THESIS

ON

STUDY ON THE BEHAVIOUR OF REINFORCED CONCRETE SLAB RESTING OVER ELASTIC SUPPORT

Submitted by

SK. AFZAL HOSSAIN

Master of Civil Engineering

(Structural Engineering)

Year-2nd

Class Roll No.-001710402006

Examination Roll No.-M4CIV19011

University Registration No.- 140633 of 2017-2018

Session-2018-2019

in partial fulfilLment of the requirement for the degree of

M.E.

in

CIVIL ENGINEERING

(Specialisation- STRUCTURAL ENGINEERING)

under the guidance of

Dr. SUBHASHISH ROYCHOWDHURY

FACULTY OF ENGINEERING AND TECHNOLOGY

DEPARTMENT OF CIVIL ENGINEERING

(STRUCTURAL ENGINEERING)

JADAVPUR UNIVERSITY

KOLKATA – 700032MAY 2019

ACKNOWLEDGEMENT

The work presented here would not have been possible without the help and support of many people. Firstly, I would like to express my sincerest appreciation to theses adviser, Dr. Subhashish RoyChowdhury, for his guidance, expert supervision, and giving me the opportunity to be involved in such interesting research. I could not have asked for an adviser more approachable and willing to help.

Secondly, I would like to thank my parents and my fellow batch mates for their constant support and encouragement. My sincere thanks to all faculty members of Civil Engineering Department of Jadavpur University for their assistance Lastly, I am grateful to the computer lab technicians for providing me the lab facilities at any hour of the day.

CERTIFICATE OF RECOMMENDATION

This is to certify that, **Sk. Afzal Hossain** (Class Roll No: 001710402006, Examination Roll No: M4CIV19011, Registration No: 140633 of 2017-2018) has carried out the thesis work titled **"Study on the Behaviour of Reinforced Concrete Slab resting over Elastic Support"**, under my supervision and guidance. He has carried out this work independently. I hereby recommend that the theses be accepted in partial fulfilment of the requirements for awarding the degree of **"Master in Civil Engineering" (Specialization – Structural Engineering).**

....

Dr. Subhashish RoyChowdhury

Asst. Professor, Department of Civil Engineering

Jadavpur University, Kolkata- 700032

Countersigned by:

....

Prof. Dr. Chiranjib Bhattacharjee

Dean, Faculty of Engineering and Technology

Jadavpur University, Kolkata-700032

.... ……

Prof. Dr. Dipankar Chakravorty

Head of the Department

Civil Engineering Department

Jadavpur University

Kolkata-700032

CERTIFICATE OF APPROVAL

This foregoing thesis titled **“Study on the Behaviour of Reinforced Concrete Slab resting over Elastic Support”**, submitted by **Sk. Afzal Hossain** is hereby approved as a credible study on an Engineering subject carried out and presented in a manner satisfactory to warrant its acceptance as a prerequisite to the degree of **Master in Civil Engineering (Specialization – Structural Engineering)** for which it has been submitted. It is understood that by this approval the undersigned do not necessarily endorse or approve any statements made, opinions expressed or conclusion drawn therein, but approve the thesis only for the purpose for which it is submitted.

Committee on final examination for evaluation of the thesis

……………………………….

……………………………….

……………………………….

(Signature of Examiners)

DECLARATION

I, **Sk. Afzal Hossain**, a student of **M.E in Civil Engineering (Structural Engineering)**, Jadavpur University, hereby declare the work being presented in the thesis work titled, **“Study on the Behaviour of Reinforced Concrete Slab resting over Elastic Support”**, is an authentic record of work that has been carried out at the **Department of Civil Engineering, Jadavpur University**, under the guidance of **Dr. Subhashish RoyChowdhury**, Asst. Professor, Department of Civil Engineering, Jadavpur University.

The work contained in the thesis has not yet been submitted in part or fully to any other university or institution or professional body for award of any degree or diploma or any fellowship.

Date: ....

Place: Jadavpur University, Kolkata

....

SK. AFZAL HOSSAIN

Class Roll No: 001710402006

Registration No: 140633 of 2017-2018

Examination Roll No: M4CIV19011

Jadavpur University

ABSTRACT

It has been aimed in the present research work to carry out the nonlinear finite element analysis of reinforced concrete slab supported over an elastic medium. The deformational characteristics of the elastic medium such as soil are represented by a series of linear springs similar to Winkler's model. The concrete part of the slab and the reinforcement bars are modelled as three dimensional solid element. The nonlinear material property of both concrete and steel are to be considered. The concrete, in tension, are considered as elastic brittle fracture material and in compression, as elastic-elasto-plastic material. The material behaviour of steel are considered as elastic-fully plastic. The slabs are analysed under the action of gradually increasing set of concentrated and uniformly distributed static loads. Initially the spring constants are considered as equal. Then different pattern of distribution of spring stiffness will be taken into account. The changes in the behaviour of the slab and crack propagation under different types of loading and different pattern of spring constants are studied in detail.

Table of Contents

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

Foundations are used to support the super-structure and also to transmit the structural loads to the soil below. Reinforced concrete foundation slab resting directly on the soil medium is very commonly used for this purpose. Mat foundation or raft foundation is well-known examples of a concrete slab being directly supported by the soil medium. The bottom slabs of storage tanks, swimming pools, silos and other structural systems are also the examples of this kind. A mat foundation is in essence a stiff slab that acts as a single foundation element that covers the entire plan area of the structure.[19] Analyzing thick plates as a construction component has been of interest to structural engineering research for several decades. In particular, thick plates resting on elastic foundations are more specific. [28]

In the last two decades the increase on computer power and software facilities, as well as a significant effort on the experimental research about material constitutive laws, contributed to the development of several computer programs for numerical simulation of concrete structures. However, numerical simulation of a slab supported on soil remains a difficult task. Accurate simulation of the behaviour of this kind of structures is only feasible if the numerical model takes into account the non-linear behaviour of the concrete, soil and reinforcement, as well as the soil-slab interaction. [30]

Soils are not linearly elastic and perfectly plastic for the entire range of loading. Actual behaviour of soil is very complicated and it demonstrates a great variety of behaviour when subjected to different conditions. Different constitutive models have been suggested to describe different aspects of soil behaviour in detail. The simplest type of idealized soil response is to assume the behaviour of supporting soil medium as a linear elastic continuum.

A variety of methods are available for the structural analysis of mat foundations, ranging from simple static analogues to elaborate three-dimensional finite element analysis. The most common method in current design practice is the Winkler spring approach, in which the soil is represented in the analysis as linear vertical springs supporting the mat. [19] The modelling of plates over elastic foundation is based on Winkler hypothesis, in which only the normal interacting forces in the foundation are considered. In this approach the foundation is treated as if it consists of many closely spaced linear springs, and the reactions are directly proportional to the plate deflection at any point. One of the major drawbacks of this model is that a plate when subjected to a uniformly or uniformly varying loads will undergo rigid body displacements without any bending moments or shear forces in the plate, leading to non-conservative design quantities. Actually, the value of *k* depends on the depth of the soil, the geometry of the structure, and even the distribution of the loading. [27]

In the past numerous researchers have studied this issue, which is referred to as "beams and slabs on elastic foundation." Most of the past work started with the Winkler model wherein the vertical deflection of the slab is directly proportional to the contact pressure. The utilization of the Winkler model includes one noteworthy issue and one huge behavioural irregularity. The issue includes the need for deciding the modulus of sub-grade response, "k," and the behavioural irregularity is that the interaction between consecutive springs is not considered. Despite being a rather rough idealization of reality and the emergence of more accurate methods, the Winkler spring approach still constitutes the state-of-practice because it can be easily applied in most commercial structural analysis computer programs.[19]

Many studies have pointed out the shortcomings of the original basic Winkler spring approach, which assumes that the modulus of subgrade reaction has the same value everywhere under the mat, and have proposed alternative methods. One of these alternatives is the pseudo-coupled approach, in which the mat still rests on vertical springs, but with spring constants that vary across the mat depending on the location of a given spring, taking larger values near the edge of the foundation compared to its center. The pseudo-coupled approach is meant to improve the accuracy of the original Winkler spring method while retaining its simplicity. [19] For example, to predict the correct variation of settlement across a perfectly flexible (zero rigidity) foundation subjected to a uniform distributed load, the k value assigned at the corners of the foundation needs to be twice as much the value assigned at the center. [19] Bowles (1988) and Coduto (1994) have indicated that the value of k has to be augmented on the boundaries of the plate in order to get a satisfactory solution, but they have not specified how much increase is necessary. [27]

Based on the literature reviewed, it is found that many literatures are reported on nonlinear behavior of soil and concrete, and considerable amount of literatures are reported on the three-dimensional nonlinear analysis of plate on elastic foundation, but very few studies adopted failure of reinforced concrete raft slab supported over spring support. Thus there is a need to investigate the behavioural study of reinforced concrete slab supported over elastic medium considering nonlinear properties of concrete, steel as well as failure criterion for them.

In the present research work, it has been aimed to analyse the reinforced concrete slab, using finite element analysis, supported over linear independent springs. As mentioned above, initially uniform spring constants, like basic Winkler's model, are considered to get the deformational behavior of reinforced concrete slab and to find the propagation of cracks within it before failure. Then, the variation of stiffness of springs, as suggested in the above mentioned literatures, are considered to assess the deformational response and cracking profile for the slab.

ABAQUS, a well-known finite element software package has been utilized here to prepare the models and to analyse them. 8 noded linear brick elements are chosen to prepare the models of the slabs. Both concrete and steel are modellled separately. The elastic as well as elastoplastic material properties are assigned for concrete and steel. Concrete Damage Plasticicty model has been adopted to idealize the post-yield behavior of concrete. Maximum principal stress criterion has been adopted to simulate the development of cracks and propagation of that. Number of case studies are undertaken to assess the applicability of the proposed approach and to investigate the effect of different parameters on the overall behaviours of the slab. In all problems, the deformational response, stress contours, crack propagation (if any) are represented here.

2. LITERATURE REVIEW

The effect of elastic medium like soil, on the structural behaviour can be taken into consideration in different ways or approaches. These different approaches are as follows:

1. Structural element and soil medium can be considered as a separate volume in the finite element. The volume representing the soil mass as suggested by previous researchers can be considered as bounded by the rigid support. In this case the soil volume and the structure volume will be considered connected through their common nodes.

2. The structure and the soil surface are considered as continuously connected between each other using some contact element.

3. Thirdly, the interaction between the soil and the structural component can be simulated by modelling the structure using Finite Element technique and the soil as boundary element technique. This is also a very useful approach as three dimensional analysis of the soil volume can be avoided.

4. The classical solution strategy can also be applied for the analytical solution of this problem of slab on elastic foundation.

5. In the fifth approach, the soil medium is be replaced by a series of springs distributed in the horizontal plane representing the deformational behaviour of the soil and the structural component (slab) is assumed to be supported over these group of springs. Winkler foundation approach is one common approach under this category.

2.1 Literature Review on the First Approach

- An analytical study was undertaken by **Rifat Bulut [1]** to develop an improved analysis method for calculating the performance of slabs on expansive soil. A Finite element method formulation of slabs on elastic continuum foundations was developed to analyze this complex soil-structure system.

- In the study conducted by **Fattah, Hamood and Abbas[2]**, nonlinear three-dimensional finite element analysis has been used to conduct a numerical investigation of the effect of applied impact load on the foundation based on sandy soil using the finite element method by ANSYS. The interface between the soil and the foundation is modelled using three dimensional surface to surface contact elements. A parametric study was carried out to investigate the effect of several parameters including: foundation dimensions (geometry) and amplitude of impact load. It was concluded that as the foundation thickness increases, the time for maximum displacement to take place increases due to geometrical damping induced by the foundation. When the length of foundation increases, the oscillation of vertical displacement decreases which means that the foundation becomes more stable.

- The work of **Shakir and Abbas [3]** confirmed that as the modulus of elasticity of soil decreases the maximum vertical deflection at the centre of the foundation increases, decrease the maximum vertical stress depending the amplitude of the load. The time at which the maximum vertical displacement takes place increases as the soil modulus of elasticity increases.

- In the study of **Hamood, Abbas and Fattah[4]**, a nonlinear three-dimensional finite element analysis has been used to conduct a numerical investigation of the effect of applied impact load on the foundation based on sandy soil using the finite element method by ANSYS. A parametric study is carried out to investigate the effect of several parameters including: foundation thickness, load eccentricity and amplitude of impact load. It was concluded that the load eccentricity increases the displacement and decreases the stress at the foundation center. This is attributed to non-uniformly distributed stresses on the loaded area, where the loads are concentrated locally within the loaded area. The presence of damping leads to a considerable decrease in the foundation displacements and stresses. The increase in the damping ratio reduces the vertical displacement of the foundation at the same time at all damping ratios.

2.2 Literature Review on the Second Approach

- In their work, **Malekova and Jendzelovsky [5]**, have studied the interaction between the foundation and the subsoil using contact elements. The contact elements have certain modifiable properties which influence the behaviour of the mentioned contact elements. The behaviour of the contact elements in turn affects the foundation and the subsoil. Three ways of solution of contacts between a concrete foot foundation and gravel subsoil were used. The first way was fixed connection, the second is the application of contact element named CONTAC52 and the third being the application of CONTA173 (TARGE170). The first way of contact was the easiest to apply but matched the least to the actual behaviour. CONTAC52 was shown to have some advantages like the possibility of determining parameters like normal or tangential stiffness, coefficient of friction etc. The disadvantage of this way of contact was that it could be defined only in points so it does not take into account results obtained outside these points. CONTA173 was shown to be the most advantageous from modelling and definition point of view. This surface contact element transferred pressing and tension forces and accounted for the interaction of the whole surface and not only in points.

- Different values of soil sub-grade reaction with variable slab thickness and stiffness modifier were parametrically studied by **Risan[6].** The variation affect of soil sub-grade reaction values on natural frequency of the concrete slab with free boundary conditions had a negligible effect for each mode shape. While the increasing of slab thickness, rested on soil with free boundary condition, resulted a decreasing in the natural frequency for the first three mode shapes and an increasing in other three mode shapes. Finally, for the first three mode shapes of concrete free edges slab-soil system, there is no any change in natural frequency as a function of the stiffness modifier due to rigid body concept. In other mode shapes, a reduction in stiffness modifier value resulted in a reduction in natural frequency of concrete slab.

2.3 Literature Review on the Third Approach

- The paper by **Paiva and Mendonca [7]** presents a Boundary Element Method formulation for the static analysis of piled rafts in which all the interactions between the plate, the pile and the soil are simultaneously considered. In this approach the soil is treated as an elastic linear homogeneous half space, the plate is assumed to be thin and both are represented by integral equations. Each pile is represented by a single element and the shear force along it is approximated by a second-degree polynomial. The pile-tip stress is assumed to be constant over the cross-section. The cap–soil interface is divided into triangular elements and the contact pressure is assumed to vary linearly across each element. The vertical displacement of each node in the plate and in the piles is represented by an integral equation so that a set of linear equations is obtained involving the tractions and displacements at all nodal points on all the interfaces.

- In the work of **Rashed [8]**, a new boundary/domain element method is developed to analyse plates resting on elastic foundations. The developed formulation is then used in analysing building raft foundations. For more practical representation, the considered raft plate is treated as thick plate with free edge boundary conditions. The soil or the elastic foundation is represented as continuous media (follows the Winkler assumption). The boundary element method is employed to model the raft plate; whereas the soil is modelled using constant domain cells or both the domain and the boundary of the raft plate are discretized. The main advantage of the present formulation is the ability of analysing rafts on non-homogenous soils.

- The objective of the paper by **Ribeiro and Paiva [9]** is to present formulations developed for soil-building interaction analysis, including foundations. The soil is modelled with the boundary element method (BEM) as a layered solid which may be finite for the vertical direction, but is always infinite for radial directions. Beams, columns and piles are modelled with the finite element method (FEM) using one dimensional elements. Slabs and rafts are also modelled with the FEM, but with two dimensional elements. The analysis is static and all materials are considered homogeneous, isotropic, elastic and with linear behaviour.

2.4 Literature Review on the Fourth Approach

- In the book titled, "Beams, Plates and Shells on Elastic Foundation," a new theory for analyzing structures on elastic foundations, based on **Vlasov's[10]** variation method, has been proposed. This theory is more accurate than the well-known theory of Winkler and Zimmermann, but is simpler than the theory of elastic semi-infinite space. This theory considers the elastic foundation as a single or double layer model whose properties are described by two or more generalized elastic characteristics. The theory of the single layer foundation was further developed by **Leon'tev [10]**. A merit of the theory proposed here is that the solution of many problems of practical importance is reduced to solving ordinary differential equations whose integrals can be found from tables. The simplicity of the mathematical problem and the clearness of the mathematical model make this theory very adaptable; not only the problem of beams and plates on elastic foundation, but also more various complex problems can be solved with its aid. These problems include the analysis of shells, taking into consideration additional transverse loads and the deformation of the underlying foundation, and problems of the dynamics and stability of structures on elastic foundations. The proposed theory can be applied to the determination of the stresses and strains in single and multi-layer strata in horizontal and inclined excavations.

