SEISMIC DESIGN OF EXPANDED POLYSTYRENE CORE PANEL BUILDINGS

A DISSERTATION

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

of

MASTER OF TECHNOLOGY

in

EARTHQUAKE ENGINEERING

(With Specialization in Structural Dynamics)

By

AVIRUP SARKAR

DEPARTMENT OF EARTHQUAKE ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE-247667, INDIA

MAY, 2016

CANDIDATE'S DECLARATION

I hereby declare that the work which is being presented in this dissertation report entitled **"Seismic Design of Expanded Polystyrene Core Panel Buildings"** in partial fulfilment of **Master of Technology** in **Earthquake Engineering** with specialization in **Structural Dynamics**, submitted in the Department of Earthquake Engineering, IIT Roorkee, Roorkee, is an authentic record of my own work under the guidance of **Dr. Yogendra Singh**, Professor, Department of Earthquake Engineering, IIT Roorkee, Roorkee.

Date:

Place: IIT Roorkee (Avirup Sarkar)

CERTIFICATE

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

Dr. Shailesh Agrawal

Executive Director, Building Material
And Technology Promotion Council
Core 5A, First Floor,
Indian Habitat Centre
Lodi Road, New Delhi-11003

Dr. Yogendra Singh

Professor
Department of Earthquake Engineering
Indian Institute of Technology Roorkee
Roorkee -247667

ACKNOWLEDGEMENT

I wish to express my deep sense of gratitude and indebtedness to my learned and enlightened guide **Dr. Yogendra Singh**, Professor, Department of Earthquake Engineering, Indian Institute of Technology, Roorkee, for his instinctive and careful guidance and continued encouragement in the completion of my Dissertation. I consider myself extremely fortunate to have got the opportunity to work under his supervision.

I wish to sincerely thank my co-guide **Dr. Shailesh Agrawal,** Executive Director, Building Material and Technology Promotion Council, without whose support and guidance my project would not have been successful.

I am thankful to Mr. Dhiraj Raj and Mr. Mitesh Surana, Research Scholars, Department of Earthquake Engineering, IIT Roorkee, for their valuable suggestions and guidance.

I am extremely grateful to my parents Mr. Sanjoy Sarkar and Mrs. Madhumita Sarkar, and to my elder brother Mr. Avishek Sarkar for their endless support and encouragement, and for always believing that I can succeed in my endeavours.

I wish to express my sincere gratitude to Miss Torsha Ghosh for being a special person to me and for her endless support. I would also like to thank Mr. Aakash Khatri, Mr. B.T.M. Rao and Mr. G.N. Prajapati for being more than just friends to me and making my stay at Roorkee enjoyable for the past two years.

I am highly indebted to all my friends, faculty members and other members of the Department for their continuous support, valuable suggestions and encouragement extended throughout this dissertation work. I thank all members, who helped directly or indirectly in bringing this report to this present form.

(**Avirup Sarkar)**

ABSTRACT

Sandwich composite panel system is an upcoming, modern day construction technology that has many advantages like low-cost, light-weight and better seismic performance. Among the different types of sandwich composite panels, the expanded polystyrene (EPS) core panel based building system is the main focus of this dissertation work. A literature review is carried out on the EPS core panel system, which includes the various numerical modelling and experimental investigations carried out on the EPS core panels. The main objective of the study was to find out the seismic performance of buildings constructed using the EPS core panels.

To find out the in-plane shear strength of the EPS core panels, diagonal compression test has been performed. To find out the out-of-plane bending strength of the EPS core panels, four-point flexural load test has been performed on the panels. Finite element modelling has been performed to investigate the behaviour of the EPS core panels due to in-plane shear loading and out-of-plane flexural loading using solid element in ABAQUS and layered shell element in SAP 2000.

In order to assess the seismic performance of buildings constructed using the EPS core panels, finite element model of a G+3 building constructed using the eps core panels has been developed in SAP 2000. Response spectrum analysis has been carried out on the developed model considering Design Basis Earthquake (DBE), Zone V and medium soil conditions. The building has been subjected to all possible load combinations according to IS 1893 (Part 1): 2002. Demand forces of all the critical sections at each storey of the building due to the most critical load combinations are computed and plotted on the P-M interaction curves of the sections. It has been observed that the demand-capacity ratios of all the sections are less than 1 by a high margin, thus indicating very good seismic performance of the building. Non-linear static pushover analysis has been performed to evaluate the performance of the building due to seismic forces.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

INTRODUCTION

1.1 NEED FOR ALTERNATIVE CONSTRUCTION MATERIALS

Due to the rapid growth of population, the construction industry is facing a new challenge to construct new, cost-effective building system to satisfy the tremendous demand for low-cost housing in the world. This type of building systems must fulfil the following requirements:

1) Structurally stable construction
2) Use of prefabricated elements produced on an industrial scale (thus, low cost)
3) Fast and easy erection with unskilled labourers
4) Economical use of local materials
5) Good thermal and sound insulation

The traditional building systems like concrete, steel or prefab only partially fulfil the above mentioned requirements. Due to these inadequacies of the existent building systems, there is need for new kind of construction techniques that will meet the present needs. In the recent times, there has been inception of a few new kind of methodologies in the construction industry. Of the various types, composite panels are one that are widely used for serving the desired purpose. Composite panels generally are made of an insulating layer sandwiched two layers of welded wire mesh and some other material.

1.2 DIFFERENT TYPES OF COMPOSITE SANDWICH SYSTEMS

Composite sandwich panels are being extensively and increasingly used in building construction because they are light in weight, energy efficient, aesthetically attractive and can be easily handled and erected. In the modern day construction industry, different types of composite systems are used for construction of external walls, load bearing walls, floor slabs, roof slabs, staircase slabs, etc. A few of the different types of sandwich composite systems that are currently used in the construction industry are:

1) Sandwich panels with aluminium facesheets and PVC foam core
2) Thermoplastic composite facesheets with Expanded Polystyrene as core

3) Sandwich panels with Expanded Polystyrene as core and Oriented Strand Board facesheets
4) Composite panels with Expanded Polystyrene as core sandwiched between two layers of concrete and two layers of welded wire mesh at the interface of concrete and expanded polystyrene.

In these type of construction systems, the use of a foam type of material in the core makes the system light-weight and acts as insulation against thermal, acoustics and vibration. Our focus in this study is on the composite system with Expanded Polystyrene as core sandwiched between two layers of concrete and two layers of welded wire mesh at the interface of concrete and expanded polystyrene. The details of the above mentioned panels are discussed in the succeeding text.

1.3 CONCRETE PANELS WITH EXPANDED POLYSTYRENE (EPS) CORE

The system is a prefabricated panel, which consists of a super-insulated core of rigid EPS sandwiched between two-engineered sheets of (2.5-3.5 mm diameter (Ø) with a tensile strength of about 880 N/mm^2) steel welded wire fabric mesh. To achieve 3D panel form, another (2.5 mm diameter) galvanized steel truss wire is pierced completely through the polystyrene core at offset angles for superior strength and integrity, and welded to each of the outer layer sheets of steel welded wire fabric mesh. Finally, two layers of concrete are sprayed externally on the system to increase the performance and durability. The system can be used for numerous building applications:

i) As load bearing walls in buildings.
ii) As high capacity vertical and shear load bearing structural walling in multi-storey construction.
iii) Non Load bearing wall panels.
iv) As partition infill wall in multi-storey framed building
v) As floor/ roof slabs
vi) As cladding for industrial building
vii) As staircase panel

In general, sandwich panels behave similarly to other present concrete members. However, due to the presence of the intervening layer of insulation, sandwich panels do exhibit some unique characteristics and behaviour. Applying reinforced concrete skin to both sides of panel takes the advantages of the sandwich concept where the reinforced concrete skins take compressive and tensile loads resulting in higher stiffness and strength and the core transfers shear loads between the faces. The exterior of the panels may be finished with weather proof coating such as plaster while interior surfaces (walls) and ceilings can either be plastered or lined with conventional lining material.

The system is made up of the following components:

i) EPS core for insulation
ii) Wire mesh on inside and outside
iii) Welded truss of wire cross pieces
iv) Sprayed concrete on both sides (shotcrete) or manual concreting.

1.3.1 Details of panel components

The components of the panel i.e. the polystyrene core, the steel wire mesh and the concrete layer generally have the following properties and dimensions.

1) **Expanded Polystyrene Core-** The density of the EPS core should be greater than 15 Kg/m^3 and thickness provided is generally not less than 30mm. The EPS provided should be in accordance with BS EN-13163:2013 or IS: 4671:1984 in India to get good insulation property.
2) **Steel wire mesh-** The used steel wire mesh are made of galvanized steel wires placed on both sides of the polystyrene panel and connected by means of joints of the same material. The diameters of the steel wires used are generally 2.5-3.5 mm. Zinc coating is provided on the steel wires used and zinc coating galvanizing should be of 60 gm. /m^2 (± 5 gm. /m^2). Yield stress should be minimum 600 MPa and elongation should be greater than 8 %.
3) **Concrete cover-** Concrete grade should not be less than M25 and thickness should not be less than 35 mm on each side.

.

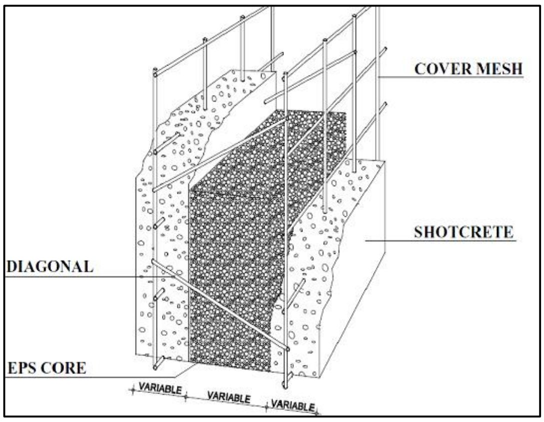

Figure 1.1: Typical section of an EPS core panel showing all three components

1.4 CONSTRUCTION METHODOLOGY OF EPS CORE PANEL BUILDINGS

The EPS core panel based buildings are a semi-precast building system and the construction process includes the following stages.

1.4.1 Construction of foundation and connection of wall to foundation

The EPS core panel buildings can be constructed on any type of foundation. For constructing the buildings on the foundation, the first step is to connect the walls to the foundation. In connecting the walls to the foundation, the first step is to cast the starter bars into the foundation as per the structural engineering requirements. The panels are placed in such a manner so that the reinforcing bars of the foundation is set between the reinforcing mesh and the polystyrene. This ensures easy and precise wall alignment. A damp proof layer is applied under the wall panel.

Figure 1.2: Erection of EPS wall panel on top of foundation

1.4.2 Erection of wall panels

Erection of wall panels always starts at the corners to achieve the required rigidity. Individual panels are connected together using a splice mesh on both sides. This is done by using a manual or pneumatic fastening tool.

1.4.3 Reinforcing panel splices

Panel walls are reinforced by splice mesh. This is required normally at the corners, between panels and around openings. This creates continuous mesh reinforcement. Reinforcing ties and bars at building element junctions to add strength to joints. Meshes are tied together using pneumatic tools.

1.4.4 Forming openings

Openings for doors and windows can be easily cut on site to specify size as per details. Splice mesh is placed generally at each corner for consolidation. Extra reinforcing bars need to be included for large openings.

1.4.5 Shoring and placing of slabs

The panels can be used as any roof or slab elements between storeys. Shoring is to be carried out using adjustable props using tripods and beams. Slab panels are reinforced with adjustable reinforcing bars at the bottom, U-bars at the supports and splice mesh at the corners. Slab panels are lifted manually and placed on top of the walls as shown in the Figure 1.3.

Figure 1.3: Placing of slab panels on top of wall panels

1.4.6 Construction of utilities

A great feature that is presented by this system is the way it accommodates utilities. Once the wall and panels are fixed in their positions, the utilities are passed between the reinforcing mesh and the polystyrene. A heat gun or propane torch is used to melt the polystyrene in places where there is a need to increase the space for constructing the utilities.

1.4.7 Shotcreting of panels and finishing

Concrete is placed on the walls and the underside of the slabs using a shotcrete pump. Hand trowel finishing of the concrete layer is required to provide the appropriate finish and surface tolerance.

Figure 1.4: Shotcreting of wall and slab panels

The final step is to apply a finishing material. Various type of finishing material can be applied, both internally and externally.

1.5 LITERATURE REVIEW

In order to find out the properties of the concrete-polystyrene-steel wire mesh system and their behaviour as a sandwich system, many researchers performed laboratory tests on the system to find out the load deflection behaviour and failure patterns of the system under compressive and flexural loading. They also analysed the different results obtained from the tests and based on that, finally presented numerical models to simulate the behaviour of the panels.

1.5.1 Compressive behaviour of concrete based EPS core panels

Carbonari *et* al. (2012) carried out two experimental programs to evaluate the compressive behaviour of panels. In the first of them, small-scale panels were tested in order to assess its local compressive strength. In the second of them, slender panels were tested to gage the global behaviour of panels subject to higher lateral instability due to buckling. In both cases, the sandwich panels tested were formed by two mortar layers with a wavy EPS layer between them. The EPS waves were 14 mm deep and 75 mm wide, being observed in a cross section perpendicular to the height of the panel and to the direction of compressive normal load applied as shown in Figure 1.5.

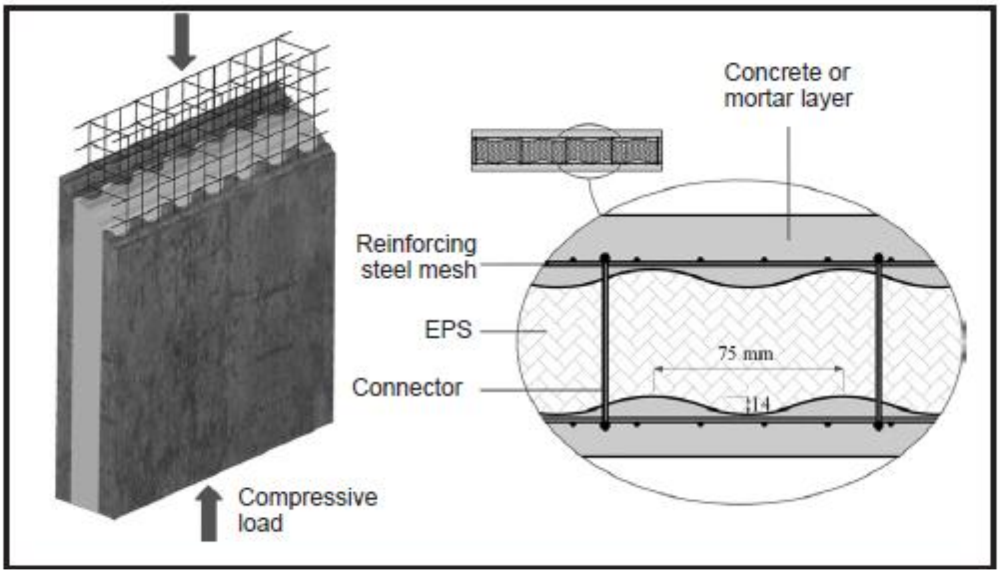

Figure 1.5: Isometric view and typical cross section (Source: Carbonari et al. (2012))

1.5.1.1 Small scale panels

A total of 38 panels with different EPS thickness (from 40 mm to 120 mm), mortar layer thickness (from 15 mm to 60 mm), panel heights (from 300 mm to 777 mm) and mortar mixes were tested. In most of the panels, both the mortar layers had the same thickness except in a few panels where the effect of variation of panel thickness was investigated by the authors. A 75 mm square reinforcing mesh composed by galvanized steel bars with 3.4 mm of diameter was placed with a cover of 7.5 mm measured from the inner side of the mortar layer. Steel connectors with the same material and 3.0 mm of diameter were placed between the reinforcing meshes at every 225 mm on the width and at every 75 mm on the length of the panels.

Table 1.1: Characteristics of materials of tested specimen (Source: Carbonari et al. (2012))

Material	Density (kg/m^3)	Elasticity modulus (MPa)	Yield stress (MPa)	Compressive strength (MPa)	Tensile strength (MPa)
EPS	25	5.9 - 7.2	-	-	0.32 - 0.41
Galvanized steel	7850	210000	620	700	700
Steel B-500 S	7850	210000	500	500	500

Table 1.2: Characteristics of small scale panels (Source: Carbonari et al. (2012))

Referen.	Mortar	Average mortar thickness (mm)		EPS thickness (mm)	Connector length (mm)	Width (mm)	Height (mm)	Number of tests
		Layer 1	Layer 2					
SC1	M1 (4.60 MPa)	15.0	15.0	80	95	500	300	2
SC2		15.0	15.0	120	135	500	300	2
SC3	M2 (39.60 MPa)	15.0	15.0	80	95	500	300	2
SC4		15.0	15.0	120	135	500	300	2
SC5		15.0	15.0	160	175	500	300	2
SC6		15.0	15.0	200	215	500	300	2
SC7		15.0	15.0	240	255	500	300	2
SC8		25.0	15.0	80	95	500	300	2
SC9		25.0	15.0	120	135	500	300	2
SC10		25.0	15.0	240	255	500	300	2
SC11	M3 (17.15MPa)	15.0	15.0	80	95	500	300	2
SC12		15.0	15.0	120	135	500	300	2
SC13	M4 (8.24 MPa)	58.5	58.5	80	95	555	475	1
SC14		60.0	60.0	80	95	530	795	1
SC15		60.0	60.0	80	95	545	483	1
SC16		60.0	60.0	80	95	525	777	1
SC17	M4-FV3.0 (9.12 MPa)	59.5	59.5	60	75	554	470	1
SC18		60.0	60.0	60	75	530	770	1
SC19		58.5	58.5	80	95	545	480	1
SC20		59.0	59.0	80	95	530	782	1
SC21	M4-FV0.6 (9.01 MPa)	58.5	58.5	60	75	550	467	1
SC22		58.5	58.5	60	75	530	770	1
SC23	M4-PF1.2 (8.51 MPa)	58.5	58.5	60	75	542	473	1
SC24		58.5	58.5	60	75	521	776	1
SC25	M5 (5.11 MPa)	22.5	22.5	40	55	550	477	1
SC26		22.5	22.5	40	55	546	776	1

Table 1.3: Results of experiments performed on small scale panels (Source: Carbonari et al. (2012))

Ref.	Mortar	Average mortar thickness (mm)		EPS thickness (mm)	Width (mm)	Height (mm)	Experimental Pmax (kN)		
		Layer 1	Layer 2				Spec. 1	Spec. 2	Avar.
SC1	M1 (4.60 MPa)	15.0	15.0	80	500	300	98.0	96.0	97.0
SC2		15.0	15.0	120	500	300	111.0	91.0	101.0
SC3	M2 (39.6 MPa)	15.0	15.0	80	500	300	493.0	468.0	480.5
SC4		15.0	15.0	120	500	300	288.0	243.0	265.5
SC5		15.0	15.0	160	500	300	419.0	352.0	385.5
SC6		15.0	15.0	200	500	300	303.0	276.0	289.5
SC7		15.0	15.0	240	500	300	293.0	237.0	265.0
SC8		25.0	15.0	80	500	300	543.0	471.0	507.0
SC9		25.0	15.0	120	500	300	253.0	228.0	240.5
SC10		25.0	15.0	240	500	300	264.0	-	264.0
SC11	M3 (17.15MPa)	15.0	15.0	80	500	300	208.8	204.0	206.4
SC12		15.0	15.0	120	500	300	191.0	152.0	171.5
SC13	M4 (8.24 MPa)	58.5	58.5	80	555	475	732.0		
SC14		60.0	60.0	80	530	795	528.0		
SC15		60.0	60.0	80	545	483	802.0		
SC16		60.0	60.0	80	525	777	589.0		

1.5.1.2 Full scale panels

In the second experimental program one panel with 100 mm thick EPS and one panel with 60 mm thick EPS were tested. Both of them had 2.55 m of height and 0.90 m of length. The thicknesses of the mortar layers were 40 mm and 50 mm each. The compressive strength of the mortar used was 25 MPa. A reinforcing steel hoop was cast in the upper and in the lower ends of the panel in order to reduce the incidence of local damages and to simulate the layout usually found in the joint between panels and slabs or foundations. The reinforcing mesh was composed by galvanized steel bars that on the vertical direction presented 3.4 mm of diameter and were placed at every 5 cm whereas on the horizontal direction presented 3.0 mm of diameter and were placed at every 15 cm. In addition to that, 5 vertical steel bars of the type B-500 S with 6 mm of diameter were disposed coinciding with the position of the steel connectors in order to provide more stability to the reinforcement during the production of the panels. Galvanized steel connectors with 3.0 mm of diameter were spaced by 21.5 cm on the horizontal direction and by 15 cm on the vertical direction. The material properties of the EPS and reinforcing bars were same as that used for small scale panels.

