University of Southern Queensland

Faculty of Engineering and Surveying

STRUCTURAL EVALUATION OF CONCRETE-EXPANDED POLYSTYRENE SANDWICH PANELS FOR SLAB APPLICATIONS

A dissertation submitted by

Rohan Muni Bajracharya

In fulfilment of the requirements of

Courses ENG8411 and ENG8412 Research Project

towards the degree of

Masters of Engineering Science (Structural Engineering)

Submitted: October 2010

University of Southern Queensland

Faculty of Engineering and Surveying

ENG8411 Research Project Part 1 & ENG8412 Research Project Part 2

Limitations of Use

The Council of the University of Southern Queensland, its Faculty of Engineering and Surveying, and the staff of the University of Southern Queensland, do not accept any responsibility for the truth, accuracy or completeness of material contained within or associated with this dissertation.

Persons using all or any part of this material do so at their own risk, and not at the risk of the Council of the University of Southern Queensland, its Faculty of Engineering and Surveying or the staff of the University of Southern Queensland.

This dissertation reports an educational exercise and has no purpose or validity beyond this exercise. The sole purpose of the course pair entitled “Research Project” is to contribute to the overall education within the student's chosen degree program. This document, the associated hardware, software, drawings, and other material set out in the associated appendices should not be used for any other purpose: if they are so used, it is entirely at the risk of the user.

Professor Frank Bullen

Dean

Faculty of Engineering and Surveying

Certification

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

Rohan Muni Bajracharya
Student Number: 0050108273

Signature

26 October 2011

Date

Abstract

This dissertation presents a structural evaluation of Concrete-Expanded Polystyrene (CEPS) sandwich panels for slab applications through the finite element analysis approach. It is based on the experimental work previously conducted at University of California, Irvine (UCI). These panels comprise of expanded polystyrene foam sandwiched between concrete-steel panels.

This research uses structural software Strand7, which is a numerical approach based on finite element method, to predict the load deformation behaviour of the CEPS sandwich slab panels. The concrete facings of the panel and expanded polystyrene (EPS) foam are modeled using brick elements whereas steel wire mesh is represented by cut-off truss elements. The model was analyzed by non linear static analysis. Finite element results are compared with experimental data to validate the numerical approach used. The verified model is used for parametric study to understand the behaviour of CEPS panels under different load combinations.

This research shows that a simple Strand7 finite element model can be used for the analysis of CEPS sandwich panels for slab applications. Analytical results using FEA show good correlation with the experimental results. Furthermore, the use of foam in the middle of the sandwich panel will reduce the weight of the structure and also acts as insulation against thermal, acoustics and vibration. The design chart developed for various thicknesses of CEPS slab panel can be used for low cost residential and commercial structures.

Acknowledgments

I would like to thank everyone who has assisted me in any form throughout the time spent doing this research. In particular I would like to make a special mention of:

- Associate Professor Karu Karunasena and Dr. Weena Lokuge for the valuable guidance they have provided to me as my research supervisor.
- My family for their continued support in all of my endeavours.
- Australian Government for supporting me financially for studying Masters degree.
- University of California, Irvine for providing the experimental data for the analysis.
- Chamila Sirimanna for teaching me Strand7 and giving me information regarding finite element modeling.

I am very grateful for the input and assistance provided by these people.

Rohan Muni Bajracharya

University of Southern Queensland

October 2011

Table of Contents

List of Figures

List of Tables

Nomenclature

A_s	Area of each steel (mm^2)
A_{ts}	Total Area of steel (mm^2)
b	Width of test specimen (mm)
C_c	Compressive concrete Force (kN)
C_s	Compressive Steel Force (kN)
D	Depth of test specimen (mm)
d	Effective depth, from the extreme fibre in compression to the resultant tensile force T, of a cross-section in bending (mm)
d_n	Depth of neutral axis of a cross-section in bending (mm)
Ec	Modulus of Elasticity (MOE) of Concrete (MPa)
Ef	Modulus of Elasticity (MOE) of Expanded Polystyrene Foam (MPa)
Es	Modulus of Elasticity (MOE) of Steel (MPa)
E_{ti}	Young's modulus of elasticity of EPS suggested by Horvath (MPa)
E_{ts}	Young's modulus of elasticity of EPS suggested by Duskov (MPa)
f'_c	Peak stress of concrete in compression (MPa)
f'_t	Peak stress of concrete in compression (MPa)
f_{c1}	Ultimate stress of concrete in compression (MPa)
f_{cm}	Mean compressive strength (MPa)
f_{sy}	Yield stress of steel (MPa)
f_{t1}	Ultimate stress of concrete in compression (MPa)
G	Shear modulus (MPa)
G_f	Fracture energy
h	Total Depth of test specimen (mm)
h_c	Crack band width
k_u	Neutral axis depth parameter for a cross-section, dn/d

l	Length of specimen (mm)
M	Bending moment (kN.m)
M_{max}	Maximum bending moment (kN.m)
M_u	Ultimate bending capacity of a cross-section (kN.m)
P	Load applied to test specimen (kN)
T	Tensile force in the steel (kN)
$\mathbf{t_c}$	Thickness of Expanded Polystyrene Foam (mm)
$\mathbf{t_f}$	Thickness of concrete layer (mm)
γ	Compressive stress block factor
δ	Central deflection of a simply supported slab (mm)
ε_{c1}	Ultimate strain of concrete in compression
ε_{co}	Peak strain of concrete in compression
ε_{ct}	Peak strain of concrete in compression
ε_{sc}	Compressive strain in steel reinforcement
ε_{sp}	Failure strain of concrete in compression
ε_{st}	Tensile strain in steel reinforcement
ε_{sy}	Yield strain of steel
ε_{t1}	Ultimate strain of concrete in compression
ε_u	Failure strain of concrete in compression
ν	Poisons ratio
ρ	Material Density (kg/m^3)

Glossary

CEPS	Concrete-Expanded Polystyrene
EPS	Expanded Polystyrene
FEA	Finite Element Analysis
FOES	Faculty of Engineering and Surveying
UCI	University of California, Irvine
UDL	Uniformly Distributed Load
USQ	University of Southern Queensland

Chapter 1

1. Introduction

1.1. Summary

Sandwich panels are being extensively and increasingly used in single storey and multi-storey building construction because they are light in weight, energy efficient, aesthetically attractive and can be easily handled and erected. The sandwich panels have been used as structural building components in various industrial and office buildings in many countries. Their uses have now been extended to residential building construction due to their ability to improve the structural and thermal performance of the houses. Sandwich panel construction in Australia has been limited to cold-storage buildings due to the lack of design methods and data. However, in recent times, the sandwich panels are extensively used in buildings, particularly as roof and wall cladding systems.

The structure of sandwich panels consists of two facings, which are relatively thin and of high strength and enclose a core which is relatively thick and light and which has adequate stiffness in a direction normal to the faces of the panel. In this research the panels comprise of expanded polystyrene foam sandwiched between concrete panels and steel wire mesh. The concrete expanded polystyrene (CEPS) sandwich panels are made of foam panels with robotically welded steel mesh on each side and three dimensional truss system steel welded through the center foam panel. Applying reinforced concrete skin to both sides of the panel takes the advantages of the sandwich concept where the reinforced concrete faces take compressive and tensile loads resulting in higher stiffness and strength and the core transfers shear loads between the faces. The expanded polystyrene (EPS) core also provides excellent insulation against heat, sound and vibration. Besides these, the expanded polystyrene (EPS) core also has construction viability as it provides a support mechanism for steel wire mesh for construction. Hence, concrete expanded polystyrene (CEPS) sandwich panels represent an excellent example of the optimum use of dissimilar materials.

Following are some of the benefits of concrete expanded polystyrene (CEPS) sandwich panels:

- Easy erection
- Light-weight on-site sandwich panels
- Reduces Carbon emissions
- Superior Thermal Insulation
- Superior Impact and Fire Resistance
- Excellent for Affordable Mass and Rapid Housing Project

1.2. Background

Prior to about 1960, sandwich technology was confined almost entirely to aerospace applications. By about 1960, the sandwich panel construction began a worldwide boom in prefabricated building elements for diverse applications (Davies 2001).

Due to considerable structural importance, a large number of publications dealing with structural sandwich panels are in existence. However, Rizzo and Fazio (1983) found that their analytical results exceed the actual values by 15% for sandwich wall and slab panels. Sokolinsky et al. (2003) found that the classical sandwich theory underestimates the vertical displacements of the sandwich beam specimens by more than 20%. All of this evidence indicates that more research work needs to be done for understanding the behaviour of sandwich panel. Hence, this research uses a numerical approach based on finite element method to predict the load deformation behaviour of the concrete expanded polystyrene (CEPS) sandwich slab panels. Finite element results are compared with experimental data to validate the numerical approach used.

1.3. Research Significance and Objectives

Most of the sandwich panel construction is confined to panelized construction. Panelized construction is a method where the building is subdivided into basic planar elements that are typically constructed under some form of mass production then shipped directly to the construction site and assembled into the finished structure. Although the panelized construction has advantages but this research focuses on the construction of such panel on site. This research is concerned with sandwich panels having concrete-steel faces and polystyrene core materials which can be casted on site and will be cheaper as well.

As the use of expanded polystyrene foam in the middle of concrete-steel facings is a relatively new concept, in such circumstances, there is a need to verify the applicability of such new panels in order to develop the necessary confidence among Australian manufacturers and designers. This clearly indicates the need for research to investigate the behaviour of concrete expanded polystyrene (CEPS) sandwich panels. Hence, this research work towards achieving accurate design recommendations for concrete expanded polystyrene (CEPS) sandwich panel's construction in the Australian environment.

This research on the structural evaluation of CEPS sandwich panels is intended to utilize the sandwich panels in a manner that is safe and reliable. There is different kind of sandwich panels that have been used however research is needed to investigate issues associated with the development of CEPS sandwich panels as a slab flooring system. Therefore, the scope of this research mainly focuses on the structural evaluation of CEPS sandwich panel as a slab flooring systems, supported by the traditional wall configuration.

Overall Objective

The main purpose of this research is to undertake a thorough investigation on the behaviour of CEPS sandwich panels and develop a finite element model of CEPS sandwich panels to predict load deformation curve. Moreover, this research can reduce the number of prototypes that need to be built.

Specific Objectives:

Briefly, the specific objectives of the project can be summarized as:

1. Conduct a literature review on sandwich panels.
2. Create a finite element model of CEPS sandwich panels using Strand7.
3. Validate the model by using the test data provided by University of California, Irvine.

4. Use the results from modeling to create a design aid for the use of CEPS sandwich panel in slab construction.

1.4. Method of Investigation

This dissertation is mainly based on a series of laboratory experiments performed at University of California, Irvine (UCI) followed by numerical studies. Laboratory experiments include the four point bending test performed in three different CEPS sandwich panels.

Finite element analyses were carried out using the finite element program Strand7. For the analysis of the CEPS sandwich panels, available theoretical and previous research papers were used to define the properties of the materials. Experimental results were used as the benchmark data to calibrate the finite element models created and analysed in this study. Finally, finite element analysis results were used to do a parametric study and develop a design chart for CEPS sandwich panels to be used as structural slabs in buildings.

1.5. Overview of the dissertation

This dissertation consists of six chapters. Chapter 1 presents an introduction to impart an understanding of the principal reasons for the commencement of this research, followed by the research significance and objectives of this dissertation. Chapter 2 provides an overview on the work related to this research including past research on sandwich panels, material properties and relevant Australian Standards.

The main body of the dissertation starts at Chapter 3 and goes through to Chapter 6. Chapter 3 described the experimental program used to obtain the behaviour of Concrete-Expanded Polystyrene (CEPS) Sandwich panels subjected to four point bending test. The experimental results of the testing in this chapter are used to get the information required for the finite element analysis modeling undertaken in the following chapter. Chapter 3 also describes the simplified method of analysis to predict the failure load. Chapter 4 is associated with determining the load displacement curve of the CEPS sandwich panels by using finite element analysis. Finite element analysis (FEA) results will be verified using the experimental result given in previous chapter. The FEA model will be used to do a parametric study and develop a design chart for CEPS slab panels in the following chapter. Chapter 5 is associated with determining the limiting design criteria for CEPS sandwich panels, comparing the experimental results to the current method of floor construction, and using the results of modeling to create a CEPS slab panel's design chart.

This dissertation concludes with Chapter 6 with the important conclusions that are based on the findings of this research. Fulfillment of the set project objectives is also presented along with recommendations for future research.

Chapter 2

2. Literature Review

2.1. Introduction

Although the Second World War ‘Mosquito’ aircraft is often quoted as the first major structure to incorporate sandwich panels, sandwich construction has been used in many earlier but less spectacular circumstances. Reviewers of the history of sandwich construction compete to name the first person to describe the principle and the record appears to be held by Fairbairn (1849).

People have been looking for light but strong materials ever since they began to use devices for transportation. After the Wright brothers invented the airplane, the search of such materials intensified because it is extremely important that light and strong materials be used for the design of a wing and fuselage structure in order for an airplane to provide a high payload under a given thrust force from the propelling system (Ueng 2001).

Sandwich panels for aircraft structures almost invariably employ metal faces with metal honeycomb or corrugated cores. The honeycomb is formed from strips of thin aluminum alloy or steel foil deformed and joined together. The corrugated core is a fluted metal sheet attached alternately to the upper and lower faces. The first extensive use of sandwich panel was during the World War II. The famous British Mosquito airplane was manufactured by adopting the sandwich idea with the use of lightweight balsa wood. Since it increases stiffness tremendously without adding too much weight, sandwich construction has been used ever since, not only in aircraft and space vehicle design, but also for ground transportation, building construction, packaging, and other engineering applications (Allen 1969).

The first successful landing of a space ship on the moon on 20 July 1969 was the result of the successful application of a number of new technologies including rocketry, computers and sandwich construction. However panels for use in the building industry have hitherto been of a mainly semi-structural character, called upon to carry relatively small loads over fairly long spans. Building panels, like aircraft panels, should be light in weight but, unlike the aircraft panels, they must be cheap. All metal panels may yet find substantial application in buildings but there is also great scope for many other materials (Davies 2001).

New materials and new combinations of old materials are constantly being proposed and used in sandwich panels. Sandwich panels have many engineering applications from wall, slab to beam. Karam and Gibson (1994) evaluated the wood-cement and natural-fibre-cement to be used as a sandwich-panel facing by performing three-point bending test. Pokharel (2003) studied the behaviour and design of sandwich panels made up of steel as a skins and polystyrene foam as a core. The author further mentioned that the structural sandwich panels generally used in Australia comprise of polystyrene foam core and thinner (0.42 mm) and high strength (minimum yield stress of 550 MPa and reduced ductility) steel faces bonded together using separate adhesives.

Schenker et al. (2005) studied the behaviour of aluminum foam protected reinforced concrete structures under impact. Vaidya et al. (2010) demonstrated the panels consisting facesheets of E-glass fibers impregnated with polypropylene matrix, while the core consists of expanded polystyrene foam developed for the exterior walls of a modularized structure. Manalo (2011) investigated the concept of glue-laminated composite sandwich beams made up of glass fibre composite skins and modified phenolic core material for railway turnout sleepers.

This research focuses to use EPS as a foam and concrete wire meshing as a facesheets which will be easier to cast on site and cheaper as well.

2.2. Concept and functions of Sandwich Panels

The concept behind sandwiched construction is to have the facings and core as shown in Figure 2-1 to act in concert as a very efficient structural element. The function of sandwich structures can be compared to that of I-sections, in which the facings of a sandwich panel can be compared to the flanges of an I-beam, as they carry the bending stresses. Core corresponds to the web of the I-beam, as it resists the shear loads and stabilizes the faces against bulking or wrinkling (Zenkert 1995).

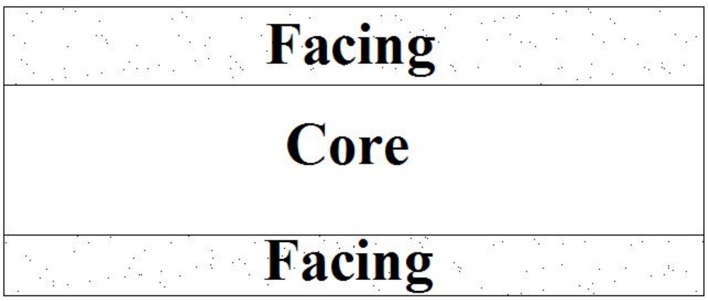

Figure 2-1 Basic Sandwich Structure

The core must be stiff enough to ensure that the facings remain at a proper distance apart. The core must also provide adequate shearing strength so that the facings will not slide relative to each other when the sandwich panel is bent. In the absence of necessary shear strength, the two thin facings would act as two independent beams or panels, and lose the sandwich effect. Finally, the core must also possess enough stiffness so that the facings will stay flat or nearly flat when they are subjected to compressive stresses that would otherwise cause buckling or wrinkling. The objective of the sandwich composite is to offer a structure that is strong and stiff but at the same time lightweight.

The major advantages of sandwich composites over conventional materials are that sandwich composites

(1) have a low overall density, a high strength-to-weight ratio, and a high stiffness-to-weight ratio

(2) are capable of providing good thermal and acoustical insulation; and

(3) have uniform energy absorption capacity.

Such overall versatility has contributed greatly to the development of lightweight sandwich composites (Ueng 2001) .

2.3. Analysis in Sandwich Panels

A great many alternative forms of sandwich construction may be obtained by combining different facing and core materials. The facings may be made of cork, balsa wood, rubber, solid plastic material or rigid foam material, mineral wood slabs or from honeycombs of metal or paper.

Analytical study of Sandwich Panel was considered by many researchers like Rizzo, Davies, Sokolinsky etc. Rizzo and Fazio (1983) used two dimensional analysis of sandwich panel, having aluminum facings and Styrofoam core; found that their analytical results will generally exceed the actual values by some 15% for sandwich wall and slab panels. Davies (1987) proposed the appropriate methods of analysis for sandwich panels subject to combined axial compressive load and bending moment and an exact finite element is derived. The author suggested that sandwich panels subject to axial load is necessary to consider the shift of the neutral axis caused by local buckling in order to obtain reasonable estimates of the failure load.

Sokolinsky et al. (2003) demonstrated four-point loading tests carried out on sandwich beam specimens with aluminum facesheets and a PVC foam core. The authors found that the classical sandwich theory underestimates the vertical displacements of the sandwich beam specimens by more than 20%. They suggested that higher-order plate theory can be efficiently used to estimate vertical displacements of soft-core sandwich beam with great accuracy. Chakrabarti and Sheikh (2005) refined higher-order shear deformation theory by using a six-noded triangular element having seven degree of freedom at each node. However, in this research, the panel comprises of concrete along with steel wire mesh and longitudinal reinforcement as a facings and EPS as a core. Numerically, the combination of concrete, wire mesh, longitudinal reinforcement and EPS as a composite material is modeled by using finite element.

2.4. Material Properties of CEPS panels

CEPS panels are made up of three materials with different characteristics, namely, concrete, steel and expanded polystyrene foam. Concrete is a heterogeneous material made up of cement, mortar and aggregates. Mature, hardened concrete has good compressive strength, typically between 30 and 60 MPa. Its mechanical properties scatter more widely and cannot be defined easily. For the convenience of analysis and design, however, concrete is often considered a homogeneous material in the macroscopic sense. Steel can be considered a homogeneous material and its material properties are generally well defined (Warner 2007). On the other hand, EPS is a lightweight material with a good insulation and energy absorption characteristics (Mousa & Uddin 2010).

The combination of these materials will be a good result for both withstanding the design load and at the same time the structure will be lighter, cheaper and thermally insulated. The properties of these materials generated by previous researcher are discussed in detail.

2.4.1. Concrete

Concrete is made by mixing coarse aggregates and sand with cement and water. After a short period of time, the fresh concrete undergoes an initial set as a result of the reaction of the cement with the water. It then goes through a hardening process that continues over weeks, months and even years. The strength of the concrete increases with time rapidly at first but at a progressively decreasing rate (Warner 2007).

Stress-strain curve of Concrete

There are several models reported in the literature to illustrate the stress-strain behaviour of concrete. Among them, one proposed by Bangash (2001) is shown in Figure 2-2. According to Bangash (2001), experimental tests show that concrete behaves in a highly nonlinear manner in uniaxial compression. The stress- strain curve of concrete is linearly elastic upto 30% of the maximum compressive strength. Above this point the curve increases gradually upto about 70-90% of the compressive strength. Eventually it reaches the peak value, then stress-strain curve descends. After the curve descends, crushing failure occurs at an ultimate strain ε_{cu}.

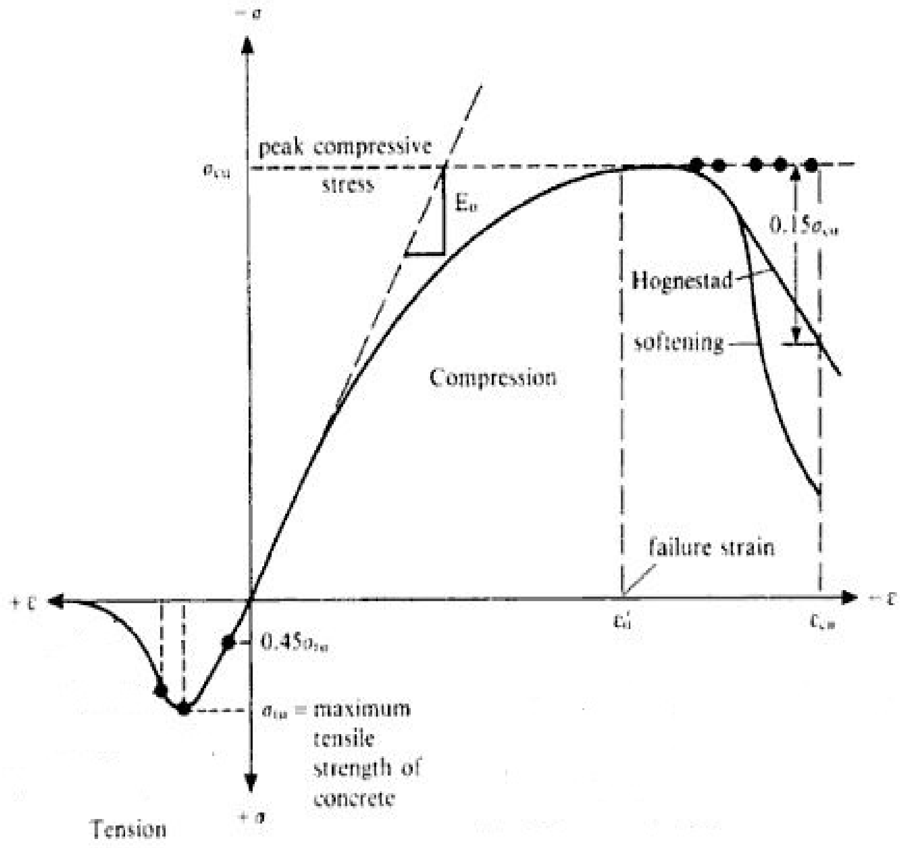

Figure 2-2 Stress-strain curve of concrete (Bangash 2001)

Several researchers have estimated the values for the stress strain curve of concrete. Collins and Mitchell (1994) suggested the stress-strain relation of concrete in compression as shown in Table 2-1.

Table 2-1 Values of the stress-strain curve for concrete in compression based on Collins and Mitchell (1994)

Paramater	Compression
Peak stress	f'_c
Peak strain	ε_{co}= 0.0015+ f'c/70000
Ultimate stress	f_{c1}=12 MPa
Ultimate strain	ε_{c1}=0.0036
Failure strain	ε_{sp}= 0.012-0.0001 f'c

Rots et al. (1985) suggested the stress- strain relation of concrete in tension as shown in Table 2-2.

Table 2-2 Values of the stress-strain curve for concrete in tension based on Rots et al. (1985)

Paramater	Tension
Peak stress	$f'_t = 0.625\sqrt{f'_c}$
Peak strain	ε_{ct}= 0.1 εco
Ultimate stress	f_{t1}= f't/3
Ultimate strain	ε_{t1}=2 εu/9
Failure strain	ε_u= 18 Gf/(5f'th)

Where, G_f= fracture energy = h_c X area under stress-strain softening diagram, and h_c= crack band width

Structural properties of Concrete

Rashid et al. (2002) found that Poisson's ratio changes from 0.15 to 0.25. Initial Poisson's ratio is defined by Candappa (2000) as 0.15. AS3600-2009 recommends to use the Poisson's ratio as 0.2 for concrete.

Bangash (2000) summarises that Young's modulus does not change substantially with time. After 28 days, in general, about 86 % of the final value is reached and, after three months, it attains 99.7 % of its value. The values of Ec defined in some codes are given below:

Australian Standard AS 3600-2009:

Ec = $(\rho)^{1.5} \times (0.043\sqrt{f_{cm}})\ MPa$

The American Concrete Institute (ACI):

Ec = $5000\sqrt{f'_c}\ \ MPa$

Table2-3 shows the structural properties of concrete as per AS3600-2009.

Table 2-3 Structural Properties of concrete as per AS3600-2009

Definition	**Values**	**Clause**
Modulus of Elasticity (E_c)	$(\rho)^{1.5} \times (0.043\sqrt{f_{cm}})$, in MPa	AS3600-6.1.2
Denisty of concrete (ρ)	2400 kg /m^3	AS3600-6.1.3
Poisson's ratio (γ)	0.2	AS3600-6.1.4

Where, f_{cm}= mean value of the compressive strength of concrete

2.4.2. Expanded Polystyrene Foam (EPS)

EPS is a closed cell lightweight cellular plastics material produced from polystyrene. Polystyrene is translated from "polymerized styrene." That is, the single styrene molecules are chemically joined together to form a large molecule which is called the polymer. Styrene is produced from benzene and ethylene, and polymerization is accomplished in the presence of catalysts, usually organic peroxides. The expandable form is produced as small beads containing a blowing agent.

Australian Standard for EPS

Australian Standard AS 1366.3-1992 sets out minimum properties for six classes which in shown in Table 2-4.