- **Pavlou, Dan, Belc, Lucaci [11]** presented an exact solution of an infinite plate on elastic foundation. The formulation is based on application of Laplace and Hankel integral transforms and Bessel functions' properties. Representative examples are studied and the obtained solutions are discussed. They concluded that this solution can be used as a Green's function in order to solve boundary-value problems of finite circular or annular plates on elastic foundation under impact axi-symmetric loads. Some real examples ware solved indicating the wave propagation for several values of the time.

- An analytical formulation using the principle of minimum potential energy is presented by **Kukreti and Ko [12]** to predict the flexural behaviour of a rectangular plate resting in smooth contact with an elastic half space (soil medium) and subjected to a uniformly distributed load. The procedure accounts for interaction between the plate and the soil medium. Compatibility at the interface of the plate und soil medium is satisfied by integrating the Boussinesq's formula, which relates the contact stress and the soil surface deformation. Analytical formulations for the following two approximations used to model the contact stress distributions are presented: u power series expansion and use of Chebychev polynomials. In both the formulations the integrations over the plate domain are analytically derived by dividing the plate surface urea into eight triangular zones and evaluating explicitly the integrals over each zone and summing the results. First, the boundary conditions at the free edges of the plate are satisfied by expressing some of the selected generalized coordinates appearing in the assumed function in terms of the other, and then the total energy of the system is minimized to evaluate the unknown independent generalized coordinates. The process of selecting these generalized coordinates to satisfy the boundary conditions is automated. Results obtained for a square plate are compared with similar results reported in the literature and with those obtained from three-dimensional finite element analyses. Results of a parametric study investigating the effect of the relative stiffness of the plate with respect to the elastic half space are also presented.

- Analytic bending solutions of free rectangular thin plates resting on elastic foundations, based on the Winkler model, are obtained by a new symplectic superposition method in the work of **Zhong and Li [13]**. The proposed method offers a rational elegant approach to solve the problem analytically, which was believed to be difficult to attain. By way of a rigorous but simple derivation, the governing differential equations for rectangular thin plates on elastic foundations are transferred into Hamilton canonical equations. The symplectic geometry method is then introduced to obtain analytic solutions of the plates with all edges slidingly supported, followed by the application of superposition, which yields the resultant solutions of the plates with all edges free on elastic foundations. The proposed method is capable of solving plates on elastic foundations with any other combinations of boundary conditions. Comprehensive numerical results validate the solutions by comparison with those obtained by the finite element method. The solution approach reveals several advantages with respect to bending problems of rectangular plates on elastic foundations. First, the symplectic superposition method provides a totally rational way to obtain analytic solutions, which starts from the basic elasticity equations of the problem and proceeds without any pre-selected solutions. The second advantage is that the method gives us a systematic solution procedure, which can be applied to plates with all possible combinations of clamped, free and simply supported boundary conditions. In addition, the method is expected to be extended to vibration and buckling problems.

2.5 Literature Review on the Fifth Approach

- **Kazakov and Karamanski [14]** have proposed a modified model of a foundation plate based on Winkler's hypothesis. The "modified Winkler's model" takes into account the action of the soil mass beyond the contour of the foundation plate via the application of an additional stiffness along its contour. The formula for the evaluation of the Winkler's constant for this additional stiffness is derived. The verification of the proposed method is performed on the structure of a residential building founded on a foundation plate. The results for the vertical deflection and the bending moments in the foundation plate using the "modified Winkler's model" are close to the results when they used "elastic media" model.

- In order to overcome the shortcomings of the Winkler model, **Musat, Vrabie and Teodoru [15]** have considered the problem of a beam resting on a two-parameter elastic foundation. In order to analyse the bending behaviour of an Euler-Bernoulli beam resting on a two-parameter elastic foundation a finite element model based on the cubic displacement function of the governing differential equation is introduced. The resulting effect of shear stiffness of the Pasternak model on the mechanical properties is discussed in comparison with the Winkler model. The main conclusions of the paper is that when the length-to-height ratio of the beam foundation is less than 4 the beam can be analysed as if it rests on Winkler's foundation and the Pasternak model is more adequate for rocky or gravelly soils.

- **Yavas and Civalek [16]** applied Discrete Singular Convolution to solid mechanics. Geometrically nonlinear static analysis of thin rectangular plates on Winkler-Pasternak Elastic foundation has been studied. The nonlinear partial differential equations obtained from Von Karman's large deflection plate theory have been solved by using the DSC method. The effects of Winkler and Pasternak foundation parameters on the displacements have been investigated. To the authors' knowledge, it is the first time the DSC method has been successfully applied to rectangular plates resting on two parameter elastic foundation problem for the geometrically nonlinear static analysis. It was observed that the parameter K and G of the Winkler and Pasternak foundation have a significant influence on the displacements of the plates. Consequently, by comparing the computed results with those available in published works, the present analysis by the DSC method is examined and a very good agreement was observed.

- Rectangular plates on distributed elastic foundations are widely employed in footings and raft foundations of variety of structures. In particular, mounted columns and single footings may partially occupy the rectangular plate of any kind. The study of **Akin, Mofid and Motaghian [17]** deals with free vibration problem of thin rectangular plates on Winkler and Pasternak elastic foundation model which is distributed over a particular arbitrary area of the plate. Closed form solutions are developed through solving the governing differential equations of plates. Moreover, a novel mathematical approach is proposed to find the exact analytical solution of free vibration of plates with mixed or fully-clamped boundary conditions. Based on the parametric studies provided in this study, it could be stated that the proposed method successfully calculate natural frequencies of rectangular, partially on two-parameter elastic foundation. An application of the problem studied herein can be utilized for mounted columns and single footings that partially occupy thin rectangular slabs. It should be noted that the problems of forced vibration of plates and vibration of Reissener–Mindlin plates can also be addressed using the same techniques discussed in this paper.

- **Rajpurohit and Sayagavi [18]** presented an analysis of beams, columns and raft, in a multi-storied building structure, supported by elastic foundation. The structure is analyzed using E-Tabs and SAFE software for three different values of modulus of sub-grade reaction 'K' pertaining to different soil types, and it has been compared with the structure having fixed supports representing rigid base. The analysis highlights the fact that significant alteration of displacements, design forces and moments occur in the beams, columns and raft. The analysis also brings out the fact that settlement in a raft foundation depends on the stiffness of the soil. The settlement of raft at different values of modulus of sub=grade reactions were analysed and compare with rigid support raft.
- Structural design of mat foundations of buildings is often done by performing static analysis of a slab resting on vertical uncoupled Winkler springs. It is already well established that the simplifying assumption of a uniform modulus of sub-grade reaction throughout the mat foundation leads to inaccurate results that significantly

underestimate the bending moments in the mat. The paper by **Tamiolakis and Loukidis [19]** examines the spatial variation of the Winkler spring stiffness constants that is necessary for the mat-on-springs analysis to produce the same slab deflections and bending moment diagrams as finite element analysis that treats the soil as continuum. For this purpose, three-dimensional parametric analyses of slabs resting on elastic soil are performed using the finite element method for various values of soil elastic properties, slab geometrical characteristics and column load configurations. The finite element analysis results were used for back-calculating analytically the equivalent Winkler spring constants at each node of the mat. Based on the numerical results, equations describing the spatial distribution of spring stiffness are proposed. The performance of the proposed equations is compared against existing spring stiffness spatial distribution approaches used in practice.

- The behaviour of the soil is represented by fictitious springs in the Winkler foundation. **Daloglu**and **Vallabhan[20]** have developed a method to evaluate the equivalent value of the spring stiffness to be used in the Winkler model. They have analysed a slab subjected to concentrated load and uniformly distributed loads. They have calculated the spring stiffness using a Vlasov model and used it in a Winkler model. This step is repeated till the difference in the maximum central deflection of the two models is negligible. This is how the modulus of sub-grade reaction for the Winkler model has been evaluated. They have suggested that in order to get realistic results for a slab subjected to uniformly distributed load, higher values of spring stiffness should be used around the edges.
- **Dey, Basudhar** and **Chandra[21]** have studied the variation in sub-grade modulus of a footing resting on compacted granular bed due to the

variation in confining pressure beneath the beam. Their study indicates that the type of loading is a factor in the sub-grade modulus profile beneath the beam. Results of models incorporating variation in sub-grade modulus show 45-50% variation in flexural responses when compared to uniform sub-grade modulus.

- **Bowles[22]** has mentioned that for beams on elastic foundation problems, we should decrease the the value of spring stiffness from the edge towards the centre. Doing this would ensure the consideration of coupling effects of the adjacent springs. This method is called the Pseudo-coupling method.
- **ACI 336.2R-88[23]** reports that doubling the outermost zone k_s value compared to the innermost zone k_s value will produce the dishing of a uniformly loaded mat and give reliable results.
- **Coduto[24]** has implemented the ACI recommendations by dividing the mat into at least two concentric zones, where the innermost zone should be about half as wide and half as long as the mat foundation.
- **Wahalathantri et al[25]** have presented a model to represent the non-linear behaviour concrete. The model has also been modified for use in Abaqus under Damage Plasticity. The proposed model has been validated with experimental results as well as existing literature.
- **Oller et al[26]** have presented a model for calculating the compressive and tensile damage parameters d_c and d_t which are used to define the stiffness degradation in concrete damaged plasticity model. The proposed approach is mesh insensitive and does not require to be validated with experimental results.

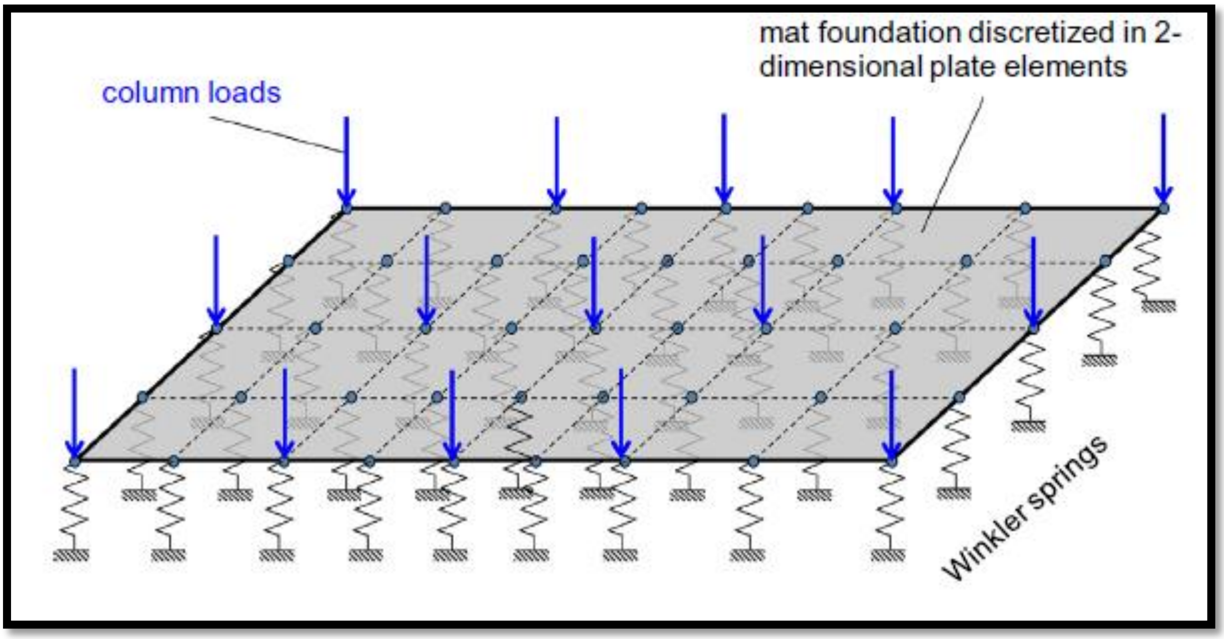

Figure 2.1 Representation of a slab on springs by [19]

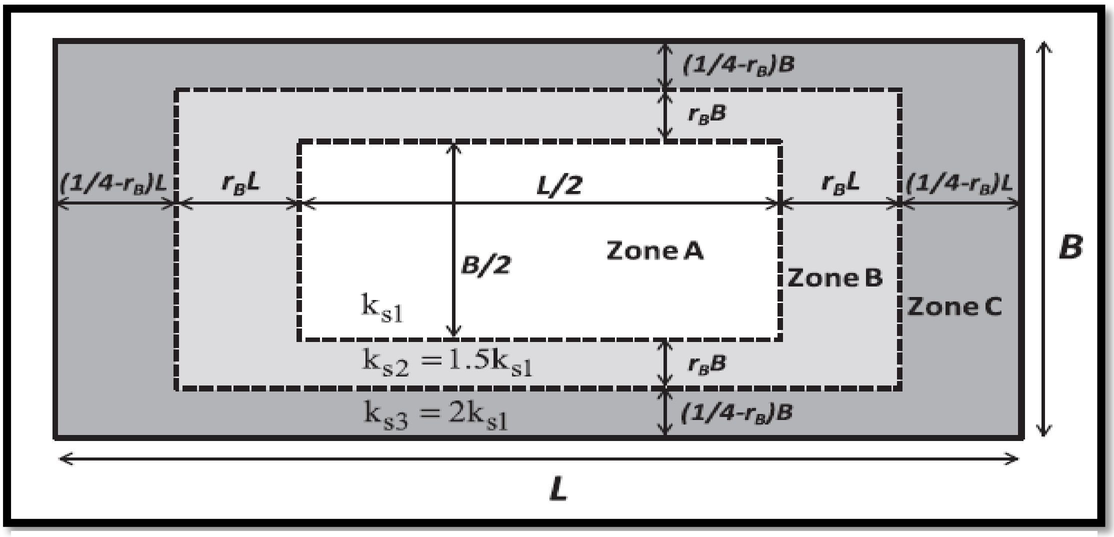

Figure 2.2 Zoning of the slab by [19] based on the recommendations of [24]

2.6 Findings from the Literature Review

The review of all these literatures reveals the fact that there are several approaches for analysis of reinforced concrete slab supported over elastic medium like soil. But out of them, the most common one is the fifth approach where the deformational characteristics of the soil are represented by the linear/nonlinear springs. Here also the choice of spring constants is also an important issue. Published literatures indicate that there may be different pattern of variation of spring constants. But the structural behaviour of the foundation slab resting over springs may be affected by the different variation of stiffness's of these springs.

There are many papers for the analysis of slabs on elastic foundation. There are even fewer papers considering the variation of spring stiffness wherein they have considered linear material properties of concrete and steel. The study on the changes in the behaviour of the foundation slab due to the variation of spring stiffness have not yet been done considering the nonlinear material properties of concrete and steel. Thus the present research is aimed to bridge this gap.

2.7 **Present Scope of work**

- The aim of the present research is to develop a Finite Element model using ABAQUS for reinforced concrete slab subjected to some concentrated and uniformly distributed static loads and supported over elastic continuum like soil.
- To model the stiffness of this slab, linear as well as non-linear material properties of concrete and reinforcement bars have been considered. The proper failure criteria for both these materials have been taken into account. Both concrete and reinforcement bar within the concrete slab are modelled using 8-noded brick elements and are assumed to be connected with the concrete elements at the common nodal points.
- The slabs are considered to be supported over a number of linear springs representing the soil stiffness. The spring stiffness should be calculated based on some typical soil properties assumed for the purpose of present study.
- The loads are considered as incremental static concentrated load acting at several nodal points on the top surface of the slab.

- In the first phase of the work uniform spring stiffness has been considered and the response of the slab under gradually increasing external load was studied. It has been tried to compare these results with the same to be obtained from other software to validate the present model.
- Then in the second phase of the work the spring stiffness will be changed in different fashion as the previous researchers suggest different form of variation of spring stiffness representing the variable soil characteristics. Under these different groups of variation of spring stiffness, the behaviour of the slab will be studied and the relationship between the changes in the behaviour of the reinforced concrete slab due to the change in the spring stiffness will be searched for.
- Crack propagation study is done with both spring supported and rigid supported slabs and these are compared to get the effect of spring stiffness over that.

3. BACKGROUND THEORY AND METHODOLOGY

A mat foundation is a large concrete slab, used to interface the columns with the underlying soil. The slab may cover the entire area of the foundation or part of it. Mats are generally considered under the following conditions:

- The soil bearing capacity is low or the structural loads are so high that a spread foundation covers more than 50 percent of the area. Thus a mat becomes more economical.
- The soil is erratic and prone to excessive differential settlements. The continuity and flexural strength of a mat is helpful.
- The structural loads are erratic which may cause differential settlements
- Lateral loads are not uniformly distributed.
- The uplift pressure cannot be resisted by a spread footing.
- When the foundation is below ground water table, thus propping water-proofing problems. Since a mat is monolithic, it is easier to waterproof.

The methods to design mat foundations are:

1. Rigid method
2. Non-Rigid method

3.1 RIGID METHOD

This method is also known as the conventional method or conventional method of static equilibrium. Here it is assumed that the mat is much more rigid as compared to the soil below. Now this implies that any distortions in the mat are negligibly small to influence the bearing pressure distribution. So the bearing pressure distribution is either uniform or has a linear variation.