Figure 1.6: Compression test setup (Source: Carbonari et al. (2012))

During the test, a group of jacks applied the compressive load to 3 cm thick steel plates that were in contact with the ends of the panels up to the failure.

1.5.1.3 Analysis of results

In the case of small scale panels, during the test, a crack was generally observed parallel to the lateral surface of the panel. The crack opening increased as the compressive load was applied, leading to the failure of the panel. It was also observed that the increase in the thickness of the EPS layer resulted in a decrease of the maximum load resisted. Furthermore, it was also observed that for the panels with same properties, the increase in the compressive strength of mortar leads to an increase of the maximum load resisted. In general, the increase of this dimension produced a considerable reduction of the maximum load. According to the authors, such response may be attributed to the increase of the bending moment due to test imperfections and second order moments in higher panels. The use of different mortar layer thicknesses (15 and 25 mm) in panels SC8, SC9 and SC10 did not cause any significant variation regarding the maximum load measured in the test of panels SC3, SC4 and SC7, which present the same thickness for both mortar layers (15 mm). Apparently, the cross section increase in the former does not affect the failure mechanism that is more dependent of the EPS thickness (length of the connectors) and the minimum mortar layer thickness.

In the case of full scale panels, the typical failure observed was similar to that of the small-scale panels with a higher tendency to lateral instability. It was verified that the increase of the EPS thickness and of the length of the connectors led to a decrease of the maximum load measured. This confirmed the trend already observed in the test with small-scale panels.

1.5.2 Flexural behaviour of concrete based EPS core slab panels

Bajracharya *et* al. (2011) performed four point flexural loading tests on the panels to find out the out-of-plane load carrying capacity of the panels.

1.5.2.1 Details of tests performed

The testing specimens included three separate sandwich panels. The diameter of the steel bar was 3 mm with the grid size of 50.8 mm x 50.8 mm. The first panel was tested without any bottom longitudinal reinforcement bars. The second and the third panels were tested with the addition of longitudinal reinforcement bars of 9.53 mm and 12.7 mm respectively.

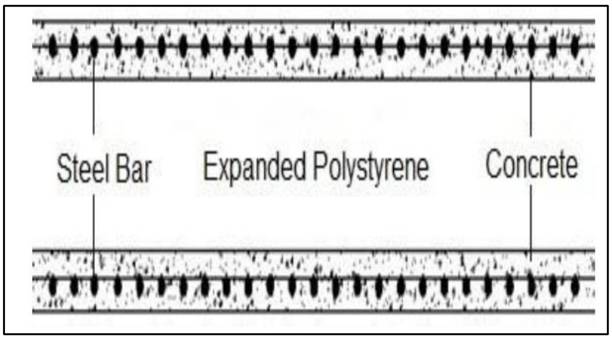

Figure 1.7: Schematic diagram of a tested panel (Source: Bajracharya et al. (2011))

Table 1.4: Details of test specimen (Source: Bajracharya et al. (2011))

Length (mm)	Breadth (mm)	Thickness of core (mm)	Thickness of skin (mm)	Total Thickness (mm)
3098.8	1219.2	127	44.45	215.9

Table 1.5: Compressive strength of concrete used in the three cases (Source: Bajracharya et al. (2011))

Case	Strength (MPa)
1. Without Reinforcement bars	19
2. With Reo bars of 9.53 mm	10
3. With Reo bars of 12.7 mm	10

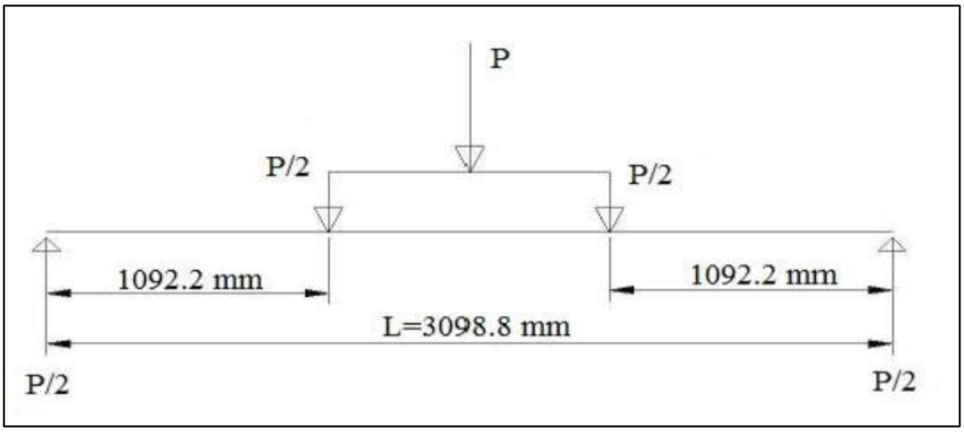

Figure 1.8: Schematic diagram of test setup (Source: Bajracharya et al. (2011))

1.5.2.2 Results obtained and comparison with numerical modelling

The authors modelled the concrete skin as 8 noded solid brick elements and the steel as one-dimensional cut-off bar elements in the finite element software Strand 7 to simulate the behaviour of the panels and made comparative study on the results obtained from the experiment and that obtained from the numerical model. The obtained graphs are shown in Figure 1.9, Figure 1.10 and Figure 1.11.

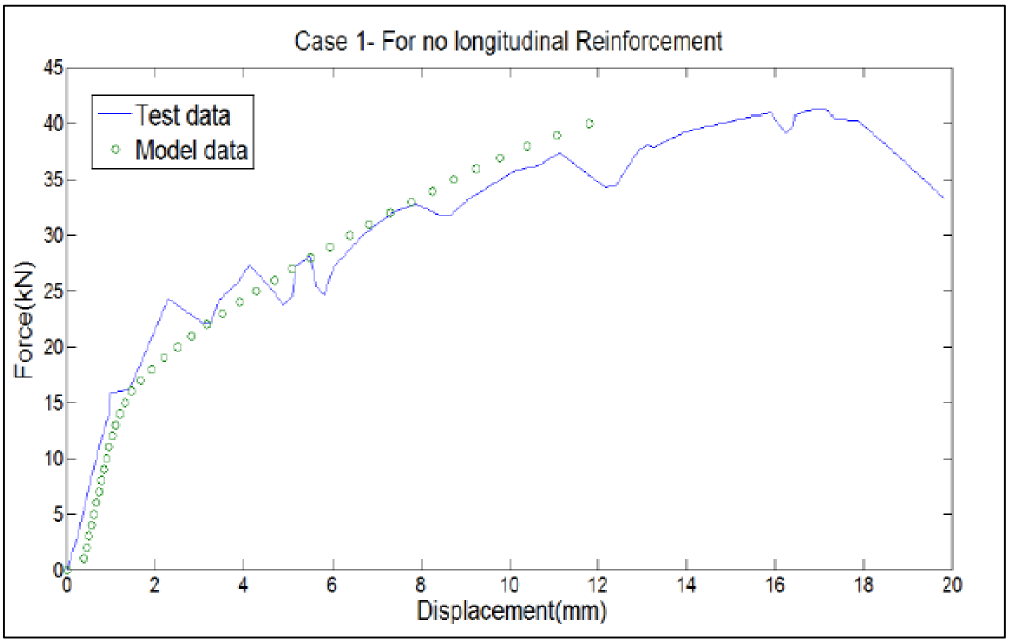

Figure 1.9: Force-displacement behaviour for the case with no longitudinal reinforcement (Source: Bajracharya et al. (2011))

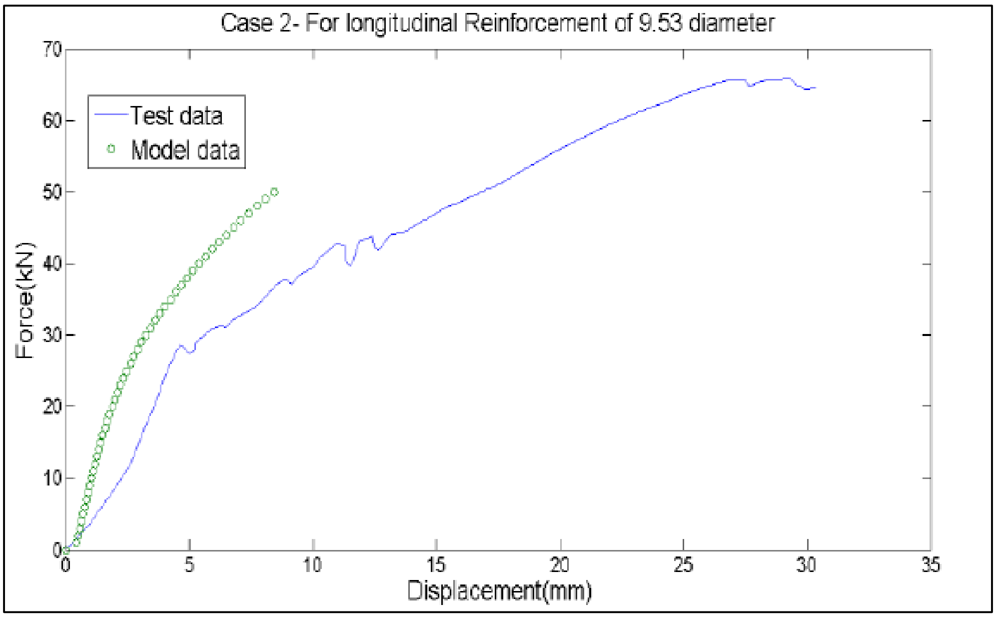

Figure 1.10: Force-displacement behaviour for the case with 9.53mm diameter longitudinal reinforcement bars (Source: Bajracharya et al. (2011))

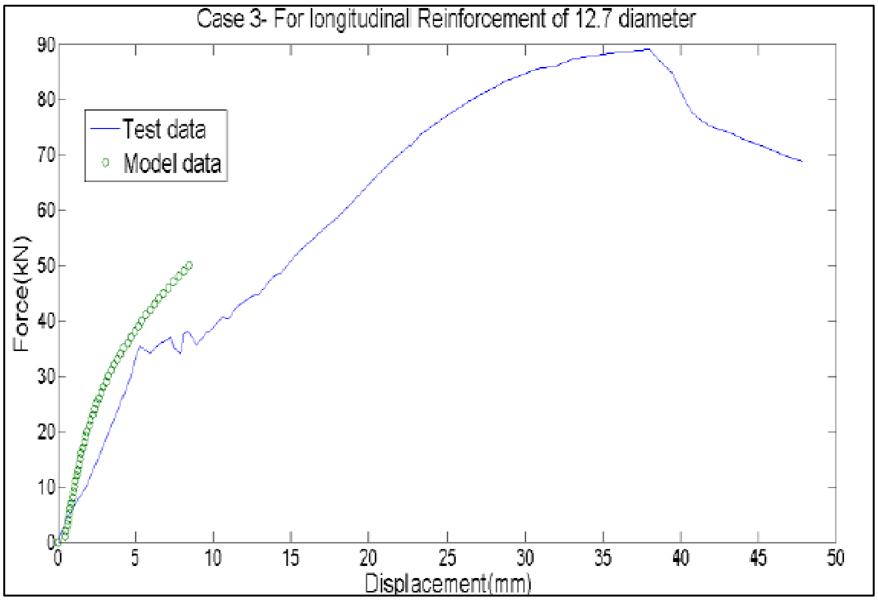

Figure 1.11: Force-displacement behaviour for the case with 12.7 mm diameter longitudinal reinforcement bars (Source: Bajracharya et al. (2011))

From the graphs obtained, it is observed that the numerical model and experimental results are almost similar. Thus, the numerical model developed by the author is validated by the experimental results.

1.6 OBJECTIVES OF THE DISSERTATION

After performing literature review, it has been observed that there is a lack of study on the compressive, out-of-plane bending and in-plane shear behaviour of EPS core panels. There is no study available on the seismic response of buildings constructed using EPS core panels. The objectives of the present study are:

1) To find out the shear strength of EPS core panels by diagonal compression test.
2) To simulate the diagonal compression behaviour of EPS core panels by finite element analysis.
3) To find out the out-of-plane bending strength of EPS core panels using four-point flexural load test.
4) To simulate the out-of-plane bending behaviour of EPS core panels using finite element analysis.

5) To estimate the seismic performance of a typical building constructed using the EPS core panels.

1.7 METHODOLGY AND SCOPE OF WORK

The dissertation work can be mainly divided into three main parts: 1) Experimental investigations of in-plane shear and out-of-plane flexural load carrying capacities of small scale concrete based EPS core panels 2) Numerical simulation of in-plane shear behaviour and out-of-plane flexural behaviour of concrete based EPS core panels by finite element analysis and then validation of the numerical analysis by the results of the experiments performed and 3) Investigation of seismic performance of a typical building constructed using the EPS core panels by finite element modelling.

1.8 ORGANIZATION OF THE DISSERTATION

The dissertation work has been organized as follows:

Chapter 1 contains an introduction on the EPS core panels, the construction methodology of the buildings constructed using the EPS core panels, a literature review on the compressive and flexural behaviour of the panels and finally presents the objectives of the dissertation.

Chapter 2 presents the evaluation of the in-plane shear force carrying capacity of the concrete based EPS core panels by diagonal compression tests performed on small scale panels and numerical modelling of the in-plane shear behaviour of the panels by finite element analysis.

Chapter 3 consists of the evaluation of the out-of-plane bending behaviour of the concrete based EPS core panels by four-point flexural load test and numerical simulation of the same by finite element analysis.

Chapter 4 presents the evaluation of seismic performance of a typical building constructed using the EPS core panels by finite element analysis.

Chapter 5 contains the concluding remarks of the dissertation.

EVALUATION OF IN-PLANE SHEAR STRENGTH OF EPS PANELS

To evaluate the in-plane shear strength of the eps core panels, diagonal compression test is performed on small scale panels. To simulate the in-plane shear behaviour of the panels, diagonal compression test is modelled in the finite element software SAP 2000 using layered shell element and later it is modelled in the finite element software ABAQUS using solid 8-noded hexahedral element.

2.1 DIAGONAL COMPRESSION TEST PERFORMED ON EPS CORE PANEL

In order to estimate the in-plane shear strength of EPS core panels and to validate the results of the numerical analysis, diagonal compression test was performed on the panels. Due to lack of codes on the testing procedure of composite panels, the procedure followed was according to ASTM E-519/E-519 M-10 (ASTM 2010a) which describes the testing procedure and setup for diagonal compression test of masonry. Three square specimen were tested, the length and breadth being 0.77 metres and thickness being 0.15 metres. The total thickness of the panels i.e. 150 mm consists of 80 mm EPS at the core sandwiched between two layers of concrete, each 35 mm thick. The concrete grade used is M25 and steel wire mesh used has yield strength of 700 MPa. The diagonal compression load was applied on the opposite corners of the specimen using the 250 T capacity INSTRON closed loop universal testing machine (UTM), available at the Structural Engineering Laboratory of Civil Engineering Department, IIT Roorkee. Displacement controlled diagonal loading was applied to the specimen through two steel loading shoes, placed at the top and bottom corners of the specimen. These loading shoes were fabricated according to the dimensions provided in ASTM E-519/E-519 M-10 (ASTM 2010a). The rate of loading used for the test was 0.2 mm /minute. In order to find out the shortening and expansion of the diagonals due to the loading, two linear variable differential transducers (LVDTs) were placed along the two diagonals at a gauge length of 380 mm. A screw and clamp arrangement was used to directly attach the LVDTs to the specimen; one LVDT was placed in the front along the vertical diagonal and the other LVDT was placed in the backside along the horizontal diagonal. The LVDTs were connected to the same data acquisition system as used for measurement of the applied load. The specimen and the loading setup is shown in the Figure 2.1.

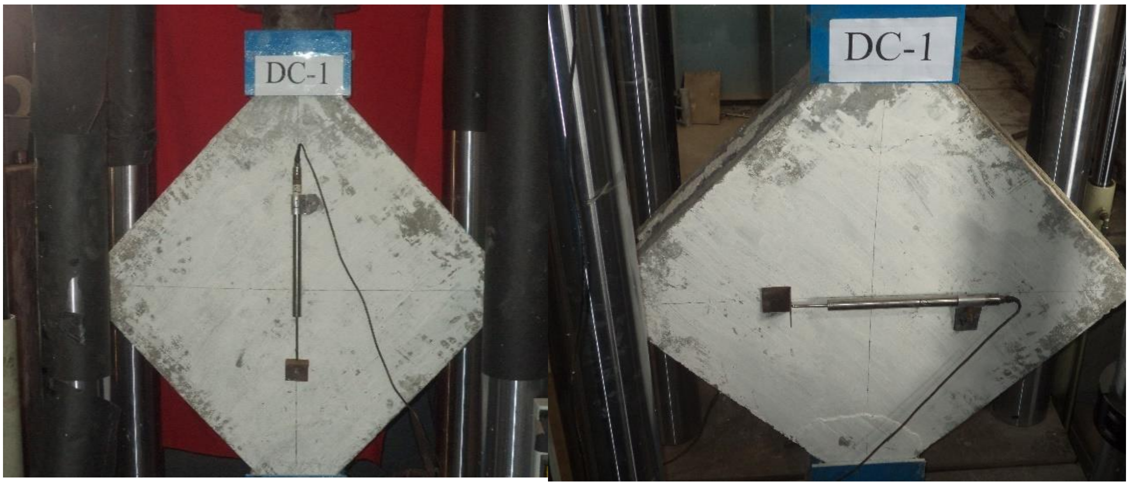

Figure 2.1: Specimen 1 of diagonal compression test

The cracks were observed along the vertical diagonal due to tension failure of concrete. Similar type of cracks were seen in all the three specimen, thus inferring that in diagonal compression test of EPS panels, the governing failure mode is tensile failure of concrete along the loaded diagonal.

Figure 2.2: Crack pattern observed in diagonal compression test

The Figure 2.2 shows the crack pattern observed in diagonal compression test. Cracks were formed along the loaded diagonal due to tensile cracking of concrete. Similar cracks were obtained in all the three tested specimen. The load versus vertical displacement curve as obtained for the three specimen are shown in the Figure 2.3.

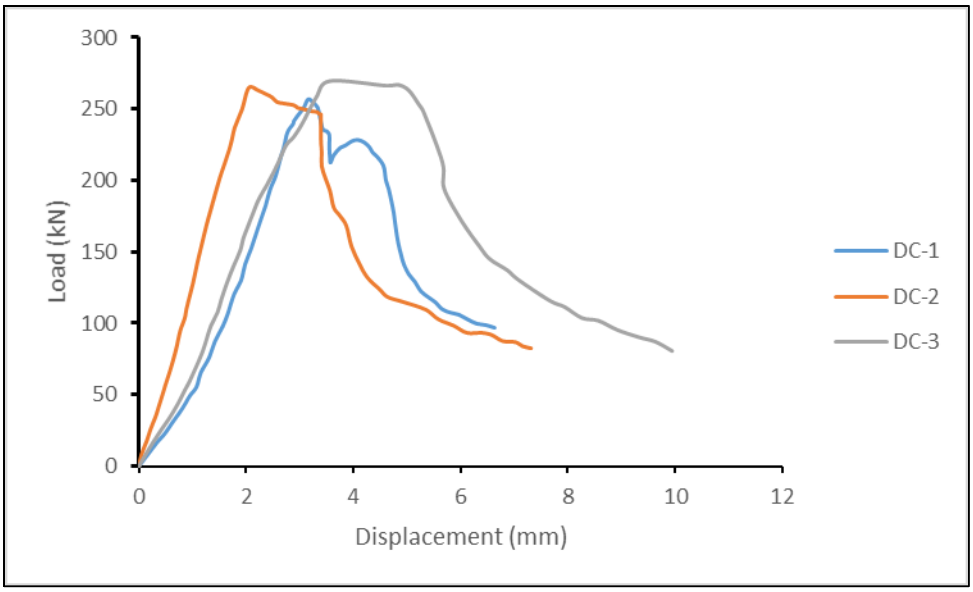

Figure 2.3: Load-displacement behaviour of three samples of diagonal compression test

The maximum diagonal force that is resisted by the three specimen are 246 kN, 257 kN and 269 kN respectively, after which softening behaviour is observed in all the three cases and the ultimate displacement observed in the three cases is approximately equal to 10 mm.