Table 2-4 Physical Properties of EPS, according to AS 1366.3-1992

Physical Property	Unit	Class						Test Method
		L	SL	S	M	H	VH	
Nominal Density (kg/m3)		11	13.5	16	19	24	28	N/a
Compressive stress at 10% deformation (min)	kPa	50	70	85	105	135	165	AS2498.3
Cross-breaking strength (min)	kPa	95	135	165	200	260	320	AS2498.4
Rate of water vapour transmission (max) measured parallel to rise at 23°C	$\mu g/m^2s$	710	630	580	520	460	400	AS2498.5
Dimensional stability of length, width, thickness (max) at 70°C, dry condition 7 days	%	1.0	1.0	1.0	1.0	1.0	1.0	AS2498.6
Thermal resistance (min) at a mean temperature of 25°C (50mm sample)	M^2K/W	1	1.13	1.17	1.20	1.25	1.28	AS2464.5 or AS2464.6
Flame propagation characteristics: - median flame duration; max - eighth value; max - median volume retained; - eighth value; min.	 S S % %	 2 3 15 12	 2 3 18 15	 2 3 22 19	 2 3 30 27	 2 3 40 37	 2 3 50 47	AS2122.1

The recommended applications of rigid cellular polystyrene as per AS 1366.3-1992 is shown in Table 2-5. However, AS 1366.3-1992 also recommend that the actual class of rigid cellular polystyrene used with respect to the load applied will be best determined by engineering assessment.

Table 2-5 Recommended applications of rigid cellular polystyrene as per AS 1366.3-1992

Class	Application
L	Decorative panels. Cavity and void forms.
SL, S	Insulation in walls, floors and ceilings; sandwich panels; insulated containers—all under low loads. Pipe and duct lagging—to operate at a maximum service temperature of 80°C.
M	Panels in walls, floors and ceilings; sandwich panels—all under medium loads.
H, VH	Insulated floors and roofs subjected to constant traffic of people and equipment.

EPS density can be considered as the main index in most of its properties. Compressive strength, tensile strength, flexural strength, modulus of elasticity, Poisson's Ratio, creep behaviour and other mechanical properties depend on the density. EPS densities for practical civil engineering applications range between 11 and 30kg/m^3. For other applications like insulation higher densities are more efficient.

In Australia, manufacturers and designers working with EPS are familiar with density classification used by AS 1366.3-1992. Table 2-6 shows the nominal densities of EPS as per AS1366.3-1992. AS1366.3-1992 also states that this table should be used a guide only. The reason for this is because of advances in technology, the physical properties specified in Table 2-6 may be achieved by EPS of other density.

Table 2-6 Nominal densities of EPS as per AS 1366.3-1992

Class	Nominal density kg/m^3
L	11
SL	13.5
S	16
M	19
H	24
VH	28

Stress- Strain Curve of EPS

For compressive strains greater than 1%, EPS behaves nonlinearly, and the Young's modulus value decreases with increasing strain. There are only a few experimental results reported in the literature that address the effect of the loading strain rate on EPS behaviour. Figure 2-3 shows a typical compressive stress-strain curves for EPS specimens having a density of 13kg/m^3. The solid line represents rapid loading conditions, whereas the dashed line was empirically estimated from rapid compression versus time data and corresponds to a very long duration of loading, i.e. it reflects the effects of creep (Horvath 1997).

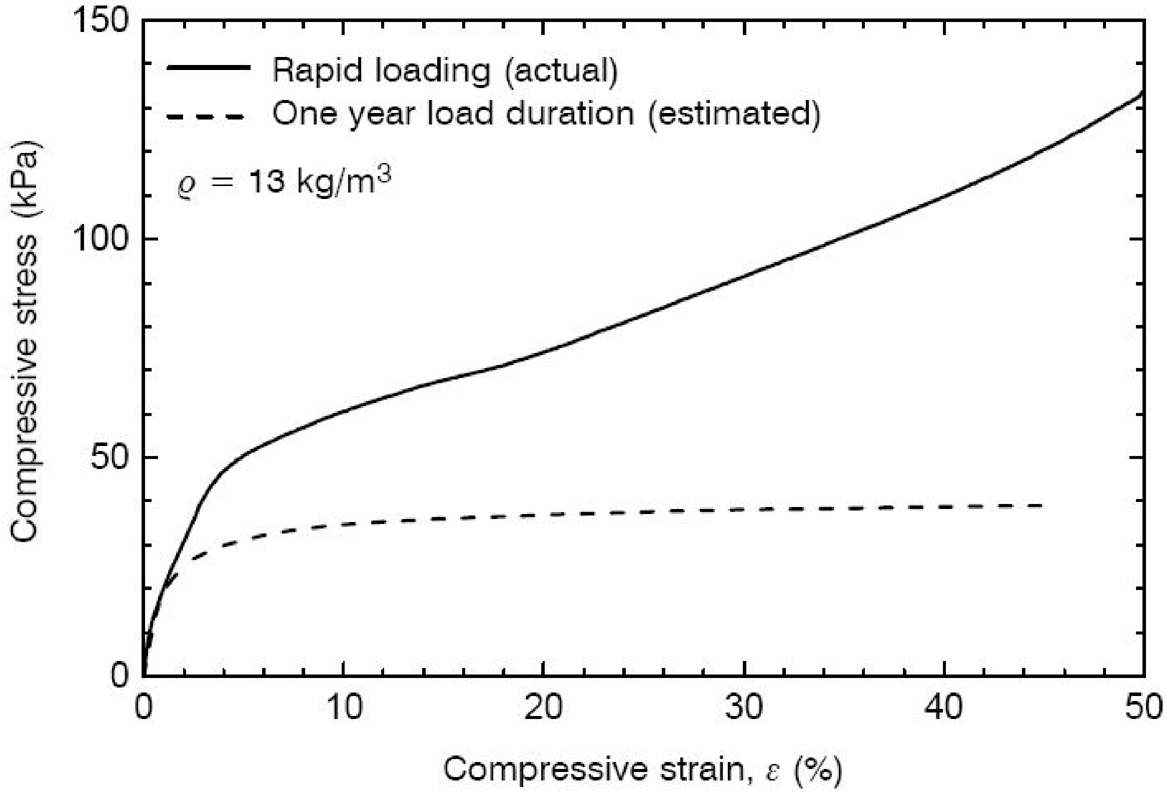

Figure 2-3 Stress- Strain curves of EPS (Horvath 1997)

Structural properties of EPS

The stress strain curve of EPS has an initial linear portion. The value of the slope of this initial portion is known as initial tangent modulus or Young's Modulus of elasticity. For low compressive strains upto approximately 1%, EPS appears to behave linearly and an initial tangent Young's modulus of elasticity, E_{ti}, can be defined, which exhibits an approximately linear correlation with the EPS density. Horvath (1995) has suggested the following empirical equation for estimating E_{ti} values:

$$E_{ti} = 0.45\ \rho - 3 \tag{2.1}$$

where Es has units of MPa and ρ is the EPS density (kg/m^3). The following second order polynomial has also been proposed for calculating E_{ts} values as a function of the EPS density (Duskov 1997):

$$E_{ts} = 16.431 - 1.645\ \rho + 0.061\ \rho^2 \tag{2.2}$$

For both researchers, initial modulus is a function of the density as shown in Figure 2-4.

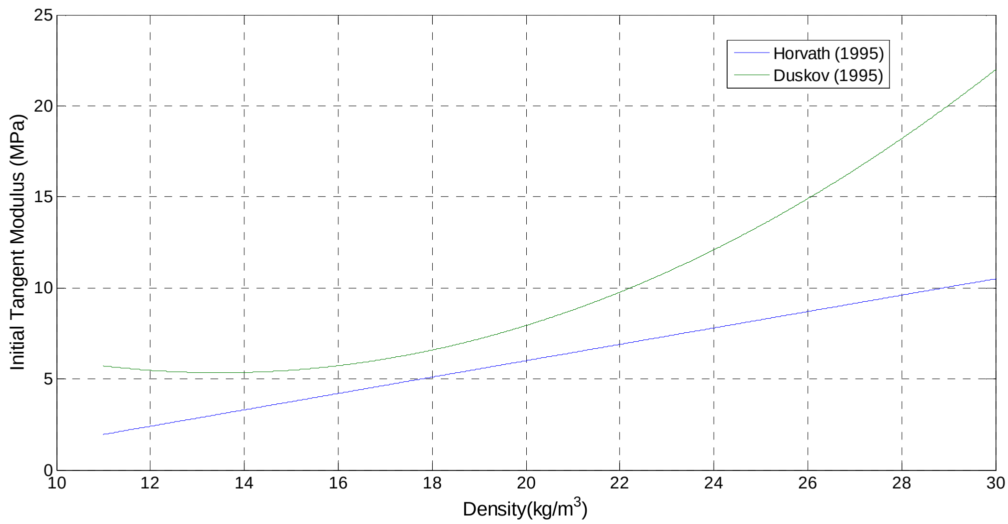

Figure 2-4 Initial Tangent Modulus for EPS

Several researchers have estimated values of Poisson's ratio for EPS. Horvath (1995) indicates that within the initial linear range of the compressive stress-strain curve, the ν value can be estimated from the following empirical relationship:

$$\nu = 0.0056\rho + 0.0024 \tag{2.3}$$

where ρ is in kg/m^3.

However, the Poisson's ratio could be a negative value or close to zero as suggested by Negussey and Jahanandish (1993).

2.4.3. Reinforcing Steel

While the compressive strength of concrete is quite adequate, its tensile strength is poor. This means that plain concrete cannot be used to construct structural members in which significant tensile stresses develop. However, small amounts of steel reinforcement can be cast in the concrete in strategic locations to carry the internal tensile forces. This results in a cheap and effective composite structural material which will be ideal for the construction of most structural members. The steel reinforcement is much more expensive than the concrete but the volume of steel used is small percentage of the volume of the concrete, so that a significant cost advantage is maintained (Warner 2007).

2.4.3.1. Stress-Strain Curve of reinforcing steel

Poh (1997) has defined the relationship between steel stress (fs) and steel strain (ε_s) as,

$$f_s = \frac{(E - E_p)\varepsilon_s}{\left[1 + \left|\frac{(E - E_p\varepsilon}{\sigma_0}\right|^n\right]^{\frac{1}{n}}} + E_p\varepsilon \qquad (2.4)$$

Where, Ep= Plastic modulus, σ_0=a reference plastic stress; n= shape parameter of the stress-strain curve. Figure 2-5 shows a stress-strain curve for steel.

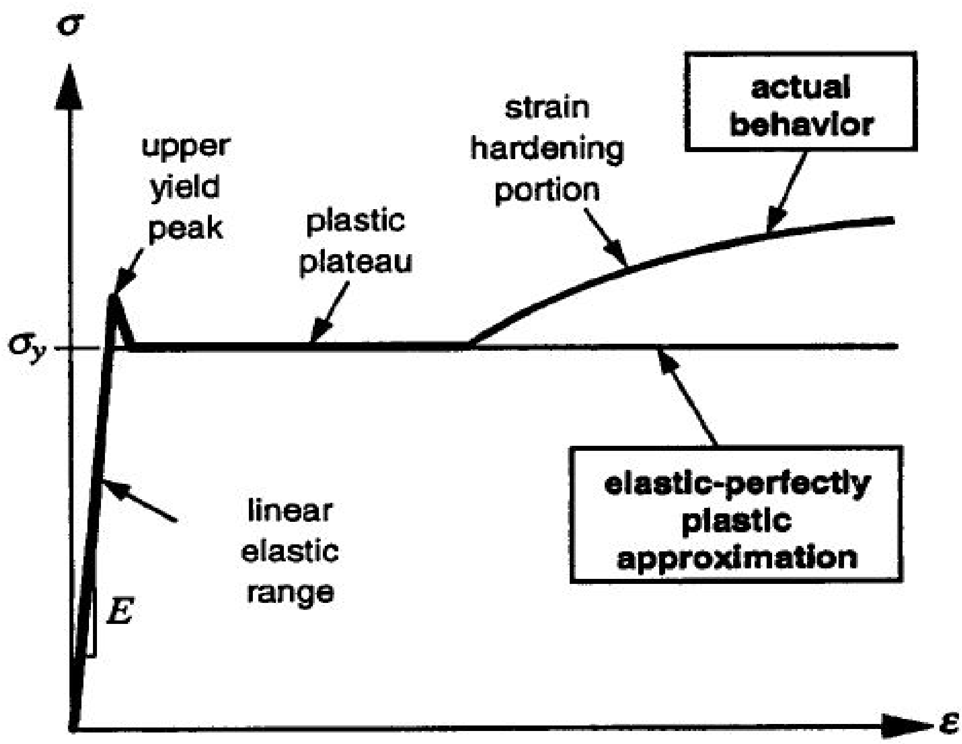

Figure 2-5 Stress-Strain curve of Reinforcing Steel (Poh 1997)

Lloyd and Rangan (1995) assumed an idealised elasto-plastic stress-strain relationship for steel as follows:

$$fs = \begin{cases} E_{st}\,\varepsilon_s & \text{if } 0 \le \varepsilon_s \le \varepsilon_y \\ f_{sy} & \text{if } \varepsilon_s > \varepsilon_y \end{cases} \tag{2.5}$$

The modulus of elasticity taken as per AS3600-2009 is 200 x 10^3 MPa for both tension and compression.

Chapter 3

3. Experimental Investigation and Simplified Analysis

3.1. Introduction

This chapter presents an overview of the experimental testing performed in University of California, Irvine (UCI) and calculation of the failure load by simplified method of analysis. The testing specimens included three separate CEPS sandwich panels. The first slab was tested without any bottom longitudinal reinforcement bars whereas the second and the third slab were tested with the addition of longitudinal reinforcement bars.

All the three slabs, each with different longitudinal reinforcement, were loaded until failure occurred. This provided some indications of the load carrying capacity of each CEPS sandwich panel, while providing some indication of the failure modes were also observed.

The purpose of the testing is to evaluate the structural performance of CEPS sandwich panels used as structural slabs. The main focus is to determine the response of CEPS sandwich panels subject to four point loading test the result of which is used for the verification of the finite element model prepared in Strand7. The result will also be used in the development of design criteria for slab construction using CEPS sandwich panels.

The information obtained will be used to create valid models of CEPS sandwich panels in order to extrapolate the information required to create design aids for use with CEPS sandwich panels used as a method of slab construction.

3.2. Test Specimens

The samples for this testing are constructed out of concrete, steel mesh and expanded polystyrene to form as CEPS sandwich panel. The skeleton of EPS foam and steel mesh is shown in Figure 3-1.

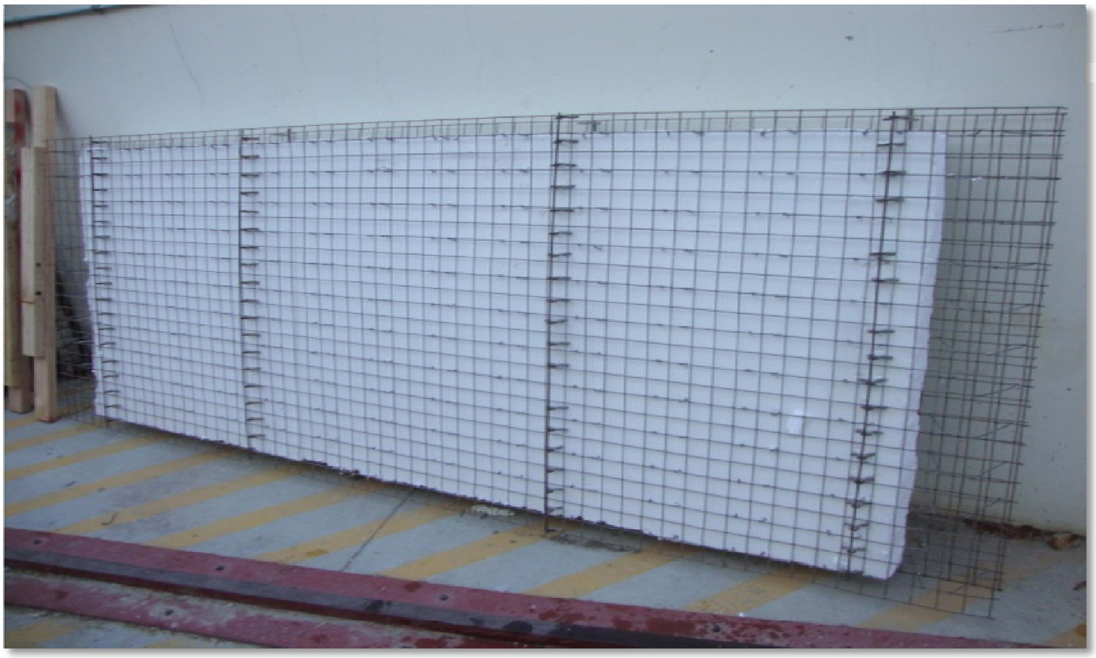

Figure 3-1 Skeleton of EPS foam and Steel mesh

The dimensions of the resulting specimen are given in Table 3-1.

Table 3-1 Dimensions of the resulting specimen

l (mm)	b (mm)	t_c (mm)	t_f (mm)	h (mm)	Steel diameter (mm)	Grid Size (mm)
3098.8	1219.2	127	44.45	215.9	3.0	50.8 by 50.8

Three test specimens were prepared. The diameter of the longitudinal bars used is shown in Table 3-2.

Table 3-2 Specimen Description

Case	Specimen Description
Case 1	First Slab without longitudinal reinforcement
Case2	Second Slab with 3 longitudinal reinforcement bars of diameter 9.53mm
Case 3	Third Slab with 3 longitudinal reinforcement bars of diameter 12.7mm

3.2.1. Slabs without Longitudinal Reinforcement Tests

Table 3-3 shows the compressive strength tests performed on the concrete poured on the panel for Case 1. The specimens were randomly taken in order to generalize the strength of the concrete. They were each 152.4mm by 304.8mm cylinders.

Table 3-3 Compressive strength test for Case 1

Specimen Size (mm)	Test Date (Days)	Force (N)	Strength (MPa)
152.4 X 304.8	14	346,143	19
152.4 X 304.8	14	346,544	19
	Average	346,344	19

3.2.2. Slabs with Longitudinal Reinforcement Tests

An additional two identical slabs to Case 1 were also tested, both reinforced by adding three extra reinforcing bars placed at the bottom to elevate the capacity of the slab due to flexure and to avoid the brittle failure that was previously witnessed. The bars were placed between the wire mesh and the polystyrene foam core and tied to the mesh as shown in Figure 3-2.

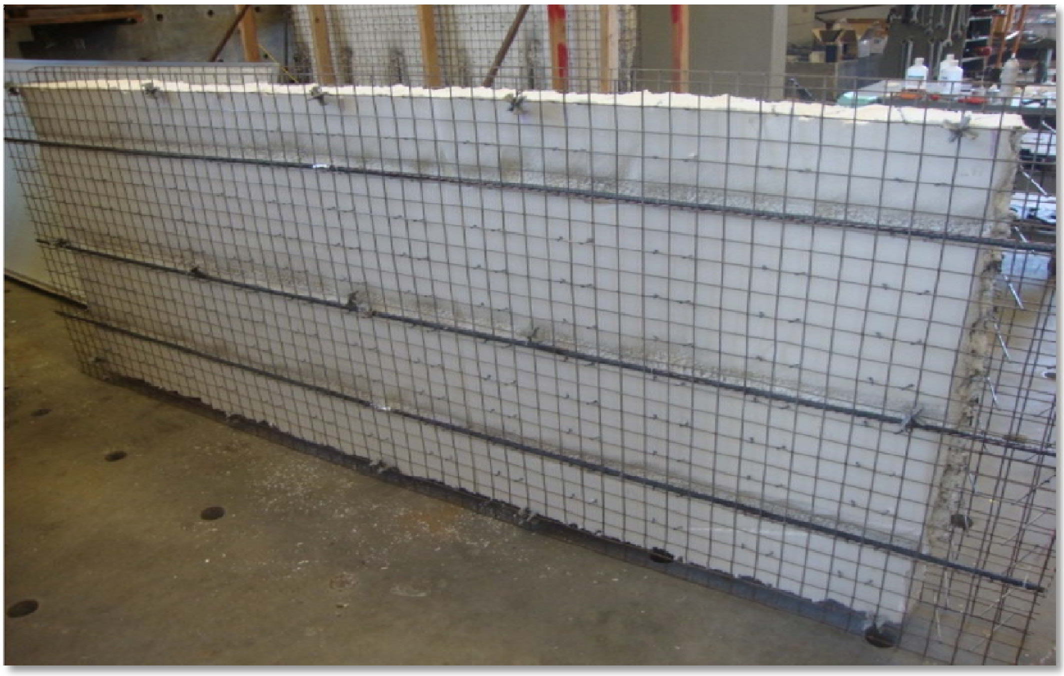

Figure 3-2 Bottom Longitudinal Reinforcement

The foam was first burnt in the destinations where the rods were to be placed to facilitate the placement of the bars and more importantly to provide an ample surface area around the bars to be completely covered in concrete. That was done to avoid any slippage that might occur when the load was applied.

For both cases, slabs were poured together and thus the cylinders taken for the compressive tests were randomly taken from the pour of both and an average obtained for both specimens as can be seen in Table 3-4.

Table 3-4 Compressive strength of concrete for Case 2 and 3

Specimen Size (mm)	Test Date (Days)	Force (N)	Strength (MPa)
152.4 X 304.8	14	212,303	12
152.4 X 304.8	14	157,904	9
152.4 X 304.8	14	205,364	11
152.4 X 304.8	14	180,945	10
152.4 X 304.8	14	187,972	10
	Average	188,898	10

It can be clearly seen that the compressive strength for the cylinders taken from Case 2 and Case 3 slabs are significantly lower than those taken for Case 1. It was noticed then when the testing for compressive strength took place, the concrete cylinders were wet when tested and that was also noticeable in the cross-section of the failed cylinders.

3.3. Test Set-up and Procedure

The slab was casted using a prefabricated steel-foam sandwich panel and a concrete mix that was created on site using a mixer and pump in order to facilitate the pouring process. The panel was placed horizontally in a mould made of wooden formwork that was manufactured at UCI as shown in Figure 3-3.

Figure 3-3 Typical Formwork that was manufactured at UCI

An increment of 25.4mm was made along the longitudinal as well as transverse direction of the formwork to facilitate the pouring and distribution of the concrete. At first the concrete was poured into the mould and then the panel was placed horizontally and pressed to ensure that the proper distribution of concrete. On second phase pouring was carried out so as to cover the rest of the panels at side and on top. With the help of a vibrator, proper distribution of concrete to fill all the voids was attained. Figures 3-4 and 3-5 show the pouring process.

Figure 3-4 Pouring Process

Figure 3-5 CEPS Panels after Pouring

The slab was tested 14 days after the initial pouring. It was tested for flexure using the 4 point loading system. The loading setup is shown in Figure 3-6.

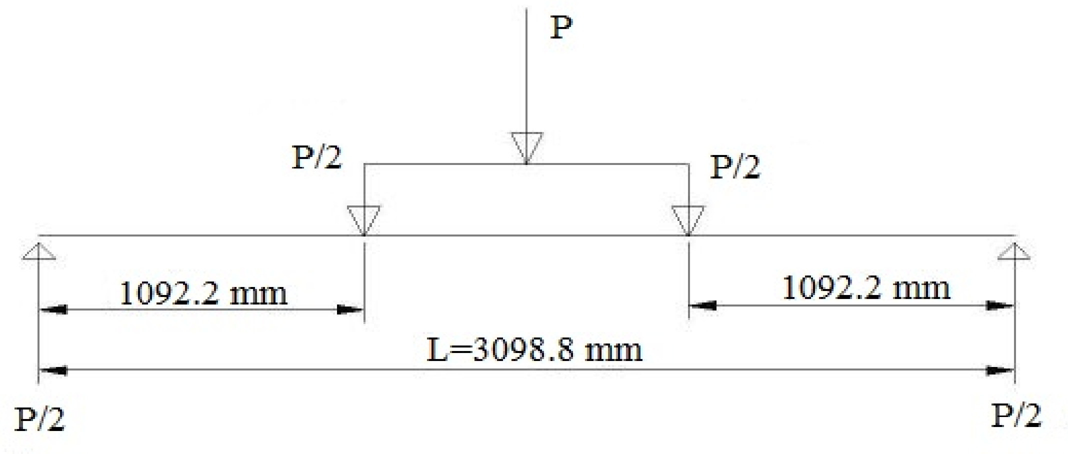

Figure 3-6 Loading Setup

The slab was placed horizontally on 2 steel beams at the ends that portrayed a hinged support at each end. A 22kN actuator placed vertically above the slab provided the load which was transferred using 2 steel cylinders connected to the actuator each 457.2 mm away from the centerline of the slab. Both of these cylinders were rested on rubber pads along the whole width of the slab to prevent the immediate crushing of the slab at the line of contact. The deflection at the mid span of the slab was measured with the help of spring pot placed beneath the centre line of the slab connected to the strong lab floor. Figure 3-7 shows the typical test setup. The slab was loaded monotonically until failure and the load and corresponding deflection were recorded.

Figure 3-7 Typical Test Setup

3.4. Test Observations and Discussions

UCI provided the load- displacement curve in the units of kips and inch. These data were converted into SI units for further analysis. The procedure and calculation are shown in Appendix C, Appendix D, Appendix E and Appendix F. These appendices mentioned the procedure for the digitization of the curve using CAD software AutoCAD. The calculation is done by Ms Excel.

3.4.1. Case 1

The test took roughly about 5 minutes till complete failure occurred. The brittle failure was observed at the maximum load of 40kN with a deflection of 17mm. The slab failed at the mid span of the slab as shown in Figure 3-8.