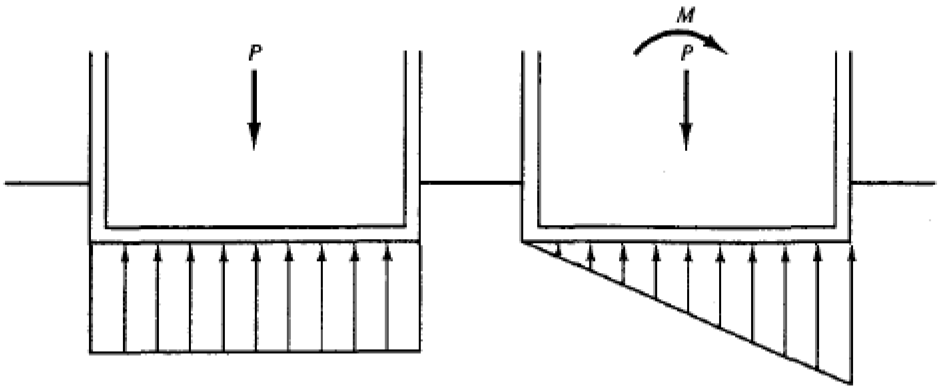

Figure 3.1Bearing pressure distribution for Rigid method

The main drawback of the method is that it does not allow for redistribution of bearing pressure. In reality, the bearing pressure is greater on the edges than in the centre. Hence the method does not give reliable results for shears, bending moments and deformations.

3.2 NON RIGID METHODS

These methods consider the deformation in the mat and the corresponding influence on bearing pressure distribution. As soil-structure relationship is involved here, so one needs to define a relation between the bearing pressure and the displacement. This is done using soil sub-grade modulus k_s where,

$$k_s = \frac{q}{\delta}$$

k_s = coefficient of sub-grade reaction

q = bearing pressure

δ = settlement

The interaction between the mat and the soil is represented by a bed of springs each having a stiffness k_s per unit area. Regions of the mat experiencing more settlement, compress the springs more which represents higher bearing pressure while those which produce lesser compression have lesser bearing pressure. The total spring forces should be equal to the applied structural loads and the weight of the mat.

3.2.1 WINKLER METHOD

The earliest use of the bed of springs to simulate soil-structure interaction is attributed to Winkler (1867). It assumes that the springs are linear and all of them have the same spring stiffness. This is an improvement over the rigid method but still suffers from drawbacks:

- The load-settlement curve of soil is non linear, so k must have some equivalent linear function.
- According to this method, if a uniformly loaded mat is resting on a perfectly uniform soil,all springs will be compressed equally i.e. the slab will suffer no differential displacement. In reality, the settlement at the centre is more compared to the edge.
- The springs act independently. But the bearing pressure induced at a point influences more than just the closest spring.

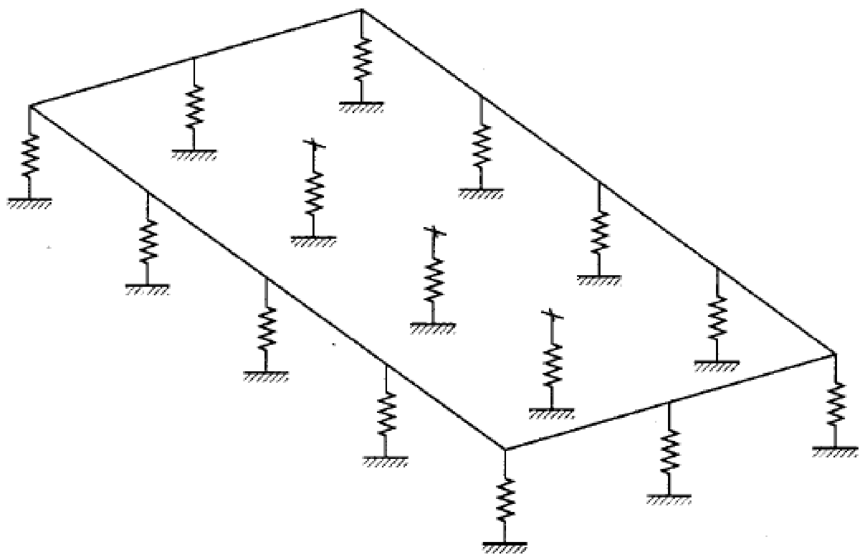

Figure 3.2 The bed of springs analogy to model soil-structure interaction.[24]

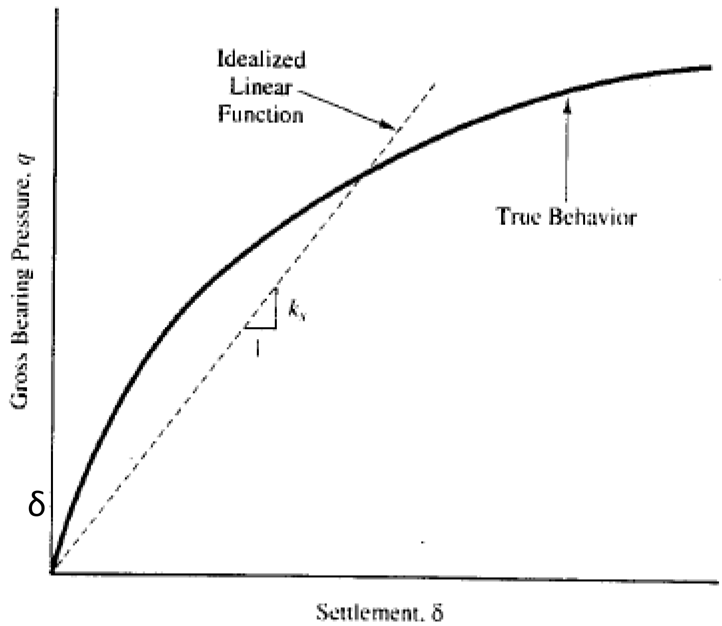

Figure 3.3 Non-linear q-δ relationship and idealised k function.[24]

3.2.2 COUPLED METHOD

This method uses additional springs so that the vertical springs do not act independently. This method is more accurate than the Winkler method as it produces the desired dishing in a uniformly loaded slab. But it is not clear how to select the k_s values for which some software may be used to analyse the given problem.

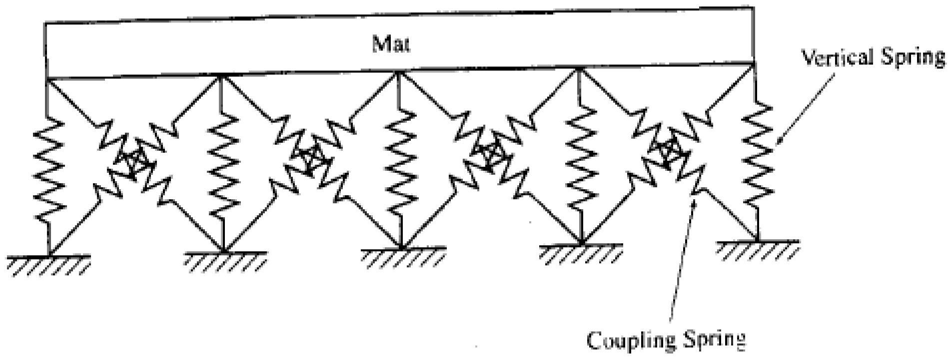

Figure 3.4 Coupled springs

3.2.3 PSEUDO-COUPLED METHOD

This method overcomes the lack of coupling in the Winkler method but also avoids the difficulties of the Coupled method. This method considers independent springs having k_s values depending on their location. According to ACI 1993, reliable results are obtained when higher k_s value, i.e. almost twice the k_s value in the centre, are assigned to the edges. This is achieved by zoning the slab as follows:

- The slab is divided into two concentric zones, with the length and breadth of the inner portion as half the dimensions of the slab.
- Progressively increasing value of k_s is assigned from the centre, with the outer zone being assigned almost twice the k_s value in the centre.
- The shears, moments and displacements are analysed using the Winkler analogy.
- The mat thickness and reinforcement are adjusted to meet safety and serviceability criteria.

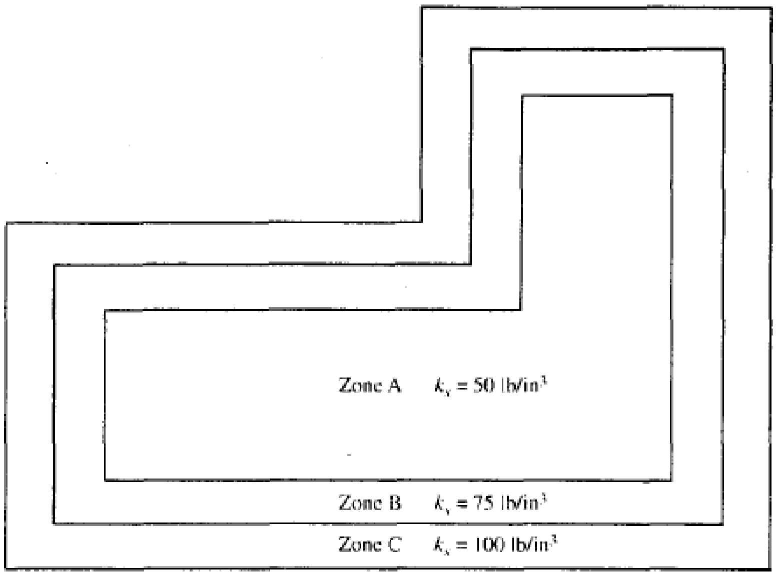

Figure 3.5 Example of a slab divided into zones for Pseudo-Coupled method.[24]

3.2.4 MULTI-PARAMETER METHOD

This is another representation of the soil-structure interaction, where the linear, single parameter springs of the Winkler method are replaced by springs which consider the coupling effects a multi parameter model (Horvath 1993). This method should be more accurate as compared to the pseudo-coupled method, but has not been put to use in available softwares.

3.2.5 FINITE ELEMENT METHOD

The above-mentioned methods have used one-dimensional springs to simulate three-dimensional soil. But using the finite element method, one can model both the mat and the soil as three-dimensional elements. Here the soil is differentiated into small elements, each having assigned soil properties and linked to adjacent soil elements through nodes. But the method has it's own drawbacks:

- The huge number of elements involved puts a strain on the computer resources. The availability of such specialised computers may be restricted to a few people only.
- Difficulty in defining the variable soil properties with precision.

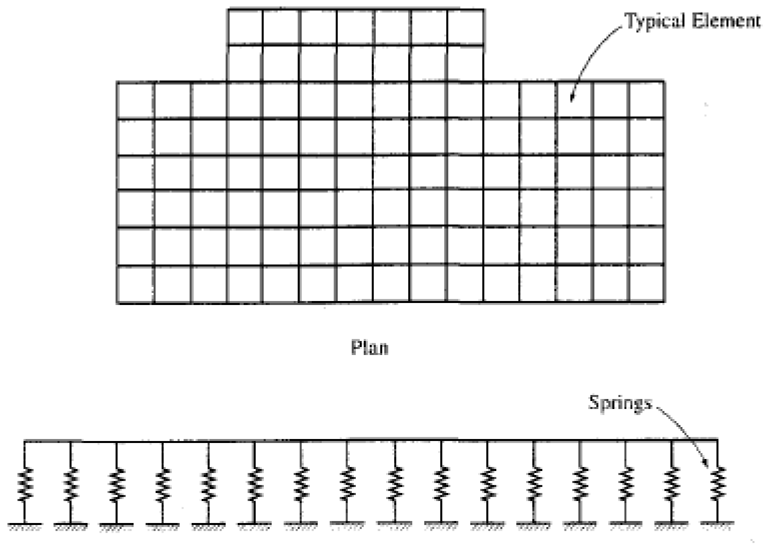

Figure 3.6 Finite element method

3.3 CONCRETE DAMAGE PLASTICITY MODEL

Reinforced Concrete displays very complex structural behaviour due to the composite action of concrete and steel. This complexity arises due to the variance in the nature of concrete and steel. Concrete is a brittle material, but under stress reversal, tensile cracks formed close up and reunite the material. Thus concrete is better represented by a damage model. Steel,on the other hand, is ductile in nature, and rarely fails by fracture but predominantly by yielding, and broken parts are not reunited. This makes the plasticity model more suitable for steel. This model considers the two main failure mechanisms as tensile cracking and compression crushing.

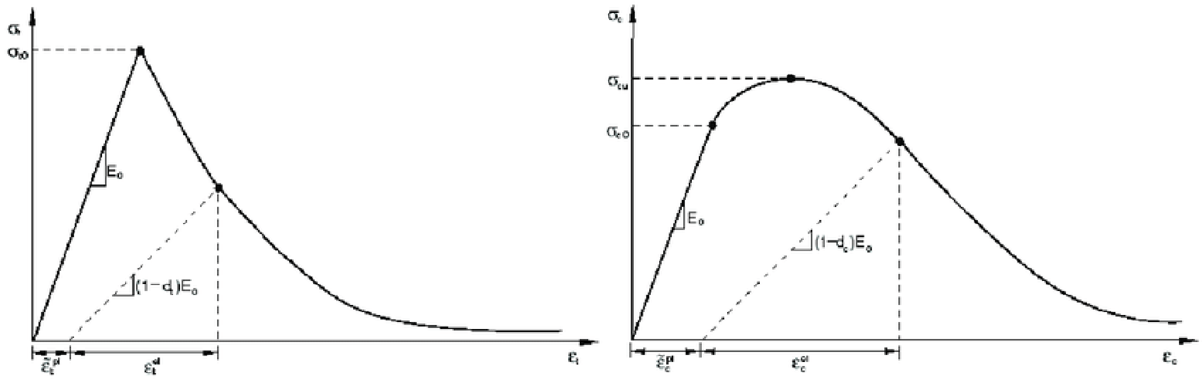

Figure 3.7 Concrete under uniaxial tension and compression (Abaqus manual 6.14)

the graph under tensile behaviour is linear upto failure stress σ_{t0}, which represents the onset of micro-cracks, after which there is strain softening. The graph for compressive behaviour is linear upto first yield stress σ_{c0} followed by strain hardening upto ultimate stress σ_{cu} and then strain softening. The stress-strain relationship is given by:

$\sigma_c = (1\text{-}d_c)E_0(\varepsilon_c - \varepsilon_c^{pl})$

$\sigma_t = (1\text{-}d_t)E_0(\varepsilon_t - \varepsilon_t^{pl})$

Here c and t sub-indexes stand for compression and tension respectively

d_c = compressive damage parameter

d_t = tension damage parameter

The above two parameters characterize the degradation in undamaged stiffness E_0 during unloading. The value range for these parameters is zero, representing no damage, to one which means complete loss of strength.

Reasons for choosing Concrete Damaged Plasticity:

- It has the potential to represent the complete inelastic behaviour of concrete
- Can be used for both plain as well as reinforced concrete.
- Can be used for modelling all types of structures like beams, shells, trusses.

Some of the models created in this work have relied on this modelling. The input data for Abaqus has been taken from the source mentioned in Table 3 and 4 in the following chapter.

3.4 **XFEM (eXtended Finite Element Method)**

This technique was developed by Ted Belytschko and collaborators in 1999. It extends the classical finite element method approach by providing solutions to differential equations having discontinuous functions. It has overcome the shortcomings of the finite element method and is mainly used to model the propagation of singularities or discontinuities like cracks, material interfaces, voids etc.

Modelling a discontinuity, like a crack, using the classical finite element method requires that the mesh comply with the geometric discontinuity. For this considerable mesh refinement is required in the vicinity of the crack. In order to model a propagating crack, it is necessary to redefine the mesh continuously to match it to the growing crack. This is quite a cumbersome process.

The XFEM allows cracks to be modelled independent of the mesh and also the initiation and propagation along an arbitrary solution-dependent path without the need for remeshing.

The traction-separation cohesive behaviour approach under XFEM has been used to model the propagating cracks in the present work. Under this the maximum principal stress Maxps criterion has been chosen. Here we have to specify the maximum principal stress and the displacement at failure for the crack initiation.

$$f = \frac{< \sigma max >}{\sigma 0}$$

Here σ^0_{max} represents the maximum allowable stress, the symbol <> represents Macaulay bracket which signifies that a purely compressive stress state does not initiate damage. The damage occurs when the ratio reaches a value of 1.

The techniques used in XFEM can be broadly classified as :

- Singularity based approach – accounts for crack tip singularities and jumps in displacements across crack surface. The crack can end inside a finite element.
- Phantom node approach – accounts only for the jumps in displacements across crack surfaces and not for crack tip singularity. The crack progresses to the edge of an element only.

The assumptions made in the XFEM method are:

- The material is assumed to be linearly elastic. The available fracture criteria are valid only for homogenous linear elastic materials.
- The analysis is assumed to be quasi-static.
- Pressure loads on faces of cracked elements are ignored.
- Contact elements should not be used in regions where the crack is defined or assumed to grow.

- The crack tip singularity effects are not taken into account in the analysis. So the stress and displacement fields around the crack tip are approximated.

