achieved beyond which it clearly shows no change in the load as the displacement increases. From the above graph an elastic range about 20N and the yield point at about 22N. Therefore, almost same curve of load vs. deformation for both the reference paper and the ABAQUS generated curve are obtained.

4.3 Compression test on Concrete Cube

In this section we display the results obtained from the FE analysis of compression test on concrete cube. The Failure behaviour of the specimen has been exhibited in the fig4.4.

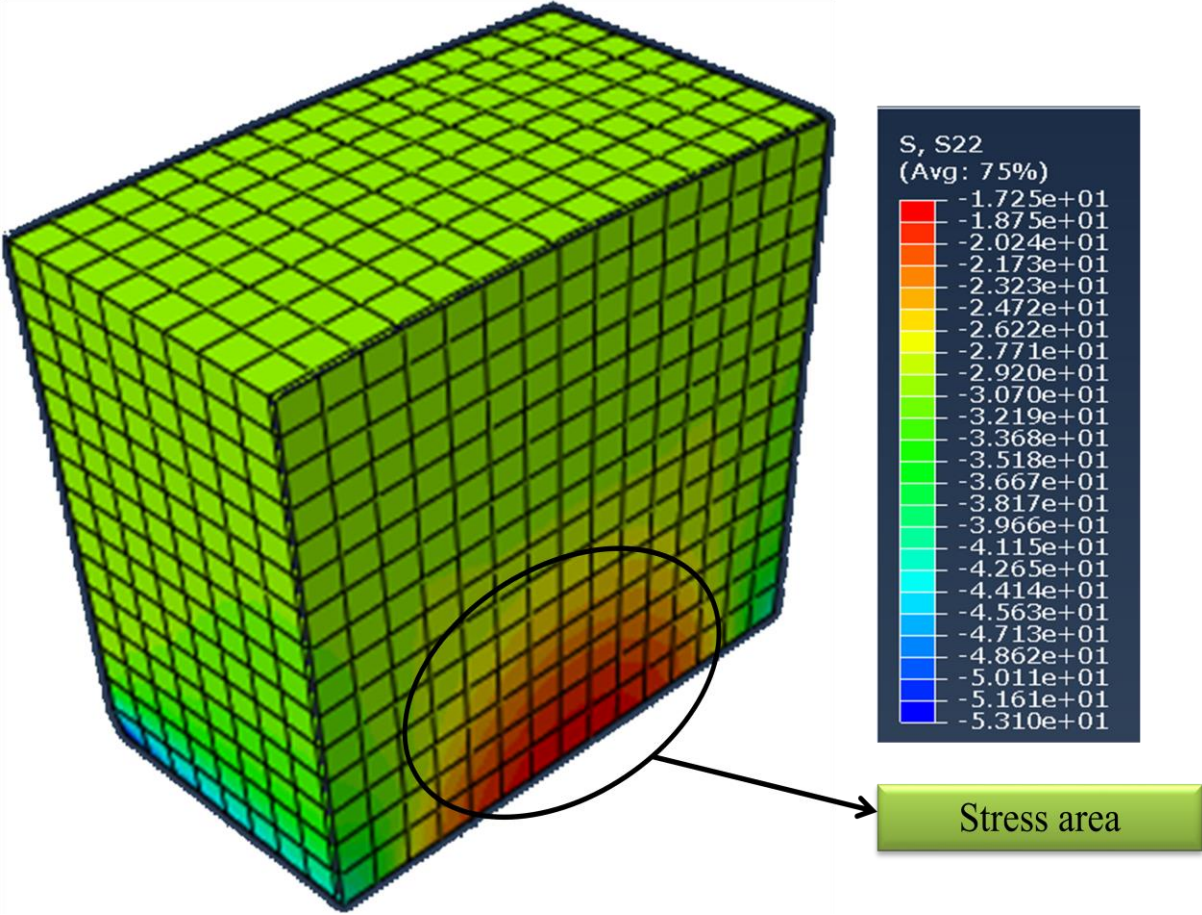


Fig 4.4: Showing the visualization of stress distribution on concrete cube at mises

The fig4.4 shows the failure mechanism of the concrete cube. The initial crack seems to show near the edges at the bottom of the concrete model and impregnating forward the centre of the concrete. The initial crack which is acquired at the bottom edges of the concrete is being designated with the red color. The green color appearing on the concrete model apart from the red color represents the areas to be meshed by the structured manner.