Figure 3-8 Failure of CEPS panels for Case 1

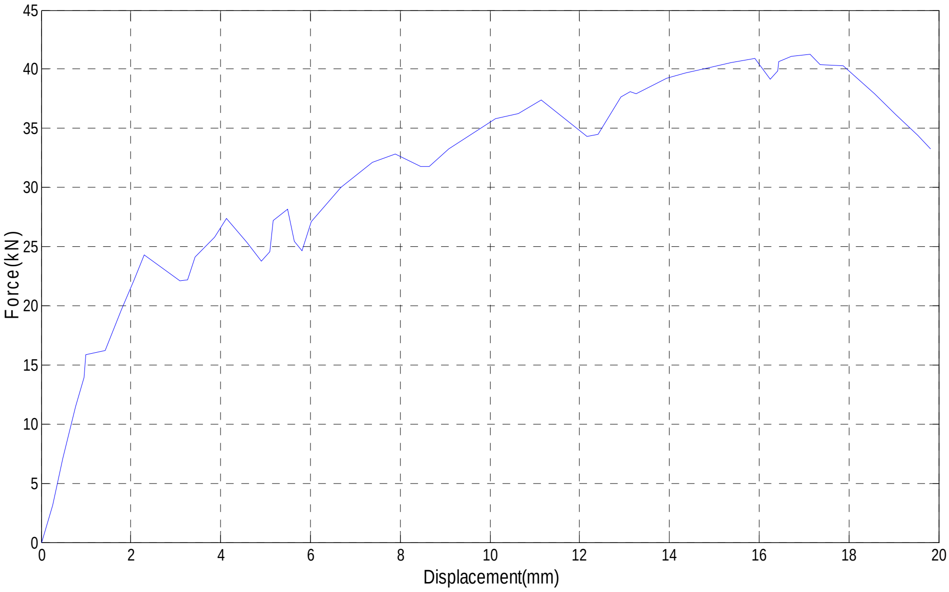

Figure 3-9 Load Vs Displacement curve for Case 1

Figure 3-9 shows a clearer look at the behaviour of the slab until failure. The failure was linear till a load of around 22 kN. It is safe to say that a design for the slab to withstand loads until 22 kN is suitable before its erratic behaviour. That is because the brittle behaviour it displayed is highly undesirable. The sudden sharp drops in load indicate that the steel mesh wires were snapping one by one. That produced a snapping sound during the test that was highly audible. The flexure test failure is shown in Figure 3-8. The failure was brittle and occurred after a few minutes of applying the monotonic load.

3.4.2. Case 2

For Case 2 which has 3 longitudinal reinforcement bars 9.53mm diameter, the slab was loaded monotonically until failure and the load and corresponding deflection were recorded. The slab withstood a maximum load of 65.38 kN with a deflection of 28 mm after which the failure occurred. The failure of the CEPS panel for Case 2 is shown in Figure 3-10.

Figure 3-10 Failure of CEPS panels for Case 2

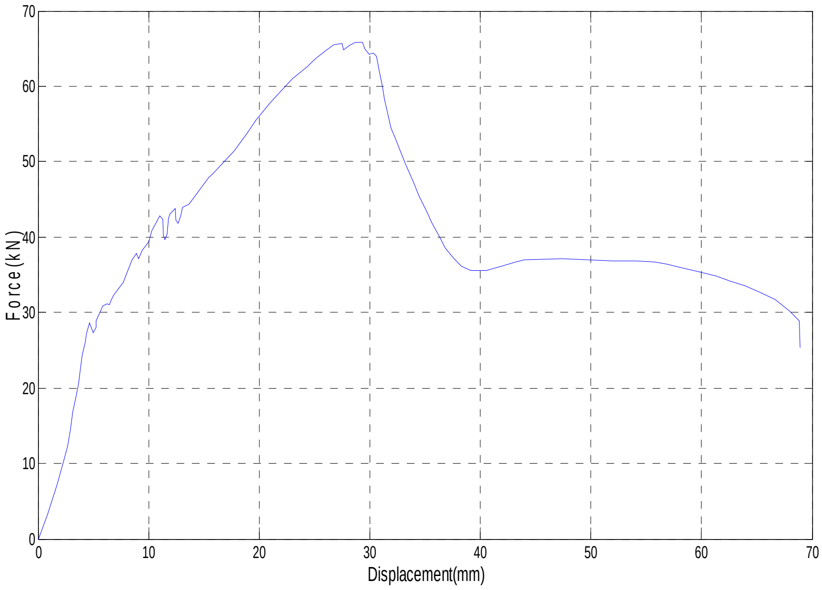

Figure 3-11 Load Vs Displacement Curve for Case 2

With the addition of 3 longitudinal bottom reinforcement, the CEPS panel shows linear behaviour till a load of around 26 kN as shown in Figure 3-11. It is safe to say that a design for the slab to withstand loads until 26kN is suitable before its erratic behaviour. The flexure test failure is shown in Figure 3-10.

3.4.3. Case 3

For Case 3 which has 3 longitudinal reinforcement bars 12.7 mm diameter, the slab was loaded monotonically until failure and the load and corresponding deflection were recorded. The slab withstood a maximum load of 89 kN with a deflection of 38 mm after which the failure was started to occur. Figure 3-12 shows the failure of the CEPS panels for Case 3.

Figure 3-12 Failure of CEPS panels for Case 3

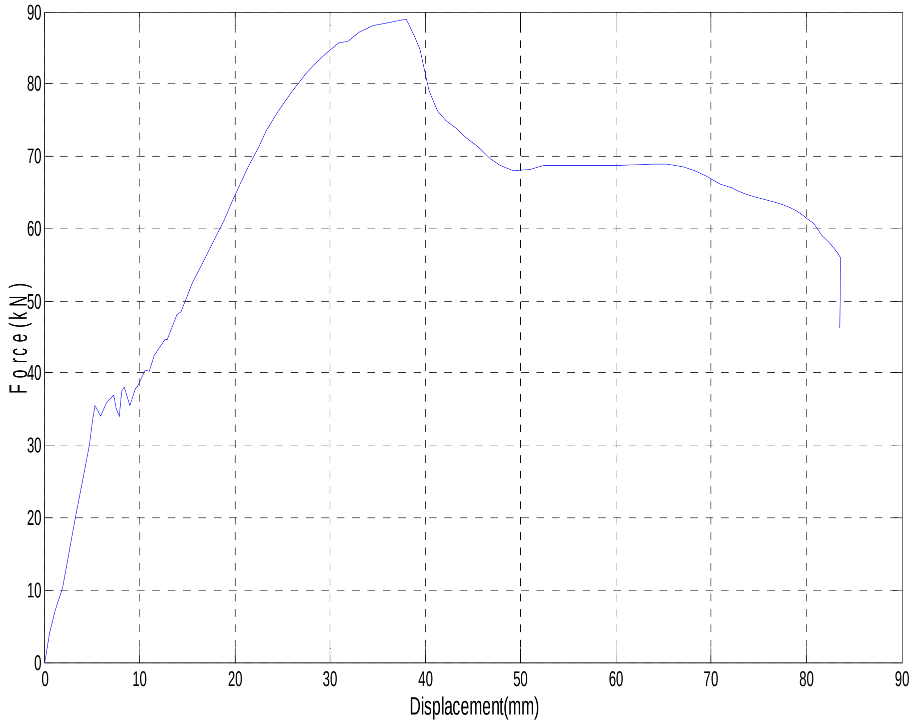

Figure 3-13 Load Vs Displacement curve for Case 3

The response of Case 3 is similar to Case 2 as shown in Figure 3-13. It shows a linear behaviour till a load of around 33kN. It is safe to say that a design for the slab to withstand loads until 33 kN is suitable before its erratic behaviour. The flexure test failure is shown in Figure 3-12. The brittle failure is checked by the addition of longitudinal reinforcement as proved by Case 2 and 3.

3.5. Method of Predicting Failure load using Simplified Analysis

This section focuses on the simplified analysis of CEPS panel to calculate the ultimate failure load. A simplified method will be used to analyse all the three cases and it will be compared with the experimental data. This section further discusses about the reason of opting finite element analysis rather than simplified analysis. This will be followed by a finite element analysis modeling undertaken using Strand7 in the next Chapter to model the performance of CEPS panels.

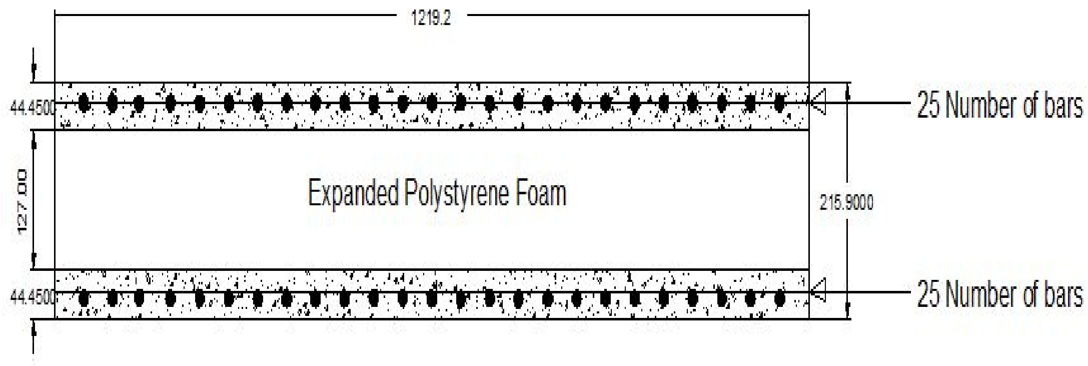

Figure 3-14 Schematic Cross section of the CEPS sandwich panels

Figure 3-14 shows the schematic cross section of the CEPS sandwich panels. Since, expanded polystyrene has a very low modulus of elasticity; hence it is assumed that it does not provide any strength in the structure. For the simplified analysis, thus the foam is neglected. Also the concrete in tension will be neglected for analysis. In that case, CEPS can be analysed as a reinforced concrete beam.

AS3600-2009 takes the ultimate concrete strain of 0.003 which is conservative but yet reasonable. Further, the maximum allowable concrete stress of 0.85 f'c is compatible with the ultimate strain. Figure 3-15 shows the free-body diagram of CEPS panels, the summation of horizontal forces shows that

$$C_c + C_s = T \tag{3.1}$$

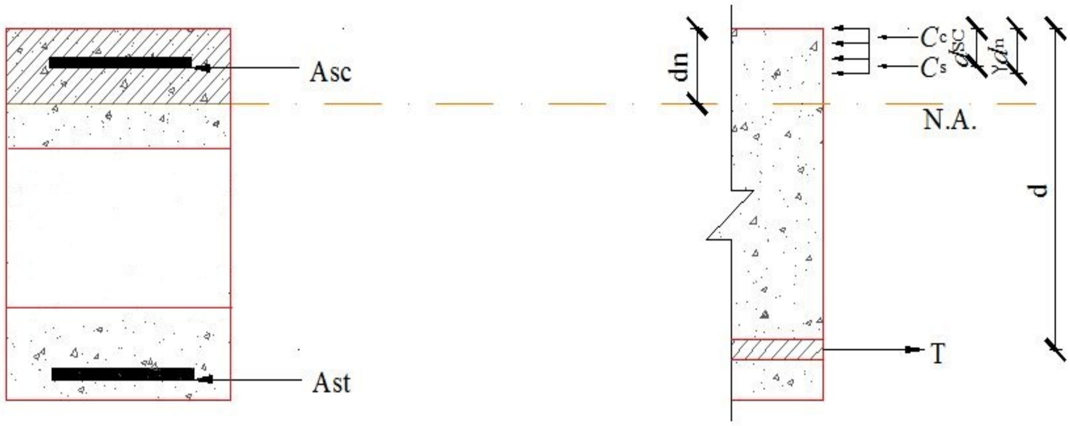

Figure 3-15 Forces on the cross-section of CEPS panels

Concrete Strength: $f'_c = 19$ MPa gives $\gamma=0.85$ and $\alpha=0.85$ as per AS3600-2009

Steel Strength: $f_{sy}= 413.68$ MPa gives $\varepsilon_{sy} = 0.00206$

$$\text{Area of each steel } (A_s) = \frac{\pi * 3^2}{4} = 7.068 \text{ mm}^2$$

$$\text{Total Area of steel } (A_{ts}) = 25 * A_s = 176.7 \text{ mm}^2$$

$$\text{Modulus of elasticity of steel } (E_s) = 2 \text{ X } 10^5 \text{ MPa}$$

We assume initially that the compressive steel is not at yield before M_u is reached but that the tensile steel is at yield. The strain in the extreme compressive fibre at M_u is $\varepsilon_{cu}=0.003$. Then:

Tensile steel force $T = 176.7 \text{ X } 413.68 \text{ X } 10^{-3} = 73.097$ kN

Compressive steel force $C_s = \varepsilon_{sc} E_s A_{sc} = 200 \text{ X } 10^3 \text{ X } 176.7\, \varepsilon_{sc} \text{ X } 10^{-3}$ kN and by similar triangles $\varepsilon_{sc} = \dfrac{0.003}{d_n} * (d_n - 19.05)$

$$C_s = \frac{106.02}{d_n} * (d_n - 19.05)\,\text{kN}$$

Concrete compressive force:

$C_c = \gamma\ d_n\ b\ \alpha\ f'c = 0.85\ d_n \text{ X } 1219.2 \text{ X } 0.85 \text{ X } 19 \text{ X } 10^{-3}$ kN

$= 16.736\ d_n$ kN

Force equilibrium ($\sum H = 0$) requires that $C_c + C_s - T = 0$ and multiplying by d_n:

$$16.736\ d_n^2 + 32.923\ d_n - 2019.681 = 0 \qquad (3.2)$$

Solving the quadratic gives $d_n = 10.04$ mm which means that the top steel lies below the neutral axis and it is also in tension.

Rewriting Equation 3.1,

Cc-Cs-T= 0

Top Steel force $C_s = \varepsilon_{st} E_s A_{sc} = 200 \text{ X } 10^3 \text{ X } 176.7\ \varepsilon_{st} \text{ X } 10^{-3}$ kN

and by similar triangles $\varepsilon_{st} = \dfrac{0.003}{d_n} * (19.05 - d_n)$

$$Cs = \frac{106.02}{d_n} * (19.05 - d_n)\,\text{kN}$$

Other terms remains the same and force equilibrium ($\sum H = 0$) requires that Cc - Cs – T = 0 and multiplying by d_n:

$$16.736\ d_n^2 + 32.923\ d_n - 2019.681 = 0$$

Numerically it is same as Equation 3.2; hence it will give us the same answer. However it will make the difference during calculation of M_u.

$k_u = 10.04/196.85 = 0.051$

By observation the strain in the tensile steel is greater than yield (ku =0.051). The strain in the top steel is :

$$\varepsilon_{st} = \frac{0.003}{10.05} * (19.05 - 10.05) = 0.00268 \qquad (> \varepsilon_{sy})$$

As the strain in the top reinforcement is greater than the yield strain the assumption is incorrect and, therefore, the calculation for the neutral axis depth is also incorrect.

Rewriting Equation 3.1,

The forces on top steel and bottom steel will be equal and both will be in tension.

Hence,

Cc= Cs + T

Or, Cc = 2 X T

Or, 16.736 d_n = 2 X 73.097

Solving,

dn = 8.735 mm

The forces are:

Cc = 16.736 X 8.73 = 146.194 kN

Cs = T = 73.097

and as a check on the calculations:

Cc = Cs + T = 73.097 + 73.097 = 146.194 kN ($\therefore$ O .K.)

With the forces calculated, Mu is obtained by

Mu = 73.097* (19.05 – 8.735/2) + 73.097 * (196.85 – 8.735/2)

= 73.097 * 14.6825 + 73.097 * 192.4825

= 15143.14 kN mm

The load that produces the ultimate bending moment Mu can be calculated from its equivalent bending moment via the bending moment diagram shown in Figure 4-3.

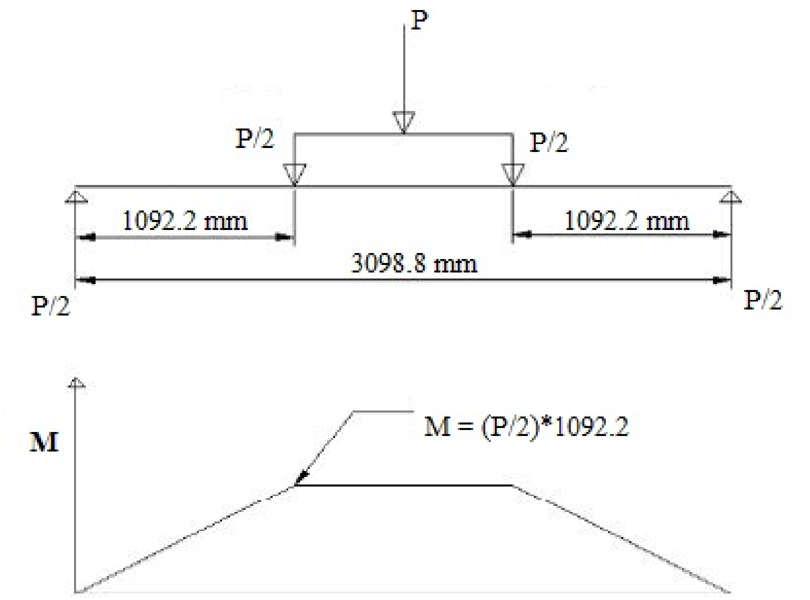

Figure 3-16 Derivation of maximum moment from loading setup

Maximum Bending moment is:

M= (P/2)*1092.2

Rewriting the equation,

Ultimate Load (P) = M_u / 546.1

= 15143.14/ 546.1

= 27.729 kN

The calculation is repeated for Case 2 and Case 3. The details of the calculation are shown in Appendix G and Appendix H.

Table 3-5 shows the comparison of the ultimate load between experimental and the simplified method.

Table 3-5 Comparison of ultimate load between experimental and simplified method

Case Number	Case Name	Ultimate load (kN)	
		Experimental Setup	Simplified Method
1.	Without Reinforcement bars	40	27.729
2.	With Reo bars of 9.53 mm dia	65	55.545
3.	With Reo bars of 12.7 mm dia	88	77.668

Table 3-5 indicates that a simplified analysis is much more conservative. In all the three cases, the value of ultimate load from simplified method lags the experimental failure load by approximately 10kN.

One of the reasons is that the tensile force of the concrete is completely ignored. Also, in the CEPS panels, the stress transfer mechanism between top concrete and bottom concrete is by the use of vertical steel connectors. This was not used in the simplified analysis.

Also, the mathematical equations characterizing CEPS sandwich panels are considerably more difficult than their counterparts where only single layer is involved. They contain a larger number of differential equations, with more variables and higher order, and often are coupled. Hence, finite element method is used to find the load deformation curve of the CEPS sandwich panel.

The finite element method is a numerical method for solving engineering problems involving complicated geometries, loading and material properties, it is generally not possible to obtain analytical mathematical solutions. Hence, we need to rely on numerical methods, such as finite element method, for acceptable solutions (Logan 1986).

As finite element method can handle such complex geometry and non linear behaviour of the material. Hence, a finite element model will be used to understand the failure mechanism of the CEPS panels.

Chapter 4

4. Development of Finite Element model for the slab analysis

4.1. Introduction

This chapter presents a structural evaluation of Concrete-Expanded Polystyrene (CEPS) sandwich panels for slab applications using finite element analysis. It is based on the experimental work previously conducted at University of California, Irvine (UCI). This section focuses on investigating the suitability of CEPS panels for slab applications with the objective of developing a finite element model to predict the load displacement curve.

In this study, a detailed experimental study on three CEPS panels with different longitudinal reinforcement was conducted. The results showed that CEPS shows a linear load displacement curve at the initial stage and can be used as flooring materials. To improve the understanding of CEPS sandwich panels behaviour further, finite element analyses (FEA) of sandwich panels were undertaken using a finite element program Strand7. Concrete, expanded polystyrene foam and reinforcing steel are represented by separate material models which are combined together with a model of the interaction between concrete, foam and steel to describe the behaviour of the sandwich material. However, the result obtained from finite element model does not guarantee its correctness. Validation is the primary method for evaluating the confidence of computer simulations. The objective of validation is to identify, remove or reduce, and quantify the errors. Validation compares the numerical solution with the experimental data. In this research, the finite element model is validated from the experimental data of UCI. The goal is to detect a model's significant discrepancies and to reduce and estimate removable and unavoidable errors in the finite element model.

Thus a finite element model was developed and validated in order to represent the real behaviour of CEPS sandwich panels. Both FEA and experimental results were then used to create a design chart for CEPS panels as flooring materials.

4.2. Types of Model

In order to develop an accurate and reliable finite element model that simulates true behaviour of CEPS sandwich panels, various types of numerical models such as full-length model or quarter-length model can be used and analysed in a finite element investigation. The full-scale model may be the easiest way to develop and used in the analysis as it uses the actual dimensions of the structure and does not require any study to scale down the sizes of the model. However, the disadvantage associated with this model is the poor level of accuracy obtained due to the smaller number of elements that can be included in the analysis. Also, it is very uneconomical as it needs large computational time.

To eliminate such difficulties, a reduced model with appropriately determined member dimensions can be used for the analysis. One such reduced model is the half-length model. In this model, only half the length (l/2) of the panel is used to create and analyse the model using appropriate boundary conditions. Also, by using the width (b/2), the half-length model can be reduced to the quarter size of the full panel. As the full panel is reduced to quarter size, a large number of elements with smaller sizes can be used in finite element meshing that will ultimately increase the level of accuracy of numerical results. Figure 4-1 shows the actual dimensions used in the quarter slab model.

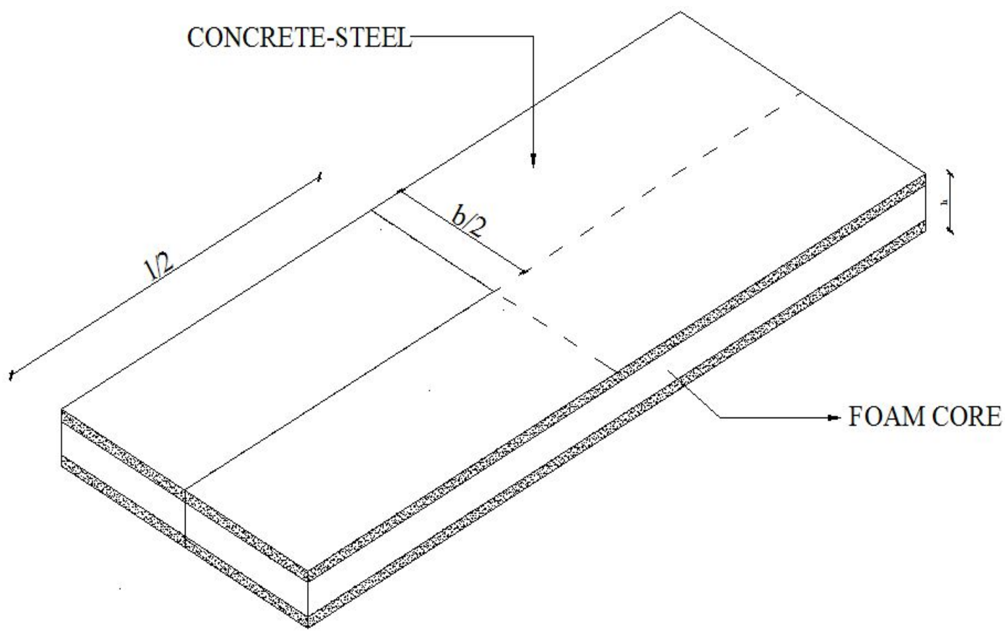

Figure 4-1 Dimensions used in the quarter slab model

In this investigation, a quarter slab model was used to compare with the experimental results. A full-length model was also used to validate the use of the quarter slab model. A detailed description of these two types of finite element models is presented in the Section 4.7 of this chapter. All the finite element models used in this investigation were based on the four point loading test performed in UCI with two parallel edges of the slab being simply supported.

4.3. Type of Elements

Concrete and EPS is modeled as an isotropic material, which means that the material elasticity property is independent of the material orientation. In other words, the material's behaviour does not change when the material is rotated or loaded in different directions. Material mechanical behaviour of an isotropic material is characterized by the following parameters.

E = Young's modulus

G = shear modulus

γ = Poisson's ratio

As Young's modulus, shear modulus and the Poisson's ratio are related by

$$G = \frac{E}{2(1+\nu)} \tag{4.1}$$

Only two of the three elasticity parameters are independent.

Since steel reinforcement is used in concrete construction in the form of reinforcing bars or wire, it is not necessary to introduce the complexities of three-dimensional constitutive relations for steel. Axial force in the steel member will more than adequately represent the contribution to the physical deformation behaviour of the overall member. Bending contribution for the overall member will automatically come through axial force of steel bar times the relevant arm from the neutral axis of overall member. So, there is no need to consider bending effects in the local coordinate system of each bar. For computational convenience it even often suffices to idealize the one dimensional stress-strain relation for steel.

In Strand7, there is a special type of truss element called a cutoff bar element that has predefined tensile and compressive strength limits. For a material linear analysis or when the element axial force is within the predefined limits, the element behaves as truss element. In a material nonlinear analysis, when the axial force exceeds the limits, its behaviour changes, and the axial stiffness is governed by the additional property parameters.

Strand 7 has three types of cutoff bar element namely:

1. Tension only
2. Compression only
3. Tension-compression

In this analysis, the steel can resists both tension and compression, but only up to the predefined cutoff values. Hence, tension- compression cutoff bar were chosen. Table 4-1 shows the cutoff force calculation for steel mesh and two longitudinal reinforcement bars used in the experimental data.

Table 4-1 Cutoff Force Calculation for Steel reinforcement

S.No	Diameter of bar (mm)	Area of Bar (mm^2)	Yield Strength (MPa)	Cutoff Force (kN)
1.	3	7.068	413.68	2.93
2.	9.53	71.33	413.68	29.57
3.	12.7	126.676	413.68	52.403

In addition of the cutoff's values, a cutoff bar element supports two different behaviour.

1. Brittle
2. Ductile

In case of steel, if the load in the bar exceeds the strength limits, the bar becomes perfectly plastic and yields. It will contribute no further stiffness but its force will remain at the same level until unloading occurs which is shown in Figure 4-2. If unloading does occur after the bar yields, the force remains at the limit value till the effective strain in the element reduces to below the yield strain. From this point, if the load is further reduced, the force in the element also reduces. To replicate this behaviour of steel, ductile behaviour of material was selected.