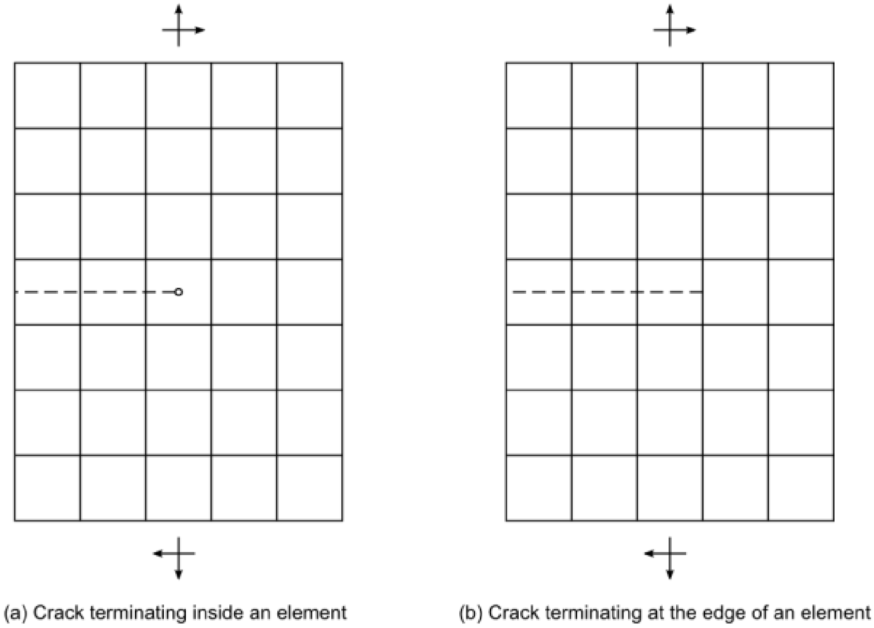

Figure 3.8 Crack representation in a Finite Element model

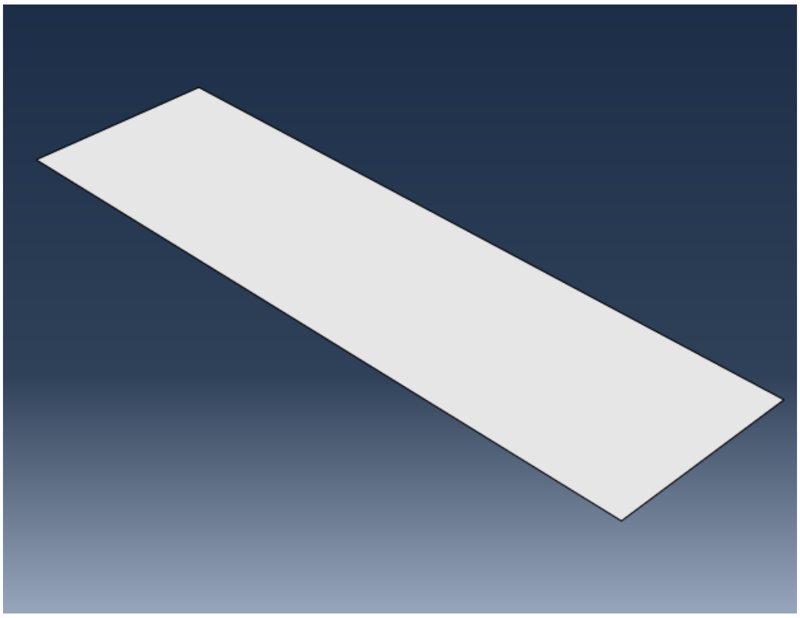

Figure 3.9 Crack element in ABAQUS

4. FE MODELLING USING ABAQUS

4.1 Finite Element Simulation using ABAQUS/CAE software

For the present research work, ABAQUS/CAE has been used to prepare the finite element model of reinforced concrete slab resting over elastic support. ABAQUS is a finite element analysis software released in 1978 by Hibbitt, Karlsson&Sorensen Inc. Currently it is called Abaqus FEA and has 5 main products, namely:

Abaqus/Cae or "Complete Abaqus Environment". It is used for modelling, analysis and viewing the results.

Abaqus/Implicit which uses implicit integration scheme.

Abaqus/Explicit which uses explicit integration scheme.

Abaqus/CFD which is used for computing fluid dynamics problems.

Abaqus/Electromagneticwhich is used for the purpose of solving electromagnetic problems.

Abaqus-CAE provides a user-friendly interface for modelling the problem (defining the geometry, material parameters, load and boundary conditions, mesh details), viewing the outputs and analysing the results. One of the main advantages of this software is the ease of modelling, revising the details, hassle-free analysis and viewing the results.

4.1.1 General Modelling steps

ABAQUS involves several modules or steps, wherein each of them deals with a particular modelling aspect.

Module 1 - **PART**: this step involves defining the geometry (i.e. 3D or 2D, solid or shell element and the dimensions) of the model. If the model consists of two or more components, then separate parts can be created for each of them. For defining the slab and the reinforcement, I have used a 3D solid homogenous section with the dimensions 20mx20mx0.8m and 12ϕ diameter respectively in two different parts. For problems with a crack, it was defined using a 3D shell extrusion element.

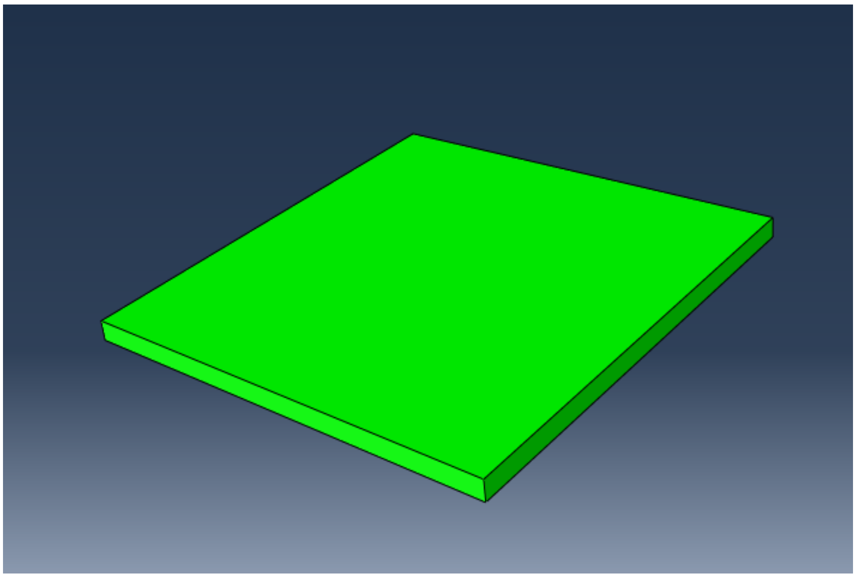

Figure 0.1 Slab model in Part module

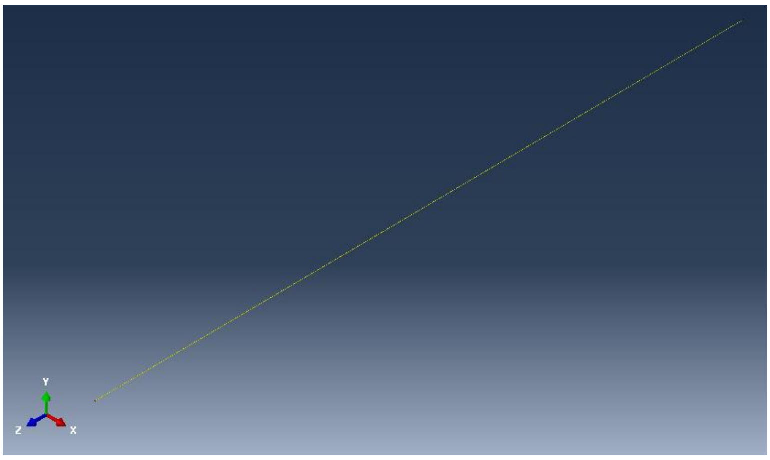

Figure 0.2 Reinforcement in Part module

Module 2 – **MATERIAL** : in this step the material parameters are input like the density, elasticity and plasticity properties, the damage criterion to be adopted with several other sub-options.

MATERIAL	DENSITY	YOUNG'S MODULUS (N/mm^2)	POISSON'S RATIO
CONCRETE	2400kg/m^3	27386.13/29100	0.2
STEEL	7800kg/m^3	210000	0.3

Table 0-1 General and Elastic properties of concrete * some models have E_c value of 27386.13 MPa while some have E_c value of 29100 MPa

Both concrete and steel were considered as elastic-isotropic. For the models used to simulate the crack propagation, the material model given by Alfarah, Oller, Almansa [26] and Wahalathantri, Chan et al [25] which in turn relies on the numerical method by Hsu and Hsu [27] has been used to simulate the Concrete Damaged Plasticity Model (CDPM). Under this there are three tabs: Plasticity, Compressive behaviour and Tensile behaviour. Under the three tabs, values of Tables 2 [26], 3 and 4 [25] were put in respectively.

DILATION ANGLE	ECCENTRICITY	f_{bo}/f_{co}	VISCOSITY PARAMETER
13	0.1	1.16	0

Table 0-2 Values for Plasticity sub-option in CDPM[26]

STRESS (σ_c) N/mm^2	INELASTIC STRAIN (ε_c^{in})	COMPRESSIVE DAMAGE PARAMETER (d_c)
25.6	0.00e+00	0.00e+00

36.4	1.00e-04	1.05e-02
44.9	2.81e-04	2.95e-02
49.7	5.87e-04	6.16e-02
51.2	1.01e-03	1.06e-01
49.0	1.76e-03	1.85e-01
44.3	2.60e-03	2.73e-01
38.9	3.46e-03	3.63e-01
33.7	4.31e-03	4.53e-01
29.2	5.14e-03	5.40e-01
25.4	5.95e-03	6.25e-01
22.2	6.74e-03	7.07e-01
19.5	7.51e-03	7.88e-01

Table 0-3 Compressive Stress-Strain values for 51.2MPa concrete[25]

STRESS (σ_t) N/mm^2	**CRACKING STRAIN (ε_t^{ck})**	**TENSILE DAMAGE PARAMETER (d_t)**
2.36	0.00e+00	0.00e+00
1.89	4.07e-05	3.85e-01
0.945	2.93e-04	9e-01
0.213	8.07e-04	9.91e-01

Table 0-4 Tensile Stress-Strain values for 51.2MPa concrete[25]

After the properties are entered, sections are created for each parts and the corresponding material is assigned to the newly created sections.

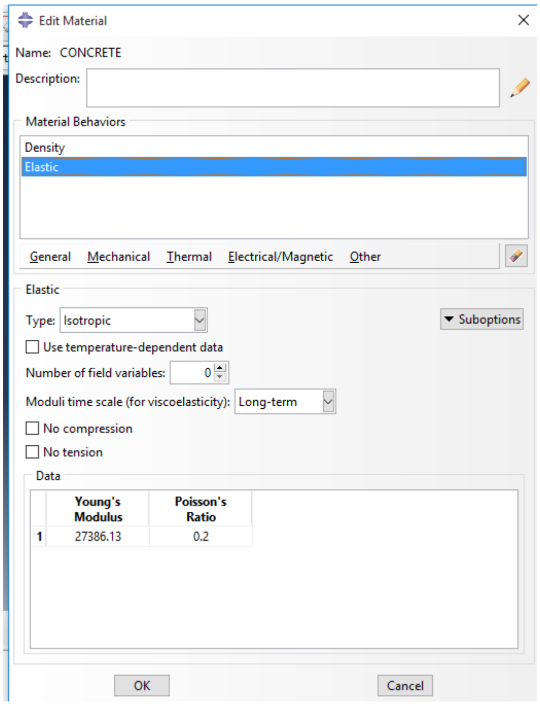

Figure 0.3 Materials Editor box

<u>Module 3 – **ASSEMBLY**</u> :now all the sections created in the previousstep are assembled into a single entity. The reinforcement for RCC models are provided in both the directions using the linear arrangement option. After achieving the proper orientation, the slab and reinforcement geometry are merged into a single instance. For problems of crack propagation, the crack part is not merged into this instance. This is done to avoid meshing and crack definition difficulties. Spring properties are also assigned in this module. By selecting Assembly=>Engineering Features=>Springs one can assign the spring properties namely the degree of freedom and the spring constant, k in Force/Displacement units. After this the software asks the user to choose the nodes where the springs are to be applied. The springs are of two types where Spring1 is between node and ground and Spring2 is between two nodes. Here the springs used are linear springs.

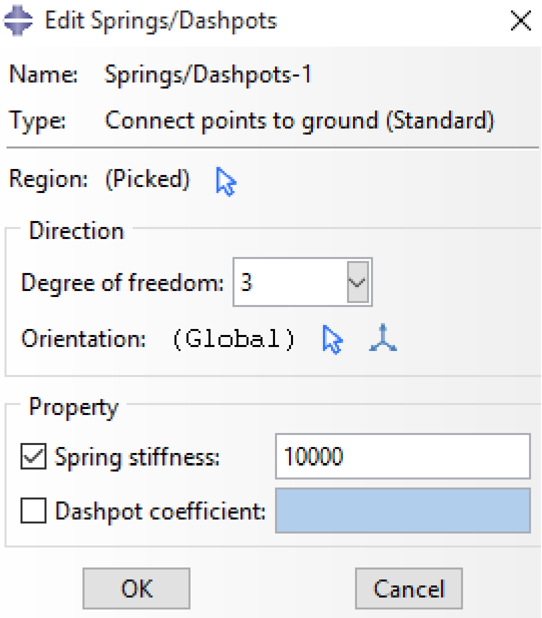

Figure 0.4Spring definition

Module 4 – **STEP** : in this step the load type is assigned. For all the models, load type "Static, General" has been used. In this step, we can request certain outputs from the software as well. For crack propagation models, PHILSM and PSILSM functions are used to define the crack location inside the body. STATUSEXFEM is a function which shows the extent of cracking in the bulk material. The value of it lies between 0 and 1, with 0 for section with no crack and 1 for fully cracked section.

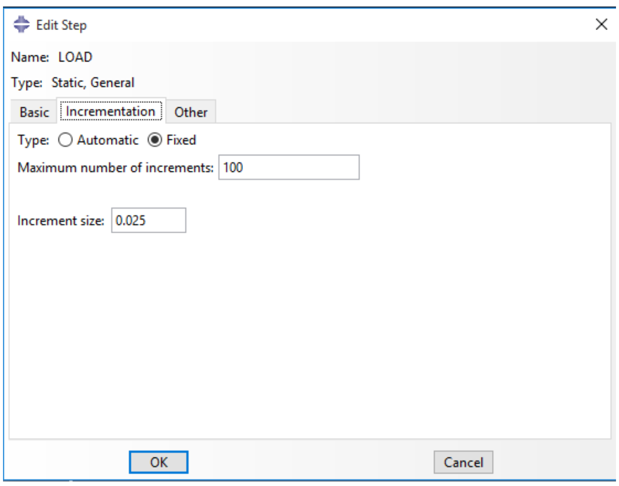

Figure 0.5 Step editor

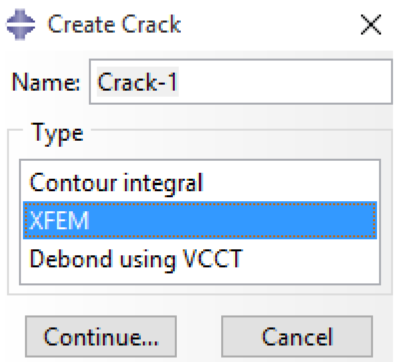

Figure 0.6 XFEM

Module 5 – **INTERACTION** : in this step XFEM (eXtended Finite Element Method) mode of crack propagation is defined. Other techniques present to simulate a crack are Contour Integral and Virtual Crack Closure technique (VCCT). But XFEM technique has some advantages over them, the main ones being that XFEM can be used to simulate crack propagation in an arbitrary, solution-dependent path in the bulk material and does away with the requirement of remeshing. Choosing Special=>Crack=>Create in the interaction module and opting for XFEM allows one to define the crack domain. One should check that the crack propagation box is ticked so that the crack propagates.

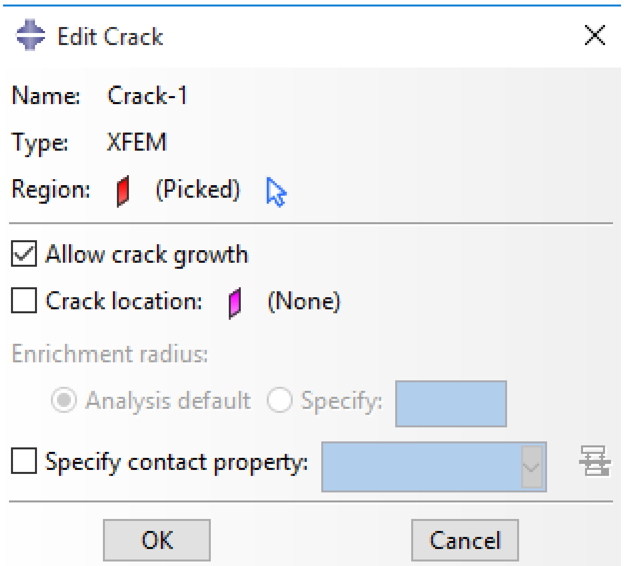

Figure 0.7 XFEM crack definition

Module 6 –**LOAD**: here the software provides a wide variety of loads which can be applied. For our models "Concentrated Force" and "Pressure" (UDL) have been used. The load values and combinations are mentioned in the respective models. This step is also used for assigning the boundary conditions. For simply supported models the boundary condition type "Displacement/Rotation" was used, where one end was assigned pinned support and the other a roller support. For models with all edges pinned U1=U2=U3=0 was assigned under the same sub-option mentioned above.