2.2 MODELLING OF DIAGONAL COMPRESSION TEST USING LAYERED SHELL ELEMENT

To simulate the in-plane shear strength of the EPS core panels, diagonal compression test is modelled in SAP 2000 using layered shell element. The dimensions of the model developed for diagonal compression test are same as those tested in the laboratory and are shown in the table below, i.e. square specimen with sides equal to 0.77 metres and thickness equal to 0.15 metres. The total thickness of 150 mm consists of 80 mm of EPS at the core and 35 mm of concrete cover on either side. Steel wire mesh is embedded in each of the concrete layers. The basic properties of each of the component layers is provided in the Table 2.1.

Table 2.1: Material properties used in SAP 2000 for diagonal compression test

Material	**Density (kg/m^3)**	**Young's Modulus (MPa)**	**Poisson's Ratio**
Concrete (M25 grade)	2400	25000	0.2
Expanded Polystyrene (EPS)	15	25	0.2
Welded Wire Mesh (f_y 700)	7850	200000	0.3

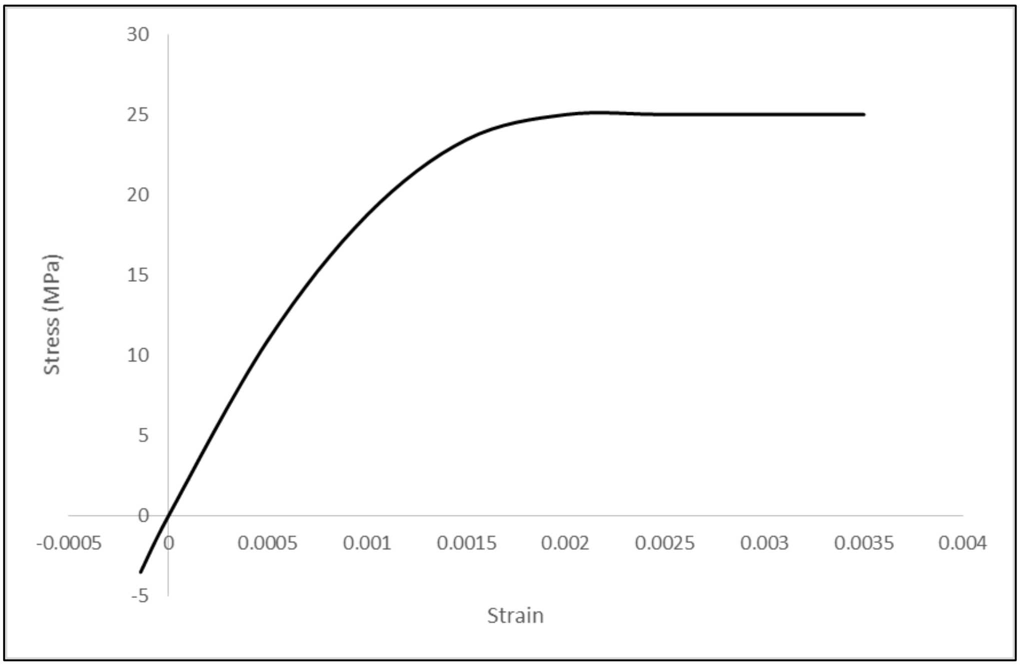

Figure 2.4: Stress-strain curve of concrete used for analysis in SAP 2000

The limiting strain in concrete used is 0.0035. For non-linear analysis, expected strength for concrete is used and the factor used is 1.5. The limiting strain in tension for welded wire mesh is considered as 0.05 and that in compression is used as 0.02. The factor used for expected strength in welded steel wire mesh is 1.25. The idealized bi-linear stress-strain curve used for welded wire mesh is given in the Figure 2.5.

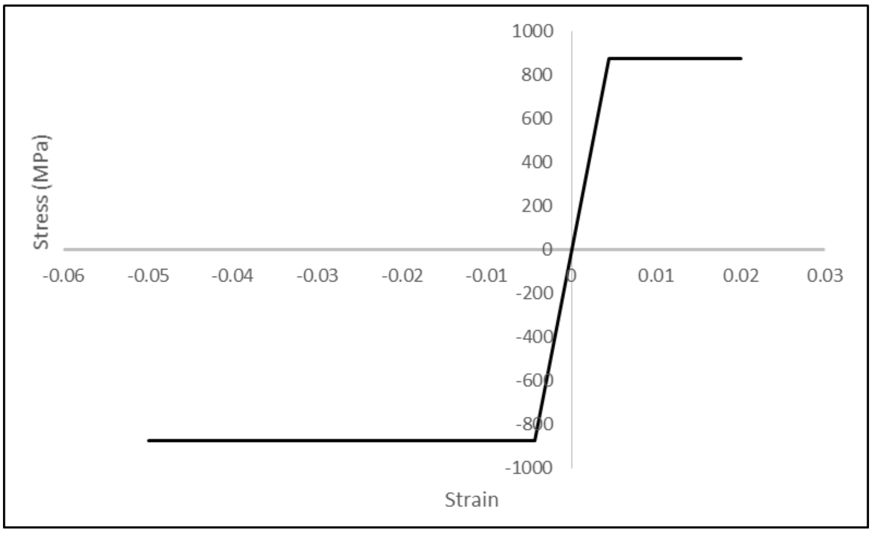

Figure 2.5: Stress-strain curve of steel wire mesh used for analysis in SAP 2000

Using the above properties for the constituent materials, the modelling is done using layered shell element in SAP 2000. The layered shell element is a finite element formulation which behaves like a thick plate (Mindlin/Reissner) and takes into account transverse shear deformations. The schematic diagram of the model developed for the diagonal compression test in SAP 2000 is shown in the Figure 2.6.

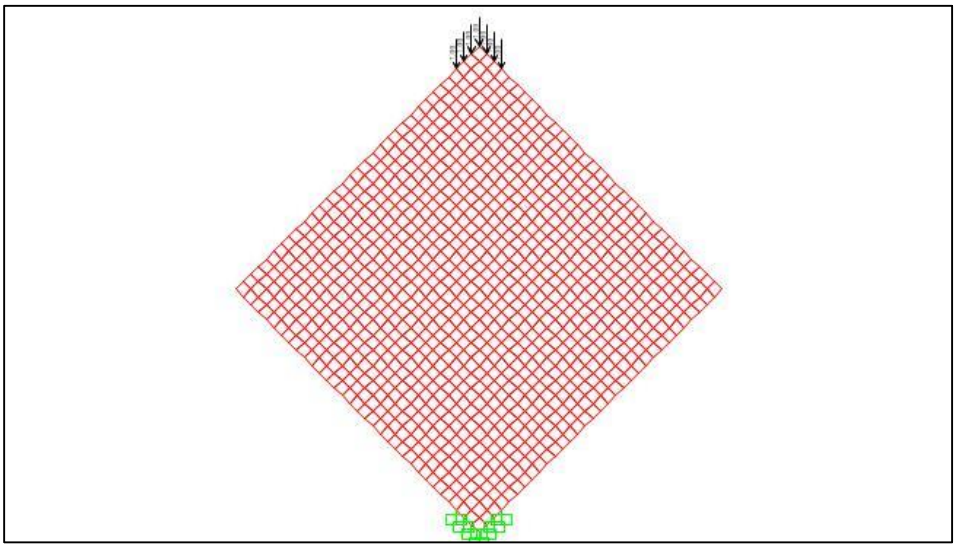

Figure 2.6: Details of mesh, loading and boundary condition in SAP model

Uniformly distributed load is applied at the top for a total length of 150 mm and the same length is kept fixed at the bottom. Non-linear static analysis is performed and the load applied is increased till failure occurs. The load-deflection behaviour obtained is shown in the Figure 2.7.

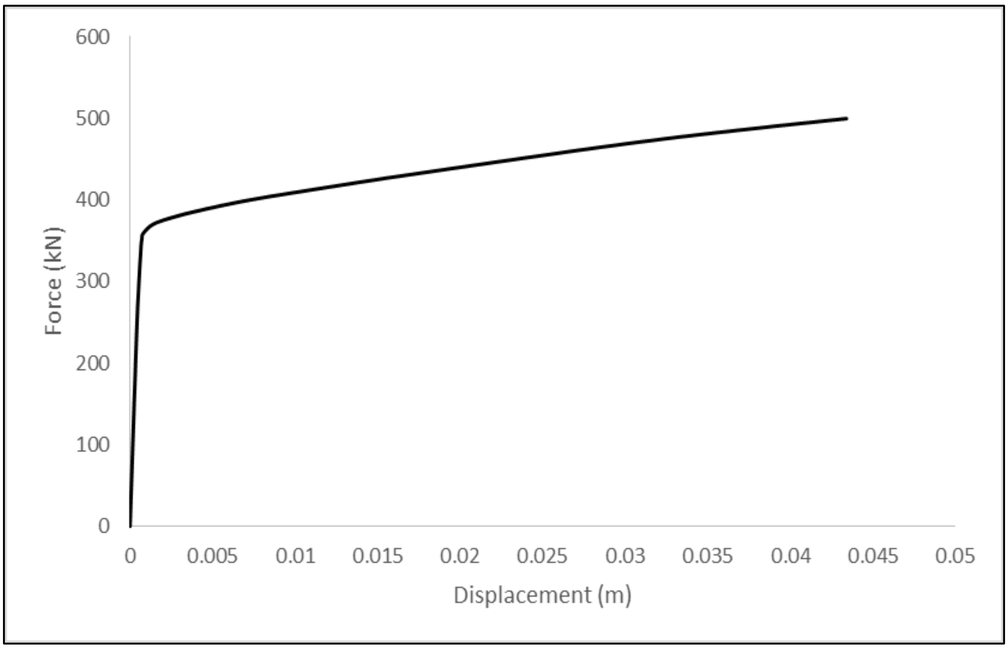

Figure 2.7: Load-displacement behaviour of EPS panels in diagonal compression test as developed in SAP 2000

2.3 MODELLING OF DIAGONAL COMPRESSION TEST USING SOLID 8-NODED HEXAHEDRAL ELEMENT

In ABAQUS, finite element simulation of the in-plane shear test of the EPS core panels was performed by modelling the diagonal compression test using the solid 8-noded hexahedral element. Concrete, expanded polystyrene and welded steel wire mesh were modelled in different layers and the interaction between them were modelled using the ‘tie constraint’ available in ABAQUS, which ensures perfect bond between the layers. By applying the tie constraint, it is ensured that all the connected nodes undergo equal displacements under the application of external load/displacement. The expanded polystyrene core and the two layers of sprayed concrete were modelled using solid 8-noded linear hexahedral element (C3D8) and the welded wire mesh reinforcement was modelled using the 2-noded linear truss element (T3D2). The welded wire mesh reinforcement was embedded inside the concrete layer.

2.3.1 Material properties used for component layers

In order to model the diagonal compression test properly, material modelling of each of the component layers is very important. The material behaviour of expanded polystyrene and concrete were modelled using the “concrete damage plasticity” model available in ABAQUS. The grade of concrete used in the study was M25 having characteristic strength (IS 456 200) f_{ck} of 25 MPa. The density of concrete used was 2400 kg/m^3 and the value of Poisson's ratio used in the analysis was 0.2. For concrete, the modulus of elasticity considered was $5000\sqrt{f_{ck}}$ as given by IS: 456 (2000). The strain corresponding to peak stress in concrete has been considered as 0.002 in the analysis and the maximum strain in concrete has been considered as 0.005 (0.003 to 0.005 according to Pillai and Menon (2010)). For concrete in tension, the cracking stress (f_{cr}) was assumed to be $0.7\sqrt{f_{ck}}$ and cracking strain was obtained from the cracking strain by dividing it by modulus of elasticity. The stress-strain variation was considered to be linear till the cracking strain and after cracking, the equation of stress-strain curve used is given below. The limiting concrete strain used is 0.002.

$$f_t = \frac{f_{cr}}{1 + 200\sqrt{\varepsilon_t}} \tag{2.1}$$

The stress-strain curves for concrete used in compression (Desayi and Krishnan, 1964) and tension are given in the Figures 2.8 and 2.9.

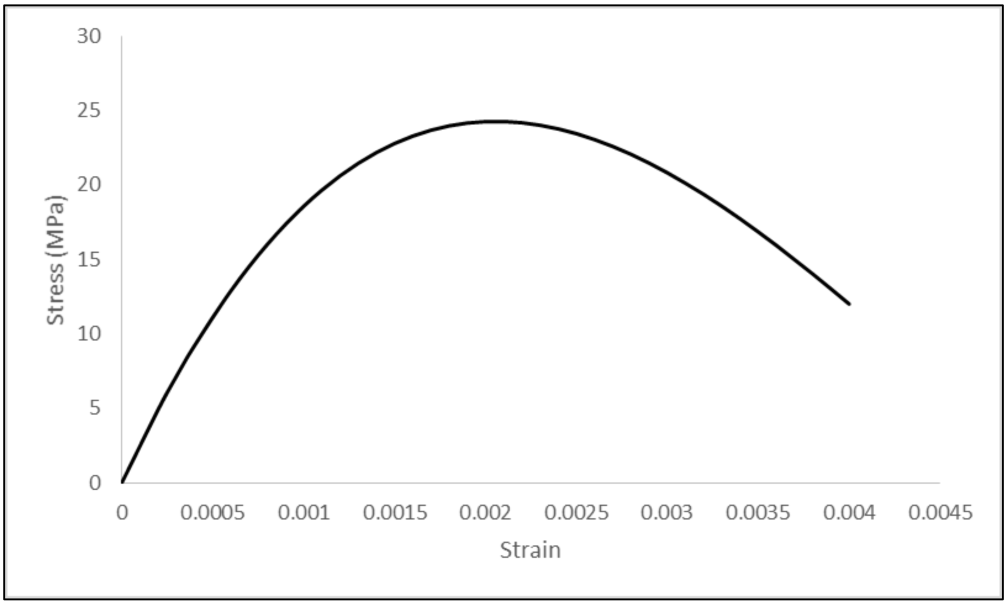

Figure 2.8: Stress-strain curve of concrete in compression

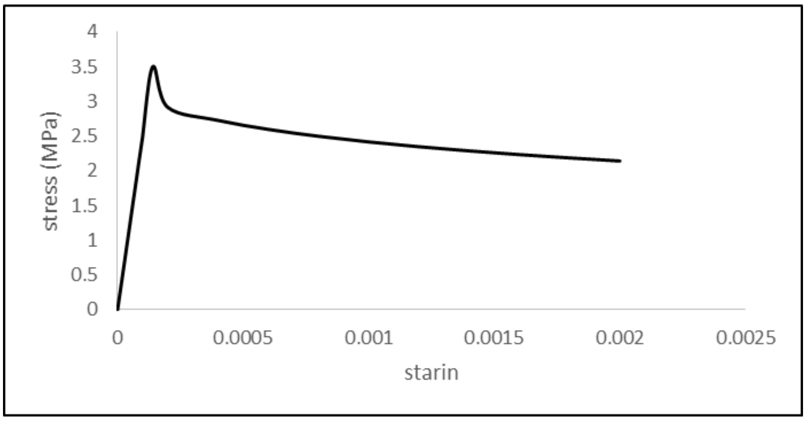

Figure 2.9: Stress-strain curve of concrete in tension

The post yield compression and tension behaviour of concrete was developed from the above curves in the same method as given by Jankowiak and Lodygowsky (2005). The yield stress is considered to be almost 0.33 times peak stress in compression and 0.7 times peak stress in tension. The post yield compression and tension curves of concrete is given in the Figures 2.10 and 2.11.

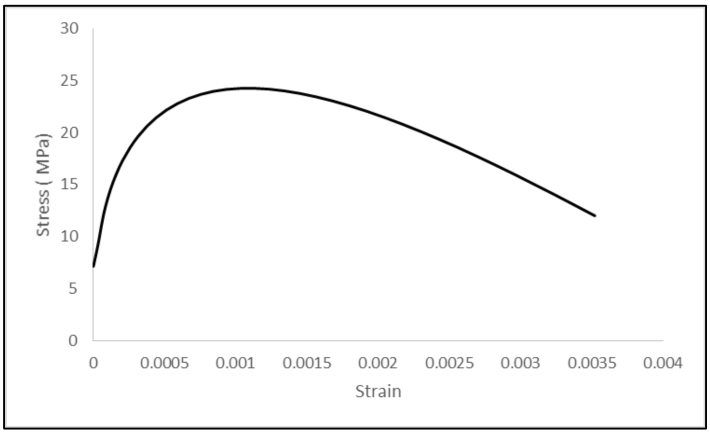

Figure 2.10: Inelastic stress-strain curve of concrete in compression

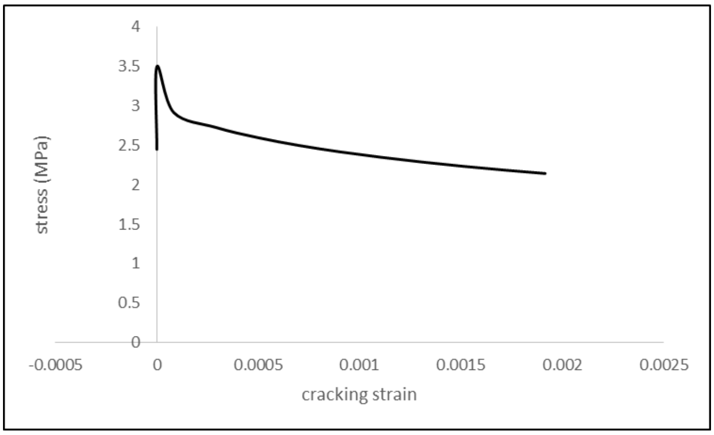

Figure 2.11: Inelastic stress-strain curve of concrete in tension

The compression and tension damage curves for concrete has been developed in the same procedure as given by Jankowiak and Lodygowsky (2005). The curves for compression and tension damage of concrete are given in the Figures 2.12 and 2.13.