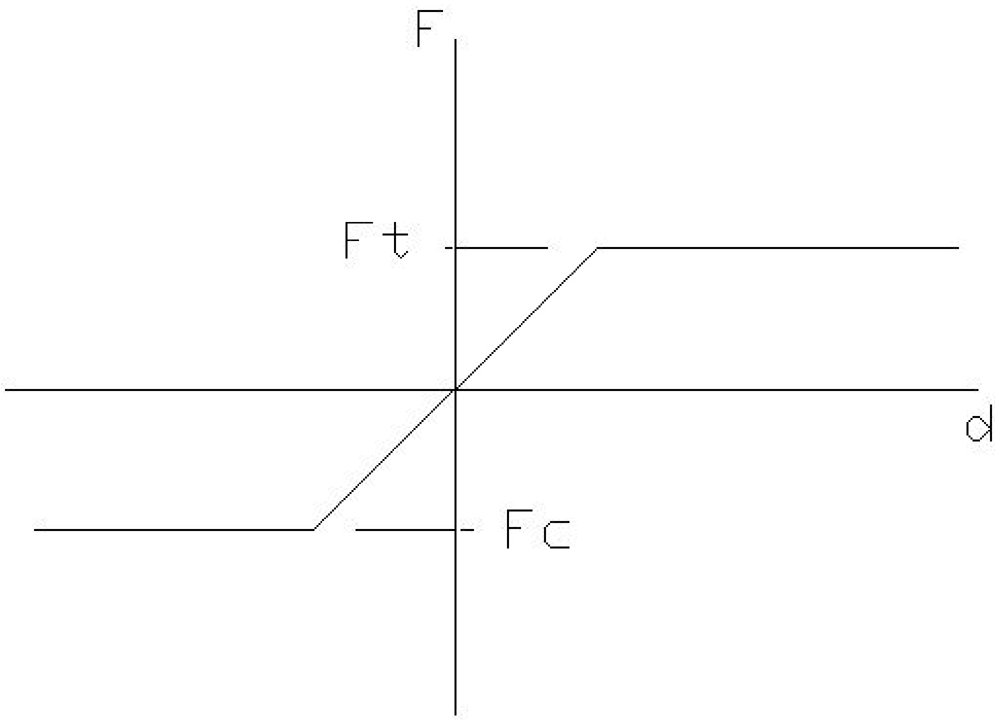

Figure 4-2 Force-displacement behaviour of a ductile cutoff bar element

4.4. Material Properties

To simulate the actual structural behaviour of CEPS sandwich panels, it is necessary to give attention to several considerations. Since the behaviour of concrete is different in compression and tension. So the concrete element must be capable of modeling structural behaviour both in compression and tension. Also concrete is weak in tension so a stress strain curve was developed as shown in Figure 4-3 to 4-4 to replicate the actual behaviour of the model.

In Strand7, a nonlinear material model is defined by a nonlinear stress-strain table in terms of an effective stress and the corresponding effective strain. Three types of effective stresses can be used : Tresca, von Mises and Max Stress. In this analysis, the Max Stress Criterion is introduced to model concrete as it can exhibits different behaviour under tension and compression. For this reason, the stress-strain relationship curve for nonlinear materials with the Max Stress criterion may cover both positive and negative ranges.

In compression, the behaviour of the concrete is taken as per Table 2-1 and, in tension, a linear elastic behaviour is assumed up to the strength of concrete in tension. The progressive loss of rigidity after cracking is quantified indirectly through an adaptation of the tension behaviour introducing a downward branch. This stress strain curve is based on the characteristic strength of the concrete. The characteristic strength of the concrete used in the experiment was 19MPa for Case 1 and 10MPa for Case 2 and Case 3. Figure 4-3 and 4-4 shows the stress-strain curve of concrete used in this analysis.

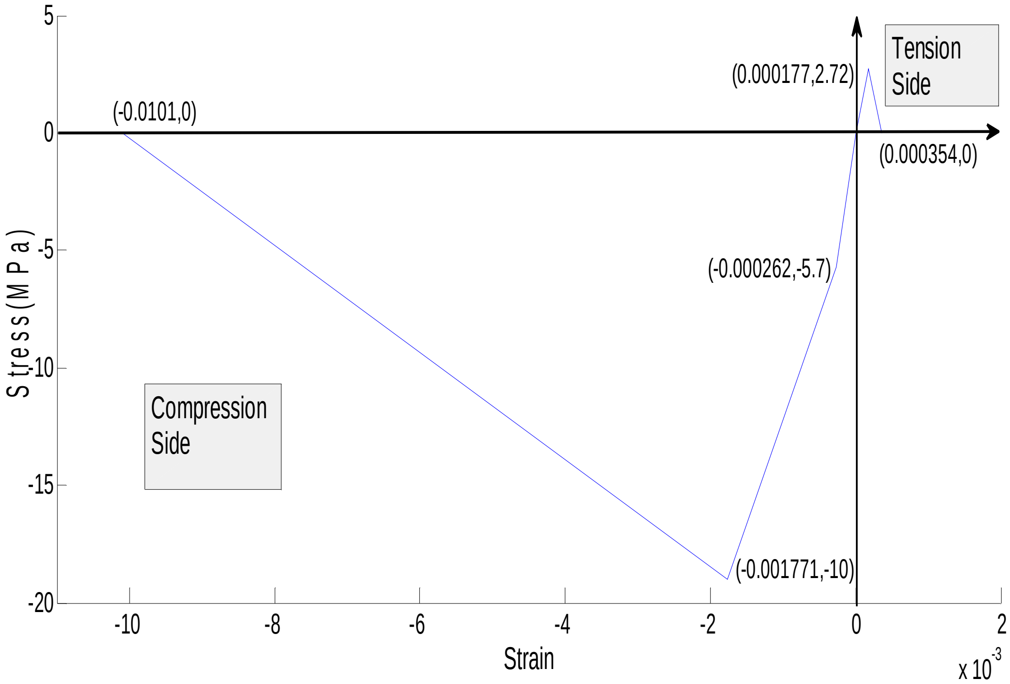

Figure 4-3 Stress-Strain of concrete for characteristic strength of 19MPa

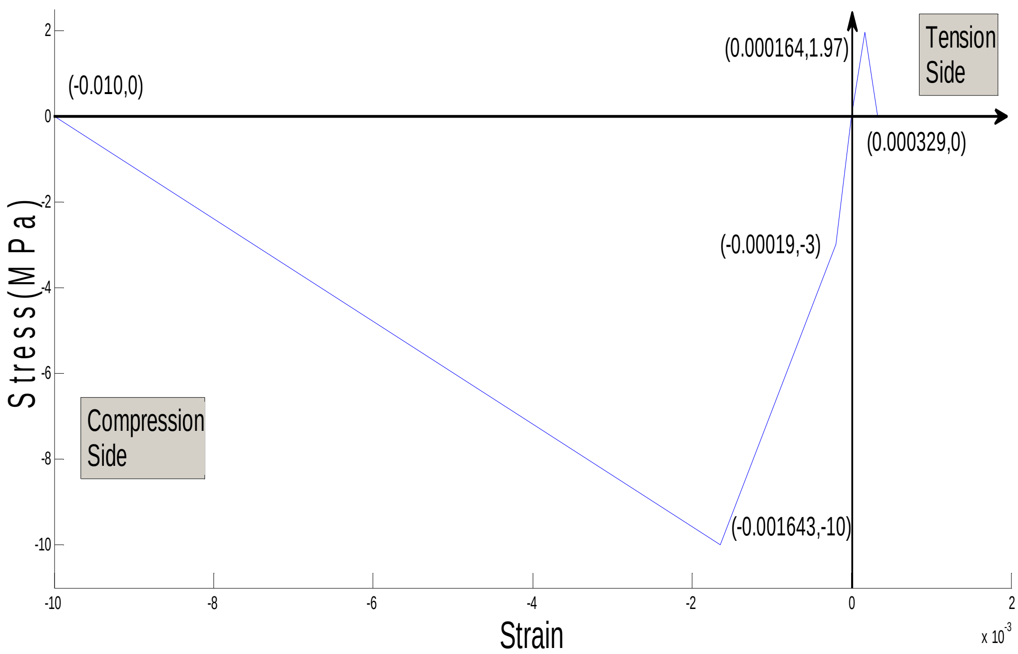

Figure 4-4 Stress-Strain curve of concrete for characteristic strength of 10MPa

Due to the unavailability of the accurate data for the density of EPS used in the experiment, an average value of 19kg/m^3 was chosen for the analysis. Equation 2.1 and 2.3 was used to calculate the modulus of elasticity and Poisson's ratio of EPS.

Since, EPS core has a very low modulus of elasticity, it was noted that the stress strain curve for EPS core does not make any difference on the model results as the value is very low compared to concrete and steel. Hence, EPS is considered as a linear material to reduce the complexity of the model.

The material properties for each of the structural elements was based on the previous research conducted. These were then applied to each of the respective materials comprising the finite element models. Each of the material properties are shown below in Table 4-2.

Table 4-2 Material Properties

Material	Density (kg/m^3)	Modullus of Elasticity (MPa)	Poisson's Ratio
Concrete	2400	$(\rho)^{1.5} \times (0.043\sqrt{f_{cm}})$	0.2
Steel	7850	200 x 10^3	0.25
Foam	19	5.55	0.1088

4.5. Loads

In the experiment, the loads were transferred using 2 steel cylinders connected to the actuator. These loads were modeled as concentrated nodal forces in the finite element analysis. Load was applied at the 457.2 mm from the centre point by distributing equally into the individual nodes. The magnitude of the load depends upon the type of the model selected i.e. full model or quarter symmetric model. In case of full model, the total load was divided by the number of the node within the width of the slab for each steel cylinder. The resulting load was then applied to each node.

For the quarter slab model, only the quarter of the slab is used for modeling hence the total number of node to transfer load will be based on the quarter of the slab. It should be noted that the applied load should also be reduced by one fourth in this case. Table 4-3shows the calculation for number of node used in each model.

Table 4-3 Calculation of number of node for experimental study

Full Model: Width of the slab = 1219.2 mm Width of each element = 25.4 mm Number of node to transfer load for each steel cylinder = 1219.2/25.4 + 1 = 49 Total number of node = 2* 49 = 98
Quarter Model: Width of the slab used for quarter model = 1219.2/2 = 609.6 mm Width of each element = 25.4 mm Total number of node to transfer load = 609.6/25.4 + 1 = 25

The various load magnitudes were applied to the model by creating the following two cases within Strand7 and then combining them and multiplying by the appropriate factor to achieve the required total load:

- Self weight (gravity)
- 10kN

These loads were applied to the slab model by having Strand7 calculate the self weight load created by a gravitational force of -9.81 m/sec^2 as one separate load case. Table 4-4 shows the calculation for nodal force used in both model.

Table 4-4 Calculation of nodal force for experimental study

Full Model: For 10 kN load, Load/ node (kN) = 10/ 98 =0.102
Quarter Model: For 10 kN load, Load/ node (kN) = 10/ (4*25) =0.1

The load is then applied to the CEPS sandwich panels to record the deflection. Figure 4-5 shows the non linear load increments inserted in Strand7.

CASES	Include	1	2	3	4	5	6
		Increment	Increment	Increment	Increment	Increment	Increment
1: Self weight	√	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}
2: Total load of 10kN	√	1.000000×10^{0}	2.000000×10^{0}	3.000000×10^{0}	4.000000×10^{0}	5.000000×10^{0}	6.000000×10^{0}
1: Freedom Case 1	√	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}	1.000000×10^{0}

Figure 4-5 Non linear load increment input in Strand7

4.6. Analysis methods

As the response of a CEPS under consideration is not a linear function of the applied load, the methods of analysis used for the investigation of behaviour of the CEPS sandwich panels are non-linear static analysis. Non linear behaviour of the structure can be due to geometric nonlinearity, material nonlinearity, boundary nonlinearity or a combination of the three. In this research, the nonlinear behaviour of the structure is due to material nonlinearity of concrete and steel. Since, the modulus of elasticity of EPS is very small compared to concrete and steel, EPS is considered as a linear material.

For a nonlinear analysis, the principle of superposition is not applicable. The results from different load situations cannot be scaled, factored or combined as is usually done in linear analysis. In Strand7, the nonlinear static solver uses an algorithm based on the modified Newton-Raphson method to solve the nonlinear equation system. The system uses an iteration procedure within each load increment to ensure that the equilibrium of forces is maintained, within a specified allowance, at the end of each load increment.

4.7. Model geometry, Mesh sizes and Boundary conditions

All the CEPS sandwich panel tested in the experiments were modeled and analysed using quarter slab model. The width of each model was b/2 (half the panel width), length l/2 (half the length of the specimen), and thickness is the sum of foam and concrete thickness ($t_c + 2*t_f$).

The model geometry created to simulate experimental CEPS panels was discretised into a number of finite elements. As the mesh density increases, the accuracy of a finite element model generally increases and converges to a numerically correct solution. Therefore it is necessary to have a fine mesh to obtain the appropriate solution. The accuracy of the model can then be compared with the experimental results. The aspect ratio of the finite element model is very important for the accuracy of a finite element model. Based on the aspect ratio which should be in the ratio of 1:1 for accurate results, solid elements with 25.4 x 25.4x25.4 mm throughout the foam depth were used. For the concrete, the mesh size was chosen as 25.4x25.4x19.05 mm for the concrete above the steel reinforcement and 25.4x25.4x25.4mm for concrete below the steel reinforcement. For the steel reinforcement, each element size was chosen as 25.4mm. Two nodes of the steel elements were connected to the respective nodes of the brick elements of the concrete. This mesh size provided satisfactory results in terms of accuracy.

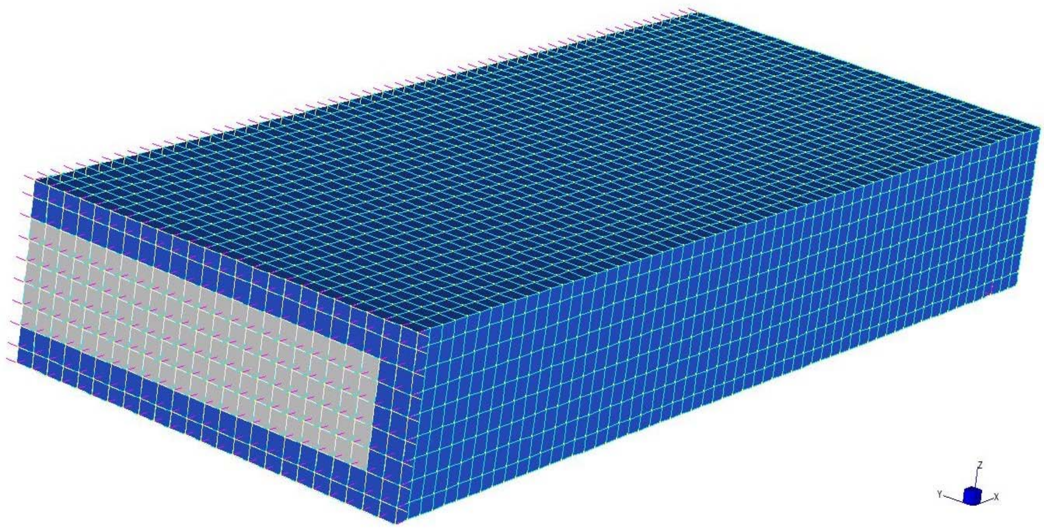

Figure 4-6 Quarter model of CEPS panel

Figure 4-6 shows the model geometry, mesh size and loading pattern along with the appropriate boundary conditions for the quarter model.

The accuracy of the results obtained from the finite element modeling largely depends on the appropriate selection of the boundary conditions. The choice of the boundary conditions depends on the type of model selected i.e. full model or quarter slab model. The size of the finite element model can be reduced significantly by using symmetry in the structure being analysed.

Appropriate boundary conditions were applied at one of the edges of the panels to simulate the experiments whereas symmetric boundary conditions were applied to the entire surface (i.e. concrete and foam core) along both the longitudinal direction and across the width to model half width and half length, respectively. As shown in Fig 4-6, the symmetric boundary condition was applied along the length and width at the centre of the panel.

For YZ symmetry plane, the boundary condition was applied such that it allows the translation in the Y and Z directions and rotation about the X axis. However, it does not allow translation in the X direction and rotation about either Y or Z axes. Similarly, for ZX symmetry plane, the boundary condition was applied such that it allows the translation in the X and Z directions and rotation about the Y axis. However, it does not allow translation in the Y direction and rotation about either X or Z axes. For the node that lies between both YZ and ZX symmetry plane, the boundary condition was applied such that it allows translation only in Z directions. Thus, the symmetric boundary condition was applied to the entire surface (concrete and foam) along the length of the panel at the centre of the panel width (b/2). Similarly, symmetric boundary condition was applied along the width at the centre of the panel length (l/2).

To validate the results obtained from the quarter model, Case 1with the application of load of 20kN was modeled and analysed using the full-length model. Figure 4-7 shows the model geometry, mesh size and the loading pattern along with appropriate boundary conditions for the full-length model. The width of full model was b, length l, and the thickness being the sum of the foam and concrete thickness. The same mesh size used in the quarter slab model was used for this full length model.

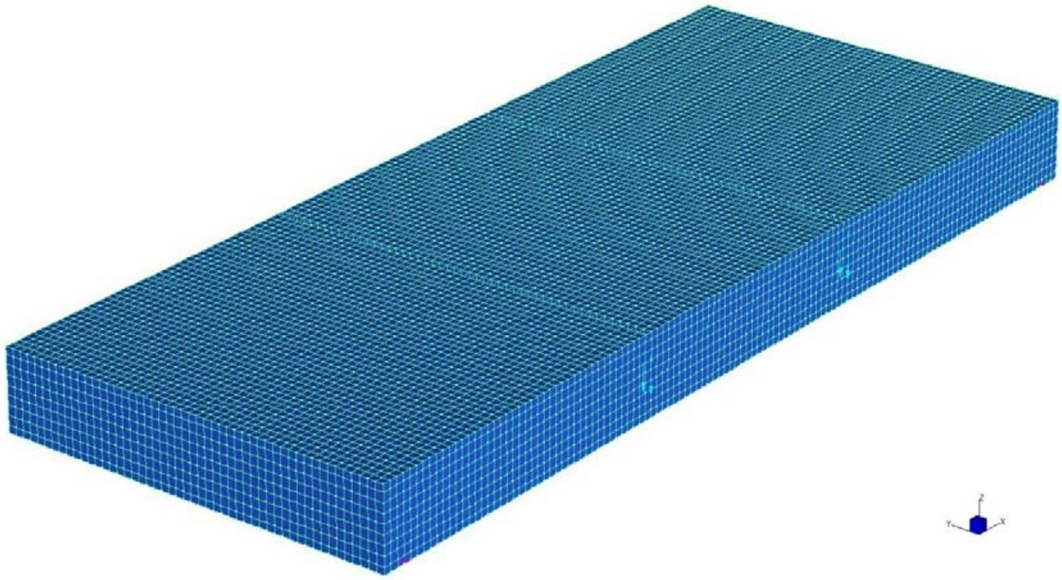

Figure 4-7 Full model of CEPS panel

In the full-length model, appropriate boundary conditions were applied at both sides of the bottom panel. For one edge of the slab panel, the boundary condition was applied such that it does not allow the translation in any directions and allows rotation about the Y axis. Similarly, for the other edge of the slab panel, the boundary condition was applied such that it allows the translation in the X direction and rotation about the Y axis. This replicates the simply supported slab that was used in the experiment.

The displacement results obtained from the full-length model and quarter slab model is compared to confirm that one model can represent the other. For comparing the result of both models, same load of 20kN was applied to Case 1. The result obtained from the full-length and the quarter slab model was very close with a value of 1.98mm and 2.01mm respectively. This comparison confirms that the full-length model can be well represented by the symmetric quarter model. Therefore further analyses in this study were conducted using the quarter model with only half length and half width to save on computational time.

4.8. Standard Model Procedure

The finite element model was broken up into three main elements. The elements available in Strand7 include beam, plates and bricks. A special type of truss element called cutoff bars with ductile failure will be used to model the steel elements. These elements are one-dimensional line elements having two nodes as shown in Figure 4-8.

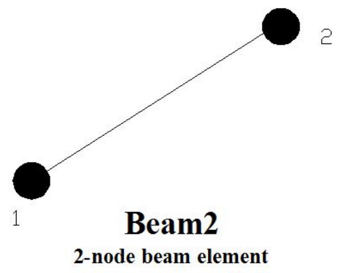

Figure 4-8 Two node beam element

Plate elements are two-dimensional surface elements, and include four and eight node quadrilateral elements. The four node quadrilateral element is shown in Figure 4-9. The plate element will be extruded to form a brick element.

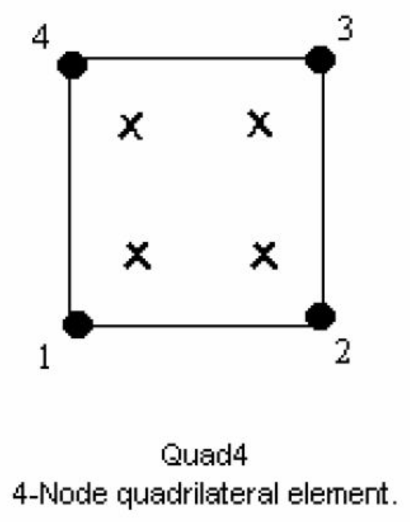

Figure 4-9 Four node quadrilateral element

The brick elements in Strand7 are three-dimensional elements that can be used to represent a wide range of brick types and shapes to enable the meshing of all possible geometries. For these models, mostly eight-node hexahedral elements will be used, as shown in Figure 4-10.

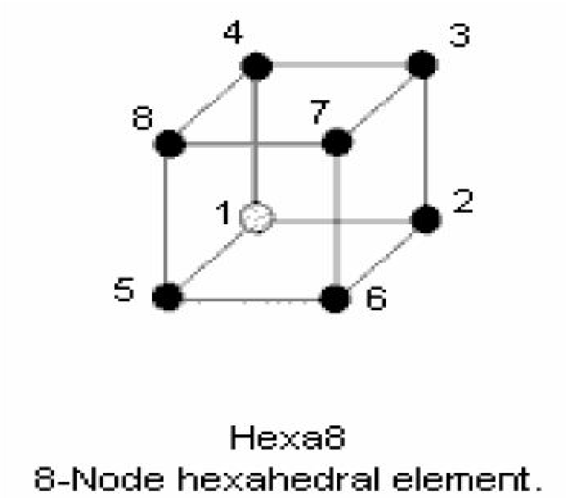

Figure 4-10 Eight node hexahedral element

Table 4-5 details which structural elements of the model were beam elements and which were modeled as brick elements.

Table 4-5 Beam and brick elements

Beam Elements	Brick element
• Steel reinforcement	• Concrete • Expanded Polystyrene Foam

The procedure outlined below was used to create each of the models in Strand 7.

Specify Units

1. Select Global / Units
2. Select ‘mm’ for Length
3. Select ‘MPa’ for Modulus/Stress
4. Select ‘N’ for Force

Strand 7 ‘Extrude’ function allows elements to be extruded into other types. For example, extruding a plate element will result in a brick element being produced.

Create Plate elements to be extruded as Brick elements

1. Select Create/Element
2. Type = Quad 4, create elements by joining appropriate nodes

Extrude elements

1. Select Tools / Extrude / By Increment
2. Select all elements using ‘Select by region’

3. Type appropriate distance in Z direction (ie. 19.05 mm for the top concrete above reinforcement)

The elements have now been extruded the required length in the z-axis direction. Therefore, the plate elements have become brick elements to represent concrete and foam. Table 4-2 previously defined the material properties.

This procedure is repeated to create different layers of the CEPS panels namely:

1. Top concrete above reinforcement
2. Top concrete below reinforcement
3. Side Concrete in the middle layer of Foam
4. EPS foam
5. Bottom concrete above reinforcement
6. Bottom concrete below reinforcement

While making the layers, each layer should be grouped together so that it would be easier to define the material properties and to create beam elements for reinforcement.

Grouping Different Layers

1. Select the required elements by using 'Select by region'.
2. Select Global/Groups
3. Create New group
4. Right click the new group and click 'Assign'
5. Rename the group as required

At this point in the model preparation, the mesh was too coarse to produce accurate results. Therefore the 'Subdivide' function in Strand7 was used to divide the bricks into smaller elements, resulting in a finer mesh and more accurate results. The number of divisions was entered based on the parameters A, B and C.

Subdivide elements

1. Select Tools / Subdivide
2. Select the element for subdividing
3. The A, B and C directions will be indicated on the element
4. Determine how many divisions are required, and enter these values in the appropriate A, B or C boxes.
5. Select Apply

This completed the mesh for the brick element of the models. For creating beam elements for steel reinforcement for top layer, as each node of the steel should be connected to the node of the brick element so all the layer except 'Top concrete below reinforcement' was turn on.

Creating Beam elements

1. Select Global Groups
2. Turn off all the layer except the layer "Top concrete below the reinforcement"
3. Select Create/Element
4. Type = Beam 2, create elements by joining appropriate nodes

This process is repeated for the bottom reinforcement as well.

This completed the mesh of the models. The mesh was then cleaned to ensure the nodes and elements were sorted and renumbered, removing any unused nodes or null elements. The models for different materials are shown below in Figure 4-11 to 4-13.