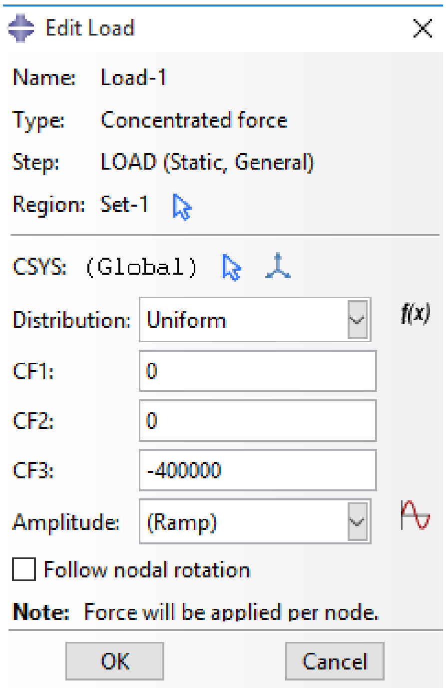

Figure 0.8 Load Editor box

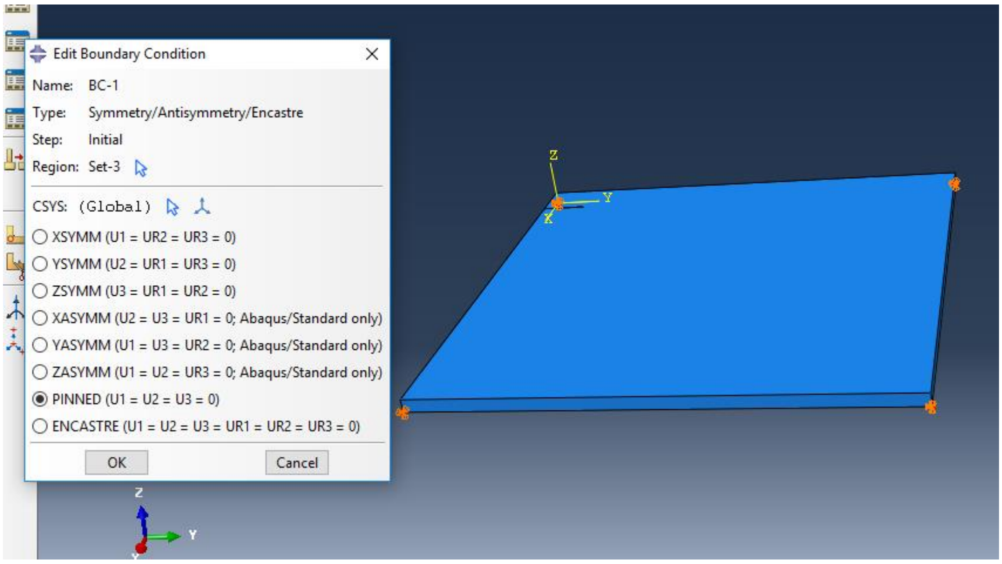

Figure 0.9 Boundary Condition assignment

Module 7 – **MESH**: here the meshing controls are assigned like the type of element, mesh size as well as the meshing technique. The variety of mesh elements depends on the region to be meshed i.e. 2D or 3D. For all the models, an 8-noded linear brick element C3D8R was used with reduced integration.

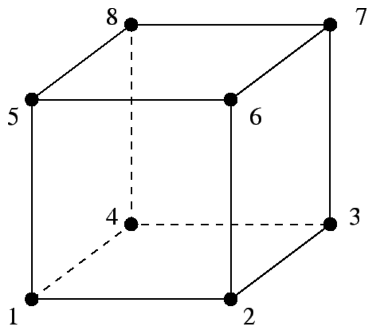

Figure 0.11 8 noded linear brick element

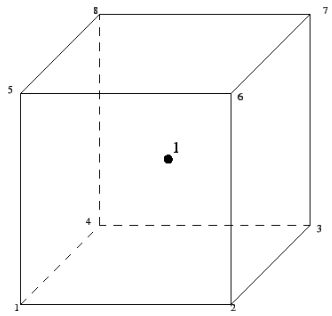

Figure 0.10 Point for reduced integration

Meshing techniques include Free, Structured, Sweep and Bottom-up. Here the models have been created using structured meshing technique which are normally used to mesh simple 2D regions (planar or curved) or simple 3D regions which employ Hex or Hex-dominated elements.

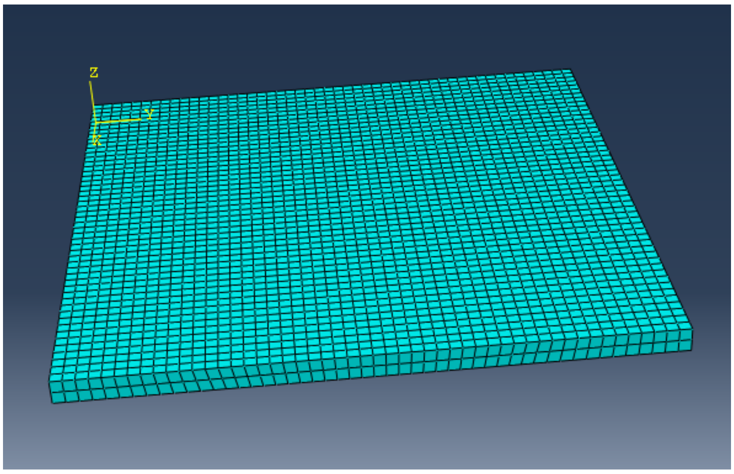

Figure 0.12 Meshed slab

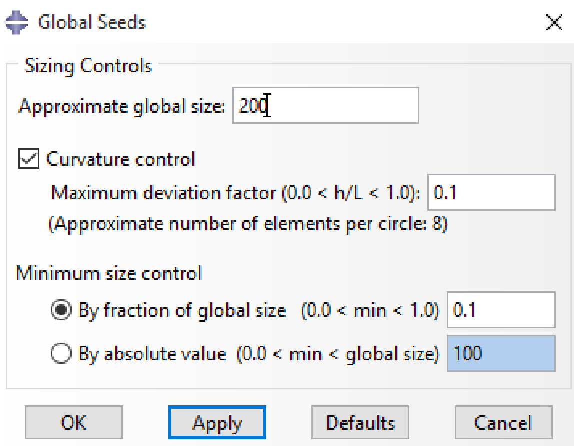

Figure 0.13 Mesh attributes

<u>Module 8 – **JOB**</u> :after the modelling is complete, a new job is created and submitted for analysis. One can monitor the progress of the analysis. Once it is complete the results are ready for viewing. It is done by clicking the results tab, then choosing Output Databases. From the list of output files (.odb), the one with the recently created job name is chosen and the outputs can be seen.

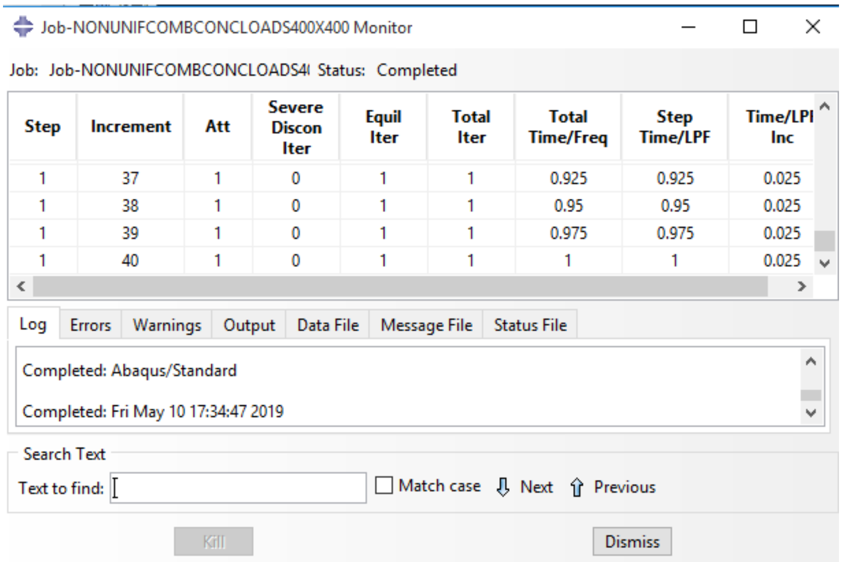

Figure 0.14 Job progress

5. RESULTS & DISCUSSION

5.1 MESH CONVERGENCE STUDY

For checking the suitability of the approach, mesh convergence study has been done. To do this, a reinforced concrete slab resting over springs supports subjected to number of concentrated loads and uniformly distributed load has been considered. In the following sets of models, the variation has been done in terms of loading type, location of the springs as well as the spring constants. Three different meshes represented by three sizes of elements are considered: 100mm x 100mm, 200mm x 200mm, 400mm x 400mm. For each of these sets, the displacement contours are plotted and the deflection of the slab at centre is tabulated. The comparison states that the displacement contours are identical but the deflections values change with the change of mesh size. As the change of the result among these three mesh sizes are considerably small, the analysis can be done with sufficient accuracy using 100 mm x 100mm.

5.1.1 SET 1 : CONCENTRATED LOAD and UNIFORM SPRING SUPPORT

Slab dimension	20m x 20m x 0.8m
Reinforcement	12ϕ HYSD

Table 0-1 Dimensions for SET 1 Models

Material parameters	Density (kg/m^3)	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	2400	27386.13	0.2
Steel	7800	210000	0.3

Table 0-2 Material parameters for SET 1 Models

Spring constant	10000 N/mm	At 5m spacing

Load	400 kN	At the centre

Table 0-3Spring and load details

Mesh details	Mesh size
Model 1	400mm x 400mm
Model 2	200mm x 200mm
Model 3	100mm x 100mm

Table 0-4 Mesh details for SET 1 Models

Mesh size	Maximum Displacement (mm)
400mm x 400mm	1.883
200mm x 200mm	1.871
100mm x 100mm	1.939

Table 0-5 Displacement results

Mesh size	Maximum Principal stress (N/mm^2)
400mm x 400mm	0.985
200mm x 200mm	3.172
100mm x 100mm	12.57

Table 0-6 Maximum Principal stress results

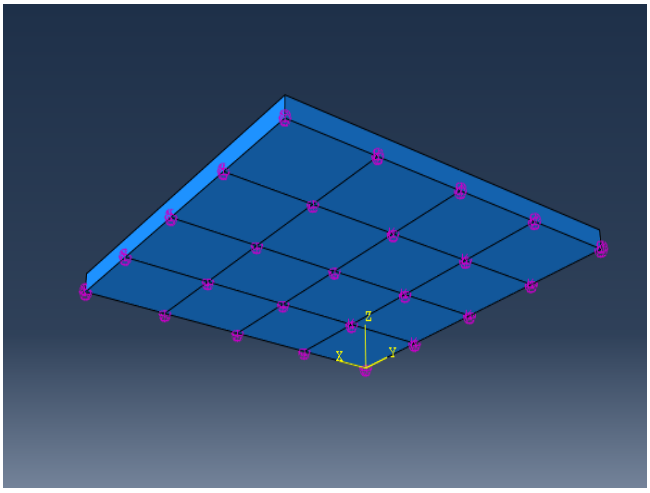

Figure 0.1 Spring locations

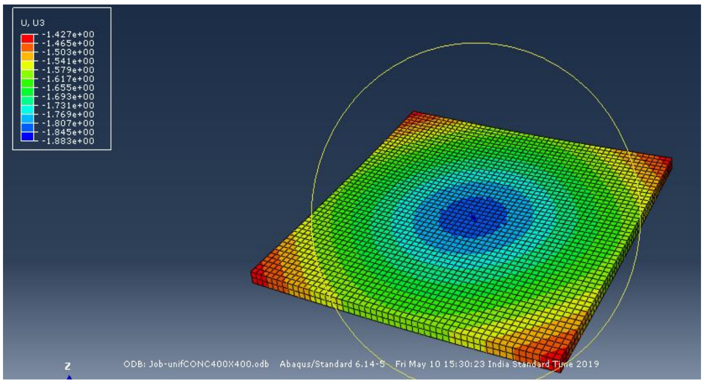

Figure 0.2 Displacement contour Model 1

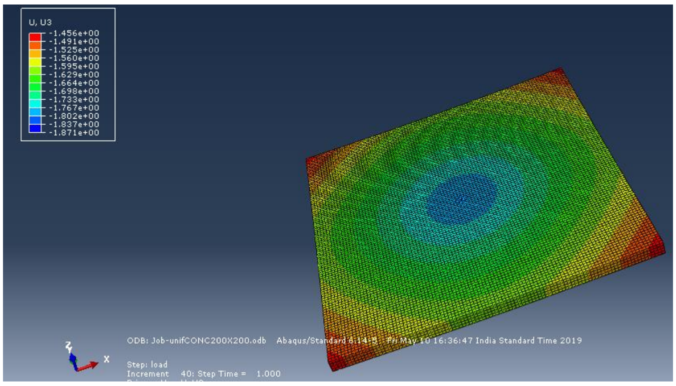

Figure 0.3 Displacement contour Model 2

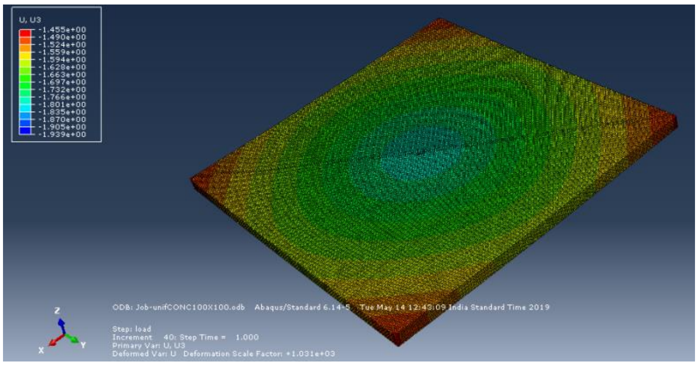

Figure 0.4 Displacement contour Model 3

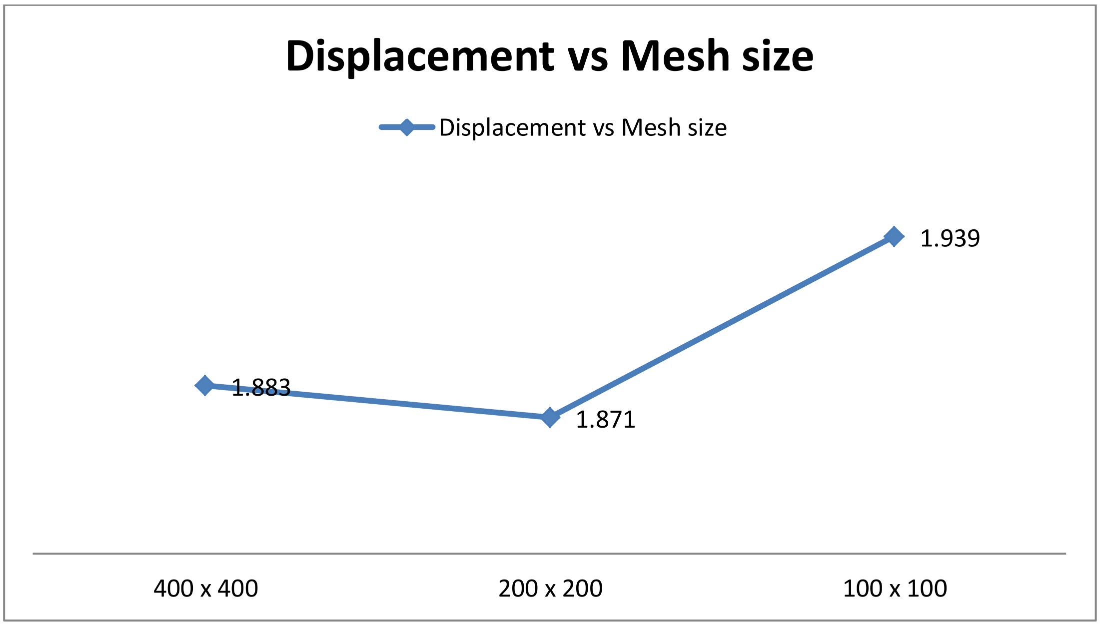

Figure 0.5 Displacement vs Mesh size

5.1.2 SET 2 : UDL and UNIFORM SPRING SUPPORT

Slab dimension	20m x 20m x 0.8m
Reinforcement	12ϕ HYSD

Table 0-7 Dimensions for SET2 Models

Material parameters	Density (kg/m^3)	Young’s Modulus of Elasticity (MPa)	Poisson’s ratio
Concrete	2400	27386.13	0.2
Steel	7800	210000	0.3

Table 0-8 Material parameters for SET 2 Models

Spring constant	10000 N/mm	At 5m spacing
Load	10 kN/m^2	UDL

Table 0-9Spring and Load details

Mesh size	
Model 1	800mm x 800mm
Model 2	400mm x 400mm
Model 3	200mm x 200mm

Table 0-10 Mesh details for SET 2 Models

Mesh size (mm x mm)	Maximum Displacement at centre (mm)	Maximum Displacement at edge (mm)	Maximum Displacement at corner (mm)
800 x 800	26.64	15.406	5.618
400 x 400	16.59	16.11	15.36
200 x 200	16.53	16.04	15.44