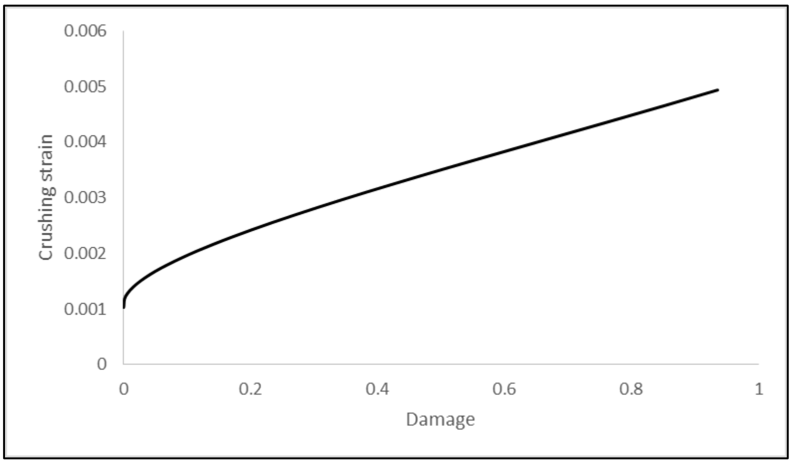

Figure 2.12: Compression damage of concrete

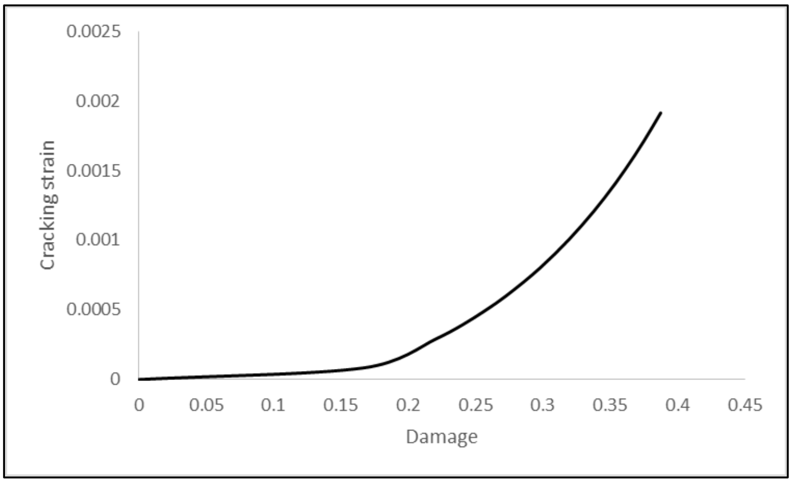

Figure 2.13: Tension damage of concrete

The parameters for concrete damage plasticity model has been taken from Jankowiak and Lodygowsky (2005). The value of the parameters used are given in the Table 2.2.

Table 2.2: Parameters used for concrete damage plasticity model

Material	Elastic Modulus (MPa)	Poisson's ratio	Dilation angle	Eccentricity	f_{bo}/f_{co}	k	Viscosity Parameter
Concrete	25000	0.19	38	0.1	1.12	0.67	0

The unit weight of welded wire mesh is considered to be 7850 kg/m^3. The tensile yield strength of welded wire mesh is considered to be 850.13 MPa and its elastic modulus is considered as 127.2 MPa as given by Sachin B. Kadam (2015). The stress-strain curve used for welded wire mesh is given in the Figure 2.14.

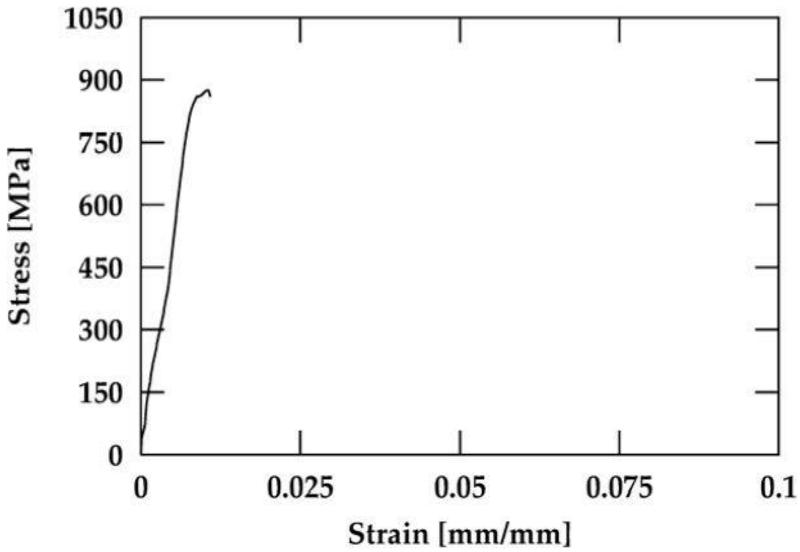

Figure 2.14: Stress-strain curve of welded wire mesh (Source: Sachin B. Kadam (2015))

The density of expanded polystyrene core is considered as 15 kg/m^3 and its modulus of elasticity is considered to be 25 MPa.

2.3.2 Details of finite element model

The model for diagonal compression test is developed using 8-noded solid hexahedral element (C3D8) for the concrete layers and expanded polystyrene (EPS) core. The panel is square with 0.77 metre sides. The internal eps core is 80 mm thick and each concrete layer is 35 mm thick. The wire mesh embedded in each concrete layer is modelled using 2-noded truss element (T3D2) and the two wire meshes are interconnected using ties at some fixed intervals which are also modelled using 2-noded linear truss element.

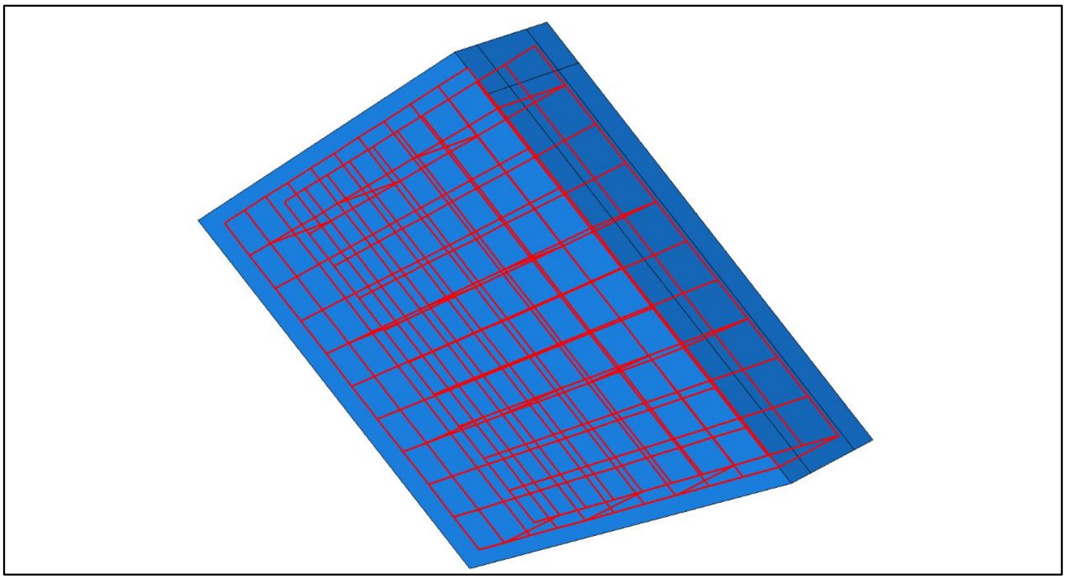

Figure 2.15: Model showing the three components of the EPS core panel

The length and breadth of the specimen is same as that used for testing in laboratory as mentioned earlier. The geometry is a square, being 0.77 metres in length. The diagonal is kept vertical and then load is applied on the top for a length of 75 mm on each side and the same length is kept fixed at the bottom. The load is increased gradually till the failure occurs. The Figure 2.16 shows the model with load applied at the top and the fixed boundary condition applied at the bottom.

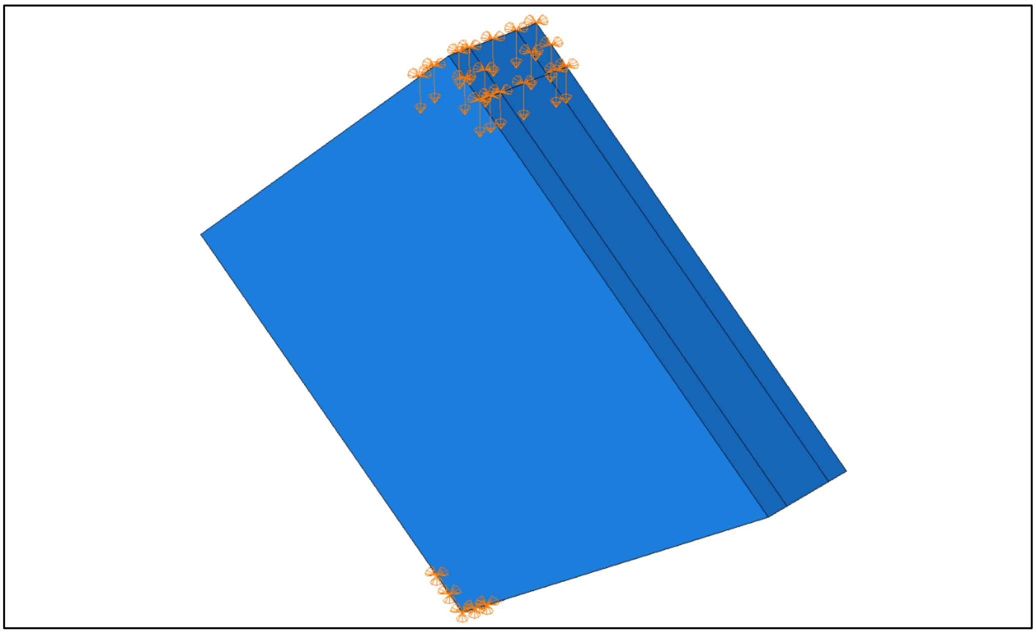

Figure 2.16: Details of loading applied and boundary conditions

The mesh that is used is checked for convergence and the final meshing used is shown in the Figure 2.17.

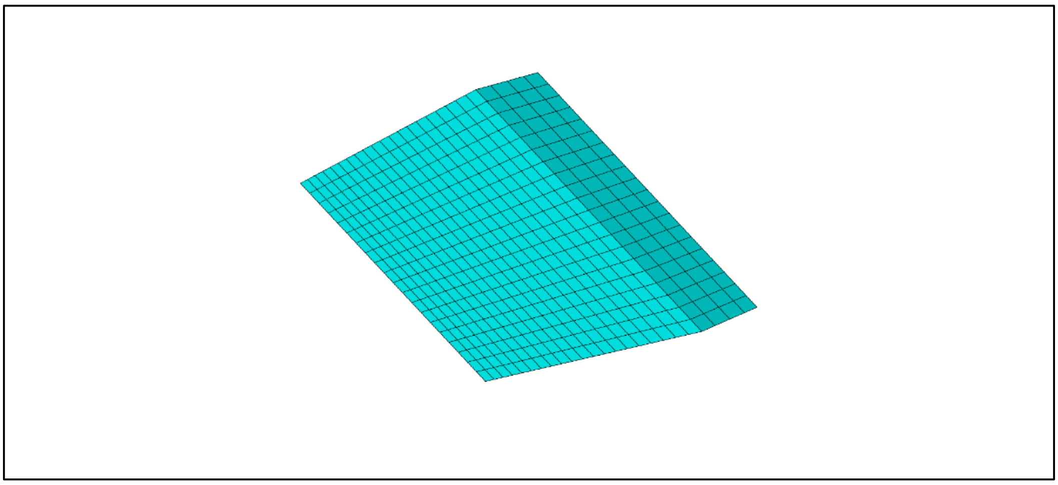

Figure 2.17: Details of finite element mesh

2.3.3 Results obtained from analysis

The load-displacement behaviour obtained from the analysis is shown in the Figure 2.18. It is observed that vertical tensile cracks occur in the specimen due to applied load and the boundary condition. The crack pattern that is obtained from the analysis is also shown, which matches with the crack patterns observed in the experiments performed, as shown earlier.

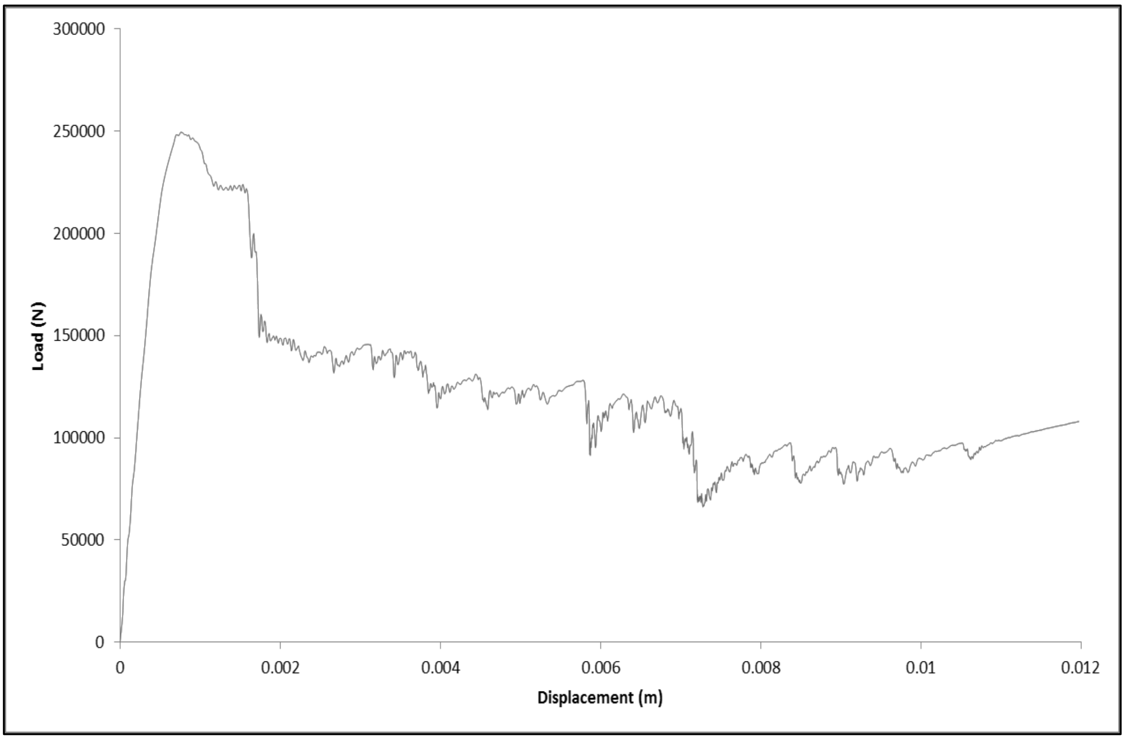

Figure 2.18: Load-displacement behaviour of EPS panels in diagonal compression test obtained from ABAQUS

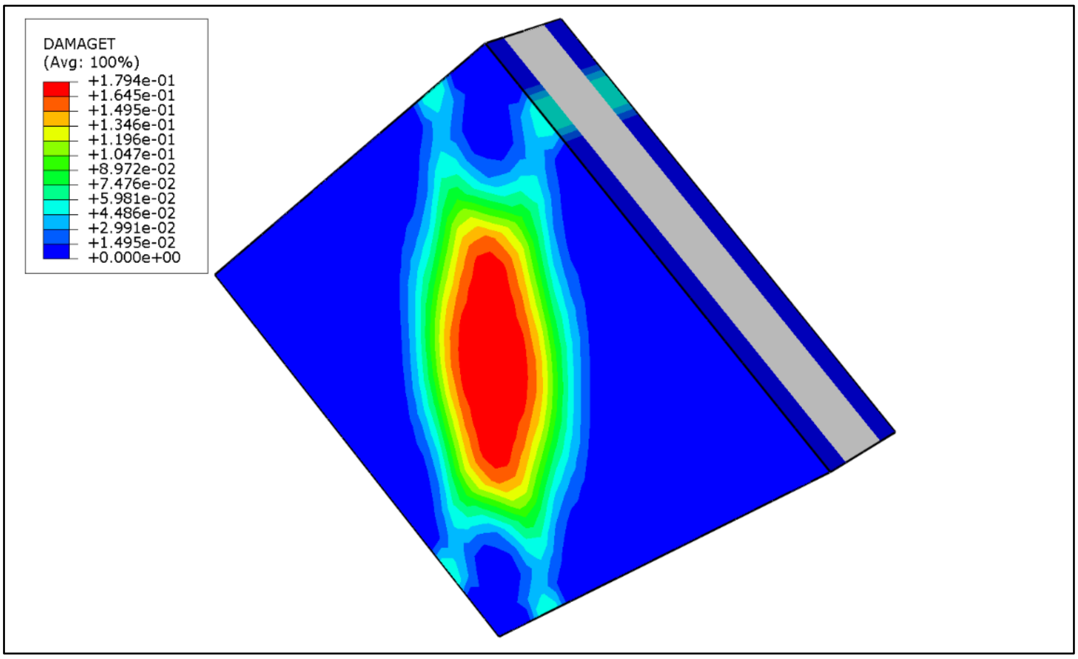

Figure 2.19: Crack pattern obtained from analysis

2.4 COMPARISON OF NUMERICAL AND EXPERIMENTAL RESULTS

The diagonal force versus vertical displacement of three tested specimen and the two types of numerical analysis are plotted together for comparison in the Figure 2.20.

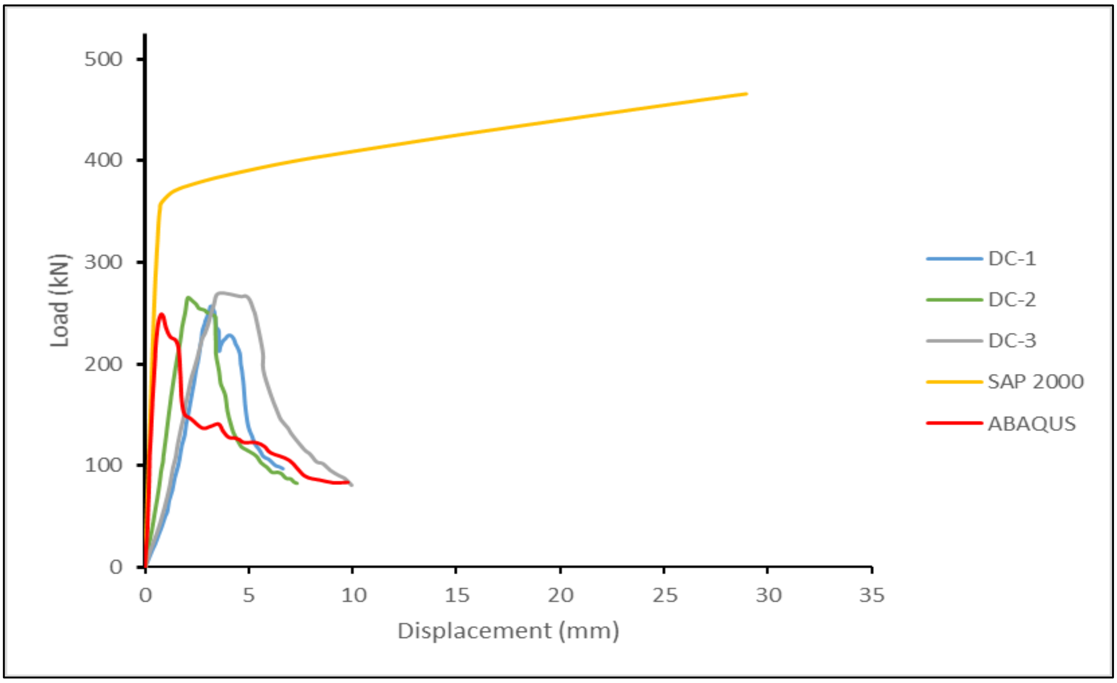

Figure 2.20: Comparative study of numerical analysis and experimental results

From the Figure 2.20, it is observed that the results of the finite element modelling done using solid 8-noded hexahedral element in ABAQUS matches with that of the experimental investigations, while the results of the finite element modelling done in SAP 2000 using layered shell element is completely different. The value of peak diagonal load resisted by the panels obtained by modelling in ABAQUS is almost equal to 250 kN which is near to the value obtained from the testing of the three specimen. To validate this result, the shear force carrying capacity of the panels is also computed by the formulation given in IS:456 (2000), considering it as a reinforced concrete section with thickness equal to 70 mm (neglecting the thickness of the eps core) and the value obtained is 171.5 kN. Thus, the diagonal force carrying capacity of the section as computed by IS: 456 (2000) method is 242.5 kN, which is in range with the values obtained from finite element modelling in ABAQUS and the experiments performed. The ultimate deflection obtained from experiment also matches with that obtained from ABAQUS. However, the modelling in SAP 2000 using layered shell element overestimates the maximum load carrying capacity of the panels by almost two times and also predicts wrong ultimate displacement value. These wrong values are estimated by modelling using layered shell element as the damage in concrete is not modelled correctly. It is also observed that the load-displacement curves of numerical analysis predicts the panels to be stiffer than the experimental results. This is due to the local crushing observed in the tested specimen at the loaded corners during the starting of the experiment. The load-displacement curves obtained from numerical analysis and experiments performed are converted to shear-stress versus shear-strain curves according to the equations given in ASTM E-519/E-519 M-10 (ASTM 2010a) as shown below.

$$\tau = \frac{0.707P}{0.5t(L+H)} \tag{2.2}$$

$$\gamma = \frac{\Delta V + \Delta H}{g'} \tag{2.3}$$

where, τ is the average shear stress and γ is the average shear strain. P is the diagonal force, t is the thickness of the specimen, L and H are the length and width of the specimen. Δ V is the decrease in length of the specimen along the loaded diagonal and Δ H is the decrease in length in the direction perpendicular to the loaded diagonal, g’ is the gauge length (kept same along both the diagonals) and in this case is equal to 380mm. The shear stress versus shear strain plots obtained by the above mentioned procedure for the two types of numerical modelling and the experiments performed are shown in the Figure 2.21.