The yellow color which is acquired slightly between the green and red represents the regions of the concrete cube to be meshed by the sweeping meshing manner. The red color is due to the highest stresses being applied to that particular region.

4.3.1 Visualization Module

The stress/strain curve procured from the results of the analysis is described in this section. It exhibits the nonlinear behaviour of the concrete under the concrete damage plasticity approach. From the FE analysis the stress vs strain graph designates the highest stress about 41N/mm² at 0.000021 strain. In this stage, the strain increases and the stress-strain curve proceed to the right.

The comparison curve between reference paper and the ABAQUS generated for both the stress vs. strain and load vs. deformation graph shows the same results obtained.

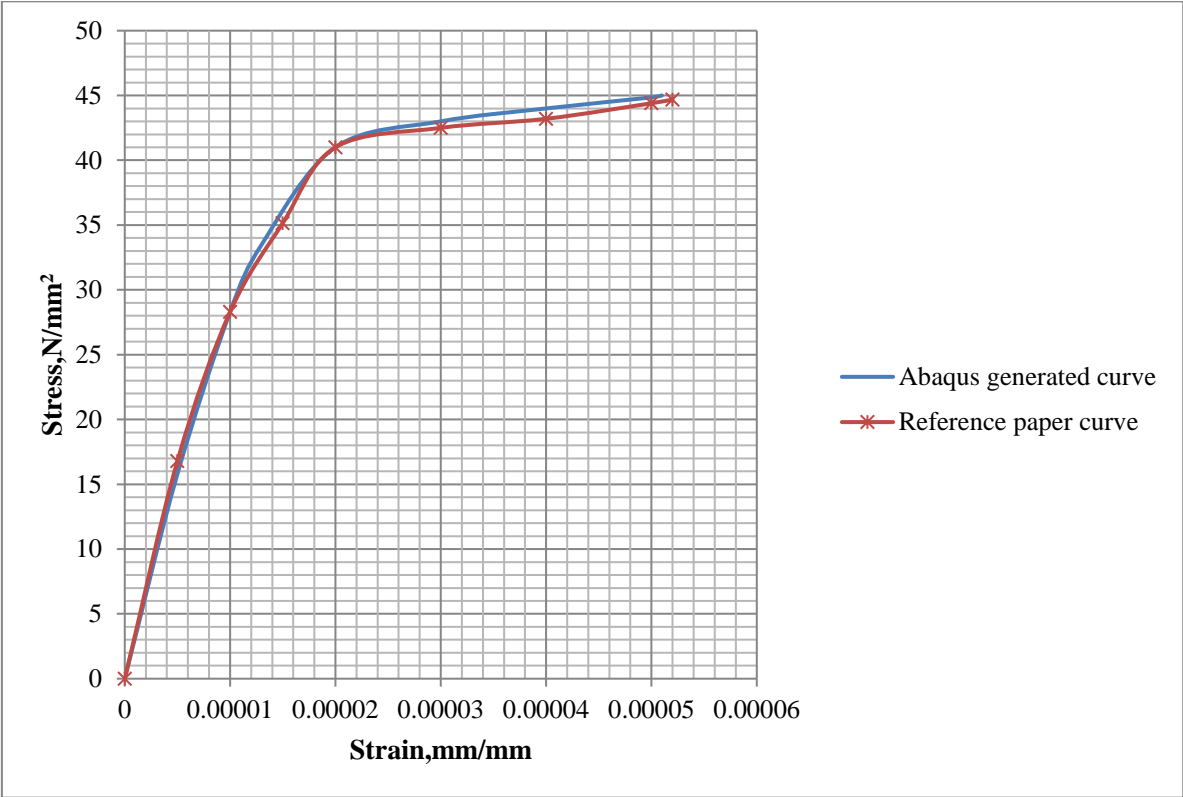


Fig 4.5 (a): Comparison between ABAQUS generated and Reference paper curve of stress vs. strain