Table 0-11 Displacement results

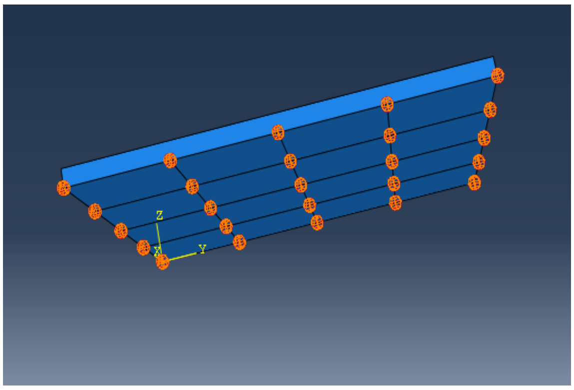

Figure 0.6 Spring locations for SET 2

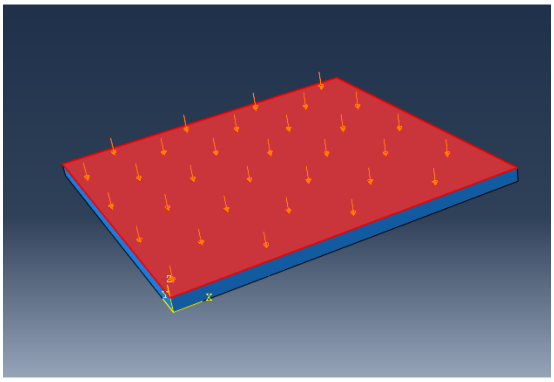

Figure 0.7 UDL for SET 2

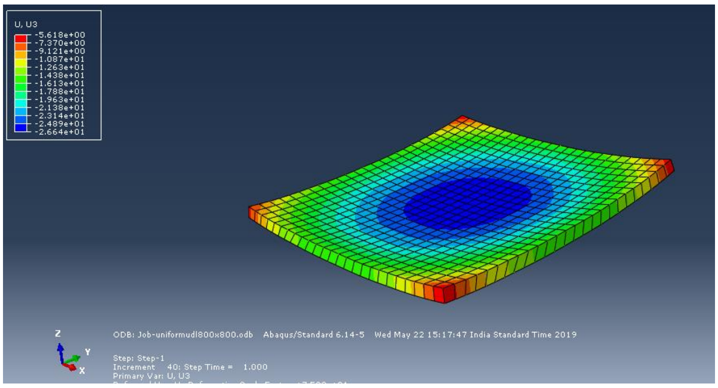

Figure 0.8 Displacement contour for Model 1

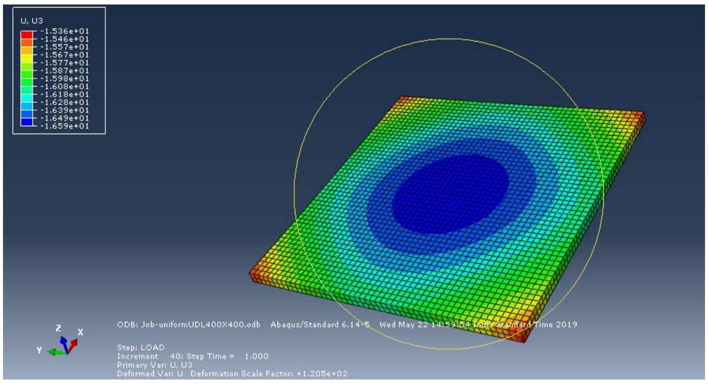

Figure 0.9 Displacement contour for Model 2

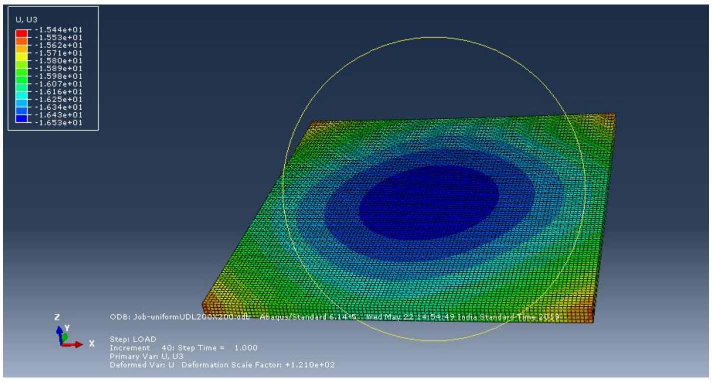

Figure 0.10 Displacement contour for Model 3

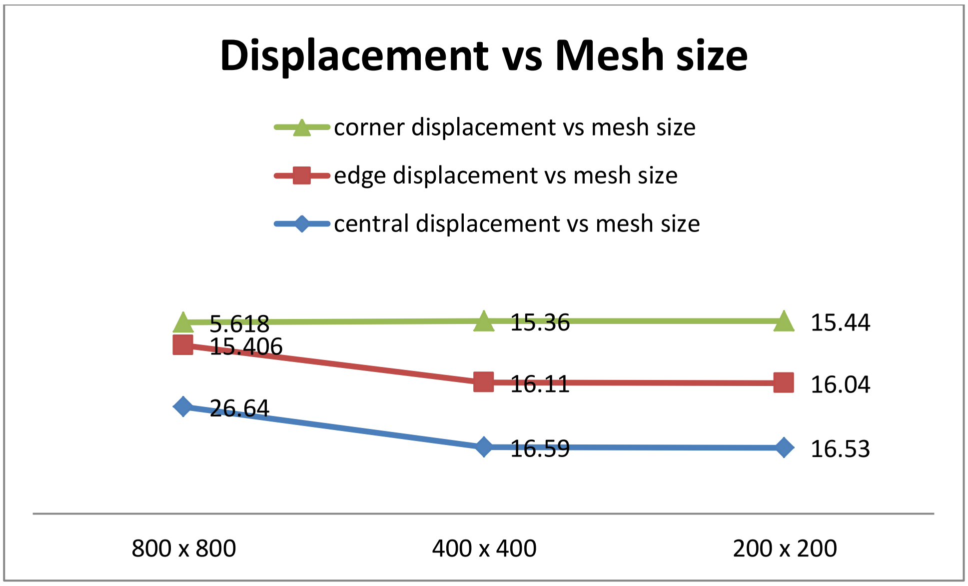

Figure 0.11 Displacement vs Mesh size for SET 2 Models

5.1.3 SET 3 : UDL and NON-UNIFORM SPRING SUPPORT

Slab dimensions	20m x 20m x 0.8m
Reinforcement	12ϕ HYSD

Table 0-12 Dimensions for SET 3 Models

Material parameters	Density (kg/m^3)	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	2400	27386.13	0.2
Steel	7800	210000	0.3

Table 0-13 Material parameters for SET 3 Models

Spring constant	Value and location	Load UDL (kN/m^2)
Outer ring	20000 N/mm at 10m spacing	10
Intermediate ring	15000 N/mm at 10m spacing	10
Inner ring	10000 N/mm at 5m spacing	10

Table 0-14 Spring and Load details for SET 3 Models *(Higher values of stiffness at the edges and lower values towards the centre)[23]

Mesh details	Size
Model 1	800mm x 800mm
Model 2	400mm x 400mm
Model 3	200mm x 200mm

Table 0-15 Mesh details for SET 3 Models

Mesh size	Maximum Displacement at centre (mm)	Maximum Displacement at edge (mm)	Maximum Displacement at corner (mm)
800mm x 800mm	25.42	15.98	6.54
400mm x 400mm	16.15	15.38	14.6
200mm x 200mm	16.09	15.14	14.67

Table 0-16 Displacement results

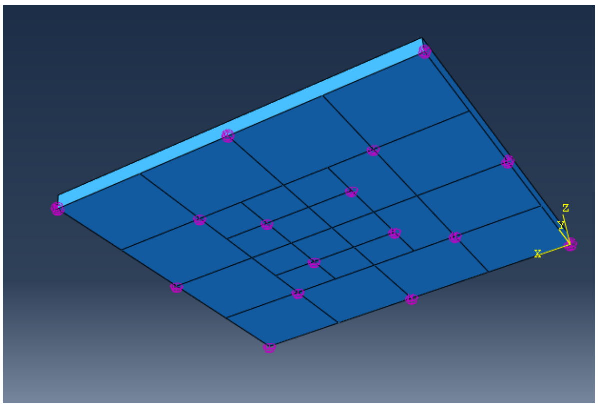

Figure 0.13 Spring location for SET 3 Models

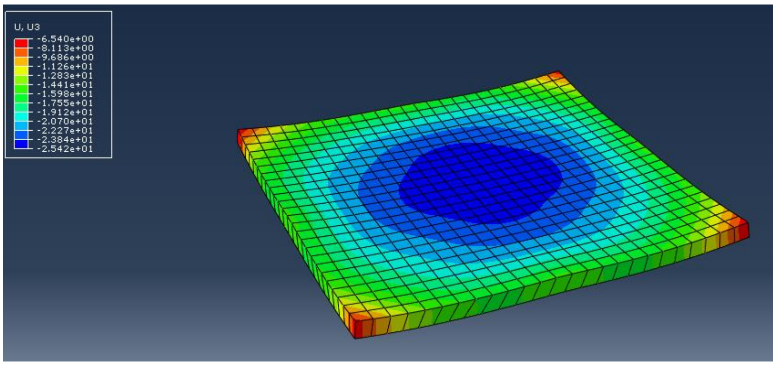

Figure 0.14 Displacement contour for Model 1

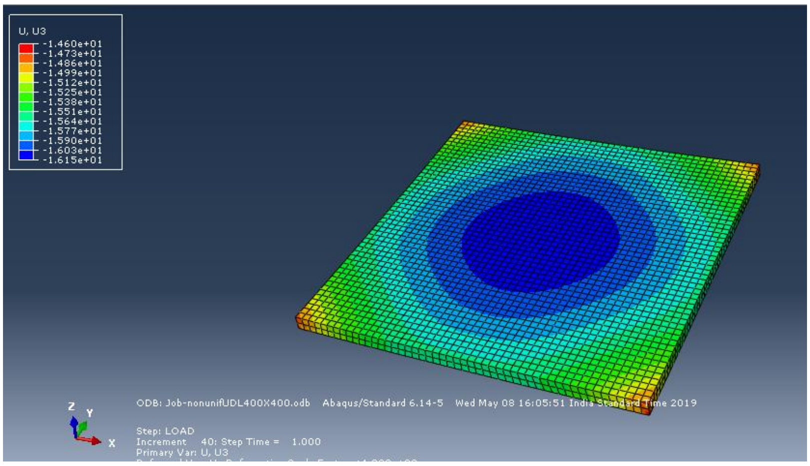

Figure 0.12 Displacement contour for Model 2

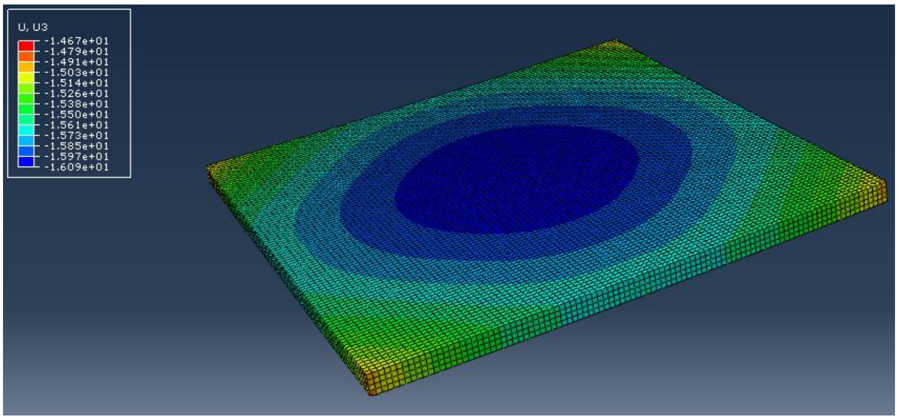

Figure 0.15 Displacement contour for Model 3

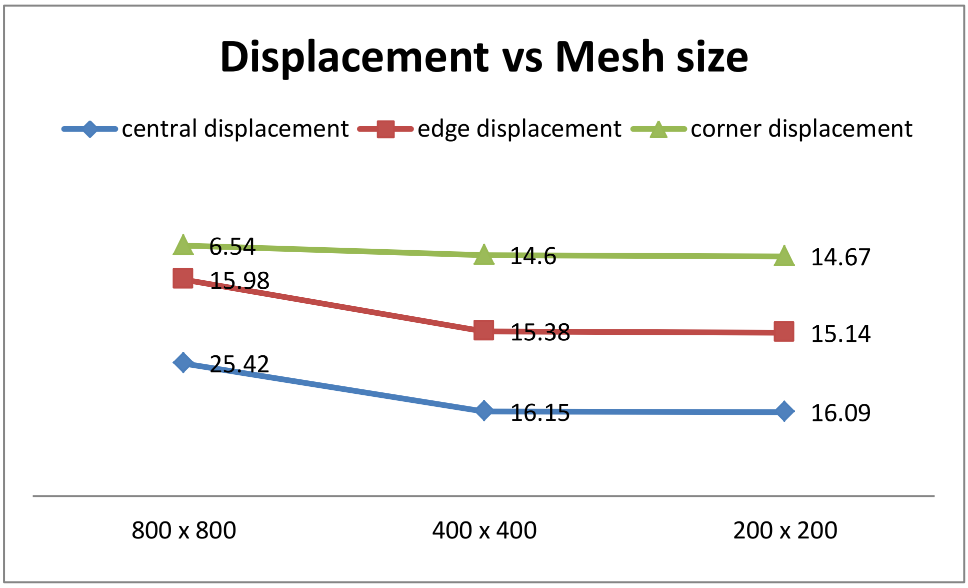

Figure 0.16 Displacement vs Mesh size for SET 3 Models

After studying the results of the models, we can say that 800mm x 800mm mesh is not suitable for modelling. Using this mesh size there is a single element across the depth of the slab, which does not, justify the true behaviour of the slab. We see that the displacement results are converging with finer meshing controls. 100mm x 100mm and also 50mm x 50mm mesh sizes were also considered for some of the cases, but due to the large computational time required for the output, they have been avoided in the study. 200mm x 200mm mesh element size can be used to get considerably accurate results.

5.2 PARAMETRIC STUDY

In the parametric study, the main parameters whose effects have been studied are the spring locations and the variation of spring constants. For this, three cases were studied, where only the spring location and the spring constants were changed, while keeping other parameters like slab dimension, material parameters, loading conditions and the mesh element size were kept constant. The spring constants were applied according to the recommendations made by [23], wherein the outer springs have been assigned twice the spring constant vale as compared to the inner zone springs. Each of the three models has been zoned into three concentric regions according to the recommendations of [24], where the inner zone has half the length and breadth of the outer zone. The models give realistic results in terms of displacements and stresses. Model 1 deflects the least and hence is subjected to the maximum stress among the three models. Model 3 on the other hand, has the maximum deflection and therefore is subjected to the least stresses. Models 1 and 2 show similarity in the deflected shape, while Model 3 shows the dishing of the slab as is mentioned by [23] which accounts for the values in Table 5-28.

5.2.1 STUDYING THE EFFECT OF CHANGING SPRING LOCATION

To assess the effect of spring locations along with the spring constants, total three models are considered. In model:1, spring locations are kept uniform, but the variation of the spring constants are chosen according the referred literature [23]. In model:2, non-uniform spring locations are considered along with non-uniform variation of spring constants, In model:3, non-uniform variation of spring constants are considered over uniform spring locations. Other parameters are kept identical as that of the previous case studies.

Figure 0.17 gives the variation of displacement at three locations vs model number. It is evident that for every model, the corner displacement is the smallest one and central displacement is the largest one. Out of three models, model:1 is showing the smallest displacement s at all locations. Table 0-17 represents percentage change in displacement with location for all three models. Maximum variation of displacement at all locations is observed between model 1and 3. Also for all models, the maximum variation of displacements are observed at the corner locations.