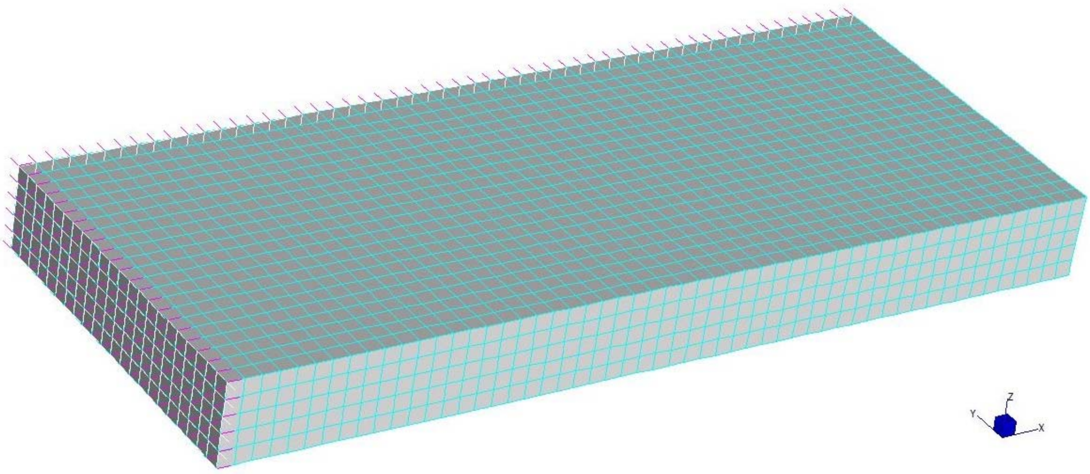

Figure 4-11 Finite element model of Expanded Polystyrene Foam

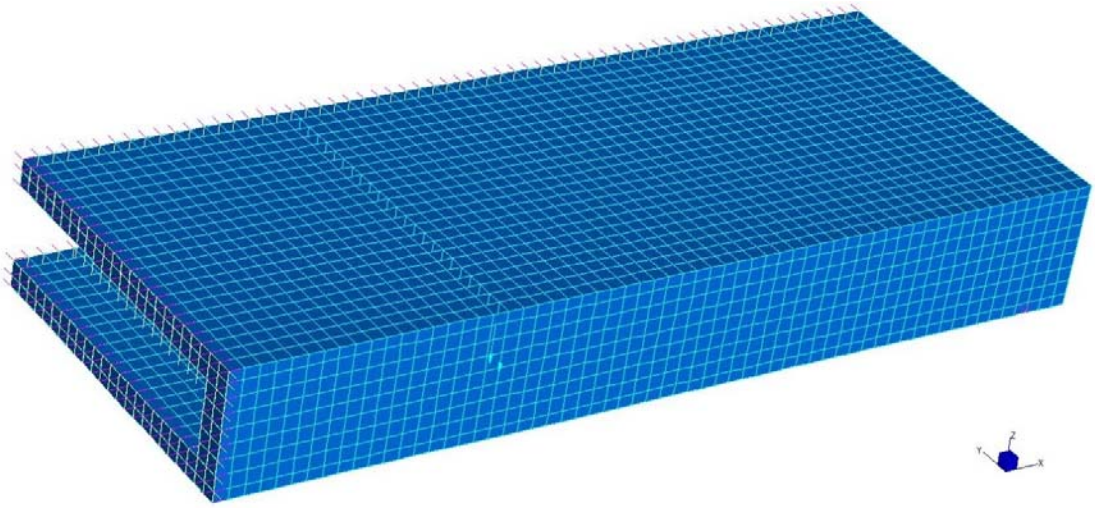

Figure 4-12 Finite element model of Concrete

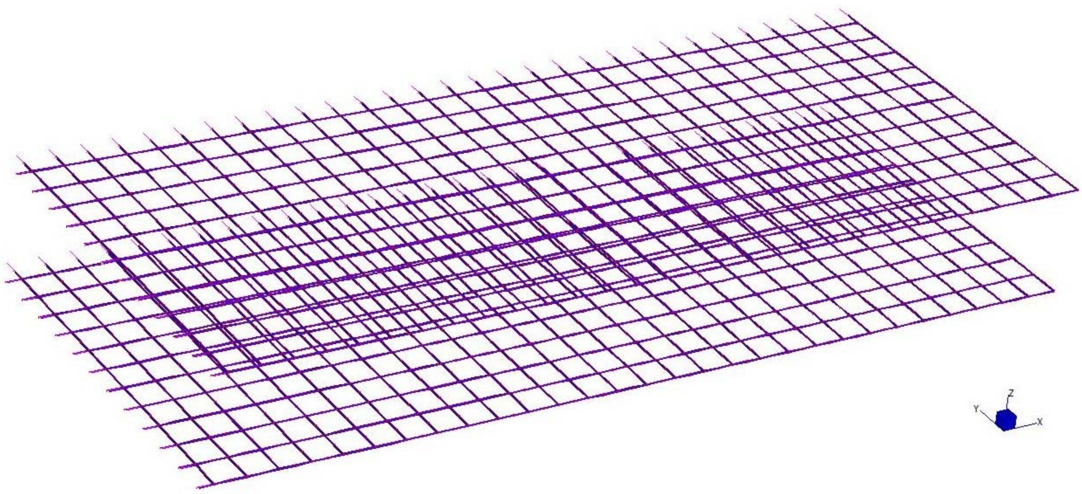

Figure 4-13 Finite element model of Steel Mesh

The next step in the process of setting up the models for analysis was to assign restraint conditions. Node restraints were used as explained in the Section 4.7 of this chapter.

Assign restraint conditions

1. Select all the required node
2. Select Attributes/Node/Restraint
3. Check the appropriate boxes next to the six degrees of freedom
4. Repeat the procedure for other nodes as well.

Links are used to define certain relationships between the displacement components of the nodes they connect. Physically a link restrains movements between the linked nodes, while mathematically; it represents a set of constraint equations on the related displacement components. In the CEPS panels, the vertical connectors were modeled as a master-slave link. Conceptually, a master-slave link will force the slave node to follow the master node in the selected displacement directions. Although the link is referred to as a master-slave link, there is no real distinction as to which node is the master and which node is the slave.

Creating Master-slave link

1. Select Create/Link
2. Use the drop down box to select 'Master-Slave link'
3. Select the required node of the top and bottom reinforcement.

The force was then applied on two sides of the slab each 457.2 mm from the centerline of the slab. The force in this case was a quarter of the full force, as the model is a quarter of the full scale model.

Apply force

1. Select Attributes/Node/ Force
2. Select the appropriate node
3. Type the appropriate force in Newton in the 'Z' box.

After the boundary and loading conditions were applied, the model was ready for analysis. The model was analysed using Non linear static analysis.

Analysis

1. Select Solver/Nonlinear Static
2. Select only Nonlinear Material
3. Select Load Increments
4. Type the load increment values as per required
5. Click Solve.

4.9. Finite Element Results

This section discusses the results from the three finite element models built in Strand7. These three models consisted of the CEPS panels without reinforcement bars and with reinforcement bars of diameters 9.53mm and 12.7 mm.

The typical deflection of the slab is shown in Figure 4-14. This shows the specimen deflected as the force is applied to the plate. As discussed, the panel was modeled to a quarter scale and the force applied was a quarter of the total force applied during testing.

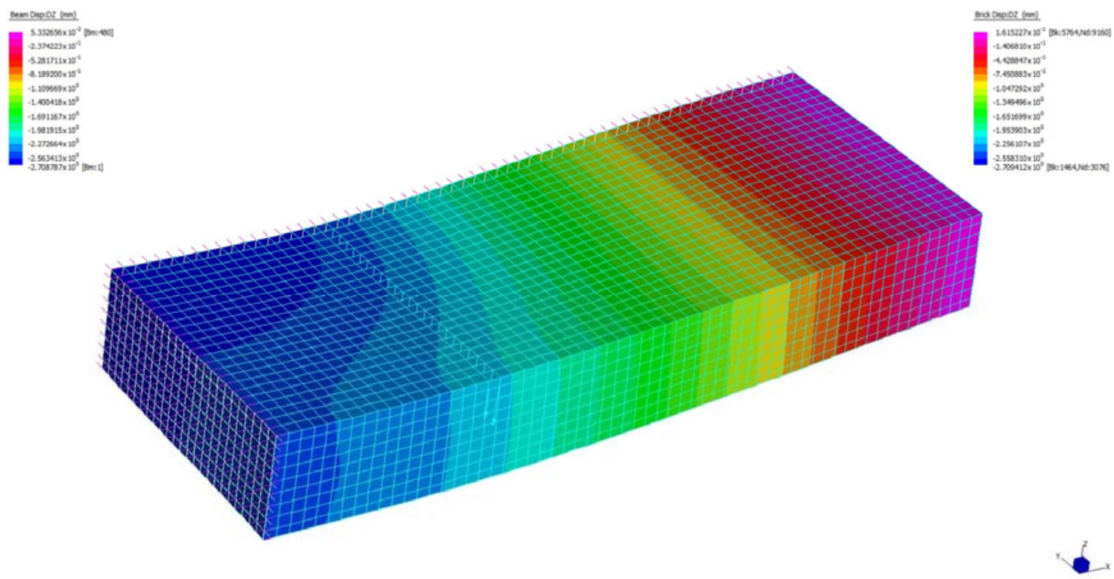

Figure 4-14 Deflection of CEPS panel

The maximum deflection occurred at the midspan of the beam, which is to be expected in the four point bending test simulation. This is shown in Figure 4-14 by the dark blue area. The pink area represents the rising of the ends of the beam, as the load is applied in the specimens. It was observed that due to the presence of the side concrete, there is less deflection in the side of the slab than in the centre part.

The stress distributions in CEPS panels for Case 1 is shown in Figure 4-15.

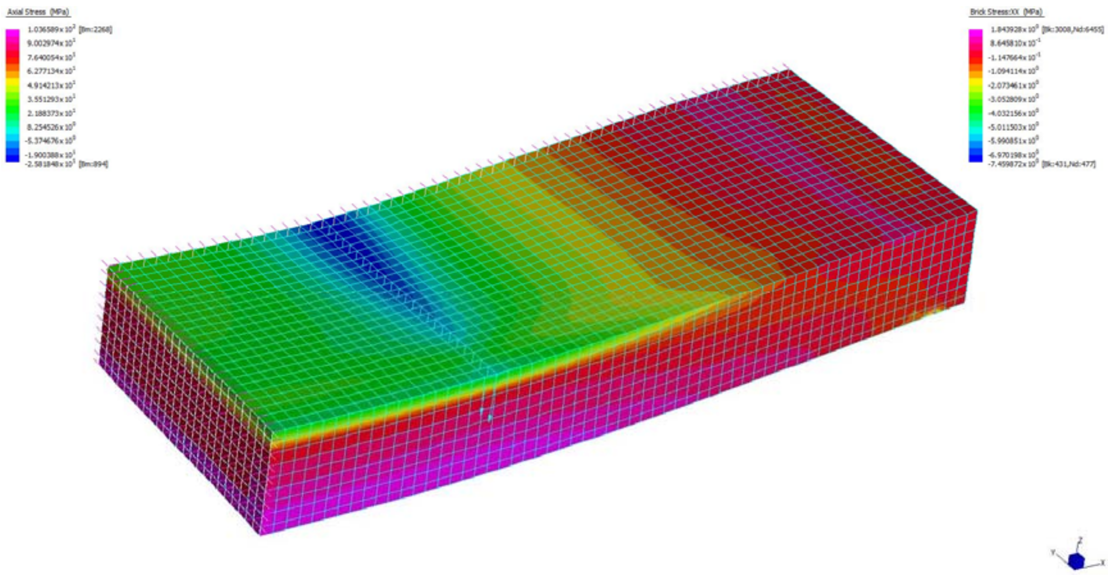

Figure 4-15 Stress Distribution in CEPS panel

Figure 4-15 shows the stresses throughout the CEPS panel as the load is applied. The pink area at the bottom of the slab represents the concrete in tensile. The dark blue and green area represents the concrete in compression.

4.10. Comparison of Results

This section provides a comparison of the experimental results and the finite element analysis results obtained using Strand7. This will provide some indication of how successful the numerical finite element analysis was compared to the experimental testing.

As previously discussed, finite element models were analysed for the testing CEPS panel specimens and experimental procedures discussed in Chapter 3. Also, a finite element was prepared to replicate CEPS panel that was built and prepared as discussed in this chapter. A comparison of typical load versus displacement curves for all the three cases from FEA and experiments are shown in Figure 4-16 to 4-18 . All these comparisons confirm that the finite element model can be satisfactorily used to analyse the load- displacement behaviour of CEPS panel used in the experiments.

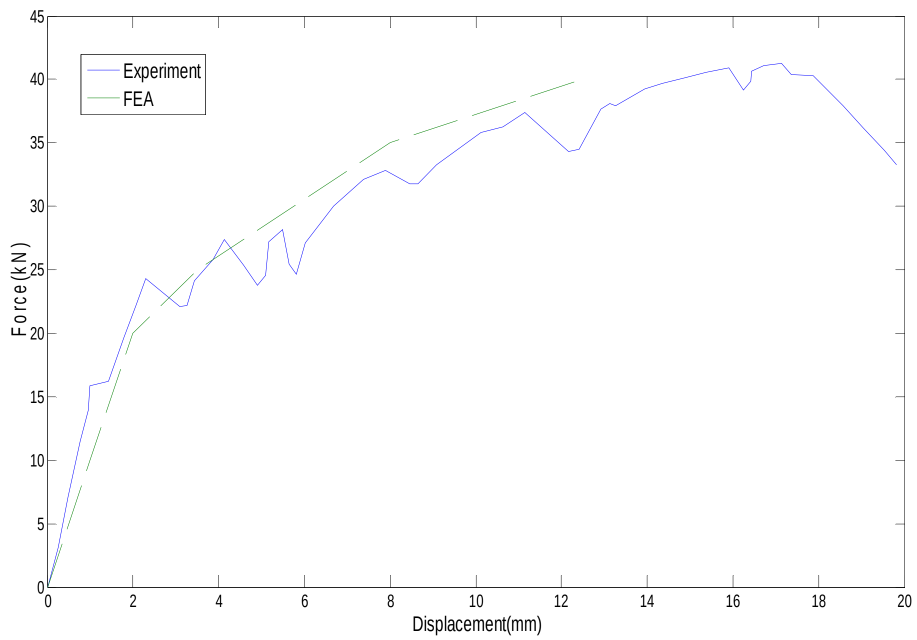

Figure 4-16 Comparison of Load Vs Displacement Curve for Case 1

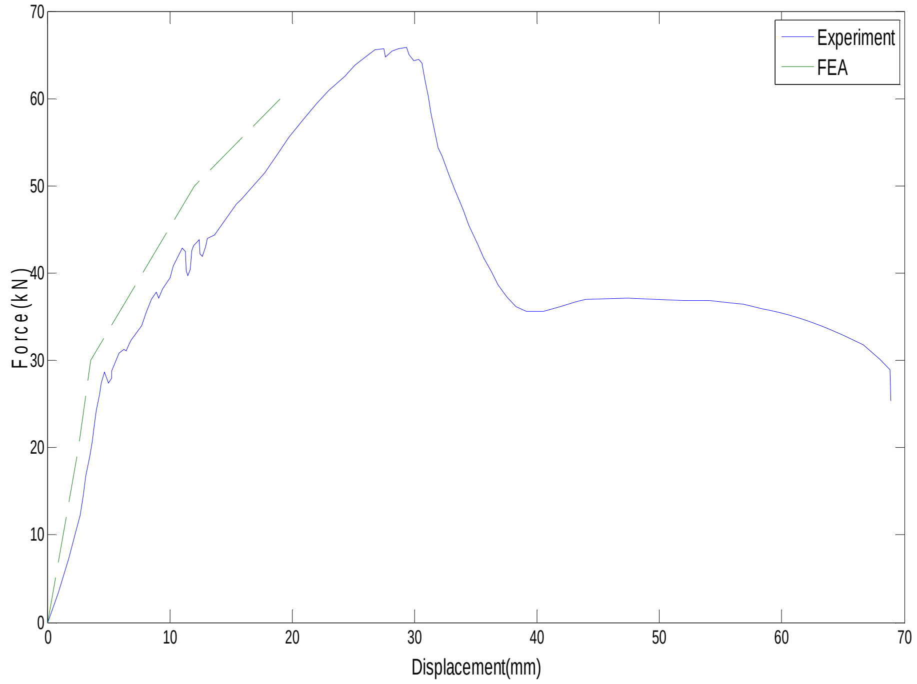

Figure 4-17 Comparison of Load Vs Displacement curve for Case 2

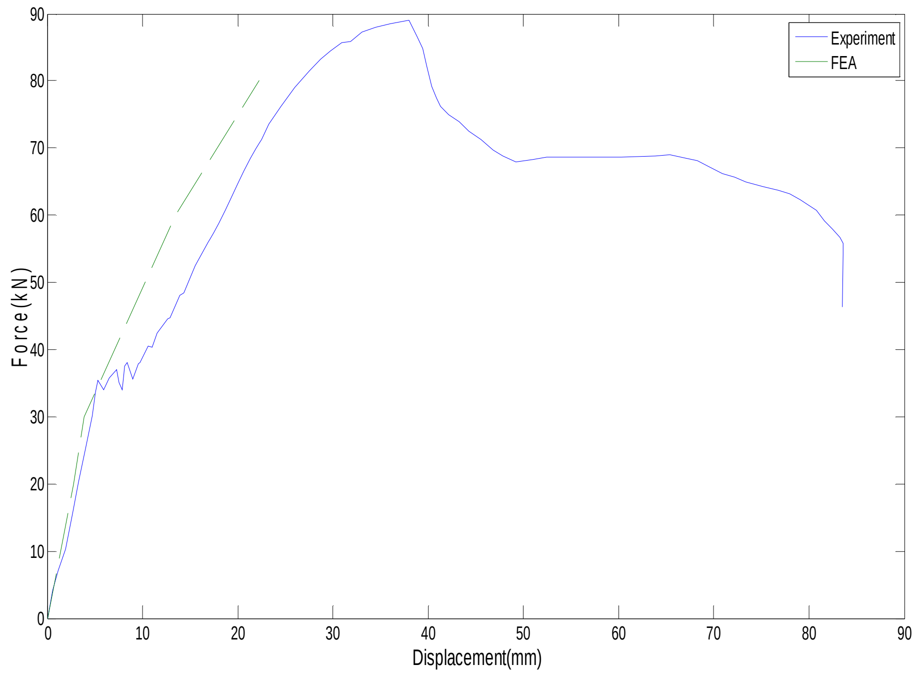

Figure 4-18 Comparison of Load Vs Displacement Curve for Case 3

As seen from Figure 4-16 to 4-18, the values compare well between the experimental testing and the finite element analysis. The non linear behaviour of CEPS panels is due to the tensile failure of the bottom concrete.

4.11. Discussion

This chapter has detailed the finite element analysis performed and also the comparisons between the experimental testing results. Overall, the experimental results compare very well with the finite element estimations.

However, the difference in values can be attributed to a number of main reasons. The first reason is the material properties used for each of the materials, concrete, EPS and steels in the Strand7 models. These properties were based on the previous researchers value as the details of the properties of the materials used in the experiment was unknown. Therefore, the finite element analysis results could differ slightly depending on the material properties assigned. In most cases, conservative material properties have been assigned.

Another reason for the inaccuracy in some places is the observation during the experimental testing of Case 2 and Case 3. As discussed in Chapter 3, during the testing for compressive strength test of concrete, the concrete cylinders were wet. This might have affected the validity of the results obtained for Case 2 and Case 3. However, these results still compare well with the finite element analysis results.

The reason for the finite element model not being able to predict the the deflection of the model at the higher load is FEA model does not include the cracking of the concrete that could occur after the tensile failure of concrete, hence after the prolongation of the wider crack the model deviate from the actual behaviour of the concrete. However, the model shows the similar behaviour in the model as that in the experimental one.

The purpose of the comparison of the experimental testing and finite element analysis results was to prove the excellent behaviour of the specimens during loading. The load deformation behaviour has been predicted very accurately by beam and brick finite element models.

Overall, the comparisons have been very promising, with the validity of the experimental tests proven with the comparison to the finite element analysis results.

Chapter 5

5. Parametric study

5.1. Introduction

This chapter deals with the parametric study of CEPS sandwich panels of different sizes and different thickness of the foam in order to determine the differences in responses when loads of various magnitudes were applied to these models.

To demonstrate the performance of CEPS sandwich panels, their structural performance was modeled with Strand7 over three spans and five separate load cases with three different foam thicknesses. The material properties used in all the models were taken from relevant Australian standards.

5.2. Practical Consideration

All the models were constructed in Strand7 based on the validated model from Chapter 4. The model was constructed to simulate real life situation. AS 1170.1-2002 gives the reference values of imposed floor actions. These values are shown in Appendix I. The load for domestic and office buildings varies from 1kPa to 5 kPa depending upon the specific uses. Hence, the design chart is developed by varying the imposed load from 1kPa to 5kPa. This allows the user to use the design chart based on their specific uses.

The various load magnitudes were applied to the model by creating the following load cases with Strand7.

- Self Weight (gravity) + 1 kN/m^2
- Self Weight (gravity) + 2 kN/m^2
- Self Weight (gravity) + 3 kN/m^2
- Self Weight (gravity) + 4 kN/m^2
- Self Weight (gravity) + 5 kN/m^2

These loads were applied to the slab model allowing the predicted deflection to be recorded.

Each model used in the parametric study was simply supported on each of the four sides. This is realistic for a slab supported by wall configuration from all four sides. The steel mesh is provided in both directions of the CEPS panels to transfer the load on both directions.

5.3. Panel Dimension and Material Property

Three different panel sizes were chosen having the dimension of 3 by 3 m, 3.5 by 3.5 m and 4 by 4 m.

According to AS3600-2009, the minimum cover recommended for steel reinforcement is 20 mm for 20MPa concrete. These values are shown in Appendix J. The diameter of steel is taken same as that of the experimental i.e. 3 mm. Hence, the total concrete thickness is taken as 50 mm with steel mesh at the centre of the concrete. This will ensure a sufficient cover for the steel mesh. The thickness of the foam is varied from 50 mm, 75 mm to 100 mm for parametric study.

In this analysis, the characteristic strength of concrete is taken as 20MPa which is the minimum concrete strength required for reinforced concrete as per AS3600-2009. The stress- strain curve is used similar to experimental one as shown in Figure 4-3.

The yield strength of steel is taken as 500MPa which is generally used in Australia. In this analysis, only the steel wire mesh is used with the diameter of 3mm. Table 5-1 shows the cutoff force calculation for steel mesh.

Table 5-1 Cutoff Force Calculation for Steel reinforcement used in Parametric study

S.No	Diameter of bar (mm)	Area of Bar (mm^2)	Yield Strength (MPa)	Cutoff Force (kN)
1.	3	7.068	500	3.534

As per AS 1366.3-1992, ‘M’ type EPS foam is used as flooring materials. This gives the density of EPS as $19kg/m^3$. The properties of material are taken similar to that of experimental shown in Table 4-2.

5.4. Load calculation

Three panel sizes of 3 by 3m, 3.5 by 3.5 m and 4 by 4 m was used for the parametric study. The quarter model of the slab was used for the analysis. Hence, the load from only the quarter of the slab was considered for the analysis.

$1kN/m^2$ was used as the base value for the ease of load case creation associated with a unit value. The load was uniformly distributed by calculating the load of $1kN/m^2$ over the area of the slab divided by the number of nodes in that floor area. The resulting load was then applied to each node within that area. For example, for the floor model of 3 by

3 m panel, only quarter of the panel was modeled which gives the panel size as 1.5 by 1.5 m. Only the nodes of the quarter model was considered which is equal to 3720 was considered. Load per node is calculated as shown in Equation 5.1. A complete list of models and applied load per node used in the parametric study is given in Table 5-2.

$$\frac{1kN \times (1.5m \times 1.5m)}{Number\ of\ \ nodes} = \frac{2.25kN}{3720} = 0.000604kN / node \qquad (5.1)$$

Table 5-2 Nodal load calculation for parametric study

Model dimension	Quarter Model dimension	Applied Load (kN)	Applied load for Quarter model (kN)	Number of nodes	Load/node (kN)
3 m x 3m	1.5 m x 1.5 m	9	2.25	3720	0.000604
3.5m x 3.5m	1.75m x 1.75 m	12.25	3.0625	5041	0.0006075
4m x 4m	2m x 2m	16	4	6561	0.0006096

The various load magnitudes were applied to the model by creating the following two cases within Strand7 and then combining them and multiplying by the appropriate factor to achieve the required total load:

- Self weight (gravity)
- 1kN/m^2

These loads were applied to the slab model by having Strand7 calculate the self weight load created by a gravitational force of -9.81 m/sec^2 as one separate load case. The load combination cases were applied to the model by creating load cases within Strand7 which take the self weight and combine it to the 1 kN/m^2 UDL multiplied by the appropriate factor to create a load case of the desired magnitude.

5.5. Mesh Size and boundary conditions

To reduce the analysis time and increase the accuracy of the test, all the panels were modeled as quarter model similar to Chapter 4.

For the parametric study, the mesh size for the concrete and EPS foam was chosen as 25 x 25 x 25 mm which gives the aspect ratio of 1:1 in all directions.

Since the slab was supported by the wall mechanism, the boundary condition for the two continous edges were taken as simply supported slabs. While on the other two sides of the panels where only the quarter of the model is used to analyse the symmetric boundary conditions similar to Chapter 4 was applied to the entire surface.

The model procedure was constructed by following the steps mentioned in Chapter 4.8 to create altogether of nine panels as follows:

1. 3m x 3 m slab of foam thickness 50 mm
2. 3.5m x 3.5 m slab of foam thickness 50 mm
3. 4m x 4m slab of foam thickness 50 mm
4. 3m x 3 m slab of foam thickness 75 mm
5. 3.5m x 3.5 m slab of foam thickness 75 mm
6. 4m x 4m slab of foam thickness 75 mm
7. 3m x 3 m slab of foam thickness 100 mm
8. 3.5m x 3.5 m slab of foam thickness 100 mm
9. 4m x 4m slab of foam thickness 100 mm

Figure 5-1 to 5-4 shows the model geometry, mesh size and the loading pattern along with appropriate boundary conditions for the quarter slab model having different foam thickness.