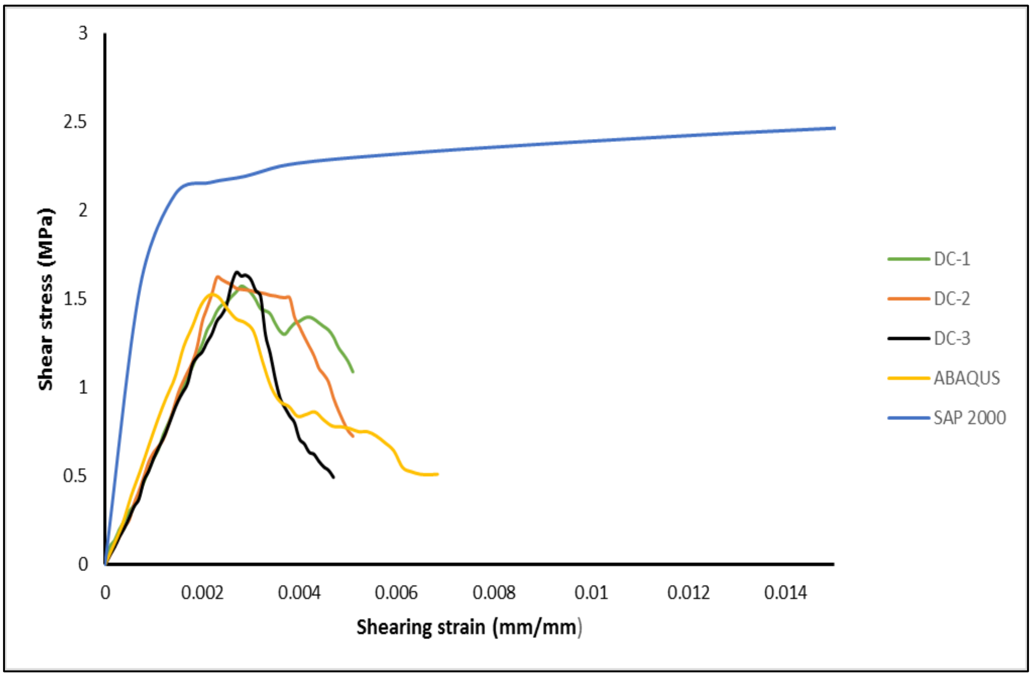

Figure 2.21: Shear-stress versus shear strain plots comparison of experimental and numerical analysis

From the Figure 2.21, it is observed that the shear stress versus shear strain plots of the numerical modelling using solid element in ABAQUS match with the experimental results exactly, but there is considerable difference in the results obtained from SAP 2000.

Chapter 3

EVALUATION OF OUT-OF-PLANE FLEXURAL STRENGTH OF THE EPS CORE PANELS

To evaluate the out-of-plane flexural strength of the EPS core panels, four-point flexural loading test of the panels is performed. Finite element simulation of four-point flexural loading test is done using 8-noded solid hexahedral element in the finite element software ABAQUS and using layered shell element in SAP 2000.

3.1 OUT-OF-PLANE FLEXURAL TEST PERFORMED ON THE EPS CORE PANELS

In order to examine the bending behaviour of the EPS panels due to out-of-plane loading, out-of-plane four-point flexural test is performed on the EPS core panels. Three samples with the same dimensions, material properties and geometry conditions are tested. The length of the panels are 1.2 metres, breadth is 0.6 metres and thickness is 0.15 metres. The total thickness of 150 mm consists of 80 mm EPS at the core and 35 mm concrete cover on each side. The effective length of the panels are 1.1 metres, as the roller supports are provided at a distance of 5 cm from the ends.

Figure 3.1: Test setup and loading condition of flexural test

Displacement controlled loading was applied on the panels. Linear variable differential transducers (LVDTs) were installed at the mid-span bottom and also under the line of load application at the bottom. The rate of loading used for the test was 0.2 mm per minute. The load versus mid-span displacement of the three specimen are given in the Figure 3.2.

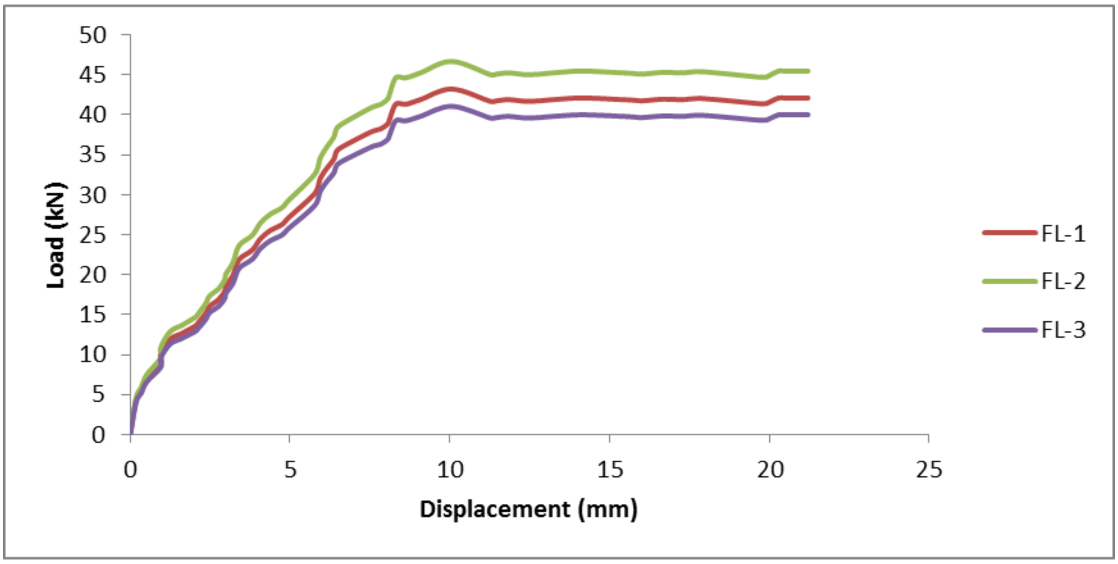

Figure 3.2: Load-displacement behaviour of EPS panels in flexural test

The maximum load resisted by the specimen ranged from 40 to 50 kN. It was observed that the peak load was reached when the concrete cracked due to tension at the bottom at mid-span and also under the lines of load application. All the three samples showed similar behaviour. It was also observed that after the maximum load was reached, the curves became flat, as there was de-bonding failure between concrete and steel. Due to de-bonding failure, there was no load transfer between concrete and steel and thus no more load could be resisted by the panels.

Figure 3.3: Cracks observed in flexural test at the mid-span and at the point of application of load

The Figure 3.3 shows the tensile cracks at the bottom developed under the mid-span and also under the line of load application. The middle one-third portion of the portion was subjected to maximum bending moment and thus the cracks occurred there at the bottom due to development of tension in concrete at the bottom.

3.2 FINITE ELEMENT MODELLING OF OUT-OF-PLANE FLEXURAL BEHAVIOUR OF EPS CORE PANELS USING LAYERED SHELL ELEMENT

In order to evaluate the out-of-plane behaviour of the eps core panels, finite element model of the panels are developed using layered shell element in SAP 2000. The properties of the constituent layers are same as that used for modelling of diagonal compression test, as mentioned in the previous chapter. The dimensions of the model developed are same as that tested in the laboratory as mentioned in the previous chapter i.e. length is 1.2 metres, width is 0.16 metres and thickness is 0.15 metres. The total thickness of 0.15 metre consists of 80 mm EPS at the core with 35 mm concrete cover on both sides. Steel wire mesh lies embedded in the concrete layer with 20 mm concrete cover. The effective length of the panel becomes 1.1 metre as the supports are provided at a distance of 5 cm from the ends. Displacement controlled loads are applied at one-third length from either side and increased till failure. The details of the model developed, loading applied and boundary conditions are shown in the Figure 3.4.

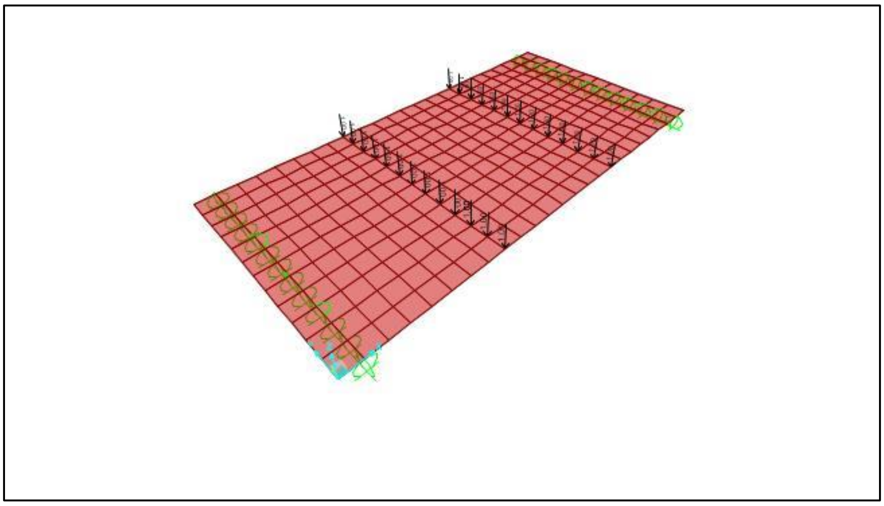

Figure 3.4: Loading, boundary conditions of the model developed for flexure test in SAP 2000

The maximum deflection occurred in the middle one-third portion of the model, as it was the portion subjected to maximum bending moment. The deflected shape observed under the applied loading conditions is given in the Figure 3.5.

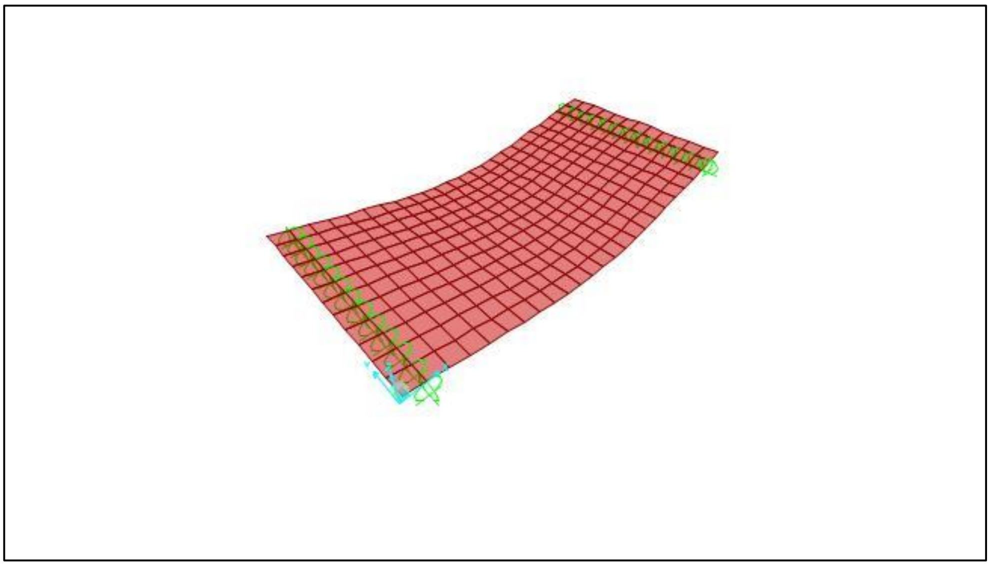

Figure 3.5: Deflected shape of the model due to flexure loading in SAP 2000

The load versus central displacement obtained from the analysis is given in the Figure 3.6.

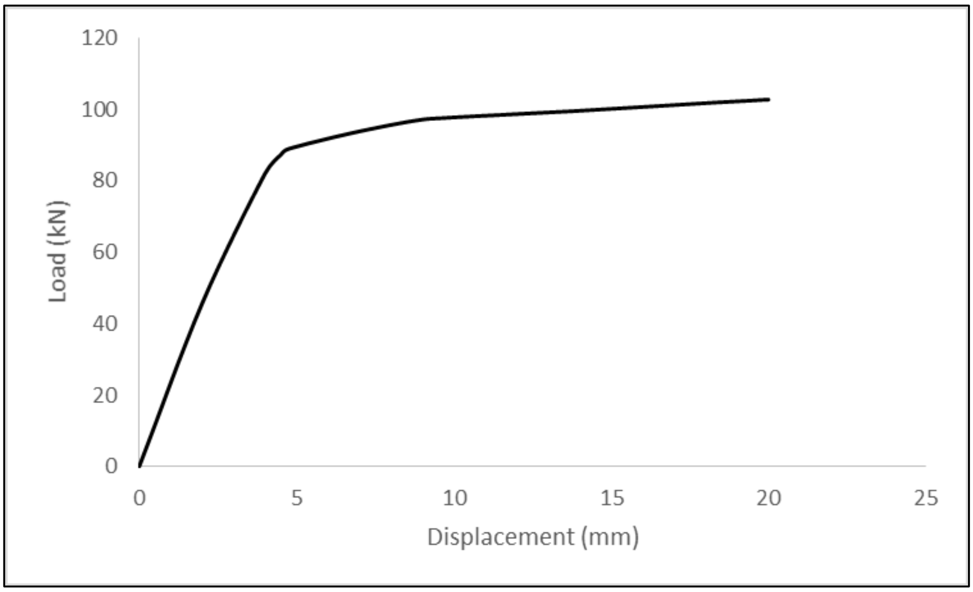

Figure 3.6: Load-displacement behaviour of EPS panels obtained for flexure test in SAP 2000

3.3 FINITE ELEMENT MODELLING OF OUT-OF-PLANE FLEXURAL BEHAVIOUR OF EPS CORE PANELS USING SOLID ELEMENT

In order to evaluate the out-of-plane flexural behaviour of the eps core panels, each of the component layers are modelled separately in ABAQUS, similar to the modelling of the panels in diagonal compression test. The EPS core and the two concrete layers are modelled using the solid 8-noded hexahedral element (C3D8) and the wire mesh is modelled using 2-noded linear truss element (T3D2). The connection between the concrete and EPS layers is modelled using the tie constraint available in ABAQUS which enables equal displacements of all the connected nodes under the application of external loads. The wire mesh is embedded within the concrete layers and the connector ties are placed at fixed intervals between the two layers of wire mesh. The connector ties are also modelled using the 2-noded linear truss element (T3D2). The material properties of concrete, steel wire mesh and EPS used are same as that used for modelling diagonal compression test, as mentioned in the previous chapter. The dimensions of the model developed are same as those on which tests are performed and are shown in the table below i.e. length is 1.2 metres, width is 0.6 metres and thickness is 0.15 metres. The total thickness of the panel i.e. 150 mm consists of 80 mm expanded polystyrene at the core and 35 mm concrete cover on each side. The effective length of the model is 1.1 metres as roller support is provided at both the sides, at a distance of 5 cm from the ends. Displacement controlled loading is applied to one-third length of the model and the load and the mid-point deflection of the model is obtained.

3.3.1 Details of the finite element model

The model developed comprising of expanded polystyrene core, the two layers of concrete cover and the welded wire mesh connected by ties are shown in the Figure 3.7.

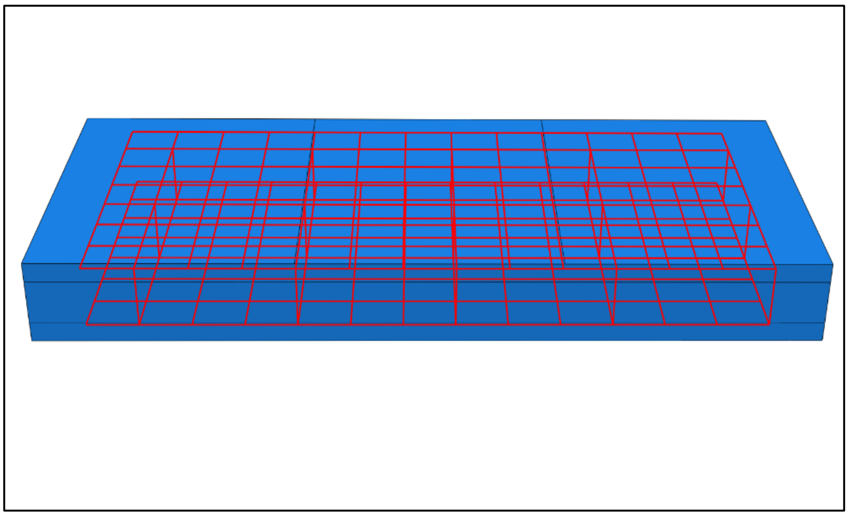

Figure 3.7: Diagram showing components of the model developed in ABAQUS for flexure test

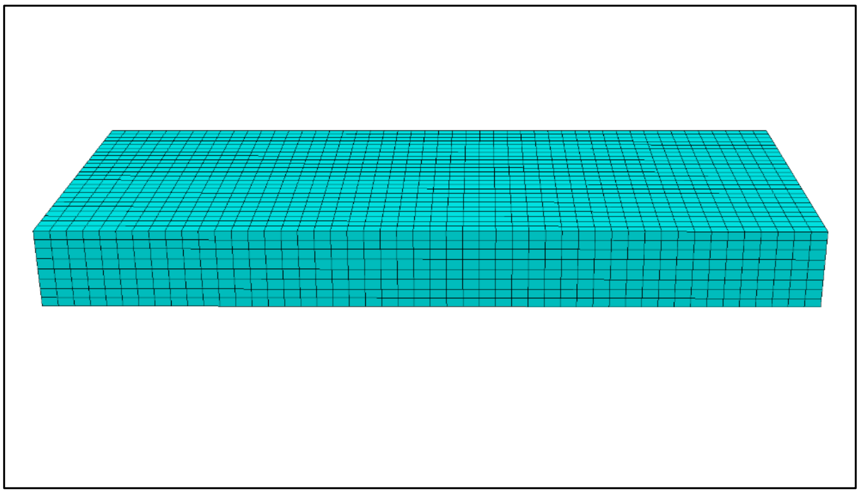

Figure 3.8: Finite element mesh

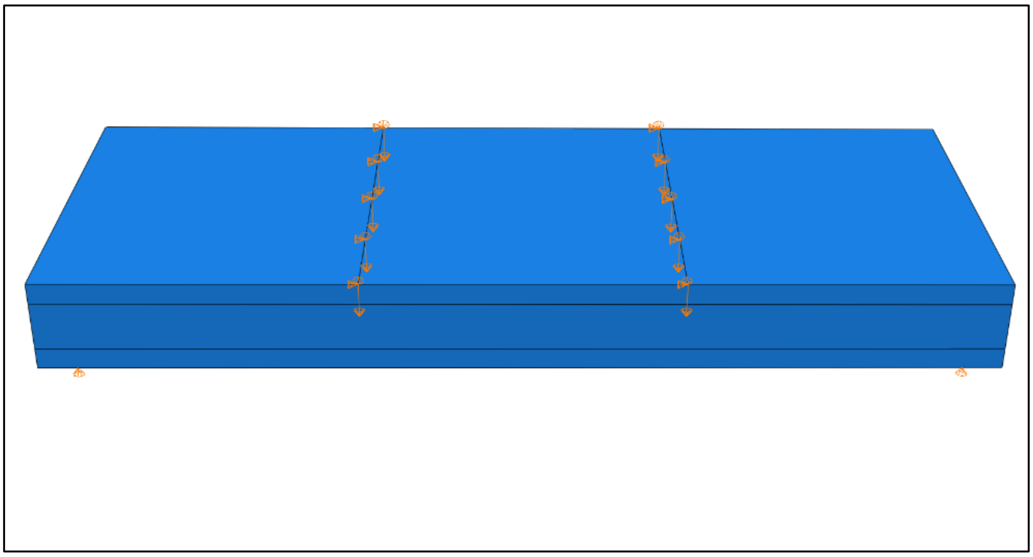

Figure 3.9: Loading applied and boundary conditions

The figures above show the details of the finite element mesh and the loading and boundary conditions applied on the model respectively. Load is applied till failure occurs.