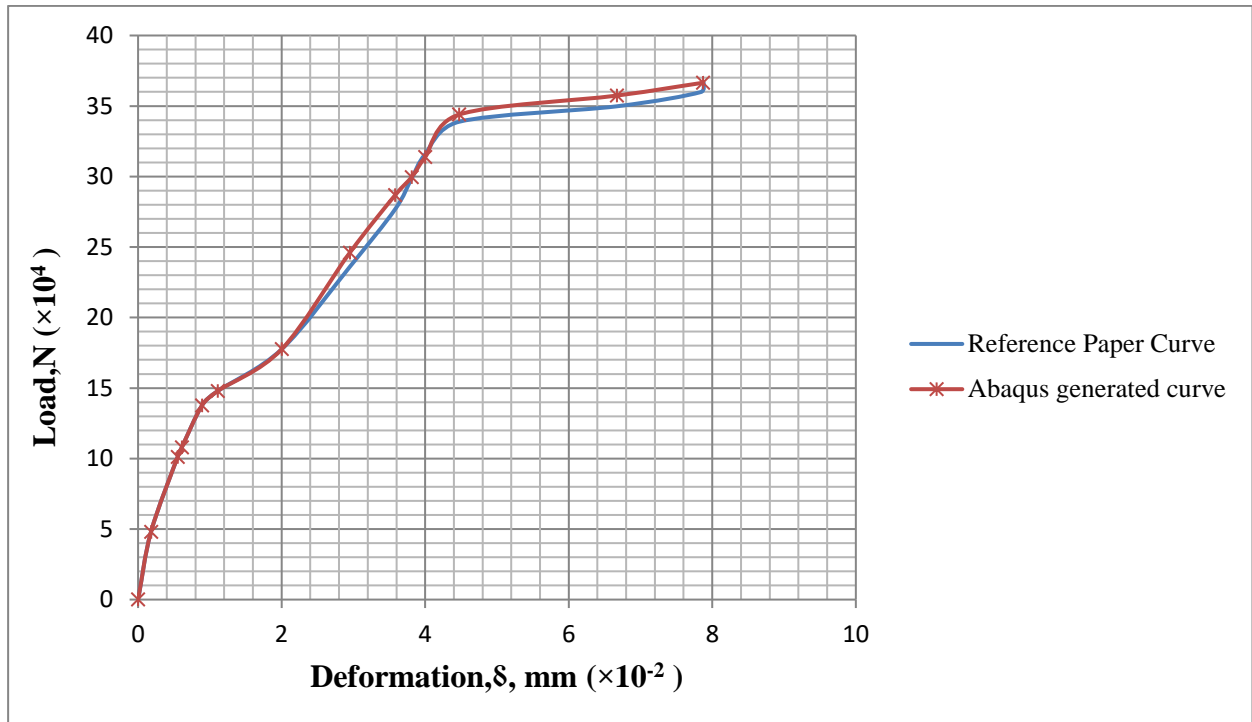


Fig 4.5 (b): Comparison between ABAQUS generated and Reference paper curve of load vs. deformation

4.4 Low Impact Velocity on Simply Supported Composite Beam

The model of the stud connected steel-concrete composite beam has been done completely considering the required loading conditions and other necessary parameters. The model of steel beam, concrete slab and of stud connector are presented in this paper along with assembled modelling which illustrates the composite structures as a whole. The boundary conditions considered and as well the interaction model of the different components has been explained properly with figures.

However, the result couldn't be shown here due to not having proper computational facility in the current situation. The trial for running the analysis in the laptop which has cracked version of software showed error of not having enough memory and recommended for running the analysis on a machine with more physical memory. Thus, the objective has been defined till here.

CHAPTER 5

CONCLUSION

Finite element analysis is a computer-aided numerical analysis technique which helps in representing the behaviour of engineering structures. In this study, the analysis of steel coupon model and concrete cube model based on Abaqus/CAE is presented. The essential properties were assigned in reference with literature survey papers and the results were validated with the datas of the reference paper as well. The theory drew a conclusion that steel-concrete composite is the most suitable combination of materials as it acquires an enhanced property of strong compressive strength from concrete beam and strong tensile strength from steel-section. The performance of steel-concrete composite member predominantly depends on connection between steel beam and concrete slab. Relatively, the very purpose of the connection at steel concrete interface is to resist the applied load as well as to transfer stresses from one component to the other.