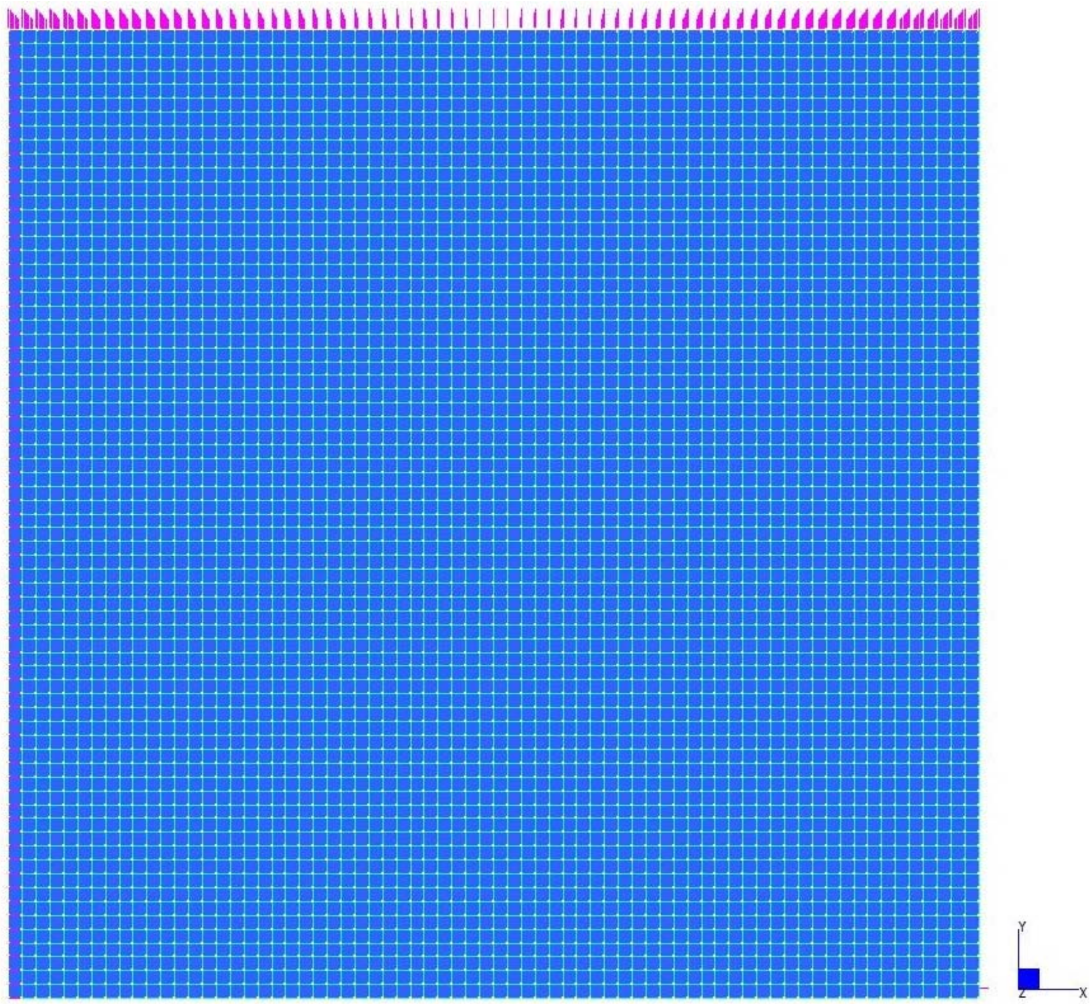

Figure 5-1 Top view of CEPS used in the parametric study

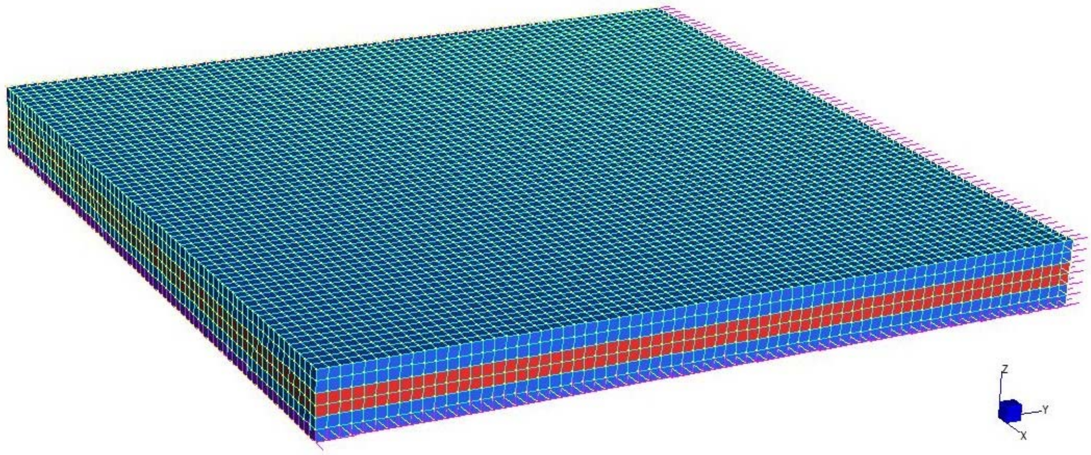

Figure 5-2 CEPS sandwich panel having thickness of 50mm

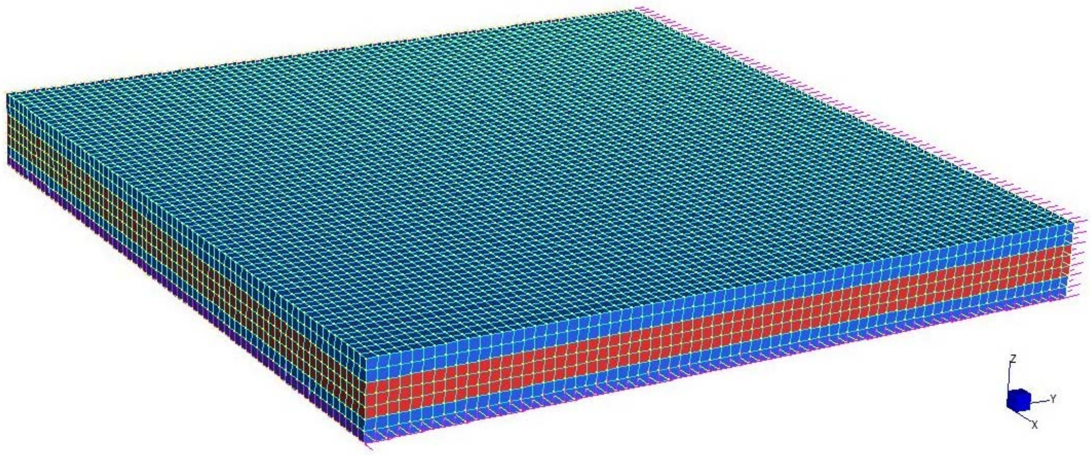

Figure 5-3 CEPS sandwich panel having thickness of 75 mm

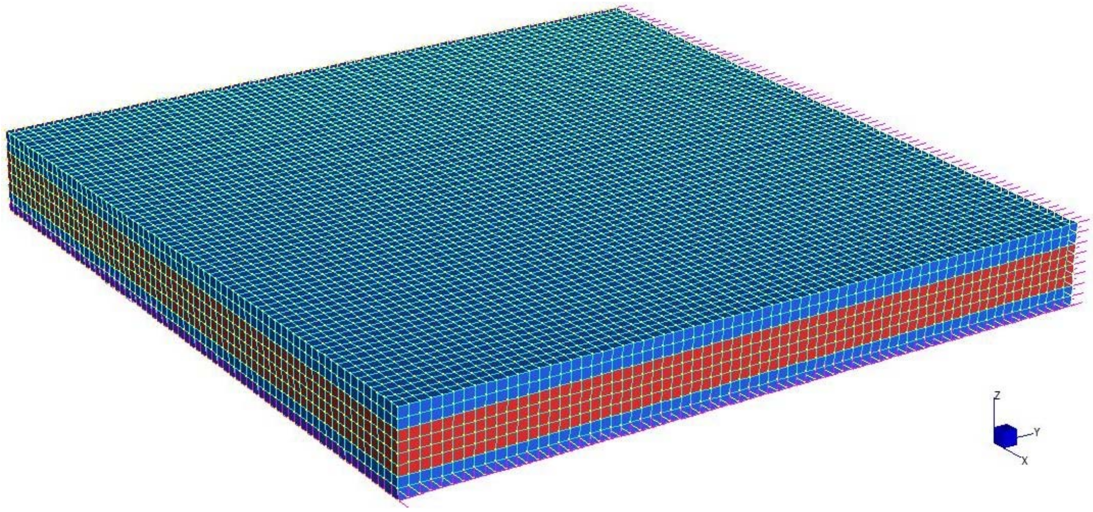

Figure 5-4 CEPS sandwich panel having thickness of 100 mm

Each model was then solve by non linear static solver to analyse the deflection caused by the applied load over the given span and width, with the vertical deflection in the centre of the slab recorded. Once the load deflection lines were plotted, further lines were created to join the distinct load case results on each span. Deflection limit lines were also superimposed on the load deflection curves for each span. This line was drawn to allow interpolation of values when used in a design situation. For example taking a required load and deflection limit and using those values to solve for the maximum clear span which can be used with CEPS panels.

5.6. Design Criteria

From the analysis of the CEPS panels, it is observed that the failure criterion is because of the crack formation in the tensile concrete that results in the de-bonding of the steel and concrete. As the static behaviour and strength of sandwich panels is based on the composite action of the three structural layers, namely the two faces and the core. But with the de-bonding of steel and concrete and de-bonding of concrete and foam, the panels loses the composite action. During the experiment, it was observed that during the failure the steel mesh wires were snapping one by one producing a snapping sound. One of the solutions for this would be use of stronger vertical connectors that can hold the top and bottom concrete.

The limitation of Strand7 is modelling of cracked beam and de-bonding of steel and concrete. However for the parametric study, only the initial linear region was considered before the failure of the concrete showing brittle behaviour. The deflection at which the brittle failure started to notice was at 6mm in the experiment having length of 3098.8 mm. Hence, a deflection limit of span/500 was used as the limiting design criteria. Although AS3600 does permit 1/250 for reinforced slab, as a conservative design, 1/500 deflection limit will be adopted for the analysis.

Based on this modeling, the span limitations required for the deflection limit is shown in Table 5-3.

Table 5-3 Deflection Limit for design chart

Panel Size	Deflection Limit (L/500) (mm)
3m x 3 m	6
3.5m x 3.5 m	7
4m x 4m	8

5.7. Design Chart

The typical deflection of the slab is shown in Figure 5-5. The panel was modeled to a quarter scales and the figure shows the deflection of quarter slab model.

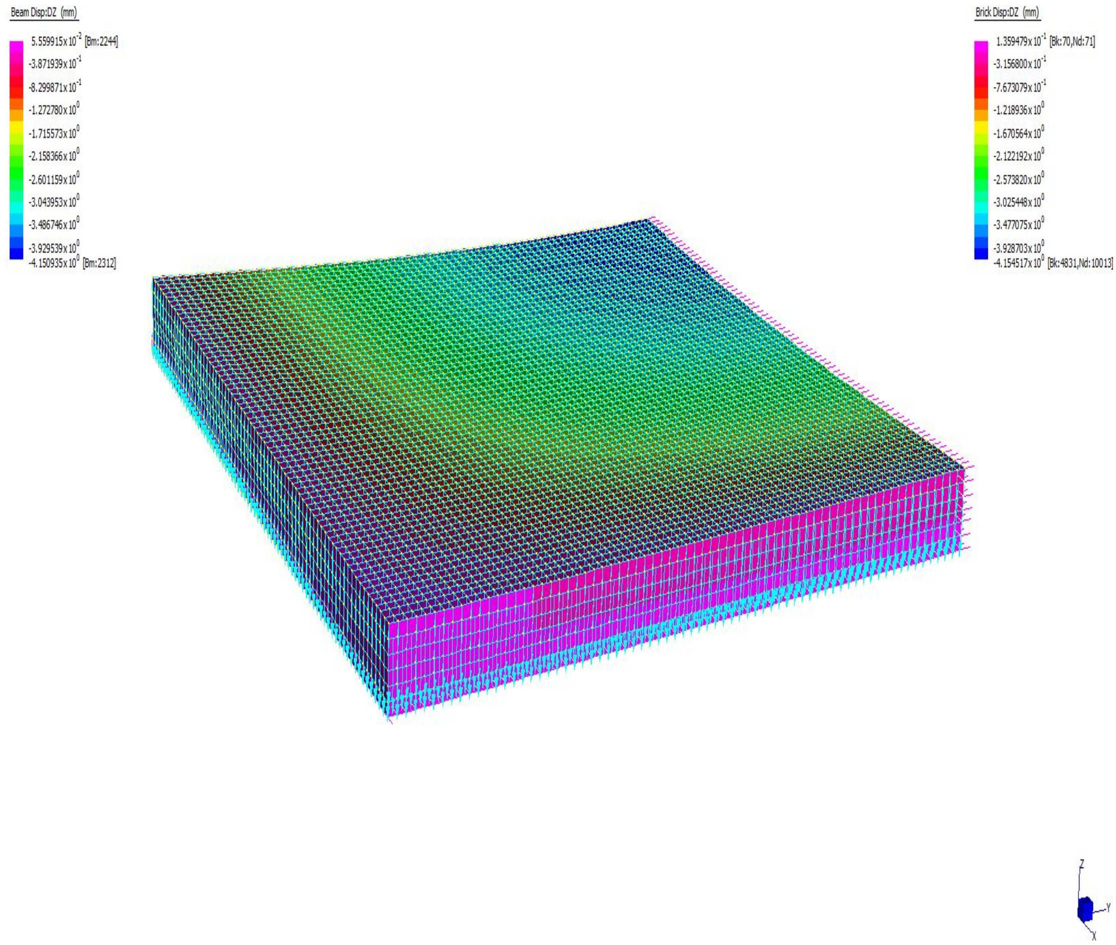

Figure 5-5 Deflection of CEPS panel for parametric study

The maximum deflection occurred at the midspan of the beam.. This is shown in Figure 5-5 by the dark blue area. The pink area represents the rising of the ends of the beam, as the load is applied in CEPS sandwich panel.

The total load and deflection for each case is shown in Table 5-4. This information is used to plot the load deflection curves for slabs having a clear span range of 3m to 4m as shown in Figure 5-6 to 5-8. All deflections recorded are the maximum value obtained at the centre of the slab. Due to the deflection limit being used as the governing criteria no evaluation was done on the variation of stress levels within the slab as the load increases.

Table 5-4 Deflections of CEPS sandwich panels for each load case

Panel Size	Load	Toal Load (kN)	50 mm Core	75 mm Core	100 mm Core
			Deflection (mm)	Deflection (mm)	Deflection (mm)
3 by 3	Self weight + 1 kN/m^2	31.5	2.557	2.123	1.487
	Self weight + 2 kN/m^2	40.5	3.330	2.759	1.926
	Self weight + 3 kN/m^2	49.5	4.100	3.395	2.366
	Self weight + 4 kN/m^2	58.5	4.885	4.037	2.805
	Self weight + 5 kN/m^2	67.5	5.667	4.678	3.244
3.5 by 3.5	Self weight + 1 kN/m^2	42.875	4.150	3.466	2.452
	Self weight + 2 kN/m^2	55.125	5.370	4.485	3.173
	Self weight + 3 kN/m^2	67.375	6.590	5.504	3.894
	Self weight + 4 kN/m^2	79.625	7.897	6.569	4.616
	Self weight + 5 kN/m^2	91.875	9.057	7.557	5.337
4 by 4	Self weight + 1 kN/m^2	56	6.080	5.111	3.656
	Self weight + 2 kN/m^2	72	7.869	6.614	4.729
	Self weight + 3 kN/m^2	88	9.660	8.118	5.803
	Self weight + 4 kN/m^2	104	11.460	9.626	6.876
	Self weight + 5 kN/m^2	120	13.260	11.135	7.949

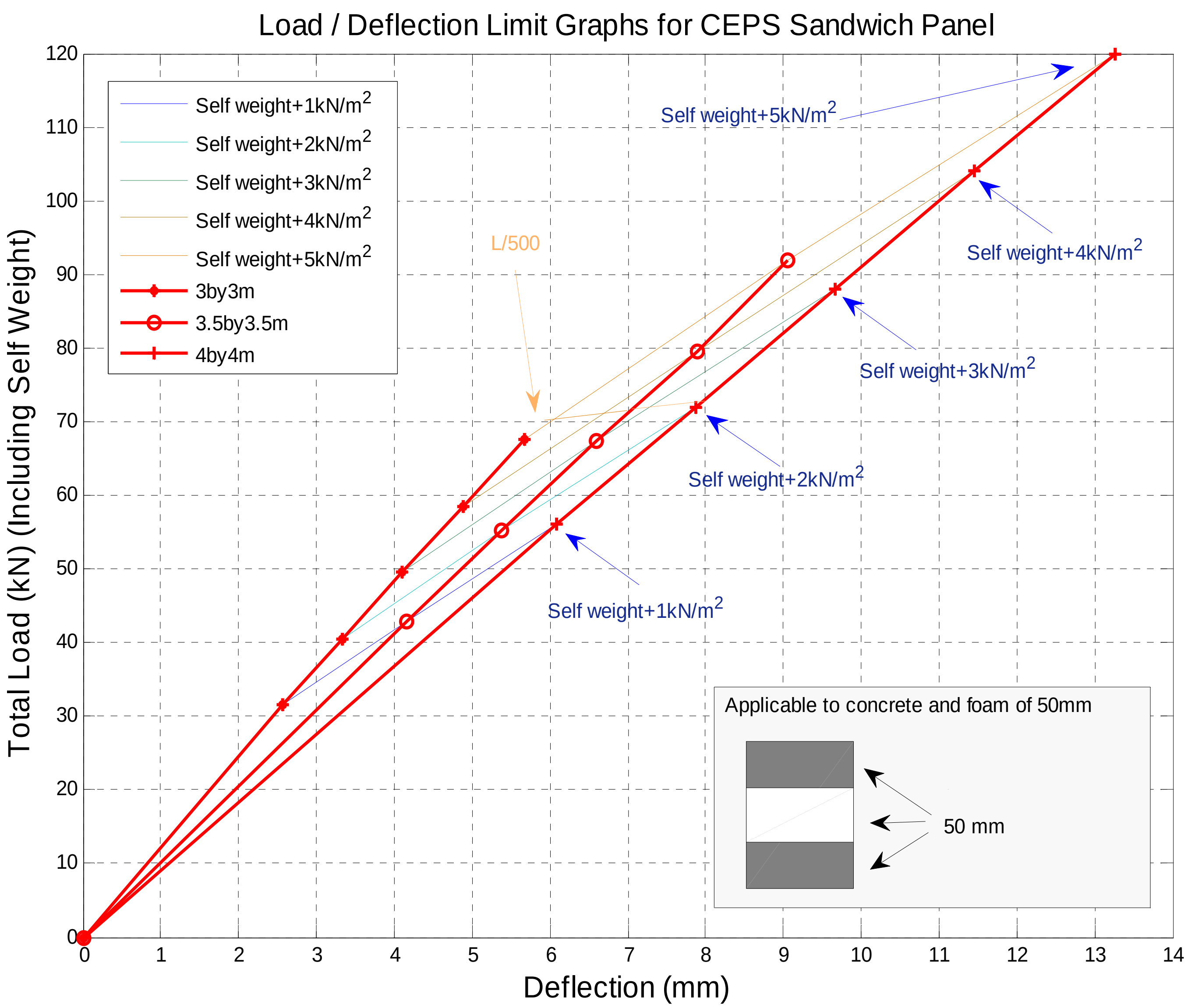

Figure 5-6 Design Chart of CEPS panel having EPS thickness of 50 mm

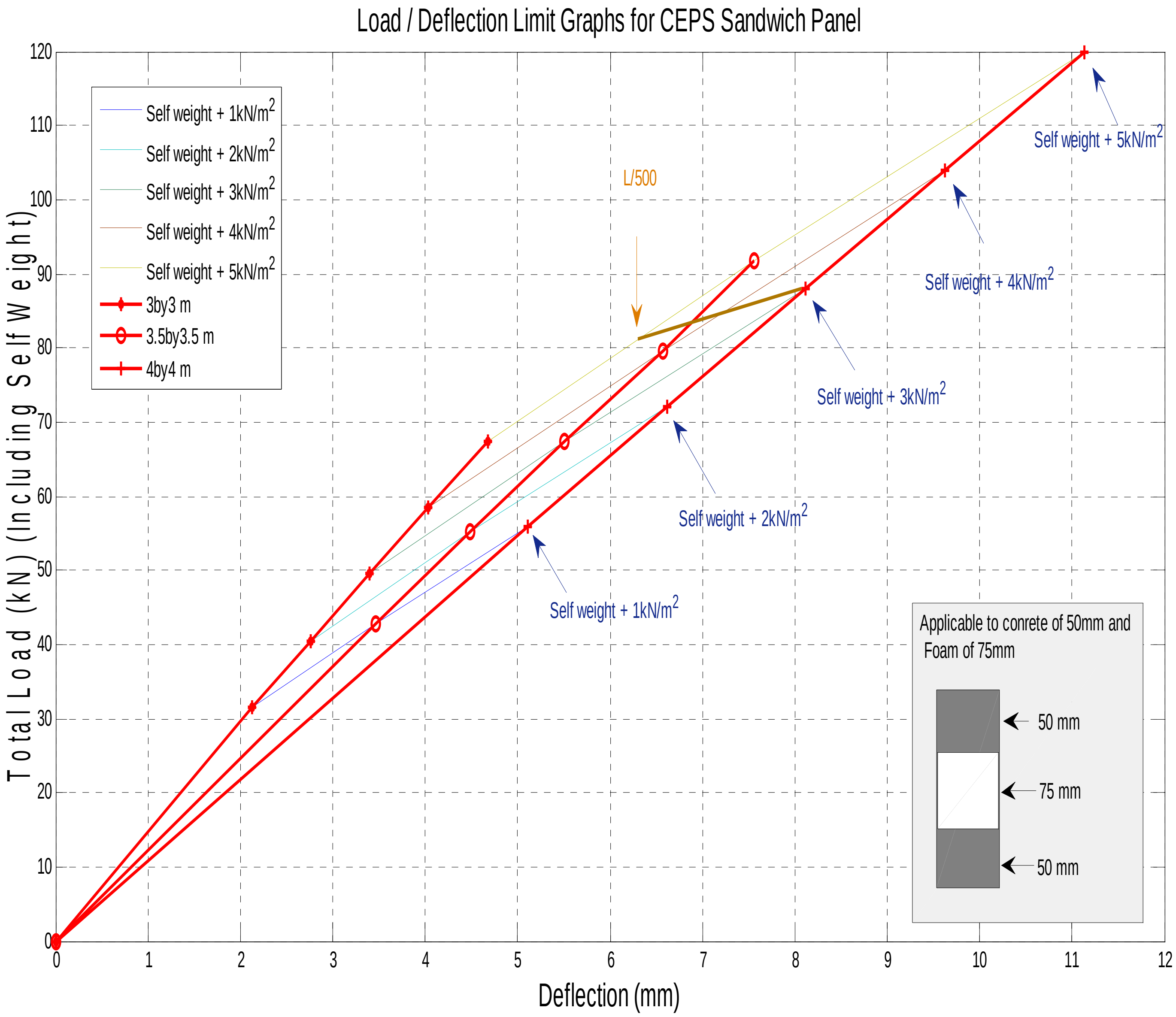

Figure 5-7 Design Chart of CEPS panel having EPS thickness of 75 mm

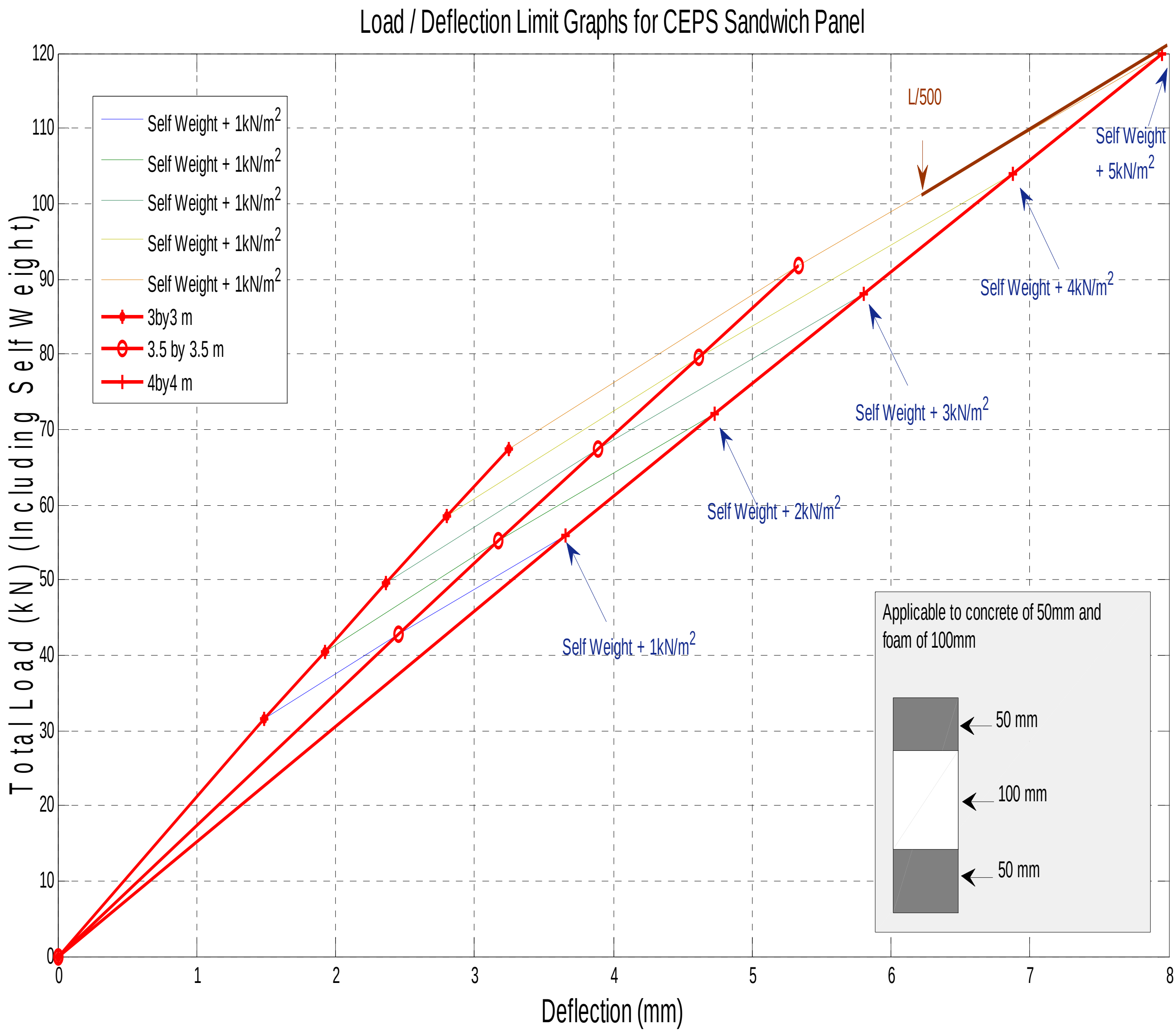

Figure 5-8 Design Chart of CEPS panel having EPS thickness of 100 mm

5.8. Discussion

The CEPS design chart was created for three different panel sizes by varying the thickness of EPS foam. In this design chart, factor of safety is not considered. Hence, an appropriate factor of safety should be included if using the base representative values shown in design chart.