3.3.2 Results obtained from analysis

The load versus the central displacement is shown in the Figure 3.10.

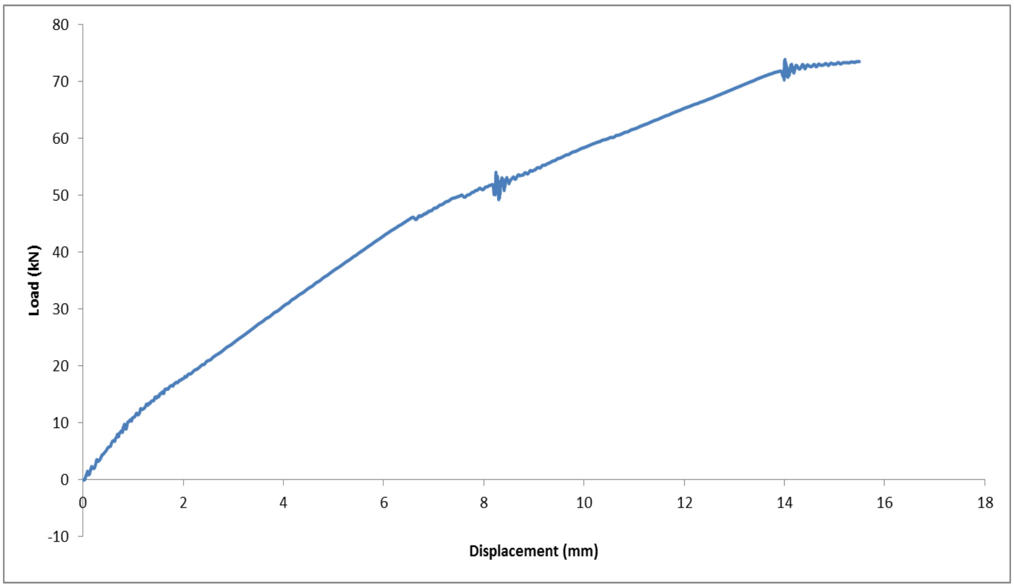

Figure 3.10: Load-displacement behaviour of EPS panels in four-point loading test as developed in ABAQUS

The deflected shape of the model showing tension damage is shown in the following figures.

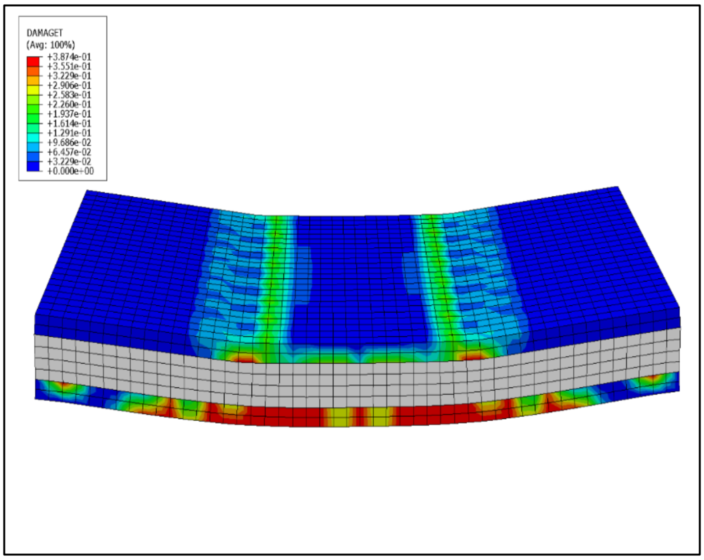

Figure 3.11: Deflected shape of the beam showing tension damage

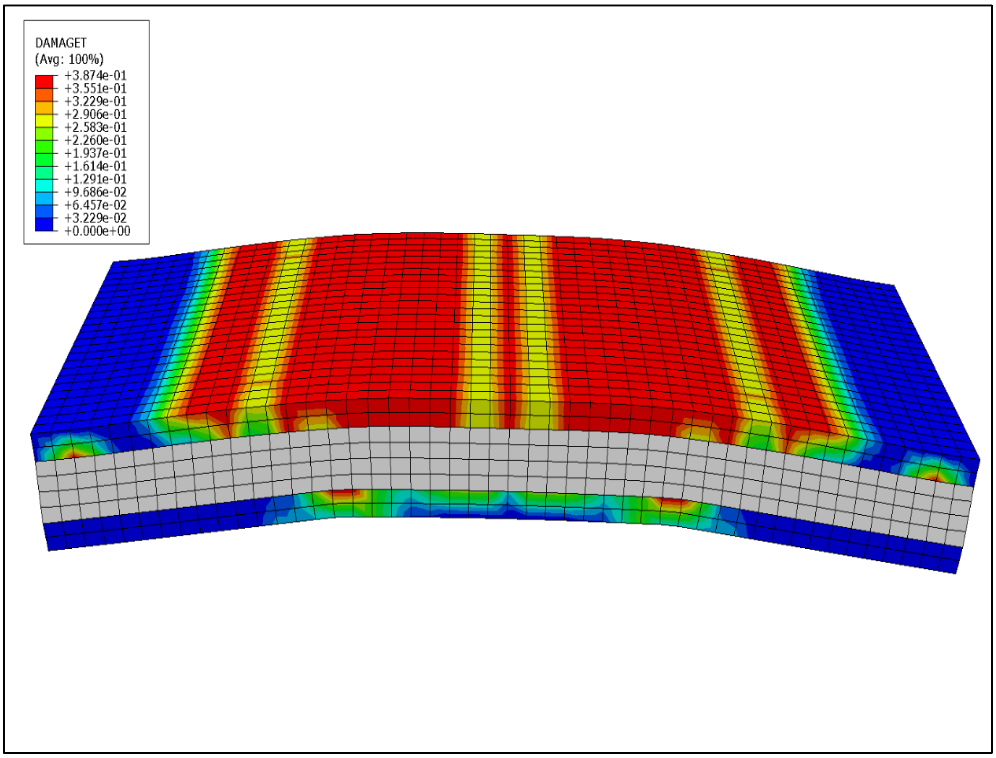

Figure 3.12: Deflected shape of the model in upside down condition to show tension damage at the bottom

3.4 COMPARISON OF NUMERICAL AND EXPERIMENTAL RESULTS

The force versus central displacement curves of the three specimen tested and that obtained from modelling in ABAQUS and SAP 2000 are plotted together for comparison in the Figure 3.13.

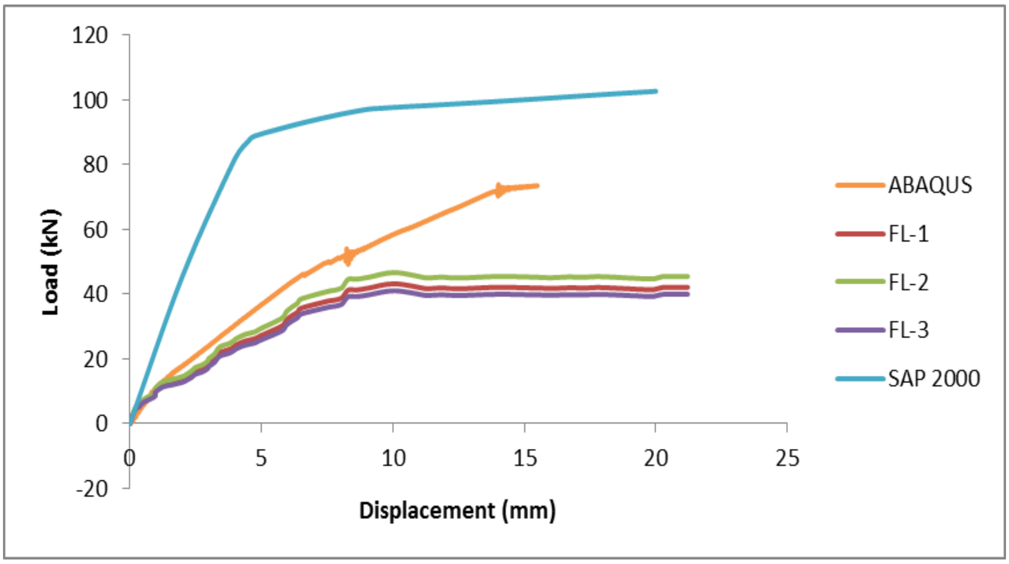

Figure 3.13: Load versus displacement behaviour of EPS panels in flexure test obtained from experiment and finite element models

From the above figure, it is observed that the initial portion of the finite element analysis in ABAQUS matches with the experimental curves, but after the tensile failure of concrete, de-bonding occurs in the tested specimen and thus, the latter part of the curves does not match as de-bonding failure has not been considered in the finite element model. The load-displacement curve of the finite element model keeps on increasing after the tensile failure of concrete as the steel has not yielded yet and has load-carrying capacity. The curve becomes flat after the steel yields and continues till the ultimate strain is reached. However, it has been observed that the results of modelling using layered shell element in SAP 2000 shows very high value of stiffness in comparison to the experimental results. The maximum load carrying capacity obtained from modelling in SAP 2000 using layered shell element is almost 2.5 times that obtained from the experimental curves.

SEISMIC ANALYSIS OF A TYPICAL EPS CORE PANEL BUILDING

To analyse the performance of the building systems constructed using the EPS core panels, finite element model of a G+3 building constructed in the North Indian city of Sonipat using the EPS core panels has been developed from the available report and drawings of the above mentioned building, in SAP 2000.

4.1 DETAILS OF BUILDING CONSTRUCTED IN SONIPAT

The G+3 building constructed using the panels have the following details:

1) The walls are made of panels total 220 mm thick (100 mm thick EPS sandwiched between two layers of concrete, each 60 mm thick).
2) The slabs are made of panels total 240 mm thick (160 mm thick EPS sandwiched between two layers of concrete, upper concrete layer thickness being 50mm and bottom concrete layer thickness being 30mm).
3) The superstructure rests on beams at the plinth level which are 300mmX 750mm.
4) The beams at the plinth level rest on columns which run to the top of the foundation level and have different dimensions: 300mmX 450mm, 300mm X 1015mm, 300mm X 550mm, 300mm X 650mm, 300mm X 900mm, 300mm X 300mm and 300mm X 230mm.
5) The panels used for walls and slabs both have steel wire mesh at the interfaces of concrete and expanded polystyrene with 3mm diameter and spacing is 80 mm in the horizontal direction and 75mm in the vertical direction. The two wire meshes are welded to each other and are also interconnected by connector bars of 3mm diameter.
6) The foundation is raft.

4.1.1 Materials Considered

1) Concrete- M25 grade concrete is used for RCC work and guniting the panels.
2) Steel- 3 mm diameter steel wire mesh of f_y 700 to have a yield strength up to 700 MPa is used for the wall and panels, f_y 500 steel having yield strength up to 500 MPa is used for normal RCC works.

3) Expanded polystyrene sheet of 100 mm thickness is used for the walls.
4) Expanded polystyrene sheet of 160 mm thickness is used for the slabs.

4.1.2 Details of the finite element model

1) The model is developed in SAP 2000 according to the available reports and architectural drawings.
2) The walls are modelled as shell elements and are taken as total 120 mm thick, considering the thickness of the two outer layers of concrete, and the thickness of the internal polystyrene layer is neglected.
3) The floor slabs are modelled as shell elements and here also the thickness of polystyrene layer is neglected.
4) Openings in the structure are modelled as per the door and window schedule.
5) Edge constraints are given to the modelled shells to form a compatibility in deformations.
6) The walls in the model are supported at the plinth level by 300 mm X 750 mm size beams as per the available architectural drawings.
7) The plinth beams are resting on columns of different dimensions as mentioned earlier and are provided in the model as per the available architectural drawings.
8) The base of the columns are fixed.
9) Live loads are applied on the model as per IS 875 part 2.
10) Earthquake loads are calculated and applied on the model as per IS 1893 (Part 1):2002.
11) The model is subjected to load combinations as per IS 1893 (Part 1):2002.

The diagrams of the developed model is given below:

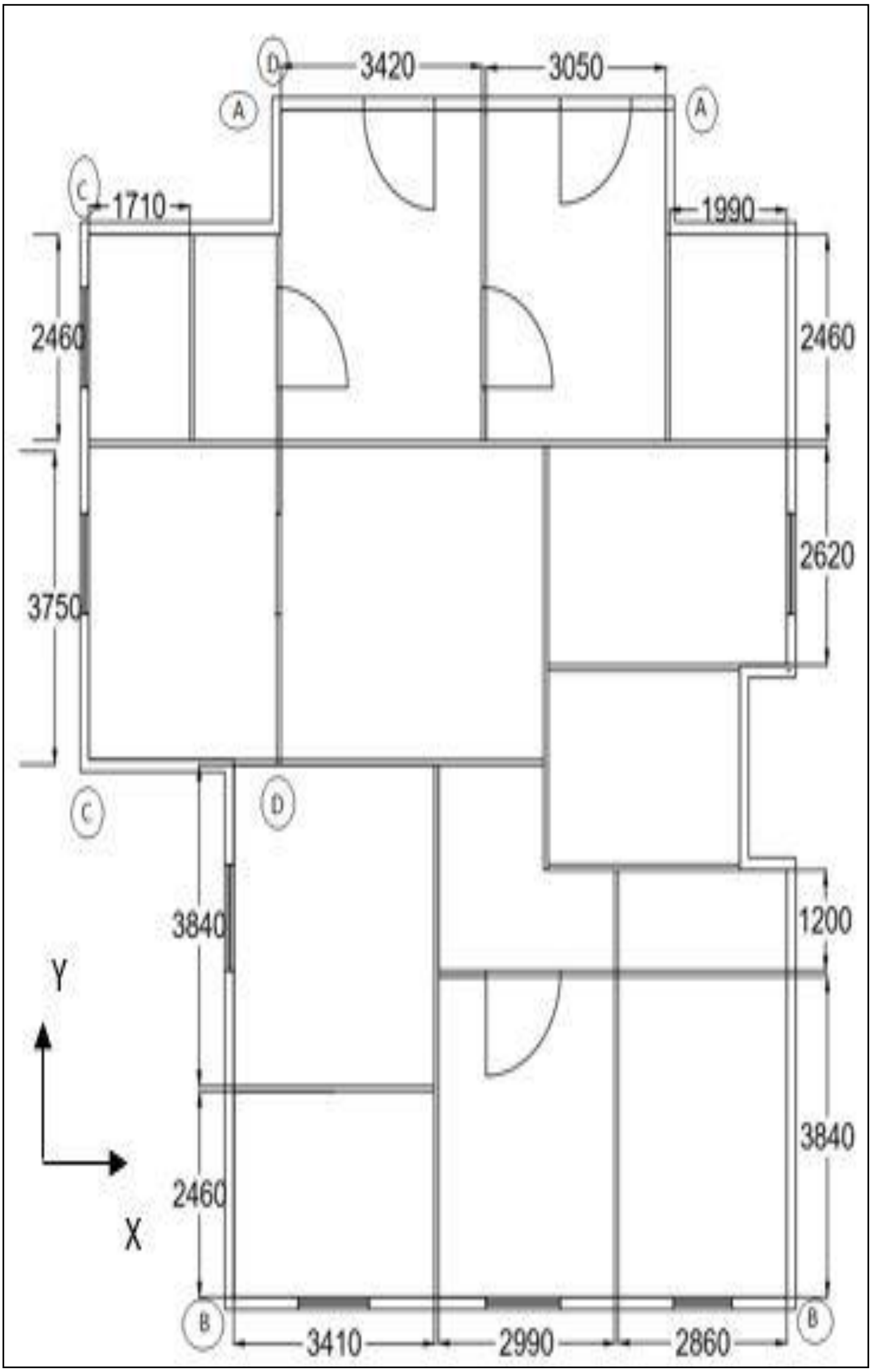

Figure 4.1: Plan of the building (all dimensions are in millimetres)

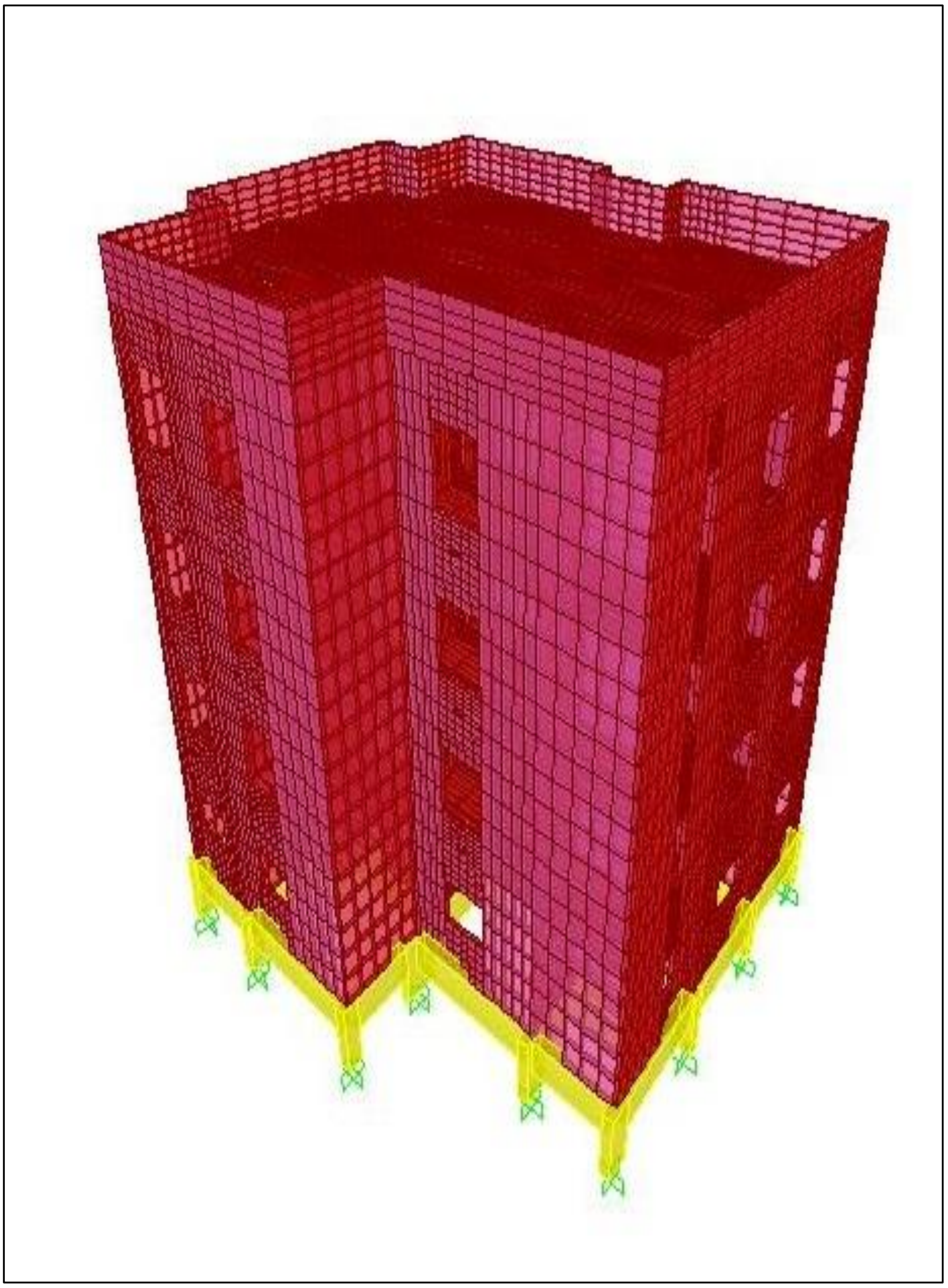

Figure 4.2: Isometric view of the model developed in SAP 2000

The dimensions of the building are given below:

1) 14.36 metres in length (along y-direction in SAP model).
2) 12 metres in width (along x-direction in SAP model).
3) 13.05 metres in height (along z-direction in SAP model).