Upon completion of objective one which has been summarized in the above paragraph, we worked on our next objective which is the design of simply supported composite beam and simulating low impact velocity on stud connected steel –concrete composite deck. The models of individual components of the composite structure which includes reinforced concrete slab, steel beam and stud connector are presented in this paper. The geometry dimensions, property definition, interaction property considered, boundary conditions taken and the assembled model has been presented in this study. Impactor which considers rigid body constraints has been modelled to simulate impact phenomenon on the composite component surface. Penalty method is being used for tangential behaviour and the coefficient of friction taken is 0.4. Once all the components are completed assembling and properties are assigned, the next very important that is to provide give connections between the separate parts in order to allow the transfer of forces and moments from one component to the other very like in the real world.

REFERENCES

- [1] CompositesLab. (2019). *What Are Composites? - Composites 101*. [online] Available at: <http://compositeslab.com/composites-101/what-are-composites/> [Accessed 10 Nov. 2019].
- [2] GS, I. (2019). *Material Science : Chapter 7: Composites – IES GS*. [Online] Iesgeneralstudies.com. Available at: <https://iesgeneralstudies.com/chapter-7-composites/> [Accessed 10 Nov. 2019].
- [3] Mbijobs.mar-bal.com. (2019). *Application*. [Online] Available at: <http://mbijobs.mar-bal.com/application/> [Accessed 21 Nov. 2019].
- [4] Mbijobs.mar-bal.com. (2019). *Application*. [Online] Available at: <http://mbijobs.mar-bal.com/application/> [Accessed 21 Nov. 2019].
- [5] TheFreeDictionary.com. (2019). *composite beam*. [online] Available at: <https://encyclopedia2.thefreedictionary.com/composite+beam> [Accessed 24 Nov. 2019]
- [6] Twi-global.com. (2019). *What is a Composite Material? – Definition and Examples*. [Online] Available at: <https://www.twi-global.com/technical-knowledge/faqs/what-is-a-composite-material> [Accessed 10 Nov. 2019].
- [7] Oosten, M. (2015). *Composite – Metal connections*. [online] pp.1-85. Available at: <https://essay.utwente.nl/69248/1/Report%20Composite-Metal%20connection%2C%20M.%20van%20Oosten.pdf> [Accessed 21 Nov. 2019].
- [8] Twi-global.com. (2019). *Welding technologies for polymers and composites*. [online] Available at: <https://www.twi-global.com/technical-knowledge/published-papers/welding-technologies-for-polymers-and-composites> [Accessed 21 Nov. 2019].
- [9] J. Carreira, D. and Chu, K. (1985). Stress-Strain Relationship for Plain Concrete in Compression. *ACI JOURNAL*, 82-72, pp.1-8.
- [10] J. Carreira, D. and Chu, K. (1986). Stress-strain Relationship for Reinforced Concrete in Tension. *Stress-strain Relationship for Reinforced Concrete in Tension*, 83-3, pp.1-8.
- [11] Ho, H., Chung, K., Liu, X., Xiao, M. and Nethercot, D. (2019). Modelling tensile tests on high strength S690 steel materials undergoing large deformations. *Engineering Structures*, 192, pp.305-322
- [12] Kumar, P., Chaudhary, S. and Gupta, R. (2017). Behaviour of adhesive bonded and mechanically connected steel concrete composite under impact loading. *Procedia Engineering*, pp.1-8.