CEPS slab panel design chart was created using the properties of concrete and steel as per Australian Standard. For EPS, the density was taken as the average value provided and is therefore a representative value. Also, since EPS has a very low modulus of elasticity compared to steel and concrete so it is considered as a linear material.

Figure 5-6 to 5-8 indicates that by increasing the foam thickness, the deflection can be decreased. It was observed that when the EPS thickness is increased to 100mm, 4 by 4 m panel shows the deflection within the permissible limit of L/500 for 5kN/m^2 load. Hence, the parametric study of the CEPS panels indicates the possibility of using CEPS panels as a flooring material.

Chapter 6

6. Conclusion and Recommendations

6.1. Summary

This research project has examined the development and behaviour of CEPS sandwich panels to be used as structural slabs. The research was based on the experimental testing of the three CEPS sandwich panels conducted by UCI. A finite element analysis of CEPS panels has been implemented with the intention of determining the limitations of using CEPS panels in slab applications.

Appropriate finite element models were developed to simulate the behaviour of CEPS panels used in the laboratory experiments. The finite element model was validated using experimental results and then used to create a design chart for different foam thickness for different panel sizes.

A combination of analyzed test results and modeling revealed that the critical limiting factor associated with CEPS panels used in slab applications is deflection. Furthermore, a parametric study was conducted on various loading situations for different panel sizes with different foam thickness to develop a deflection based design chart for CEPS panels applied to slab applications.

This section outlines the achievement of objectives, conclusions of the research undertaken and possible areas of further research.

6.2. Achievement of Project Objectives

This section details the major objectives of this research and the outcomes of each objective.

1. *Conduct a literature review on sandwich panels*

The use of sandwich panels, in civil engineering construction, is an efficient and economic way of material utilization. As the industry continues to change with advancement in technology, the improvement of sandwich panels have also continued. These materials will have a major effect on the engineering, architectural and building industry in future years. The literature review was conducted on the history of sandwich panel along with its concept. This is covered in Chapter 2. A literature review has been undertaken to understand the prior work done on different types of sandwich panels used as a civil engineering materials.

As the use of expanded polystyrene foam in the middle of concrete-steel facings is, relatively, a new concept as there was limited literature available on the actual uses of CEPS panels as appropriate materials for slab applications. However, the material properties were review individually, and then the available literature was researched. These are covered in Chapter 2.

2. *Create a finite element model of CEPS panels using Strand7*

Chapter 4 focused on the selection of appropriate modeling parameters and the creation and use of Strand7 finite element analysis computer models to analyse the limitations of using CEPS panels as a structural slabs. They were prepared based on the testing specimens. The analysis was able to successfully provide behaviour of the model specimens from a finite element analysis perception.

Results from this modeling highlighted the use of quarter model instead of full model that can reduce the time of analysis. The non linear behaviour of concrete and steel were also incorporated in the analysis.

3. *Validate the model by using the test data provided by University of California, Irvine.*

Chapter 3 presents the four point loading test performed in UCI. The dimensions of the CEPS panel and loading conditions were also described. A simplified method of analysis was presented in Chapter 3 which calculates the failure load for all the three cases. The results from simplified analysis underestimate the actual failure load. Hence, finite element analysis was used in Chapter 4 to predict the load-displacement curve. The results from finite element analysis provided much closer results which validate the models.

4. *Use the results from modeling to create a design aid for the use of CEPS panel in slab construction*

The valid models of CEPS were obtained from Chapter 4. This information was incorporated in a range of Strand7 models as described in Chapter 5. These models were then used to collect load and maximum deflection data points over various spans length of different foam thickness. The loads were considered consistent throughout the analysis. The results is deflection based design chart for three different foam thickness of 50 mm, 75 mm and 100 mm having a concrete facing of 50mm subjected to a variety of different load cases.

Since, the deflection limit of L/500 was used as the limiting criteria in the design of CEPS panels, deflection limit lines are superimposed on that chart so that it can be interpreted in accordance with the required deflection limit used in design.

6.3. Conclusions

The overall results from this research project were promising with respect to the behaviour of CEPS panels when used in a slab application.

As global environmental issue is large, CEPS panels can be consulted as an alternate structural material to be used as structural slabs. With the use of CEPS, it not only reduces the self weight of the structure but also provides excellent insulation against sound, thermal heat and vibration. Structurally, it is important to ensure that these materials will provide the qualities of structural slabs that is required. These qualities included strength and deflection of this slab which should be within tolerable limits.

Overall, the experimental testing and finite element analyses confirmed the future potential of CEPS panels. It has been shown and proven that the strength and deflection limit of these specimens is very promising from the parametric study that was conducted.

6.4. Recommendations

Since the use of sandwich panels as a mainstream product in buildings is relatively new in Australia, further research should be undertaken to improve the understanding of the various behavioural aspects of sandwich panels under specific Australian conditions. While this research project have confirmed the excellent potential of CEPS panels as a structural slabs, there is a need to further investigate these materials to ensure they will be safe and adequate replacements for fully reinforced concrete slabs.

Recommendations for further studies include:

- Conduct more extensive testing on CEPS panels and see if there are other combinations and modifications that will provide better results. This will ensure that a number of designs are considered and the possibility of more promising results is investigated. Some of the modifications may involve use of higher strength of vertical bars.
- Investigate the shear failure of these specimens. This research focuses more on the displacement behaviour of the CEPS panels. Hence, further research is needed to understand the shear mechanism of CEPS panels. This would help to understand the behaviour of CEPS panels and provide some indication of the failure modes.
- Establish simplified design models and design procedures based on the experimental testing and finite element model. After conducting more experiments with different types of vertical stirrups, a simplified design models should be created which includes the stress transfer from the vertical stirrups as well.

- Expose CEPS panels to some form of dynamic loading. This would ensure that a number of loading conditions are performed on the specimens and provide some indication of which loading cases will be more critical.
- Conduct some typical optimization studies and analyses to determine the most efficient and cost effective designs to provide the highest performance and behaviour characteristics.

References

ACI318-05 2005, *Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary,* American Concrete Institute, Farmington Hills, Mich.

Allen, HG 1969, *Analysis and design of structural sandwich panels*, [1st ed.] edn, Pergamon Press, Oxford.

Bangash, MYH 2001, *Manual of numerical methods in concrete : modelling and applications validated by experimental and site-monitoring data*, Thomas Telford, London.

Candappa, D 2000, 'The constitutive behaviour of high strength concrete under lateral confinement', PhD Thesis, Monash University, Australia.

Chakrabarti, A & Sheikh, AH 2005, 'Analysis of Laminated Sandwich Plates Based on Interlaminar Shear Stress Continuous Plate Theory', *Journal of Engineering Mechanics*, vol. 131, no. 4, pp. 377-84

Davies, JM 1987, 'Axially Loaded Sandwich Panels', *Journal of Structural Engineering*, vol. 113, no. 11, pp. 2212-30

Davies, JM (ed.) 2001, *Lightweight sandwich construction*, Blackwell Science, Oxford.

Duskov, M 1997, 'Materials Research on EPS-20 and EPS-15 Under Representative Conditions in Pavement Structures', *Geotextiles and Geomembranes*, vol. 15, nos. 1, 2, and 3, pp. 147-181.

Fairbairn, W 1849, *An Account of the Construction of the Britannia and Conway Tubular Bridges*, Red Lion Court, London, viewed 12 April 2011, <http://books.google.com.au/books?hl=en&lr=&id=UlEEAAAAQAAJ&oi=fnd&pg=PR1&dq=an+account+of+the+construction+of+the+britannia+and+conway+tubular+bridge&ots=HYKDcYNDV9&sig=nPzGWVnpaIQpjGJn2Z8UUIAIH_0#v=onepage&q=an%20account%20of%20the%20construction%20of%20the%20britannia%20and%20conway%20tubular%20bridge&f=false>.

Horvath, JS 1995, *Geofoam Geosynthetic*, New York, USA.

Horvath, JS 1997, 'The Compressible Inclusion Function of EPS Geofoam', *Geotextiles and Geomembranes*, vol. 15, nos. 1, 2, and 3, pp. 77-119.

Karam, GN & Gibson, LJ 1994, 'Evaluation of Commercial Wood-Cement Composites for Sandwich-Panel Facing', *Journal of Materials in Civil Engineering*, vol. 6, no. 1, pp. 100-16 .

Lloyd, NA & Rangan, BV 1995, 'High strength concrete columns under eccentric compression', Technical Report No. 1/95, Curtin University of Technology, Perth, Western Australia.

Logan, DL 1986, *A first course in the finite element method*, PWS Engineering, Boston.

Manalo, AC 2011, 'Behaviour of fibre composite sandwich structures: a case study on railway sleeper application', PhD thesis, University of Southern Queensland, Toowoomba.

Mousa, MA & Uddin, N 2010, 'Experimental and Analytical Study of Composite Structural Insulated Floor Panels', Honolulu, HI.

Negussey, D & Jahanandish,M 1993, 'Comparison of Some Engineering Properties of Expanded Polystyrene with Those of Soils', *Transportation Research Record 1418*, pp. 43-48.

Poh, KW 1997, 'General Stress-Strain Equation', *Journal of Materials in Civil Engineering*, vol. 9, no. 4, pp. 214-7

Pokharel, N 2003, 'Behaviour and Design of Sandwich Panels Subject to Local Buckling and Flexural Wrinkling Effects', PhD thesis, Queensland University of Technology, Brisbane.

Rashid, M. A., Mansur, M. A., & Paramasivam, P. (2002), 'Correlations between mechanical properties of high-strength concrete', *Journal of Materials in Civil Engineering,* vol.14, no. 3, pp. 230-238.

Rizzo, S & Fazio, P 1983, 'Sandwich-Panel Assemblies: Analytical Model', *Journal of Structural Engineering*, vol. 109, no. 11, pp. 2715-32

Schenker, A, Anteby, I, Nizri, E, Ostraich, B, Kivity, Y, Sadot, O, Haham, O, Michaelis, R, Gal, E & Ben-Dor, G 2005, 'Foam-Protected Reinforced Concrete Structures under Impact: Experimental and Numerical Studies', *Journal of Structural Engineering*, vol. 131, no. 8, pp. 1233-42

Sokolinsky, VS, Shen, H, Vaikhanski, L & Nutt, SR 2003, 'Experimental and analytical study of nonlinear bending response of sandwich beams', *Composite Structures*, vol. 60, no. 2, pp. 219-29

Standards Australia 2009, *Concrete structures*, AS 3600-2009, Standards Australia, Sydney, viewed 30 July 2011, <http://www.saiglobal.com/online/autologin.asp>.

Standards Australia 1992, *Rigid cellular plastics sheets for thermal insulation - Rigid cellular polystyrene - Moulded (RC/PS – M)*, AS 1366.3-1992, Standards Australia, Sydney, viewed 15 August 2011, <http://www.saiglobal.com/online/autologin.asp>.

Standards Australia 2002, *Structural design actions Part 1: Permanent, imposed and other actions*, AS/NZS 1170.1-2002, Standards Australia, Sydney, viewed 15 August 2011, <http://www.saiglobal.com/online/autologin.asp>.

Ueng, CES 2001, 'Sandwich Composites', in AM Robert (ed.), *Encyclopedia of Physical Science and Technology*, Academic Press, New York, pp. 407-12.

Vaidya, A, Uddin, N & Vaidya, U 2010, 'Structural Characterization of Composite Structural Insulated Panels for Exterior Wall Applications', *Journal of Composites for Construction*, vol. 14, no. 4, pp. 464-9

Warner, RF 2007, *Reinforced concrete basics : analysis and design of reinforced concrete structures*, Pearson Prentice Hall, Frenchs Forest, N.S.W.

Zenkert, D 1995, *An introduction to sandwich construction*, Engineering Materials Advisory Services, Cradley Heath, West Midlands, England.

Bibliography

Barbero, EJ 2007, *Finite element analysis of composite materials*, Taylor & Francis, Boca Raton FL.

Computer aided design in composite materials technology IV, 1994, Computational Mechanics Publications, Southampton..

Crisfield, MA 1991, *Non-linear finite element analysis of solids and structures*, Wiley, Chichester.

Everard, NJ 1966, *Schaum's outline of theory and problems of reinforced concrete design*, McGraw-Hill, New York.

Gallagher, RH 1974, *Finite element analysis: fundamentals*, Prentice Hall, Englewood Cliffs NJ.

Kotsovos, MD 1995, *Structural concrete : finite-element analysis for limit-state design*, Thomas Telford, London.

Palm, WJ 2005, *Introduction to MATLAB 7 for engineers*, 2nd edn, McGraw-Hill, Dubuque IA.

Appendix A- Project Specification

University of Southern Queensland

FACULTY OF ENGINEERING AND SURVEYING

ENG 8411/8412 Research Project
PROJECT SPECIFICATION

FOR:	ROHAN MUNI BAJRACHARYA
TOPIC:	STRUCTURAL EVALUATION OF CONCRETE-EXPANDED POLYSTYRENE SANDWICH PANELS FOR SLAB APPLICATIONS
SUPERVISORS:	Associate Prof. KARU KARUNASENA
	Dr WEENA LOKUGE
ENROLMENT:	ENG 8411-S1, 2011
	ENG 8412-S2, 2011
PROJECT AIM:	To undertake a thorough investigation on the behaviour of concrete expanded polystyrene sandwich panels and develop a finite element model of Concrete expanded polystyrene sandwich panels to predict load deformation curve

PROGRAMME:

1. Conduct a literature review on sandwich panels.
2. Create a finite element model of CEPS sandwich panels using Strand7.
3. Validate the model by using the test data provided by University of California, Irvine.
4. Use the results from modeling to create a design aid for the use of CEPS sandwich panel in slab construction
5. Remarks and preparation of the thesis.

If time permits:

7. Perform an experiment to verify the model.

AGREED:

26/10/11	26/10/11	26/10/11
Rohan Muni Bajracharya	Associate Professor Karu Karunasena	Dr. Weena Lokuge
Student	Supervisor	Supervisor

Appendix B - Risk assessment

Most of the test has already been done in University of California. Since, this is the theoretical study, no safety issue regarding the experiment is concerned with this research project except that of the long use of computer and safety issue of office comprises.

Table B-1 Risk Management Chart

Hazard Category: Computer Use

Description of Hazards	**People at Risk**	**Number of People**	**Parts of body**	**Risk Level**
Excessive Use of Computer	Computer user	1	• Back pain • Eye Fatigue	Significant

Risk Control:

- Shall not be engaged to work for a period longer than five hours continuously without a recess of at least 30 minutes for refreshment.

Appendix C – Procedure for digitization

The graph provided by UCI is digitized using AutoCAD 2007. Following are the procedure:

1. Open AutoCAD.
2. Go to Insert, Raster Image Reference
3. Browse and open the image file
4. Go to command prompt line
5. Type pline
6. Click on the line as shown in Figure D-1 to Figure D-3.
7. Go to prompt line and type ‘li’
8. Get the coordinates of the line and import it to Ms Excel
9. Go to Tools, Inquiry, Distance to find the distance of 1inch and 1kips.
10. Use this data to convert the digitized coordinates to find the real experimental data in Ms Excel.
11. Appendix E shows the calculation of Ms Excel.

Appendix D – AutoCAD Digitization of Experimental Data

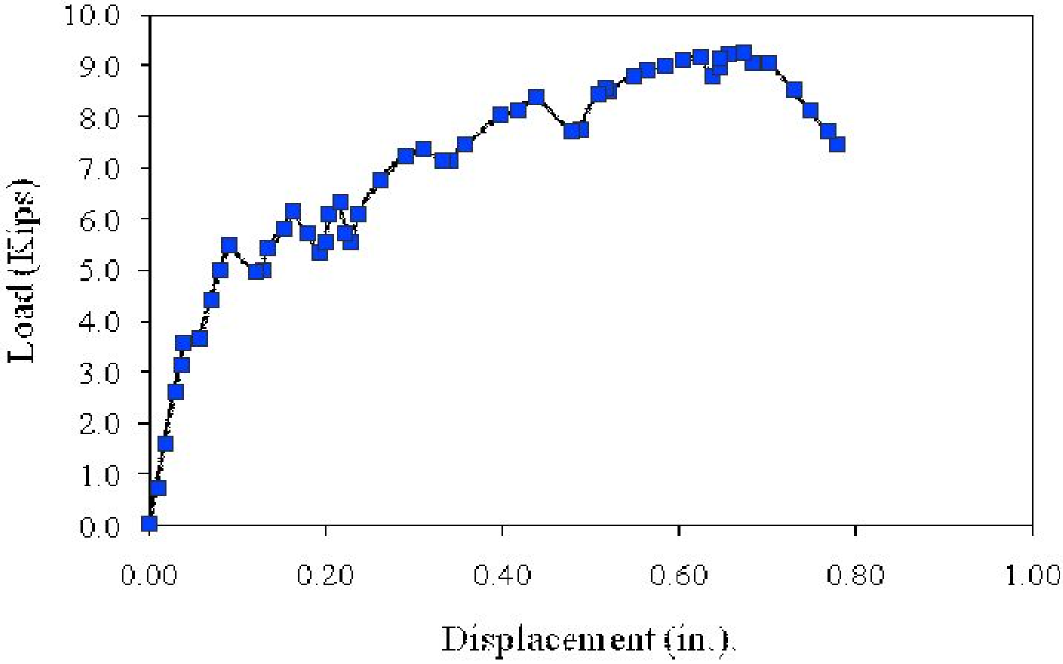

Figure D-1 Autocad Digitization of the UCI data for Case 1

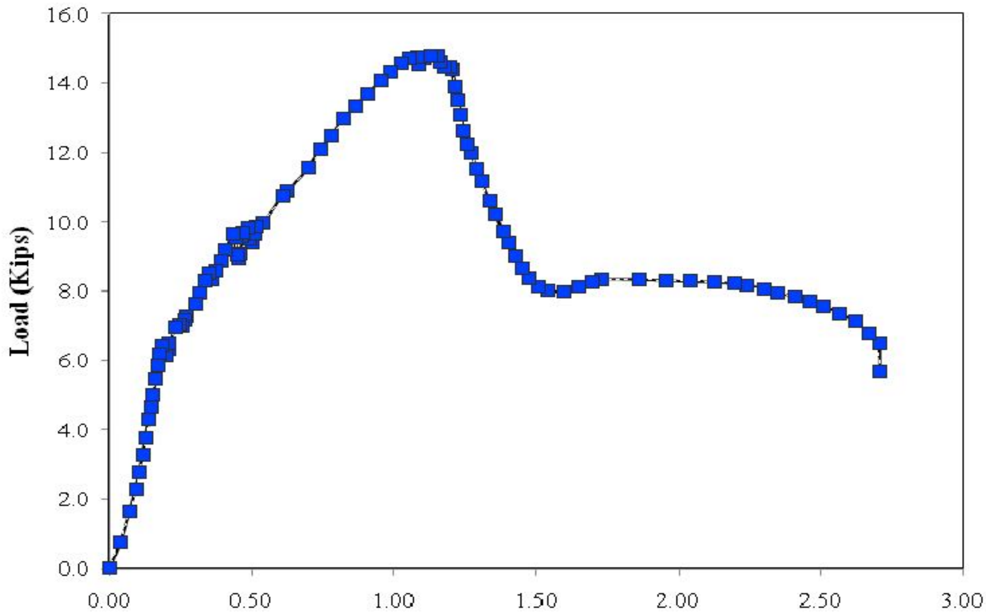

Figure D-2 Autocad Digitization of the UCI data for Case 2

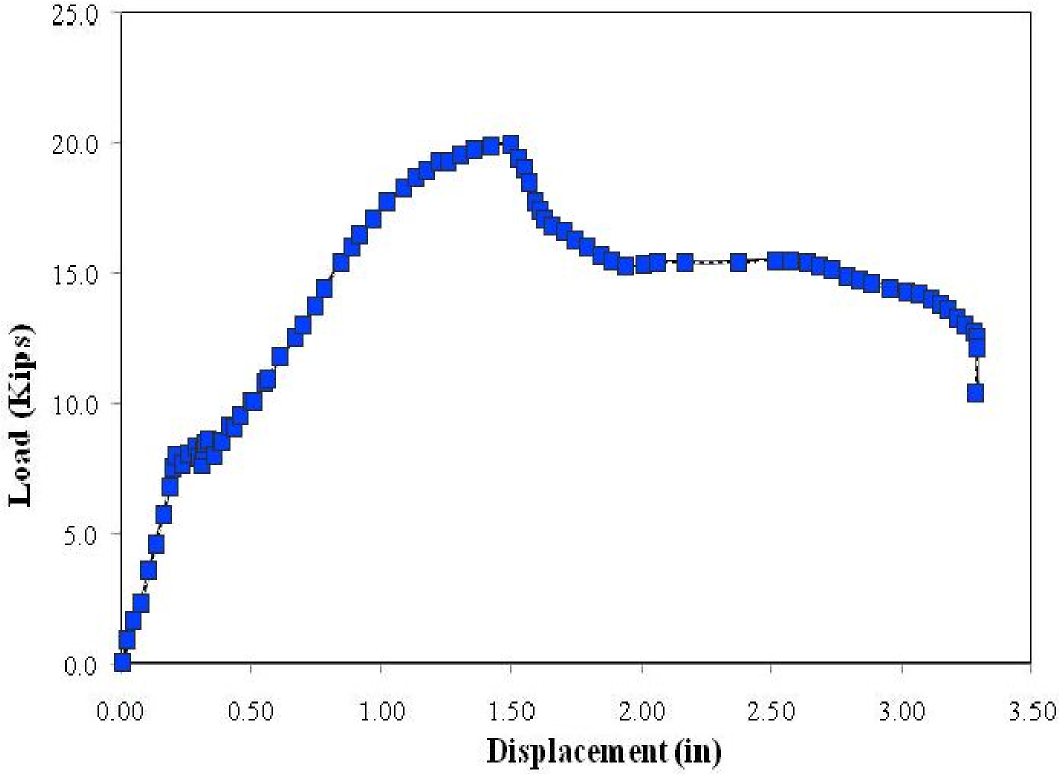

Figure D-3 Autocad Digitization of the UCI data for Case 3

Appendix E - Sample Calculation for Case 1 for Digitization in Ms Excel

S.No.	Coordinates Digitized from AutoCAD		Difference		Transferring to base coordinate (0,0)		Transferring base coordinate to Real Experimental Data	
	X	Y	Xn-Xn-1	Yn-Yn-1	X	Y	Displacement (inch)	Force (kips)
1	84.316	31.881	0.000	0.000	0.000	0.000	0.000	0.000
2	84.899	34.210	0.583	2.329	0.583	2.329	0.010	0.695
3	85.409	37.194	0.510	2.984	1.093	5.312	0.019	1.585
4	86.065	40.541	0.656	3.348	1.749	8.660	0.030	2.584
5	86.502	42.360	0.437	1.819	2.186	10.479	0.038	3.127
6	86.575	43.816	0.073	1.456	2.259	11.935	0.039	3.561
7	86.575	43.816	0.000	0.000	2.259	11.935	0.039	3.561
8	87.595	44.107	1.020	0.291	3.279	12.226	0.056	3.648
9	88.396	46.654	0.802	2.547	4.080	14.773	0.070	4.408
10	89.052	48.619	0.656	1.965	4.736	16.737	0.081	4.994
11	89.562	50.220	0.510	1.601	5.246	18.338	0.090	5.472
12	91.383	48.546	1.822	-1.674	7.067	16.665	0.121	4.973
13	91.797	48.595	0.414	0.049	7.481	16.714	0.128	4.987
14	92.152	50.065	0.355	1.470	7.836	18.184	0.135	5.426
15	93.167	51.332	1.015	1.267	8.851	19.451	0.152	5.804
16	93.751	52.523	0.584	1.191	9.435	20.642	0.162	6.159
17	94.791	51.003	1.040	-1.521	10.475	19.121	0.180	5.706
18	95.552	49.761	0.761	-1.242	11.237	17.880	0.193	5.335
19	95.958	50.420	0.406	0.659	11.643	18.539	0.200	5.532
20	96.136	52.346	0.178	1.926	11.820	20.465	0.203	6.107
21	96.897	53.081	0.761	0.735	12.581	21.200	0.216	6.326
22	97.252	51.079	0.355	-2.002	12.937	19.197	0.222	5.728
23	97.633	50.445	0.381	-0.634	13.317	18.564	0.229	5.539
24	98.090	52.320	0.457	1.875	13.774	20.439	0.236	6.099
25	99.590	54.525	1.500	2.205	15.274	22.644	0.262	6.757
26	101.197	56.112	1.607	1.587	16.881	24.231	0.290	7.230
27	102.393	56.629	1.196	0.517	18.077	24.748	0.310	7.385
28	103.660	55.791	1.268	-0.838	19.344	23.910	0.332	7.135
29	104.124	55.844	0.464	0.053	19.809	23.963	0.340	7.150
30	105.106	56.932	0.982	1.088	20.790	25.051	0.357	7.475
31	107.485	58.850	2.379	1.918	23.170	26.968	0.398	8.047
32	108.646	59.171	1.160	0.321	24.330	27.289	0.418	8.143
33	109.806	60.026	1.160	0.856	25.490	28.145	0.438	8.398

34	112.145	57.762	2.339	-2.264	27.829	25.881	0.478	7.723
35	112.716	57.851	0.571	0.089	28.400	25.970	0.488	7.749
36	113.894	60.240	1.178	2.389	29.578	28.359	0.508	8.462
37	114.376	60.615	0.482	0.374	30.060	28.734	0.516	8.574
38	114.653	60.444	0.277	-0.170	30.338	28.563	0.521	8.523
39	116.242	61.425	1.589	0.981	31.926	29.544	0.548	8.816
40	117.188	61.799	0.946	0.374	32.872	29.918	0.564	8.927
41	118.331	62.085	1.143	0.285	34.015	30.204	0.584	9.013
42	119.509	62.441	1.178	0.357	35.193	30.560	0.604	9.119
43	120.723	62.691	1.214	0.250	36.407	30.810	0.625	9.193
44	121.491	61.407	0.768	-1.284	37.175	29.526	0.638	8.810
45	121.901	61.924	0.411	0.517	37.585	30.043	0.645	8.965
46	121.937	62.530	0.036	0.606	37.621	30.649	0.646	9.146
47	122.580	62.816	0.643	0.285	38.264	30.935	0.657	9.231
48	123.526	62.958	0.946	0.143	39.210	31.077	0.673	9.273
49	124.079	62.281	0.553	-0.678	39.763	30.400	0.683	9.071
50	125.222	62.245	1.143	-0.036	40.906	30.364	0.702	9.060
51	126.814	60.473	1.593	-1.772	42.498	28.592	0.730	8.532
52	127.903	59.118	1.089	-1.355	43.587	27.237	0.748	8.127
53	129.046	57.799	1.143	-1.320	44.730	25.918	0.768	7.734
54	129.688	56.925	0.643	-0.874	45.372	25.044	0.779	7.473

Conversion Scale

1inch =	58.2546	Division
1 Kips =	3.35126	Division

Note: Conversion scale may depend on the size of image file so it may differ. However, the actual experimental data should be same.