4.2 PIER ANALYSIS METHOD

Pier analysis is a method by which simplified analysis of masonry structures under the combined action of gravity and seismic forces are performed, as given by Drysdale, Hamid and Baker (1999). The procedure is based on the following assumptions:

1) In-plane and out-of-plane behaviour of masonry walls is considered independently.
2) The in-plane behaviour of walls is simulated by an assemblage of piers (i.e. the vertical members consisting of the masonry between door and window openings, and below and above the doors and windows).
3) The spandrels (i.e. the horizontal members connecting different piers) are assumed to be rigid.
4) The end conditions of piers are assumed to be either fixed or free, depending on the restraint expected to be provided by the foundation and the walls above. In EDRM, all the piers have been assumed to be fixed at both the ends, except the upper end of the topmost pier of the building, which does not have restraining force, and has been assumed to be free.
5) In out-of-plane action the walls have been assumed to span vertically between roof/floor slabs or bands/bandages, if provided.

4.2.1 Wall and pier forces

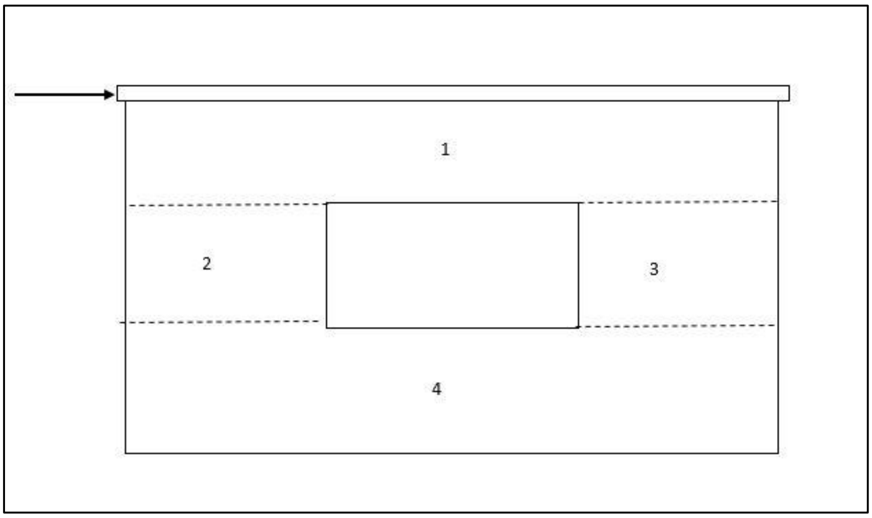

Figure 4.3: Schematic diagram of force distribution among piers by pier analysis method

The in-plane stiffness (Ri) of a pier is a function of aspect ratio of pier (h/L), thickness of pier (t), elastic modulus of concrete (E) and boundary condition.

The stiffness of a cantilever pier is expressed as given in the equation below:

$$R_i = \frac{E_c t}{4(\frac{h}{L})^3 + 3(\frac{h}{L})} \tag{4.1}$$

Similarly, the stiffness of a fixed end pier is given by the equation below.

$$R_i = \frac{E_c t}{(\frac{h}{L})^3 + 3(\frac{h}{L})} \tag{4.2}$$

The total stiffness of a wall is the combination of stiffness of all the piers either in parallel or in series.

The total shear force in each direction is distributed along the height of the structure by the method used in Indian code and from there the shear in each storey is calculated by the equations given below.

$$Q_i = V_b \frac{w_i h_i}{\sum w_i h_i^2} \tag{4.3}$$

$$V_i = \sum_{j=1}^{n} Q_j \tag{4.4}$$

The in-plane forces in individual walls are estimated as given by the equations below

$$F_{xi} = V_{xi} R_{xi} \left(\frac{1}{\sum R_{xi}} + \frac{e_d}{R_\theta} X_i\right) \tag{4.5}$$

$$F_{yi} = V_{yi} R_{yi} \left(\frac{1}{\sum R_{yi}} + \frac{e_d}{R_\theta} Y_i\right) \tag{4.6}$$

$$R_\theta = \sum R_{xi} Y_i^2 + \sum R_{yi} X_i^2 \tag{4.7}$$

Where, F_{xi} = force in i-th wall along x-direction,

F_{yi} = force in i-th wall along y-direction,

e_d = design eccentricity as per IS 1893 (Part 1):2002.

R_Θ= rotational stiffness of storey

R_{xi}= lateral stiffness of walls oriented along x-direction

R_{yi}= lateral stiffness of walls oriented along y-direction

X_i, $Y_{i=}$distances of walls from centre of stiffness

4.3 FINITE ELEMENT ANALYSIS

The response of the building due to combination of gravity and seismic loading has been evaluated using finite element analysis in SAP 2000. At first linear analysis is performed, and the stresses in walls due to most critical load combinations are computed. The axial force, bending moment and shear forces in the critical sections are computed and results are compared with the pier analysis results. The P-M interaction curves of the critical sections are calculated and it is checked whether the demand capacity ratios of the sections are less than 1 or not. Non-linear static pushover analysis is performed to estimate the performance of the building due to seismic forces.

4.3.1 Linear Analysis

After the model is developed, it is analysed in SAP 2000. Dead, live, modal and response spectrum analysis is done. The response spectrum taken is according to Seismic Zone V (Design Basis Earthquake) and soft soil conditions of IS 1893 (Part 1): 2002 (considering R=3). The shear force, bending moment and axial forces at all the critical sections are considered due to all the load combinations as given in IS 1893 (Part 1): 2002.

4.3.2 Load calculations and modal analysis

It is observed that the total dead load is 6270.7 kN which matches with the manual calculation, the total live floor load is 879.8 kN and total live roof load is 219.9 kN which also match with the manual calculations. The base shears in x-direction and y-direction are 839.9 kN and 806.4 kN respectively.

The time period of the building in the first mode is observed to be 0.111 seconds and in that in the second mode is 0.108 seconds.

4.3.3 Stresses in walls

The vertical and shear contours of all the walls are plotted due to the most critical load combinations as given in the following figures.

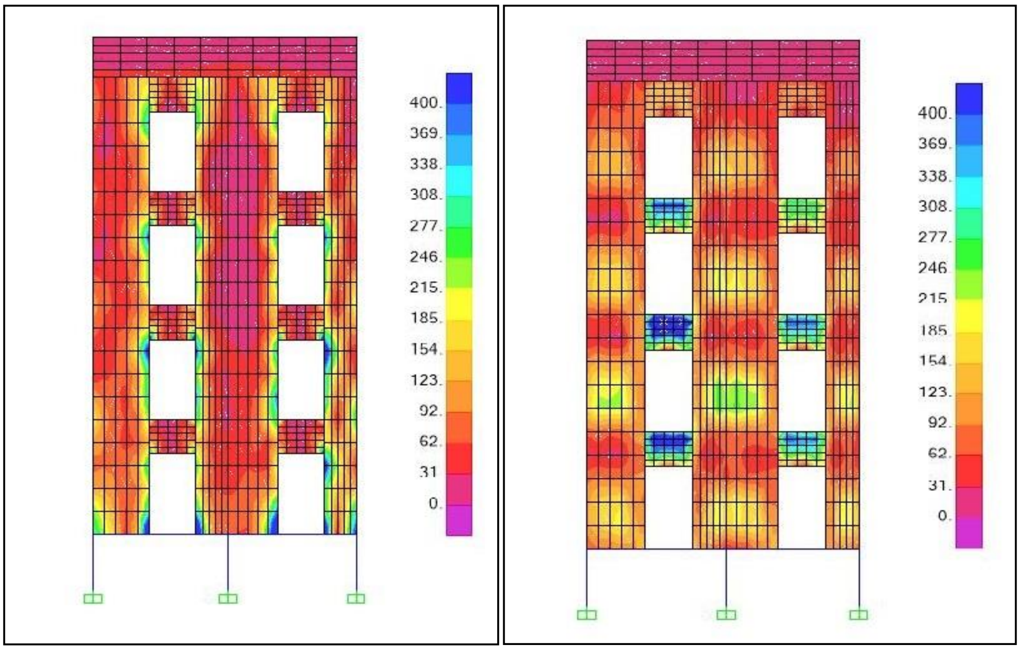

Figure 4.4: Stresses in wall A-A due to most critical load combinations a) Vertical b) Shear in kN/m^2

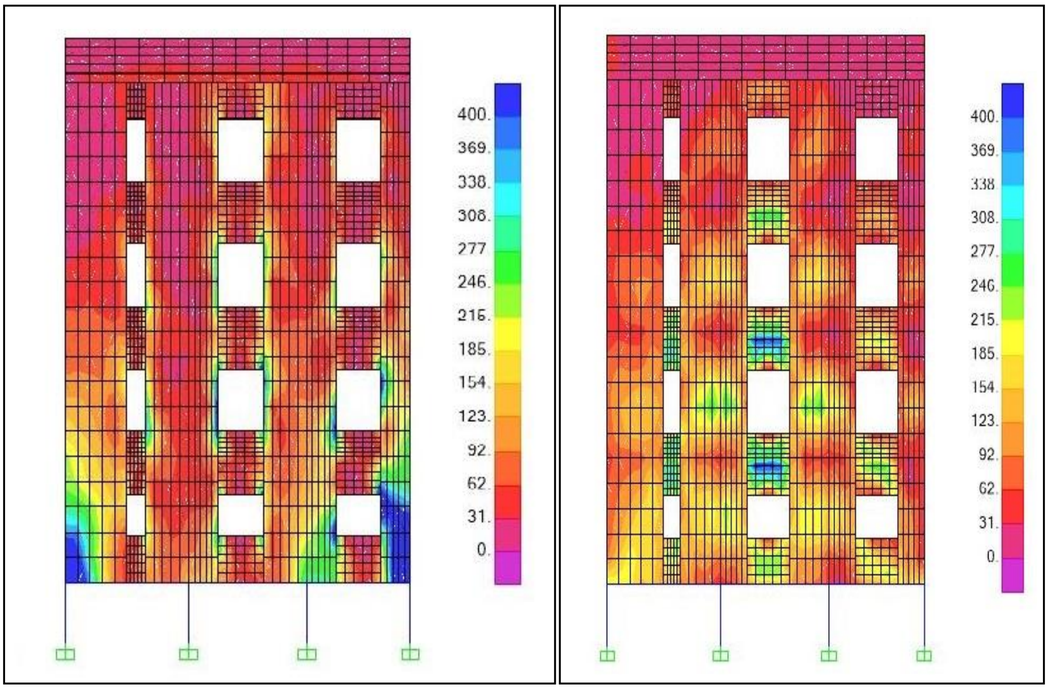

Figure 4.5: Stresses in wall B-B due to most critical load combinations a) Vertical b) Shear in kN/m^2

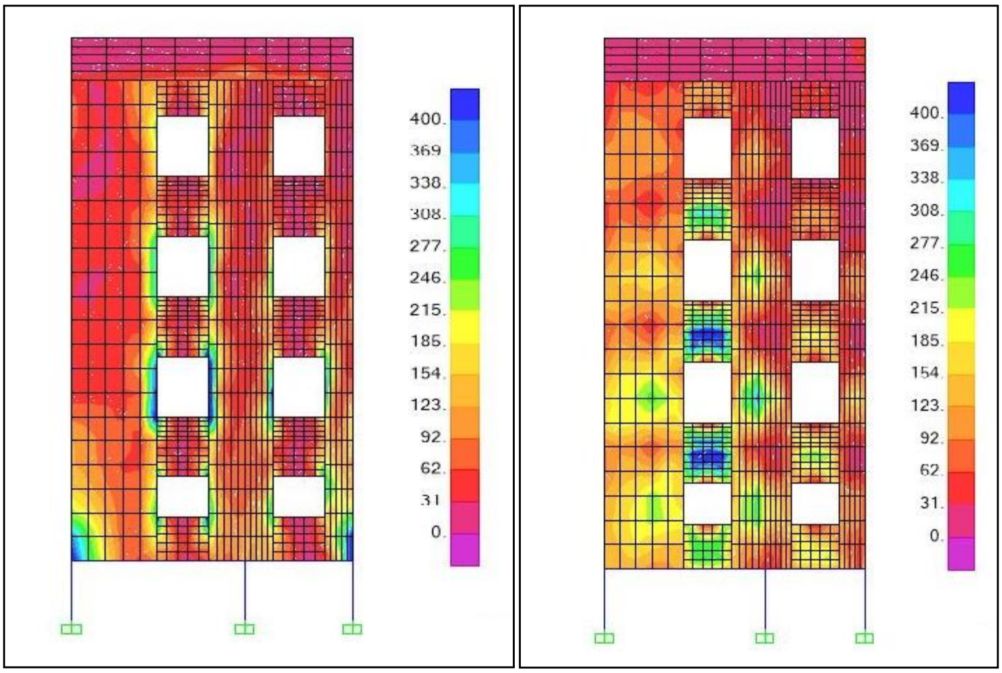

Figure 4.6: Stresses in wall C-C due to most critical load combinations a) Vertical b) Shear in kN/m^2

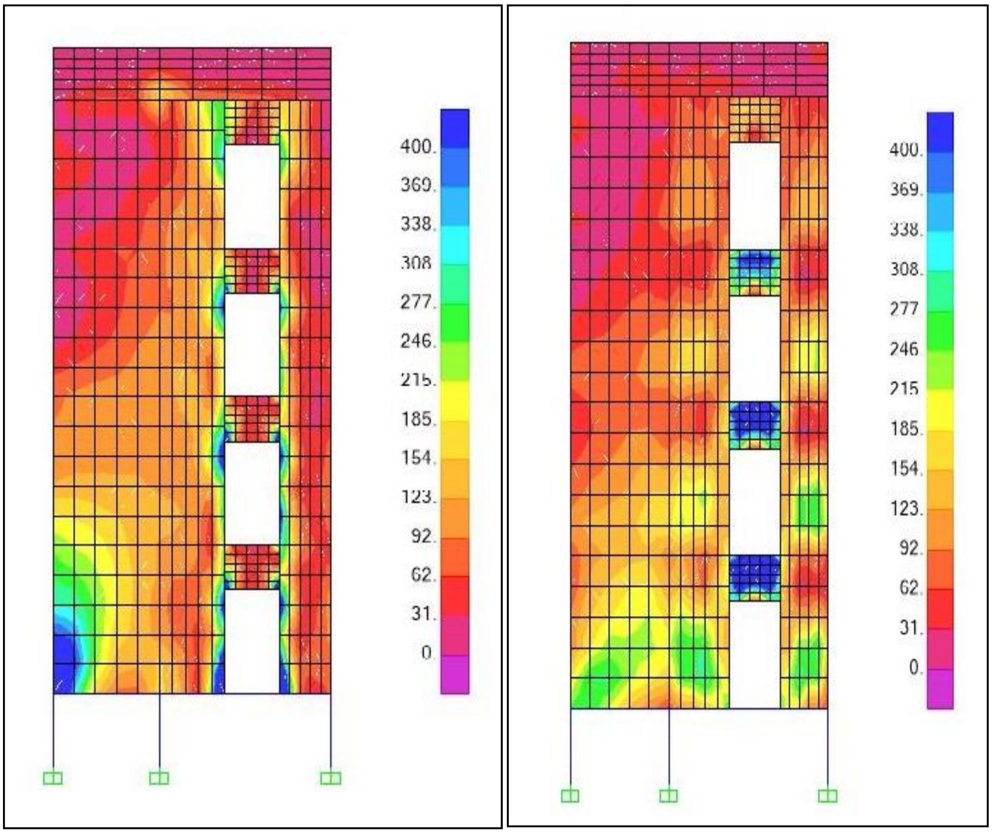

Figure 4.7: Stresses in wall D-D due to most critical load combinations a) Vertical b) Shear in kN/m^2

From the stress contours of all the walls due to the most critical load combinations, it is observed that the maximum values are within the permissible stress limits of RCC walls as mentioned in IS:456 (2000). Thus, from the stress contours of the walls, it can be stated that the design of the building is safe to withstand all critical seismic load combinations.

4.3.4 Comparison with pier analysis results

To compare the results of pier analysis method and finite element modelling, shear forces are computed in all the walls (at each storey) and then in the individual piers due to all the load combinations and are compared. The table below shows the calculated shear forces in all the walls at ground storey oriented along the x-direction by both pier analysis and finite element method. For the pier analysis method, values are shown for two cases i.e. considering the effect of torsion and also ignoring the effect of torsion. The wall B-B shown in Fig. 44 is numbered as WALL-1 and the order is increased along the direction of y-axis in Fig. 44, thus terminating at wall A-A, which is numbered as WALL-9.

Table 4.1: Comparison of pier analysis and finite element analysis (FEA) results for shear force in X-direction walls due to earthquake forces in X-direction

Wall number	Shear force from FEA in SAP 2000 (kN)	Shear force from pier analysis ignoring effect of torsion (kN)	Shear force from pier analysis considering torsion (kN)
WALL 1	117.6	131.2	144.1
WALL 2	46.6	35.3	37.6
WALL 3	48.5	43.4	45.3
WALL 4	71.3	61.1	62.7
WALL 5	134.1	125.4	126.8
WALL 6	43.1	43.9	44.1
WALL 7	186.3	191.1	195.5
WALL 8	62.7	66.9	68.5
WALL 9	66.3	74.2	69.5

The building has very low value of eccentricity, thus values obtained by pier analysis method considering the effect of torsion and ignoring the effect of torsion does not have much difference. It can be observed from the above table, that the values of shear forces obtained in the walls by finite element modelling in SAP 2000 are almost same as those obtained by pier analysis method. This resemblance is observed for walls of all the storeys due to all load combinations, in both x and y directions, thus validating the finite element model.

4.3.5 Capacity and demand at the critical sections

By using SAP 2000, the axial force and moment carrying capacity of all the critical sections are calculated by plotting the P-M interaction curves of all the critical sections. The shear force carrying capacity of the sections are calculated using Clause 32.4 of IS 456: 2000. The stress-strain curve of concrete and steel wire mesh (considering partial safety factors) used for calculating the capacities of the sections is given below.

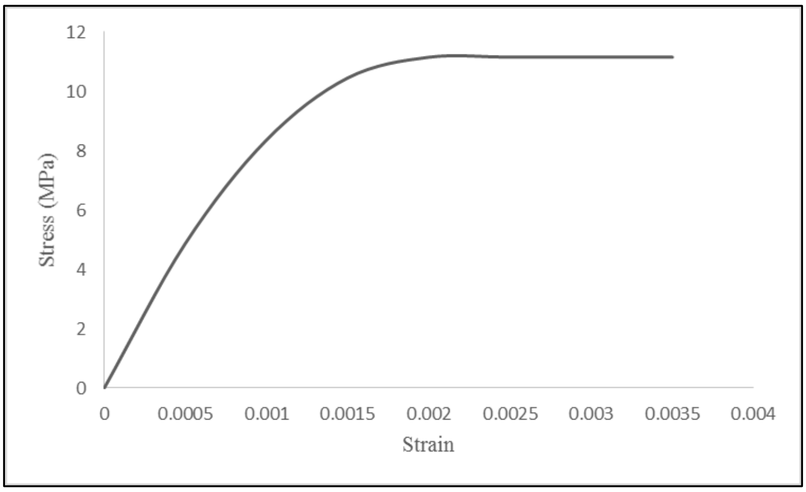

Figure 4.8: Stress-strain curve of concrete used for estimating capacities of sections

The stress-strain values of steel wire mesh of f_y 700 grade is tabulated below.