Slab dimension	1.83m x 1.83m x 0.051m
Reinforcement	12ϕ HYSD

Table 0-18 Dimensions for Parametric study

Material parameters	Density (kg/m^3)	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	2400	27386.13	0.2
Steel	7800	210000	0.3

Table 0-19 Material parameters for Parametric study

Load	Magnitude (kN)	Location
Load 1 (Q)	50	At the centre
Load 2 (Q/2)	25	At the 4 mid-points of the edges
Load 3 (Q/5)	10	At the 4 corners

Table 0-20 Load details

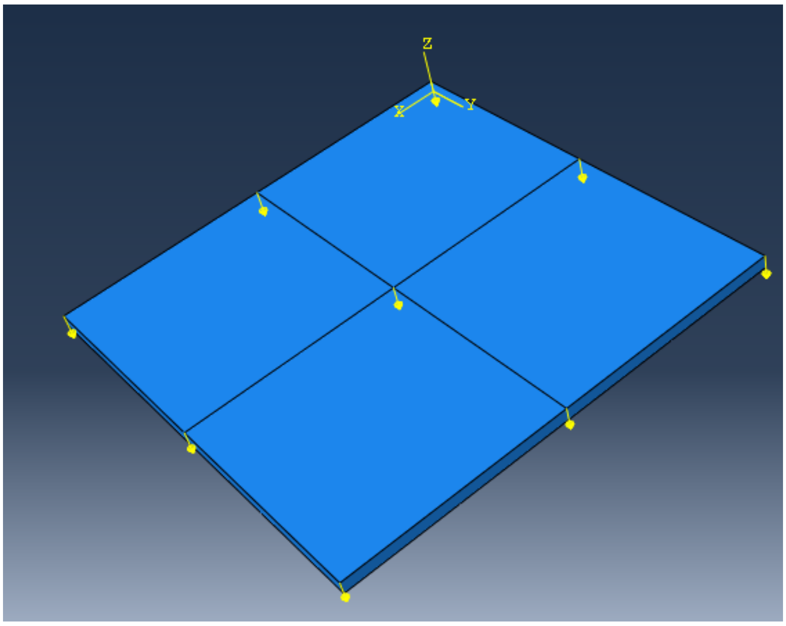

Figure 0.18 Location of Concentrated loads for Parametric study

Mesh size	152.5mm x 152.5mm x 51mm

Table 0-21 Mesh details for Parametric study

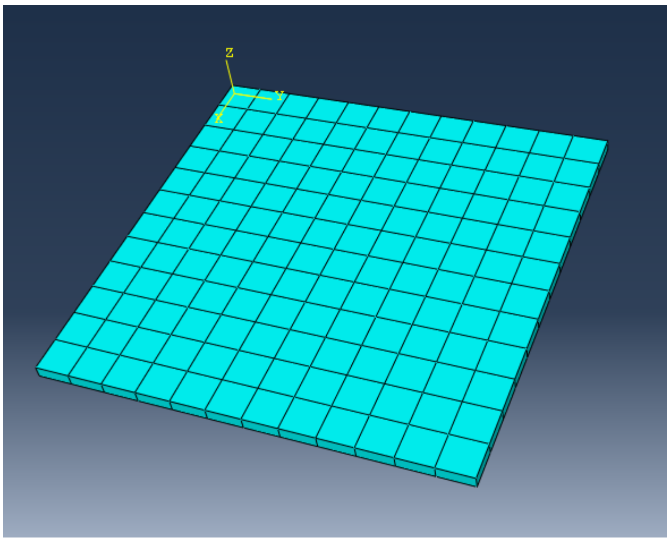

Figure 0.19 Meshed slab

5.2.1.1 MODEL 1 : UNIFORM SPRING LOCATION, NON-UNIFORM SPRING CONSTANT

Spring	Outer zone	Intermediate zone	Inner zone
Spring constant (N/mm)	20000	15000	10000
Spacing	5m (total 16nos.)	5m (total 8nos.)	1 no.

Table 0-22Spring location and spring constant details according to [24] and [23] respectively

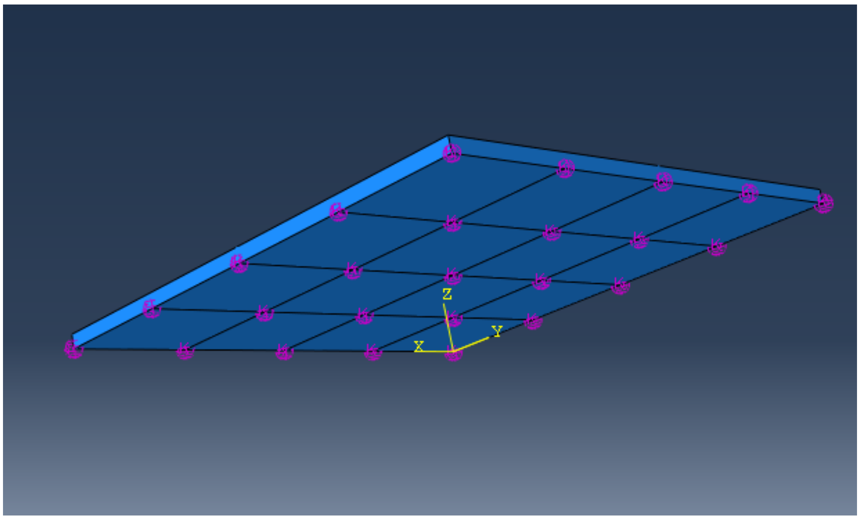

Figure 0.20 Spring location for Model 1

Displacement	Magnitude (mm)
Centre	2.302
Mid-point of edge	0.6646
Corner	0.197

Table 0-23 Displacement results for Model 1

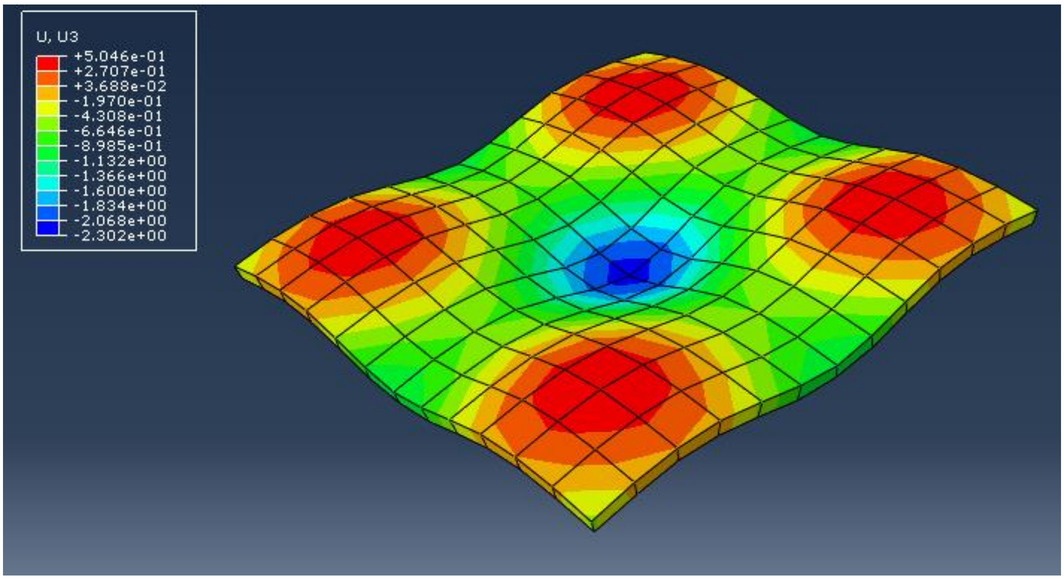

Figure 0.21 Displacement contour Model 1

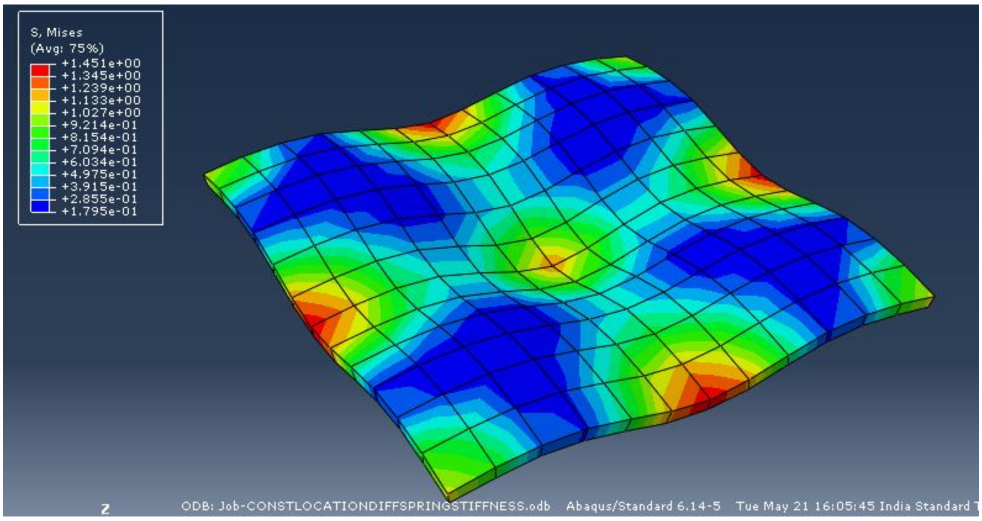

Figure 0.22 Von Mises stress contour

5.2.1.2 MODEL 2 NON-UNIFORM SPRING LOCATION, NON-UNIFORM SPRING CONSTANT

Spring	Outer zone	Intermediate zone	Inner zone
Spring constant (N/mm)	20000	15000	10000
Spacing	10m (total 8nos.)	10m (total 4nos.)	1no.

Table 0-24Spring location and spring constant details according to [24] and [23] respectively

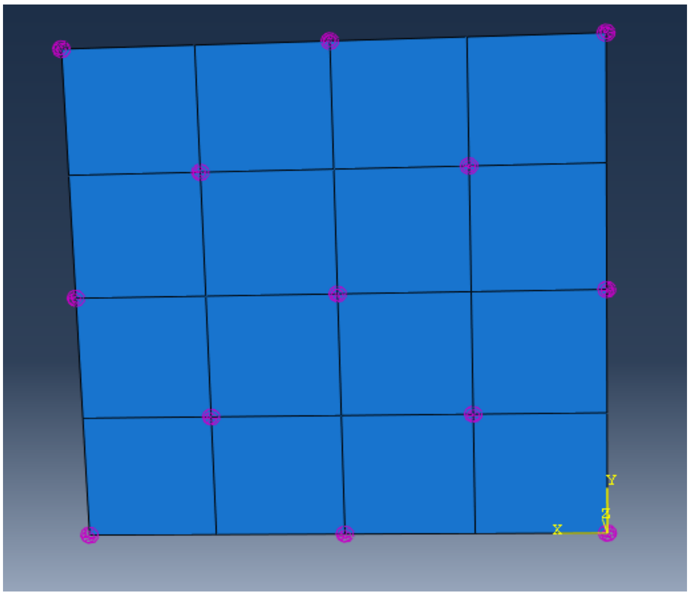

Figure 0.23Spring location for Model 2

Displacement	Magnitude (mm)
Centre	3.397
Mid-point of edge	1.272
Corner	0.6647

Table 0-25 Displacement results for Model 2

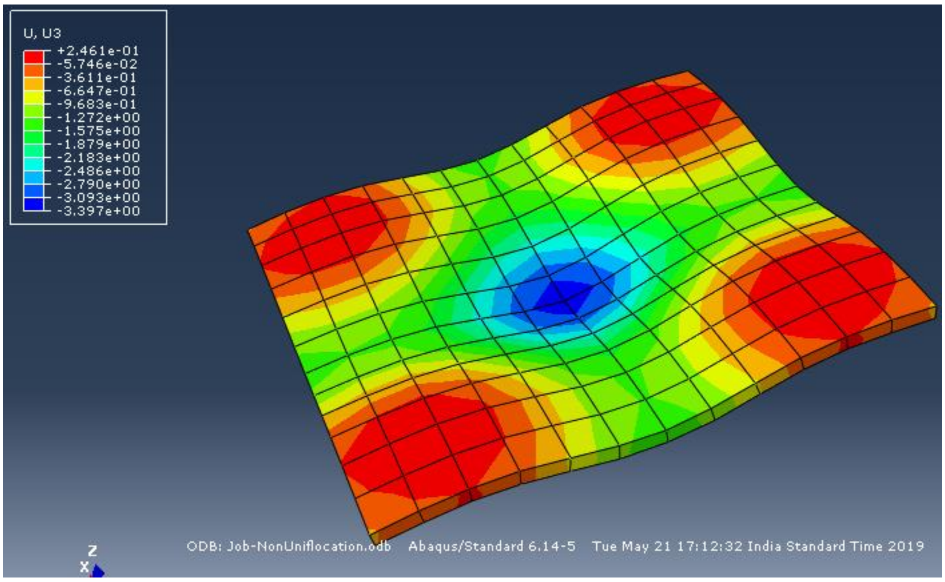

Figure 0.24 Displacement contour Model 2

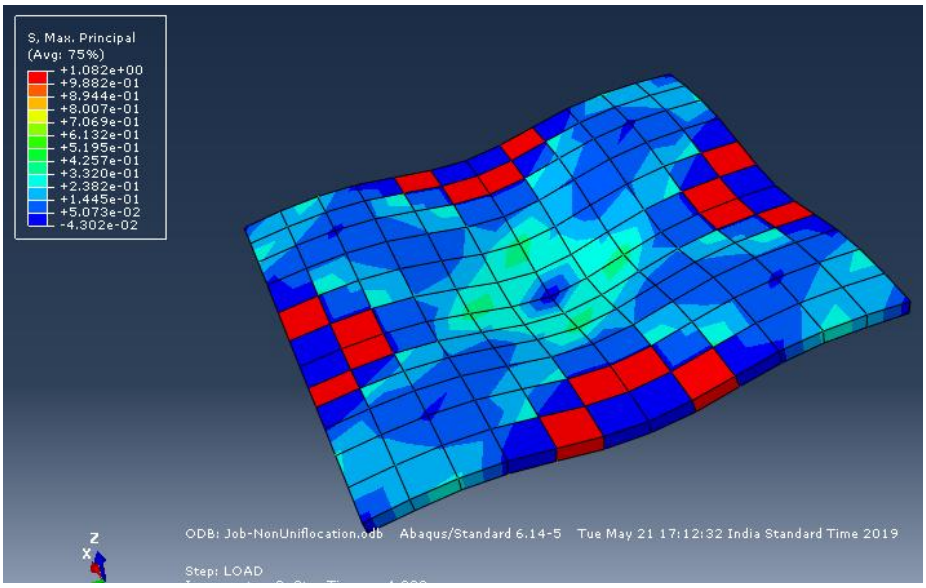

Figure 0.25 Maximum Principal stress contour Model 2

5.2.1.3 UNIFORM SPRING LOCATION, NON-UNIFORM SPRING CONSTANT

Spring	Outer zone
Spring constant (N/mm)	20000
Spacing	10m (total 9nos.)