Appendix F - Results of Experimental Data

Case 1		Case 2		Case 3	
Displacement (inch)	**Force (kips)**	**Displacement (inch)**	**Force (kips)**	**Displacement (inch)**	**Force (kips)**
0.000	0.000	0.000	0.000	0.000	0.000
0.010	0.695	0.036	0.751	0.022	0.911
0.019	1.585	0.068	1.630	0.045	1.608
0.030	2.584	0.091	2.278	0.074	2.304
0.038	3.127	0.104	2.749	0.105	3.537
0.039	3.561	0.116	3.277	0.130	4.555
0.039	3.561	0.124	3.769	0.160	5.691
0.056	3.648	0.136	4.288	0.185	6.763
0.070	4.408	0.144	4.648	0.198	7.549
0.081	4.994	0.151	4.990	0.209	7.977
0.090	5.472	0.158	5.469	0.234	7.656
0.121	4.973	0.167	5.859	0.256	8.049
0.128	4.987	0.173	6.159	0.287	8.317
0.135	5.426	0.183	6.436	0.295	7.888
0.152	5.804	0.197	6.147	0.309	7.656
0.162	6.159	0.206	6.298	0.317	8.174
0.180	5.706	0.206	6.484	0.318	8.442
0.193	5.335	0.231	6.946	0.329	8.567
0.200	5.532	0.245	7.012	0.353	7.995
0.203	6.107	0.255	6.988	0.376	8.531
0.216	6.326	0.262	7.150	0.384	8.549
0.222	5.728	0.270	7.276	0.416	9.102
0.229	5.539	0.304	7.630	0.433	9.067
0.236	6.099	0.318	7.939	0.453	9.531
0.262	6.757	0.335	8.317	0.497	10.031
0.290	7.230	0.350	8.503	0.508	10.049
0.310	7.385	0.359	8.353	0.547	10.799
0.332	7.135	0.370	8.599	0.563	10.906
0.340	7.150	0.394	8.854	0.610	11.800
0.357	7.475	0.406	9.178	0.665	12.574
0.398	8.047	0.433	9.634	0.696	13.038
0.418	8.143	0.444	9.544	0.742	13.735
0.438	8.398	0.446	9.052	0.780	14.396
0.478	7.723	0.452	8.932	0.842	15.418
0.488	7.749	0.459	9.082	0.886	16.044

0.508	8.462	0.465	9.562	0.914	16.526
0.516	8.574	0.469	9.682	0.966	17.115
0.521	8.523	0.488	9.844	1.023	17.746
0.548	8.816	0.492	9.496	1.082	18.318
0.564	8.927	0.499	9.412	1.129	18.711
0.584	9.013	0.508	9.664	1.172	18.997
0.604	9.119	0.515	9.874	1.219	19.283
0.625	9.193	0.539	9.961	1.253	19.300
0.638	8.810	0.608	10.759	1.300	19.604
0.645	8.965	0.622	10.879	1.358	19.783
0.646	9.146	0.699	11.579	1.418	19.890
0.657	9.231	0.744	12.089	1.495	20.015
0.673	9.273	0.778	12.491	1.528	19.479
0.683	9.071	0.824	12.968	1.553	19.050
0.702	9.060	0.866	13.352	1.570	18.479
0.730	8.532	0.906	13.700	1.590	17.800
0.748	8.127	0.956	14.073	1.608	17.425
0.768	7.734	0.990	14.331	1.625	17.121
0.779	7.473	1.027	14.573	1.658	16.853
		1.055	14.741	1.702	16.603
		1.082	14.753	1.744	16.318
		1.088	14.561	1.793	16.014
		1.108	14.711	1.843	15.675
		1.129	14.783	1.884	15.460
		1.155	14.795	1.939	15.282
		1.163	14.615	2.009	15.335
		1.178	14.459	2.064	15.433
		1.195	14.489	2.170	15.433
		1.204	14.393	2.372	15.446
		1.215	13.925	2.515	15.483
		1.226	13.532	2.575	15.508
		1.234	13.105	2.640	15.395
		1.245	12.613	2.687	15.294
		1.257	12.229	2.734	15.131
		1.271	11.988	2.792	14.892
		1.291	11.528	2.841	14.754
		1.308	11.159	2.888	14.603
		1.338	10.616	2.958	14.440
		1.356	10.223	3.021	14.314
		1.384	9.736	3.071	14.188
		1.403	9.386	3.114	13.997

		1.428	9.010	3.151	13.796
		1.451	8.668	3.178	13.646
		1.477	8.371	3.214	13.294
		1.508	8.123	3.246	13.030
		1.542	8.003	3.278	12.753
		1.597	7.995	3.291	12.565
		1.650	8.123	3.291	12.175
		1.699	8.259	3.289	10.412
		1.730	8.328		
		1.865	8.338		
		1.959	8.304		
		2.045	8.295		
		2.128	8.278		
		2.198	8.236		
		2.240	8.176		
		2.302	8.065		
		2.352	7.962		
		2.412	7.843		
		2.466	7.689		
		2.514	7.552		
		2.567	7.355		
		2.624	7.133		
		2.676	6.774		
		2.710	6.501		
		2.712	5.691		

Appendix G - Simplified Analysis for Case 2

Concrete Strength: f'c = 10 MPa gives γ=0.85 and α=0.85 as per AS3600

Steel Strength: fsy= 413.68 MPa gives ε_{sy} = 0.00206

Area of each steel (A_s) = $\dfrac{\pi * 3^2}{4}$ = 7.068 mm^2

Total Area of steel (A_{ts}) =25*A_s = 176.7 mm^2

Area of Longitudinal Reinforcement steel (A_{s1}) = $\dfrac{\pi * 9.53^2}{4}$

= 71.33 mm^2

Total Area of Longitudinal Reinforcement steel (A_{ts1}) =3*A_{s1}

= 213.99 mm^2

Total Steel Area (A_s) = A_s + A_{ts1} = 390.69 mm^2

Modulus of elasticity of steel (E_s) = 2 X 10^5 MPa

We assume initially that the compressive steel is not at yield before M_u is reached but that the tensile steel is at yield. The strain in the extreme compressive fibre at M_u is ε_{cu}=0.003. Then:

Tensile steel force T = 390.69 X 413.68 X 10^{-3} = 161.62 kN

Compressive steel force $C_s = \varepsilon_{sc} E_s A_{sc}$ = 200 X 10^3 X 176.7 ε_{sc} X 10^{-3} kN and by similar triangles $\varepsilon_{sc} = \frac{0.003}{d_n} * (d_n - 19.05)$

$$Cs = \frac{106.02}{d_n} * (d_n - 19.05) \text{ kN}$$

Concrete compressive force:

$$Cc = \gamma\, d_n\, b\, \alpha\, f'c = 0.85\, d_n \text{ X } 1219.2 \text{ X } 0.85 \text{ X } 10 \text{ X } 10^{-3} \text{ kN}$$

$$= 8.808\, d_n \text{ kN}$$

Force equilibrium ($\sum H = 0$) requires that Cc + Cs – T = 0 and multiplying by d_n:

$8.808\, d_n^2 - 55.6 d_n - 2019.681 = 0$

Solving the quadratic gives dn = 18.624 mm which means that the top steel lies below the neutral axis and it is also in tension.

Rewriting Equation 4.1,

Cc-Cs-T= 0

Top Steel force $C_s = \varepsilon_{st} E_s A_{sc} = 200 \text{ X } 10^3 \text{ X } 176.7\, \varepsilon_{st} \text{ X } 10^{-3}$ kN

and by similar triangles $\varepsilon_{st} = \frac{0.003}{d_n} * (19.05 - d_n)$

$$Cs = \frac{106.02}{d_n} * (19.05 - d_n) \text{ kN}$$

Other terms remains the same and force equilibrium ($\sum H = 0$) requires that Cc - Cs – T = 0 and multiplying by d_n:

$8.808\ d_n^2 - 55.6 d_n - 2019.681 = 0$

Numerically it is same as above; hence it will give us the same answer. However it will make the difference during calculation of Mu.

ku = 18.624/196.85 = 0.0946

By observation the strain in the tensile steel is greater than yield (ku =0.043). The strain in the top steel is :

$$\varepsilon_{st} = \frac{0.003}{18.624} * (19.05 - 18.624) = 0.0000686 \quad (< \varepsilon_{sy})$$

As the strain in the top reinforcement is less than the yield strain, the assumption is correct and, therefore, the calculation for the neutral axis depth is also correct.

The forces are:

Cc = 8.808 X 18.624 = 164.04 kN

Cs = $0.0000686 \text{X } 2 \text{ X } 10^5 \text{ X } 176.7 = 2.424$ kN

and as a check on our calculations:

Cc =Cs+ T = 2.424 + 161.62 = 164.044 kN ($\therefore$ O .K.)

With the forces calculated, Mu is obtained by

Mu = 2.424* (19.05 – 18.624/2) + 161.62 * (196.85 – 18.624/2)

= 2.424 * 9.738 + 161.62 * 187.538

= 30333.496 kN mm

The load that produces the ultimate bending moment Mu can be calculated from its equivalent bending moment via the bending moment diagram shown in Figure G-1.

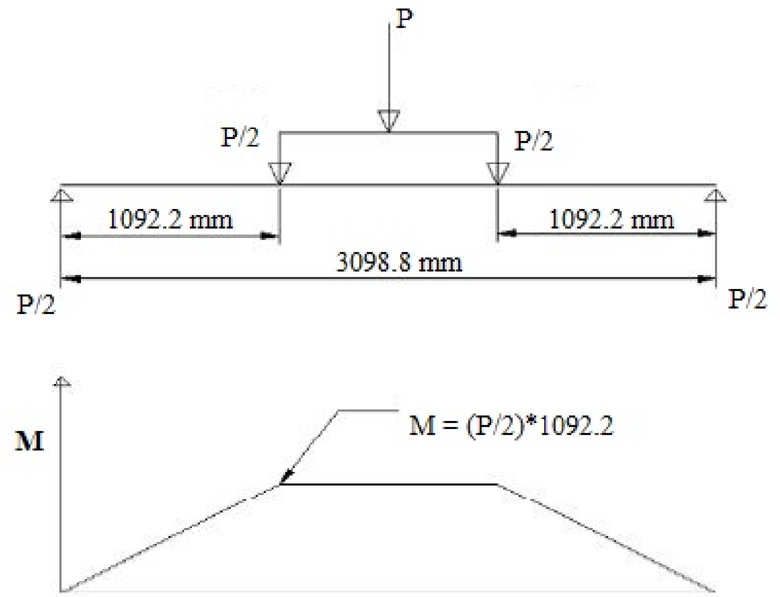

Figure G-1 Derivation of maximum moment from loading setup

Maximum Bending moment is:

M= (P/2)*1092.2

Rewriting the equation,

Ultimate Load (P) = Mu / 546.1

= 30333.496/ 546.1

= 55.545 kN

Appendix H - Simplified Analysis for Case 3

Concrete Strength: f'_c = 10 MPa gives γ=0.85 and α=0.85 as per AS3600

Steel Strength: f_{sy}= 413.68 MPa gives ε_{sy} = 0.00206

Area of each steel (A_s) = $\frac{\pi * 3^2}{4}$ = 7.068 mm^2

Total Area of steel (A_{ts}) =25*A_s = 176.7 mm^2

Area of Longitudinal Reinforcement steel (A_{s1}) = $\frac{\pi * 12.7^2}{4}$

= 126.676 mm^2

Total Area of Longitudinal Reinforcement steel (A_{ts1}) =3*A_{s1}

= 380.03 mm^2

Total Steel Area (A_s) = A_s + A_{ts1} = 556.73 mm^2

Modulus of elasticity of steel (E_s) = 2 X 10^5 MPa

We assume initially that the compressive steel is not at yield before Mu is reached but that the tensile steel is at yield. The strain in the extreme compressive fibre at Mu is ε_{cu}=0.003. Then:

Tensile steel force T = 556.73 X 413.68 X 10^{-3} = 230.308 kN

Compressive steel force $C_s = \varepsilon_{sc} E_s A_{sc}$ = 200 X 10^3 X 176.7 ε_{sc} X 10^{-3} kN

and by similar triangles $\varepsilon_{sc} = \frac{0.003}{d_n} * (d_n - 19.05)$

$$Cs = \frac{106.02}{d_n} * (d_n - 19.05)\,\text{kN}$$

Concrete compressive force:

$$Cc = \gamma\, d_n\, b\, \alpha\, f'c = 0.85\, d_n \text{ X } 1219.2 \text{ X } 0.85 \text{ X } 10 \text{ X } 10^{-3} \text{ kN}$$

$$= 8.808\, d_n \text{ kN}$$

Force equilibrium ($\sum H = 0$) requires that Cc + Cs – T = 0 and multiplying by d_n:

$8.808\ d_n^2 - 124.108 d_n - 2019.681 = 0$

Solving the quadratic gives dn = 23.746 mm

ku = 23.746/196.85 = 0.1206

By observation the strain in the tensile steel is greater than yield (ku =1206). The strain in the top steel is :

$$\varepsilon_{st} = \frac{0.003}{23.746} * (23.746 - 19.05) = 0.000593 \quad (< \varepsilon_{sy})$$

As the strain in the top reinforcement is less than the yield strain the assumption is correct and, therefore, the calculation for the neutral axis depth is also correct.

The forces are:

Cc = 8.808 X 23.746 = 209.154 kN

Cs = $0.000593 \times 2 \times 10^5 \times 176.7 = 20.956$ kN

and as a check on our calculations:

T=Cc +Cs = 209.154 + 20.956 = 230.11 kN ($\therefore$ O .K.)

With the forces calculated, Mu is obtained by

Mu = 209.154* (196.85 – 23.746/2) + 20.956* (196.85 – 19.05)

= 209.154 * 184.977 + 20.956 * 177.8

= 42414.65 kN mm

The load that produces the ultimate bending moment Mu can be calculated from its equivalent bending moment via the bending moment diagram shown in Figure H-1.

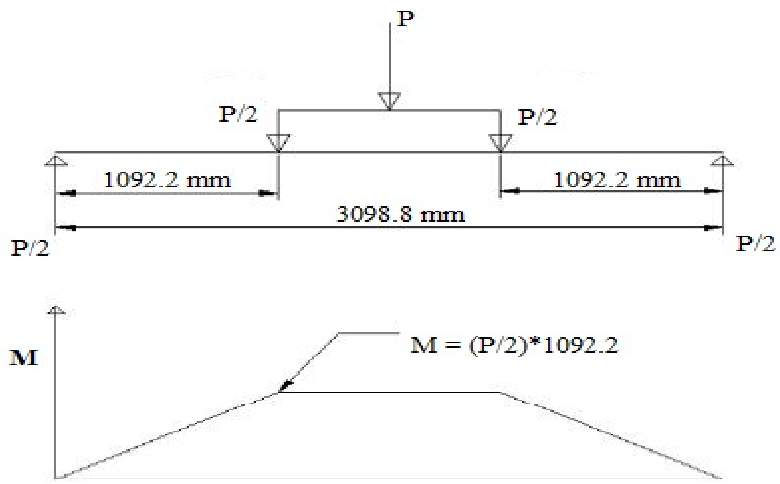

Figure H-1 Derivation of maximum moment from loading setup

Maximum Bending moment is:

M= (P/2)*1092.2

Rewriting the equation,

Ultimate Load (P) = Mu / 546.1

= 42414.65/ 546.1

= 77.668 kN

Appendix I- Reference Values of Imposed floor Actions as per AS1170.1-2002

Type of activity/occupancy for part of the building or structure	Specific uses	Uniformly distributed actions kPa	Concentrated actions kN
A Domestic and residential activities (also see Category C)			
A1 Self-contained dwellings	General areas, private kitchens and laundries in self-contained dwellings	1.5	1.8[1]
	Balconies, and roofs used for floor type activities, in self-contained dwellings— (a) less than 1 m above ground level (b) other	 1.5 2.0	 1.5 kN/m run along edge 1.8[1]
	Stairs[2] and landings in self-contained dwellings	2.0	2.7
	Non-habitable roof spaces in self-contained dwellings	0.5	1.4[8]
A2 Other	General areas, bedrooms, hospital wards, hotel rooms, toilet areas	2.0	1.8[1]
	Communal kitchens	3.0	2.7
	Balconies, and roofs used for floor type activities, with community access	same as areas providing access but not less than 4.0	1.8
B Offices and work areas not covered elsewhere	Operating theatres, X-ray rooms, utility rooms	3.0	4.5
	Work rooms (light industrial) without storage	3.0	3.5
	Offices for general use	3.0	2.7[3]
	Communal kitchens	3.0	2.7
	Commercial/institutional kitchens	5.0	4.5
	Laundries	3.0	4.5
	Laboratories	3.0	4.5
	Factories, workshops and similar buildings (general industrial)	5.0	4.5
	Balconies, and roofs used for floor type activities	same as areas providing access but not less than 4.0	1.8
	Fly galleries (in theatres, etc.)	4.5 kN/m run uniformly distributed over the width	—
	Grids (over the area of proscenium width by stage depth)	2.8	—

Appendix J- Required Cover for Steel reinforcement as per AS3600-2009

Exposure classification	Required cover, mm				
	Characteristic strength (f'_c)				
	20 MPa	25 MPa	32 MPa	40 MPa	≥ 50 MPa
A1	20	20	20	20	20
A2	(50)	30	25	20	20
B1	—	(60)	40	30	25
B2	—	—	(65)	45	35
C1	—	—	—	(70)	50
C2	—	—	—		65

Appendix K- Matlab code used to analyse data

K.1. Matlab code to calculate initial tangent modulus of EPS

```
% This code calculates the initial Tangent modulus of EPS based on two

% reseachers Horvath (1995)  and Duskov (1997)
clear all;
clc;
close all;
%
% Density of EPS which varies from 11 to 30 kg/m3
density = [11:0.01:30];
% Calculation of initial tangent modulus based on Horvath (1995)
Ehorvath=0.45.*density-3;
% Calculation of initial tangent modulus based on Duskov (1997)
Eduskov= 16.431-1.645.*density+ 0.061.*density.^2;
% Plots the data
plot(density,Ehorvath,density,Eduskov);legend('Horvath (1995)','Duskov
(1995)');xlabel('Density(kg/m^3)');ylabel('Initial Tangent Modulus (MPa)'); grid on
```

K.2. Matlab code used to plot experimental results for Case 1

```
% This code plots the experimental data of UCI in SI units for Case 1
clc;
clear;
close all;
% Load the data for Case 1
Data1=load('data1.txt')
% Convert inch to mm and kips to Newton
dis1=Data1(:,1).*25.4
force1=Data1(:,2).*4448.2216
% Plots the data
plot(dis1,force1./1000); grid on
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
```

K.3. Matlab code used to plot experimental results for Case 2

```
% This code plots the experimental data of UCI in SI units for Case 2
clc;
clear;
close all;
% Load the data for Case 2
Data2=load('data2.txt')
% Convert inch to mm and kips to Newton
dis2=Data2(:,1).*25.4
force2=Data2(:,2).*4448.2216
% Plots the data
plot(dis2,force2./1000); grid on
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
```

K.4. Matlab code used to plot experimental results for Case 3

```
% This code plots the experimental data of UCI in SI units for Case 3
clc;
clear;
close all;
% Load the data for Case 3
Data3=load('data3.txt')
% Convert inch to mm and kips to Newton
dis3=Data3(:,1).*25.4
force3=Data3(:,2).*4448.2216
% Plots the data
plot(dis3,force3./1000); grid on
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
```

K.5. Matlab code used to plot the comparison of Case 1

```
% This code plots and compare the experimental and finite element
% method's data for Case 1
clc;
clear;
close all;
% Loads the experimental and finite element method's data
Data1=load('data1.txt')
result1= load('Result1.txt')
% Convert the unit of experimental data into SI units
dis1=Data1(:,1).*25.4
force1=Data1(:,2).*4448.2216
%Plots the data
figure
plot(dis1,force1./1000,result1(:,2),result1(:,1)./1000,'--');
legend('Test data','Model data');
grid on;
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
```

K.6. Matlab code used to plot the comparison of Case 2

```
% This code plots and compare the experimental and finite element
% method's data for Case 2
clc;
clear;
close all;
% Loads the experimental and finite element method's data
Data2=load('data2.txt')
result1= load('Result2.txt')
% Convert the unit of experimental data into SI units
dis2=Data2(:,1).*25.4
force2=Data2(:,2).*4448.2216
%Plots the data
figure
plot(dis2,force2./1000,result1(:,2),result1(:,1)./1000,'--');
legend('Test data','Model data');
grid on;
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
```

K.7. Matlab code used to plot the comparison of Case 3

```
% This code plots and compare the experimental and finite element
% method's data for Case 3
clc;
clear;
close all;
% Loads the experimental and finite element method's data
Data3=load('data3.txt')
result3= load('Result3.txt')
% Convert the unit of experimental data into SI units
dis3=Data3(:,1).*25.4;
force3=Data3(:,2).*4448.2216
%Plots the data
figure
plot(dis3,force3./1000,result3(:,2),result3(:,1)./1000,'--');
legend('Test data','Model data');
grid on;
xlabel(' Displacement(mm)');
ylabel('Force(kN)');
title('Case 3- For longitudinal Reinforcement of 12.7 diameter');
```

K.8. Matlab code used to plot design chart for foam thickness of 50 mm

```
% This code create a design chart for concrete thickness of 50mm and foam
% thickness of 50 mm. The plot will then be edited using editor toolbox.
clear;
clc;
close all;
% Input the load and deflection based on Strand 7
LOAD_DATA=
[0,0,0;31.5,42.875,56;40.5,55.125,72;49.5,67.375,88;58.5,79.625,104;67.5,91.875,120];
DEFLECTION_DATA=
[0,0,0;2.557,4.15,6.08;3.33,5.37,7.869;4.1,6.59,9.66;4.885,7.897,11.46;5.667,9.057,13.2
6];
%This loops plots the data
for i = 2:6
plot(DEFLECTION_DATA(i,:),LOAD_DATA(i,:),'-b'),grid on
hold on
end
for i = 1:3
plot(DEFLECTION_DATA(:,i),LOAD_DATA(:,i),'-r')

hold on
end
% Labels the x axis, y axis and gives title
ylim([0 120])
grid on
ylabel('Total Load (kN) (Including Self Weight)','fontsize',14)
xlabel('Deflection (mm)','fontsize',14)
title ('Load / Deflection Limit Graphs for CEPS Sandwich Panel','fontsize',14)
```

K.9. Matlab code used to plot design chart for foam thickness of 75 mm

```
% This code create a design chart for concrete thickness of 50mm and foam
% thickness of 75 mm. The plot will then be edited using editor toolbox.
clear;
clc;
close all;
% Input the load and deflection based on Strand 7
LOAD_DATA=
[0,0,0;31.5,42.875,56;40.5,55.125,72;49.5,67.375,88;58.5,79.625,104;67.5,91.875,120]
DEFLECTION_DATA=
[0,0,0;2.123,3.466,5.111;2.759,4.485,6.614;3.395,5.504,8.118;4.037,6.569,9.626;4.678,
7.557,11.135]
%This loops plots the data
for i = 2:6
plot(DEFLECTION_DATA(i,:),LOAD_DATA(i,:),'-b'),grid on
hold on
end

for i = 1:3
plot(DEFLECTION_DATA(:,i),LOAD_DATA(:,i),'-r')
hold on
end
% Labels the x axis, y axis and gives title
ylim([0 120])
grid on
ylabel('Total Load (kN) (Including Self Weight)','fontsize',14)
xlabel('Deflection (mm)','fontsize',14)
title ('Load / Deflection Limit Graphs for CEPS Sandwich Panel','fontsize',14)
```

K.10. Matlab code used to plot design chart for foam thickness of 100 mm

```
% This code create a design chart for concrete thickness of 50mm and foam
% thickness of 100 mm. The plot will then be edited using editor toolbox.
clear;
clc;
close all;
% Input the load and deflection based on Strand 7
LOAD_DATA=
[0,0,0;31.5,42.875,56;40.5,55.125,72;49.5,67.375,88;58.5,79.625,104;67.5,91.875,120]
DEFLECTION_DATA=
[0,0,0;1.487,2.452,3.656;1.926,3.173,4.729;2.366,3.894,5.802;2.805,4.616,6.876;3.244,
5.337,7.949]
%This loops plots the data
for i = 2:6
plot(DEFLECTION_DATA(i,:),LOAD_DATA(i,:),'-b'),grid on
hold on
end

for i = 1:3
plot(DEFLECTION_DATA(:,i),LOAD_DATA(:,i),'-r')
hold on
end
% Labels the x axis, y axis and gives title
ylim([0 120])
grid on
ylabel('Total Load (kN) (Including Self Weight)','fontsize',14)
xlabel('Deflection (mm)','fontsize',14)
title ('Load / Deflection Limit Graphs for CEPS Sandwich Panel','fontsize',14)
```