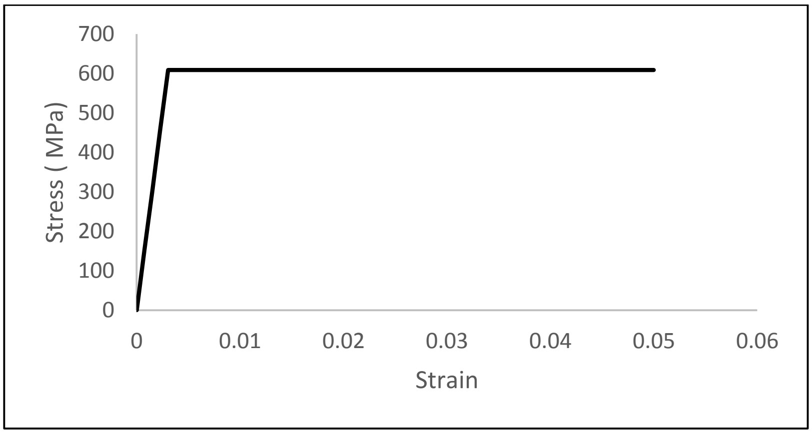

Figure 4.9: Stress-strain curve of steel wire mesh used for estimating capacities of sections

The demand forces that is the axial force, bending moment and shear force coming on the critical sections under all the considered load combinations are then compared with the capacity values obtained. In all the cases, it is observed that the load carrying capacity of the sections are greater than the demand forces coming on the sections. Thus, the thickness of the sections and the reinforcement provided are adequate to carry the design forces on the building. The P-M interaction curves and the demand forces of a few sample critical sections are shown in figures below.

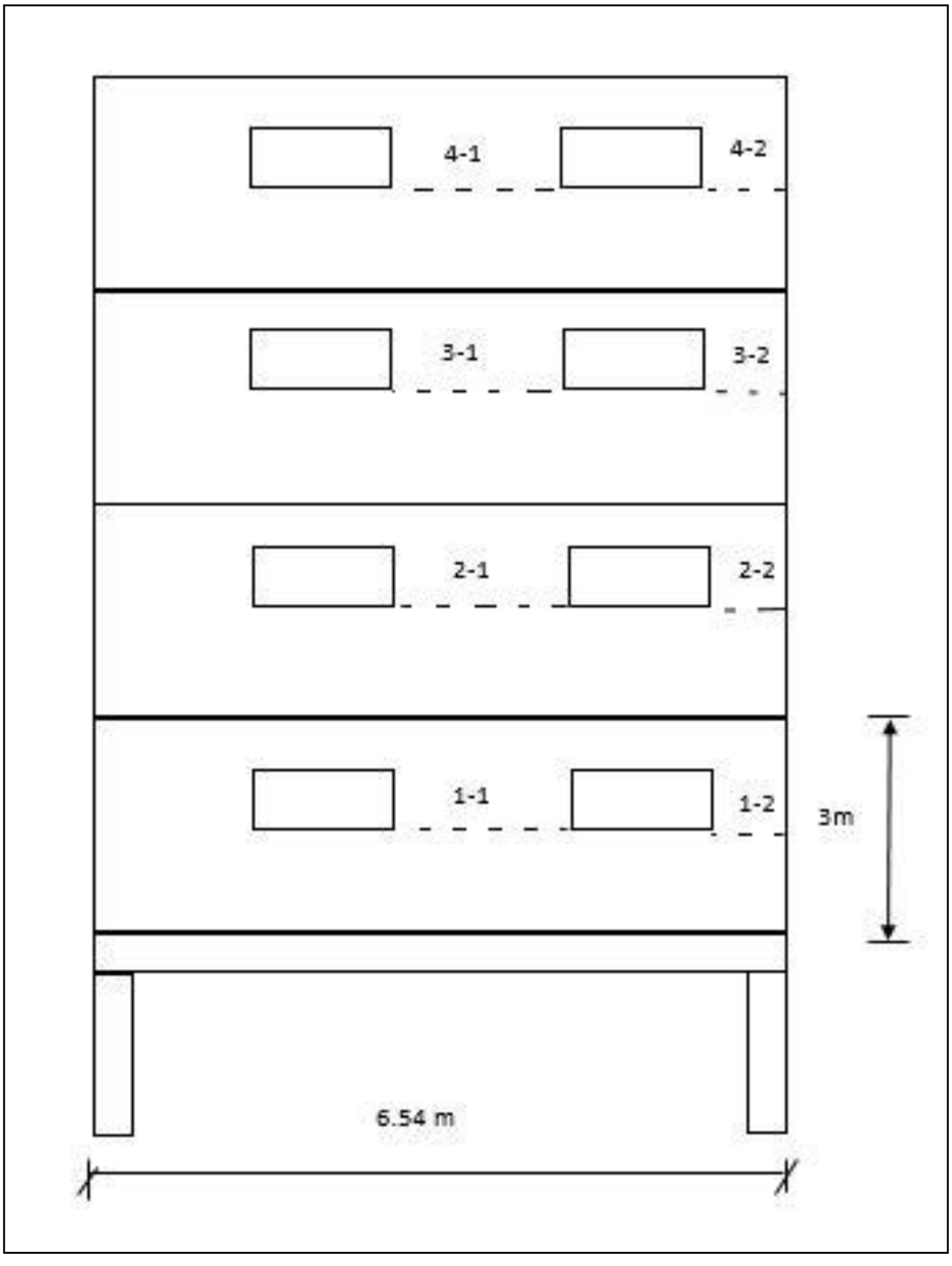

Figure 4.10: Schematic diagram showing the critical section cuts of wall C-C marked in the plan view of the building

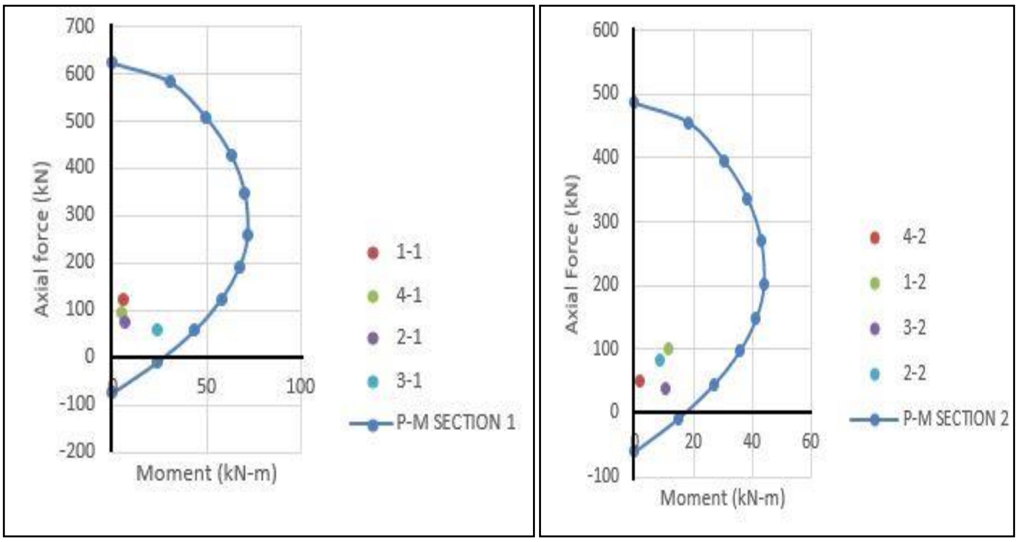

Figure 4.11: P-M interaction curves of a few critical sections showing demand forces

The P-M interaction curves and the plot of the demand forces on them show that the demand capacity ratio is less than 1 for all the critical sections of the building. Thus, from the results of linear analysis, it is concluded that the design of the building is adequate enough to withstand the earthquake forces.

4.4 NON-LINEAR STATIC PUSHOVER ANALYSIS

To understand the behaviour of the structure due to earthquake force, non-linear static pushover analysis is performed using layered shell element in SAP 2000. It is the simple way of assessing the post yield behaviour of a structure by modelling the material non-linearity in the structural members. Non-linear static pushover analysis gives the relationship between the lateral base shear and the corresponding roof displacement at each successive loading step. In the considered building, there are no beams and columns in the superstructure. The structural components in the superstructure are the walls and the slabs to which non-linear material property has been assigned using the non-linear layered shell element. The beams and columns are present below the plinth level and non-linear properties have been assigned to them as per the specifications of FEMA 356. Two sets of analysis load cases (gravity load case and lateral load case) have been created in SAP 2000. The lateral load cases have been defined in both X and Y directions and have been continued from the end of the gravity load case which is a combination of dead, live and seismic loads in the seismic analysis. The lateral loads are applied as mode proportion to storey masses, as per the governing modes in each direction. It was observed from the results of the pushover analysis that the maximum lateral deflection occurred at the beams and columns present at the substructure, while the superstructure almost remained rigid above the substructure. The failure occurred due to failure of the beams and columns due to earthquake forces. Thus, another pushover analysis was performed on the same building with the foundation as raft. It was observed that the building behaved as a more rigid structure in this case, i.e. time period of the building reduced. Deflected shape of both the buildings due to lateral load case is shown in the Figures 4.12.

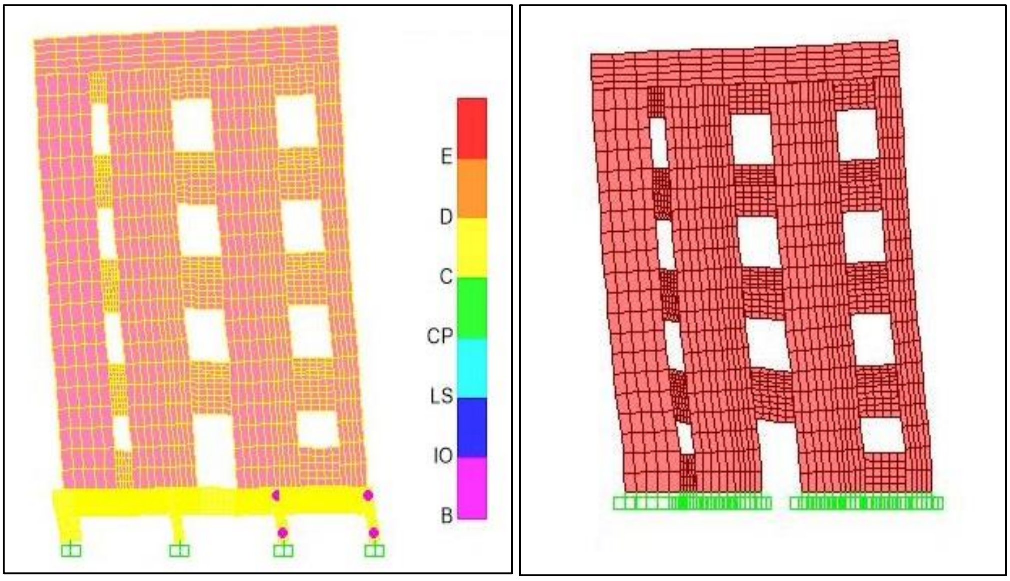

Figure 4.12: Deflected shape of the building with beam-column and raft foundation

It was also observed that there was increase in the capacity curves of the building when the beam-column foundation was replaced by raft foundation. The pushover (capacity) curves obtained for both the cases in each X and Y direction are shown in the Figures 4.13 and 4.14.

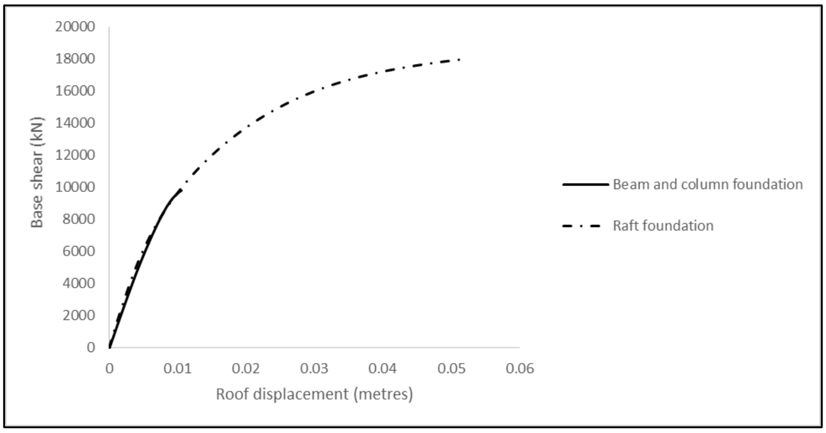

Figure 4.13: Pushover curve of the building in X-direction

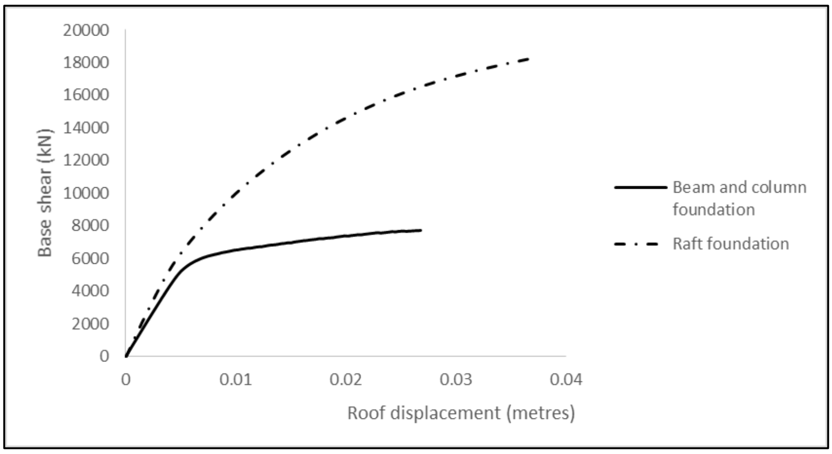

Figure 4.14: Pushover curve of the building in Y-direction

The shear force carrying capacity of the individual walls in each direction have been calculated by considering them as RCC sections as per IS: 456 (2000) and then the total lateral force carrying capacity of the building in each direction has been obtained by adding the capacity of all the walls in a particular direction. It has been observed that the lateral force carrying capacity of the building obtained from the non-linear static pushover analysis in SAP 2000 overestimates the values by almost two times.

The performance point of the building for Design Basis Earthquake (DBE) and Maximum considered Earthquake (MCE) in both X and Y directions have been calculated by ASCE 41 and given in the Table 4.2.

Table 4.2: Roof drift at performance points calculated using ASCE 41 methodology

	DBE	MCE	DBE	MCE
Pushover Load case	Beam column Foundation	Beam column Foundation	Raft Foundation	Raft Foundation
X	0.745	1.38	0.68	1.18
Y	0.3	0.8	0.24	0.68

It can be observed from Table 4.2 that the roof drifts at performance points are lower for the case with raft foundation. This is due to the fact that there is increase in rigidity for the case with raft foundation in comparison to the building with beam-column foundation. Though the values obtained from non-linear static pushover analysis in SAP 2000 using layered shell element are wrong, it is used as performing non-linear analysis of the building using solid element needs a very high computation power which was not available.

Chapter 5

CONCLUSIONS

1) The results of modelling of in-plane shear behaviour of the EPS core panels using solid-8 noded hexahedral element (C3D8) for the EPS and concrete layers and 2-noded linear truss elements (T3D2) for wire mesh in ABAQUS matched satisfactorily with the results obtained from the experiments performed on the panels. The maximum diagonal compressive load that can be carried by the 0.77 metre by 0.77 metre panels was observed to be around 260 kN. But, the modelling using layered shell element in SAP 2000 did not give satisfactory results in non-linear analysis, as the tension damage and failure could not be modelled properly.
2) The results of modelling of out-of-plane flexural behaviour of the panels using the solid 8-noded hexahedral element (C3D8) for the EPS core and concrete layers and 2-noded linear truss element (T3D2) for wire mesh in ABAQUS matched satisfactorily with the experimental results till yielding of the member. In the experiments performed, it was observed that there was bond failure between concrete and steel wire mesh after yielding, thus the post-yield behaviour could not be well predicted by finite element model.
3) The forces calculated at different sections of the building using finite element analysis in SAP 2000 matched satisfactorily with the same computed by pier analysis method.
4) The stresses in the walls due to the most critical load combinations showed that the maximum stresses developed were within the permissible limits of IS: 456 (2000).
5) The P-M interaction curves of the critical sections of the building developed and the plot of demand forces on them due to the most critical load combinations on them showed that the demand-capacity ratio was less than 1 in all the cases thus implying good seismic performance of the building.
6) The non-linear static pushover analysis of the building using the layered shell element in SAP 2000 overestimates the lateral force carrying capacity of the building by almost two times as calculated by computing the shear force

carrying capacity of the walls as per IS: 456 (2000). The values obtained by this process are not correct, but still used in this dissertation as performing non-linear analysis of the building using solid element requires a very high computation power which was not available during the study.

7) From the non-linear analysis results, it is observed that the time period of the structure reduced when the beam-column foundation was replaced by raft foundation and the capacity also increased up to two times.

REFERENCES

1) ASCE/SEI 41-13, “Seismic Evaluation and Retrofit of Existing Buildings”, American Society of Civil Engineers, Virginia, 2015.
2) ASTM (2010a). “Standard Test Method for Diagonal Tension (Shear) in Masonry Assemblages.” ASTM E-519/E-519M.
3) ATC 40. 1996. Seismic Evaluation and Retrofit of Concrete Buildings, Applied Technology Council, Redwood City, California, USA.
4) Bajracharya, R.M., Lokuge, W.P., Karunasena, W., Lau, K.T., Mosallam, A.S., (2011). “Structural evaluation of Concrete Expanded Polystyrene sandwich panels for slab applications”, Journal of Engineering Structures, Volume 37, January 2012.
5) Carbonari, G., Cavalaro, S.P.H, Cansario, M.M., Aguado, A. (2012). “Experimental and analytical study about the compressive behaviour of eps sandwich panels”, Journal of Construction and Building Materials, Volume 35, 2012.
6) Carbonari, G., Cavalaro, S.P.H., Cansario, M.M., Aguado, A. (2012). “Flexural behaviour of light-weight sandwich panels composed by concrete and EPS”, Journal of Construction and Building Materials, Volume 35, 2012, 792-799.
7) Desayi, P., and Krishnan, S. (1964). "Equation for the Stress-Strain Curve of Concrete." ACI Journal Proceedings, 61(3).
8) Drysdale, R.G., Hamid, A.A., Baker, L.R. (1999). “Masonry structures: behavior and design: The Masonry Society”.
9) FEMA 356 (2000). "Prestandard and Commentary for the Seismic Rehabilitation of Buildings. “Federal Emergency Management Agency, Washington, D.C., USA.
10) IS-1893 (Part 1):2002-“Criteria for Earthquake Resistant Design of Structures”-Part 1-General Provisions and Buildings-Fifth Revision-Bureau of Indian Standards-New Delhi-June 2002.
11) IS-456:2000-“Plain and Reinforcing Concrete-Code of Practice”-Fourth Revision-Bureau of Indian Standards-New Delhi-July 2000

12) IS 875 Part I (1987). "Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures.” Bureau of Indian Standards, New Delhi; India.

13) IS-875 (Part 2):1987-“Code of Practice for Design Loads (Other than earthquake) for buildings and structures”-Part 2 Imposed Loads-Second Revision-Sixth Revision-June 1998-Bureau of Indian Standards-New Delhi.

14) Jankowiak, T., and Lodygowski, T. (2005). "Identification of parameters of concrete damage plasticity constitutive model." Foundation of Civil and Environmental Engineering, 06, 53-69.

15) Kadam, S.B. (2015). “Seismic Evaluation and Retrofit of Masonry Buildings.” Ph.D. Thesis, IIT Roorkee, Roorkee.

16) Pillai, S. U., and Menon, D. (2010). Reinforced Concrete Design, Tata McGraw-Hill Education Private Limited, New Delhi, India.

17) SAP 2000 (2010). "Integrated Software for Structural Analysis and Design-Analysis Reference Manual”, Computers and Structures, Inc., Berkeley, California, USA.