- [13] Zheng, Y., Robinson, D., Taylor, S. and Cleland, D. (2009). Finite element investigation of the structural behaviour of deck slabs in composite bridges. *Engineering Structures*, 31(8), pp.1762-1776
- [14] Kumar, P., Patnaik, A. and Chaudhary, S. (2018). Effect of bond layer thickness on behaviour of steel-concrete composite connections. *Engineering Structures*, 177, pp.268-282
- [15] Yang, G. and Su, Q. (2011). Discussion on “Composite (steel–concrete) highway bridge fatigue assessment” by F.N. Leitão, J.G.S. da Silva, P.C.G. da S. Vellasco, S.A.L. de Andrade, and L.R.O. de Lima [J Constr Steel Res 2011; 67(1): 14–24]. *Journal of Constructional Steel Research*, 67(9), pp.1411-1412
- [16] E. Spacone, S. El-Tawil. Nonlinear analysis of steel-concrete composite structures: State of the art. *Journal of Structural Engineering*, 2004, 130(2), 159-168
- [17] Anandavalli, N., Madheswaran, C., Rajasankar, J. and Lakshmanan, N., 2012. Three Dimensional FE Model of Stud Connected Steel-Concrete Composite Girders Subjected to Monotonic Loading. *International Journal of Mechanics and Applications*, 1(1), pp.1-11.
- [18] H. Hadi, D. and H. Khaleel, H. (2015). The Low-Velocity Impact Response of Laminated Composite Plates with Holes. *Journal of Multidisciplinary Engineering Science and Technology*, Vol. 2(: 3159-0040), pp.1-8.
- [19] Yidris, N., Mustapha, F. and Sultan, M. (2014). Low Velocity and High Velocity Impact Test on Composite Materials – A review. *The International Journal Of Engineering And Science (IJES)*, || Volume || 3 ||, pp.1-11
- [20] Abdelkarim, O. and ElGawady, M. (2016). Performance of hollow-core FRP–concrete–steel bridge columns subjected to vehicle collision. *Engineering Structures*, 123, pp.517-531.
- [21] Khalili, S., Soroush, M., Davar, A. and Rahmani, O., 2011. Finite element modeling of low-velocity impact on laminated composite plates and cylindrical shells. *Composite Structures*, 93(5), pp.1363-1375.
- [22] Mirza, O. and Uy, B., 2009. Behaviour of headed stud shear connectors for composite steel–concrete beams at elevated temperatures. *Journal of Constructional Steel Research*, 65(3), pp.662-674.
- [23] El-Lobody, E. and Lam, D., 2003. Finite Element Analysis of Steel-Concrete Composite Girders. *Advances in Structural Engineering*, 6(4), pp.267-281.
- [24] Fabbrocino, G., Manfredi, G. and Cosenza, E., 2002. Modelling of continuous steel–concrete composite beams: computational aspects. *Computers & Structures*, 80(27-30), pp.2241-2251
- [25] B. Jurkiewicz, C. Meaud, L. Michel. Non linear behaviour of steel–concrete epoxy bonded composite beams. *Journal of Constructional Steel Research*, 2011, 67(3), 389-397.

JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY, WAKNAGHAT

PLAGIARISM VERIFICATION REPORT

Date: 05/06/2020.....

Type of Document (Tick): PhD Thesis M.Tech Dissertation/ Report B.Tech Project Report Paper

Name: Pema Zangmo, Parmila Gurung Department: Civil Engineering Enrolment No 161630,161695

Contact No. +97517516810, +97577203470 E-mail. zangmop113@gmail.com, parmilagurung033@gmail.com

Name of the Supervisor: Dr. Tanmay Gupta

Title of the Thesis/Dissertation/Project Report/Paper (In Capital letters): _____

INFLUENCE OF LOW VELOCITY IMPACT ON STEEL-CONCRETE COMPOSITE BRIDGE DECK

UNDERTAKING

I undertake that I am aware of the plagiarism related norms/ regulations, if I found guilty of any plagiarism and copyright violations in the above thesis/report even after award of degree, the University reserves the rights to withdraw/revoke my degree/report. Kindly allow me to avail Plagiarism verification report for the document mentioned above.

Complete Thesis/Report Pages Detail:

- Total No. of Pages =39
- Total No. of Preliminary pages =10
- Total No. of pages accommodate bibliography/references =02



(Signature of Student)

FOR DEPARTMENT USE

We have checked the thesis/report as per norms and found **Similarity Index** at15.....(%). Therefore, we are forwarding the complete thesis/report for final plagiarism check. The plagiarism verification report may be handed over to the candidate.

(Signature of Guide/Supervisor)

Signature of HOD

FOR LRC USE

The above document was scanned for plagiarism check. The outcome of the same is reported below:

Copy Received on	Excluded	Similarity Index (%)	Generated Plagiarism Report Details (Title, Abstract & Chapters)	
	<ul style="list-style-type: none"> • All Preliminary Pages • Bibliography/Images/Quotes • 14 Words String 		Word Counts	
Report Generated on			Character Counts	
		Submission ID	Total Pages Scanned	
			File Size	

Checked by
Name & Signature

Librarian

.....

Please send your complete thesis/report in (PDF) with Title Page, Abstract and Chapters in (Word File) through the supervisor at plagcheck.juit@gmail.com