Table 0-26Spring location and spring constant details

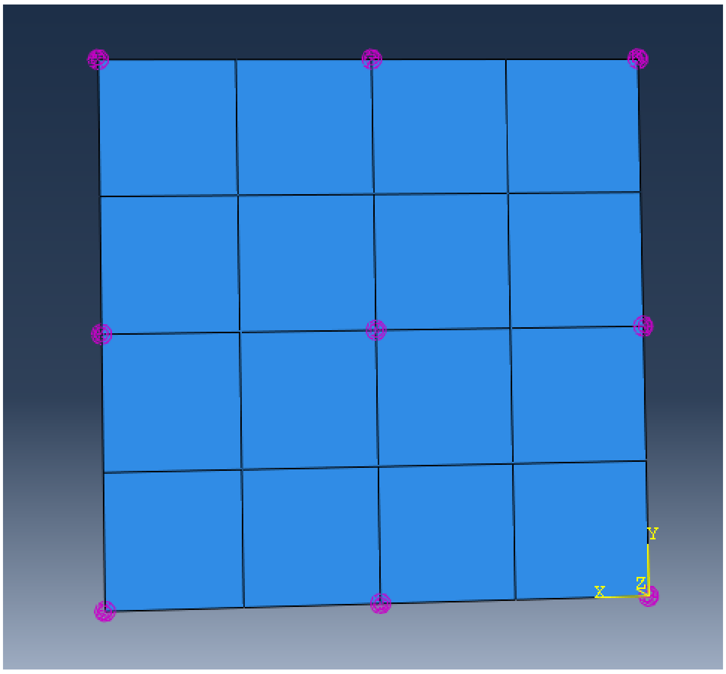

Figure 0.26Spring location for Model 3

Displacement	Magnitude (mm)
Centre	4.681
Mid-point of edge	2.5
Corner	0.9426

Table 0-27 Displacement results for Model 3

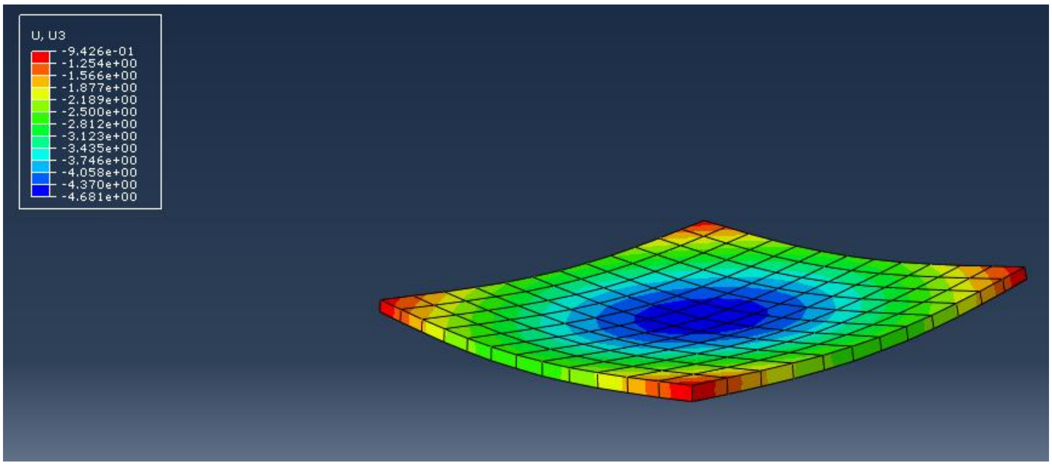

Figure 0.27 Displacement contour Model 3

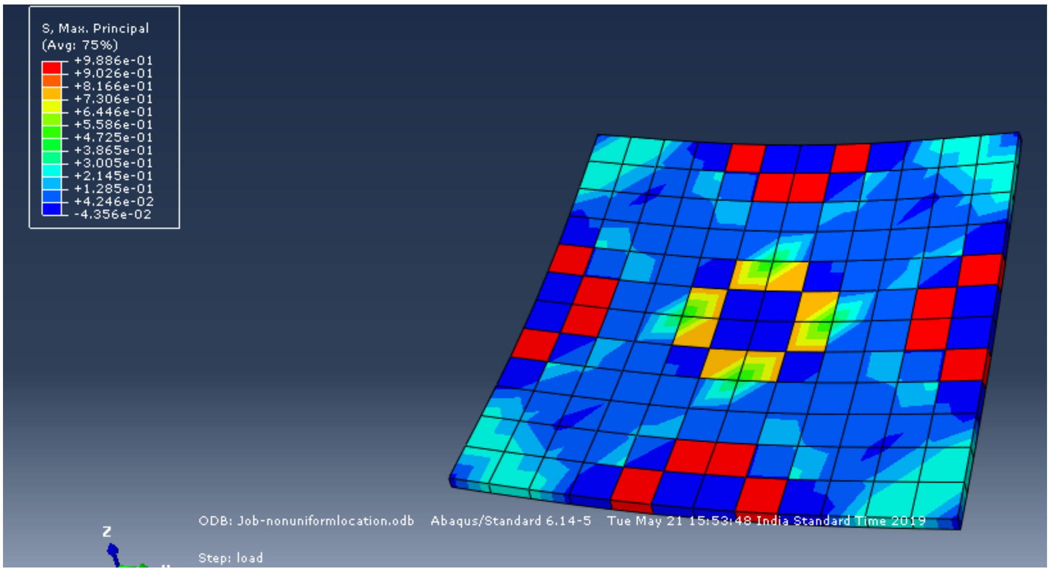

Figure 0.28 Maximum Principal stress contour Model 3

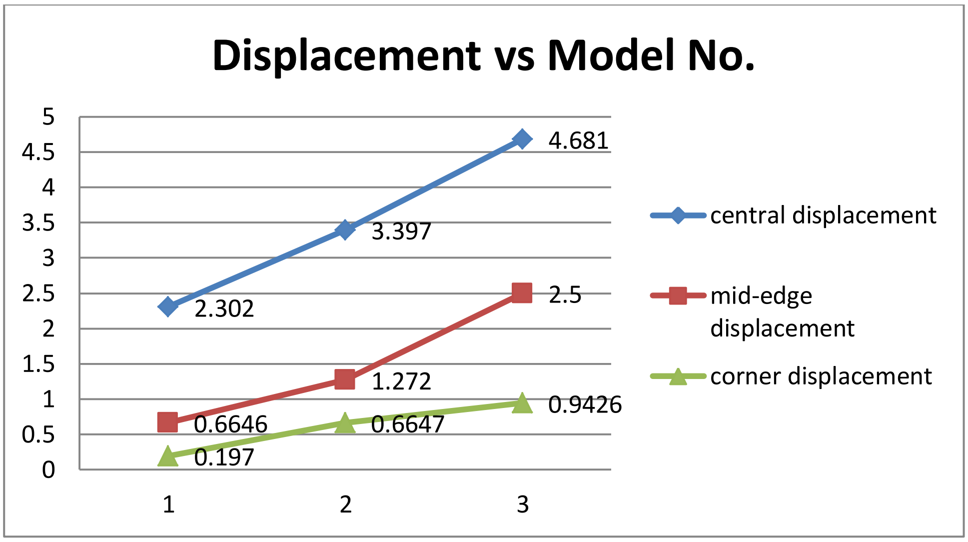

Figure 0.29 Variation of displacement vs model number

Percentage change	Model 1-2	Model 2-3	Model 3-1
Central displacement	47.57%	37.8%	103.34%
Mid-point of edge displacement	91.39%	96.54%	276.17%
Corner displacement	237.41%	41.81%	378.48%

Table 0-28 Percentage change in displacement with location

5.3 **VALIDATION**

To validate the results coming from present finite element analysis using ABAQUS, the results i.e. the displacements at central point, midpoint along edges and corner points obtained from ABAQUS are compared with the results coming from the models prepared using the software STAAD Pro. Though it is well established that STAAD Pro is not a finite element package in all respect, the comparison has been done to establish the approximate accuracy of the present results. For the purpose of validation, four models are analysed and compared. Out of these, two models (one with uniform location of springs along with uniform spring constants and other with non-uniform location of springs along with non-uniform spring constants) are considered in Set:1 where the mesh size is 200mm x 200mm. Same has been done in set:2 with mesh size 400mm x 400mm. The results i.e the displacement values at different locations are tabulated in Table:5-35,5-36,5-37 and in Table:5-39,5-40,5-41.

It is observed that the corner displacements are showing least variation and the central displacements are showing maximum variation in both set of results i.e. it is independent of mesh size. Also, the models with non-uniform location of springs along with non-uniform spring constants are showing more variation of displacement at all location compared to the model having uniform location of springs along with uniform spring constants. As the maximum difference obtained in the displacement values are considerably small (in the range of 10%), the results coming from the present ABAQUS models can be considered as realistic and can be used for other numerical experiments.

5.3.1 CONCENTRATED LOAD COMBINATION 1

5.3.1.1 SET 1 – 200mm X 200mm MESH

Slab	20m x 20m x 0.8m
Reinforcement	12ϕ HYSD

Table 0-29 Dimension details

Material parameters	Density (kg/m^3)	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	2400	27386.13	0.2
Steel	7800	210000	0.2

Table 0-30 Material parameters

Load	Magnitude (kN)	Location
Load 1 (Q)	400	At the centre
Load 2 (Q/2)	200	At the mid-point of the edges
Load 3 (Q/4)	100	At the 4 corners

Table 0-31 Load details

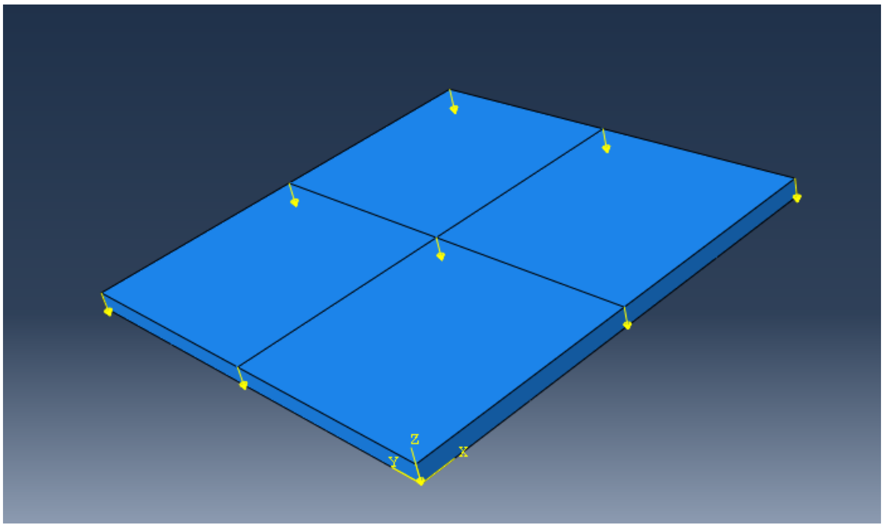

Figure 0.30 Load location

Springs in Model 1	Spring constant (N/mm)	Spacing
Outer zone	20000	10m (total 8nos.)
Intermediate zone	15000	7.5m (total 8nos.)

Inner zone	10000	5m (total 9nos.)

Table 0-32Spring details *(Slab zoning according to [24]) *(Spring constant according to [23])

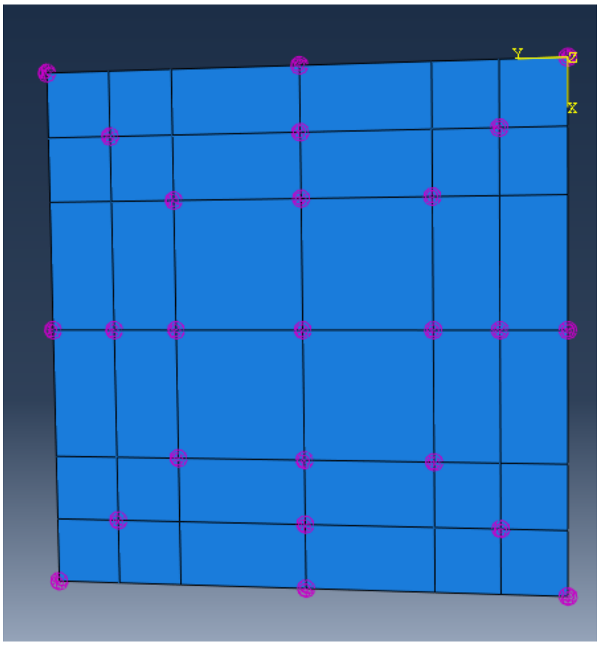

Figure 0.31Spring locations for Model 1

Springs in Model 2	Spring constant (N/mm)	Spacing
Uniform throughout	10000	5m (total 25nos.)

Table 0-33Spring details for Model 2 *(No zoning of the slab)

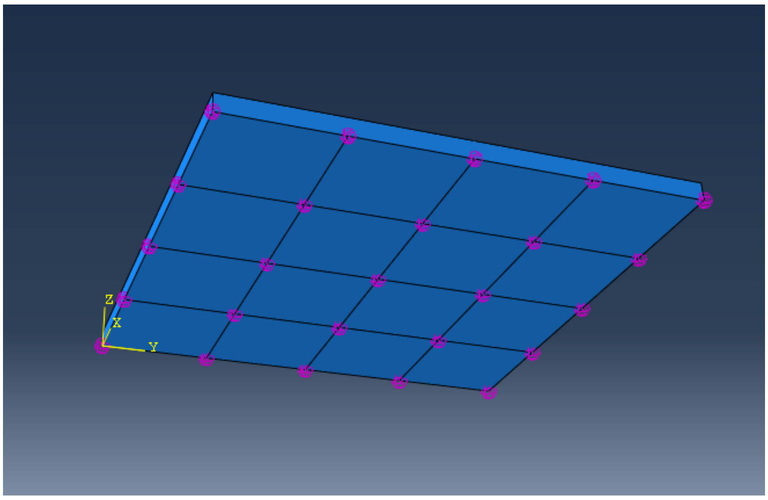

Figure 0.32Spring locations for Model 2

Mesh	Mesh element size
Model 1	200mm x 200mm

Model 2	200mm x 200mm

Table 0-34 Mesh details for SET 1

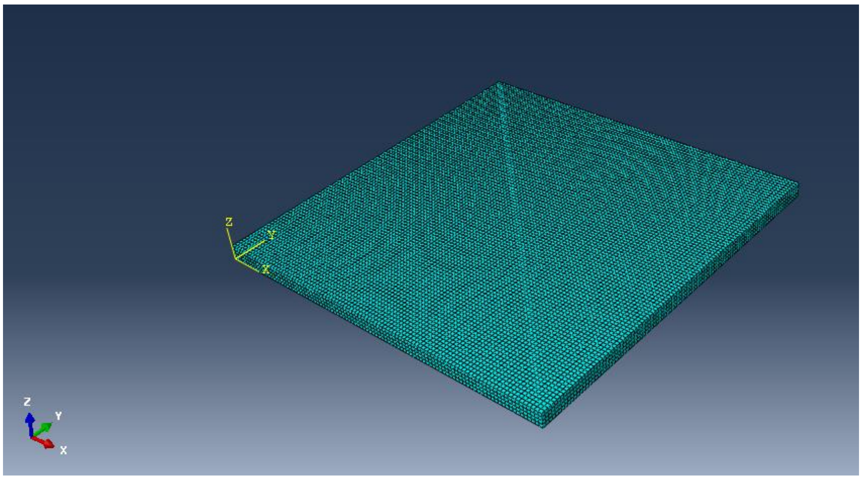

Figure 0.33 Meshing using 200mm x 200mm mesh element

Model in ABAQUS	Central displacement (mm)	Mid-point of edge displacement (mm)	Corner displacement (mm)
Model 1	4.5	4.335	4.216
Model 2	6.562	6.448	6.334

Table 0-35 Displacement results for SET 1 from ABAQUS

Model in Program	Central displacement (mm)	Mid-point of edge displacement (mm)	Corner displacement (mm)
Model 1	4.964	4.632	4.099
Model 2	6.984	6.787	6.359

Table 0-36 Displacement results for SET 1 from STAAD Pro

Percentage change w.r.t ABAQUS	Central displacement	Mid-point of edge displacement	Corner displacement
Model 1	10.31%	6.85%	2.77%
Model 2	6.43%	5.26%	0.40%

Table 0-37 Variation in displacement results from ABAQUS and STAAD Pro at different locations for SET 1

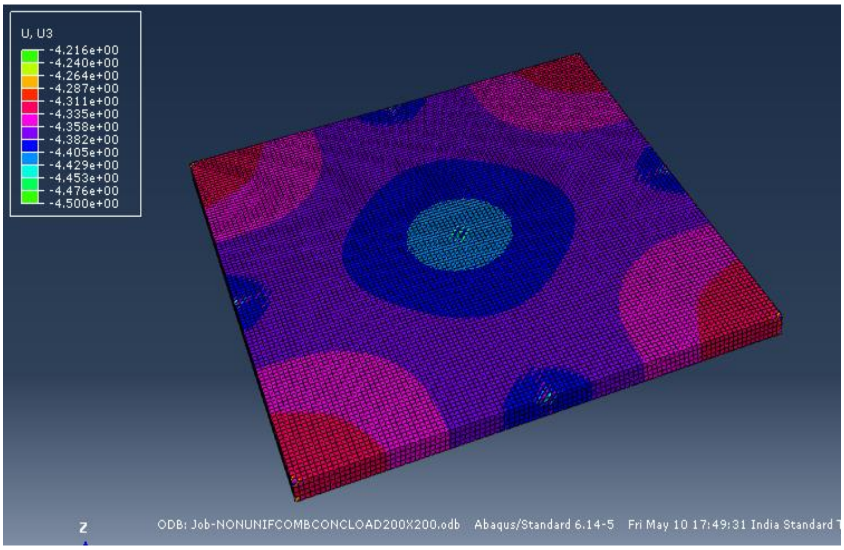

Figure 0.34 Displacement contour for Model 1

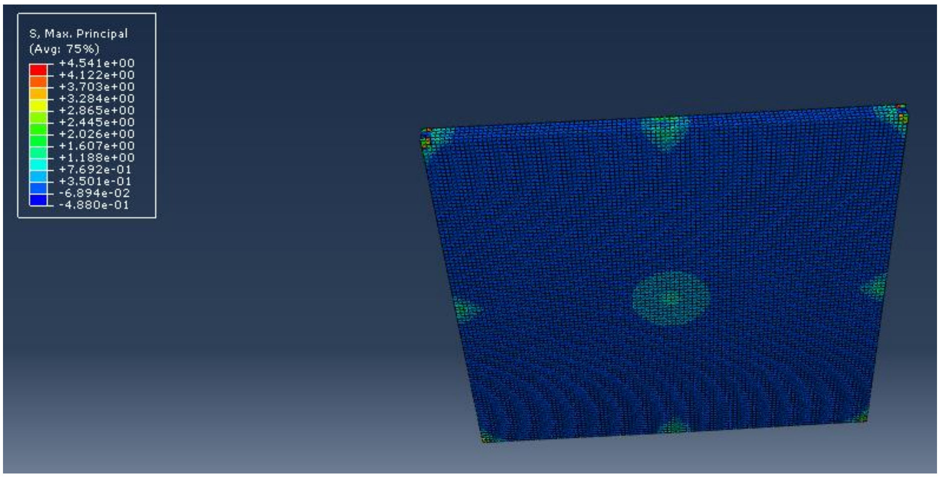

Figure 0.35 Maximum Principal stress contour for Model 1

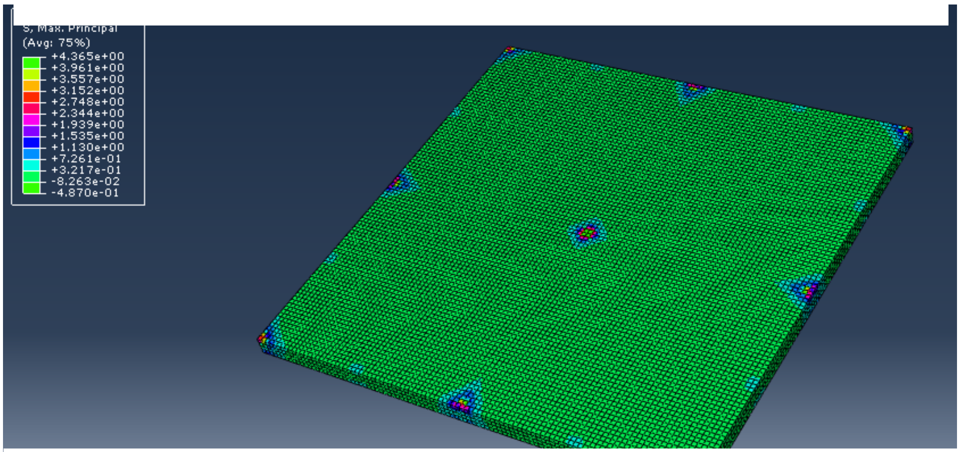

Figure 0.36 Maximum Principal stress contour for Model 2

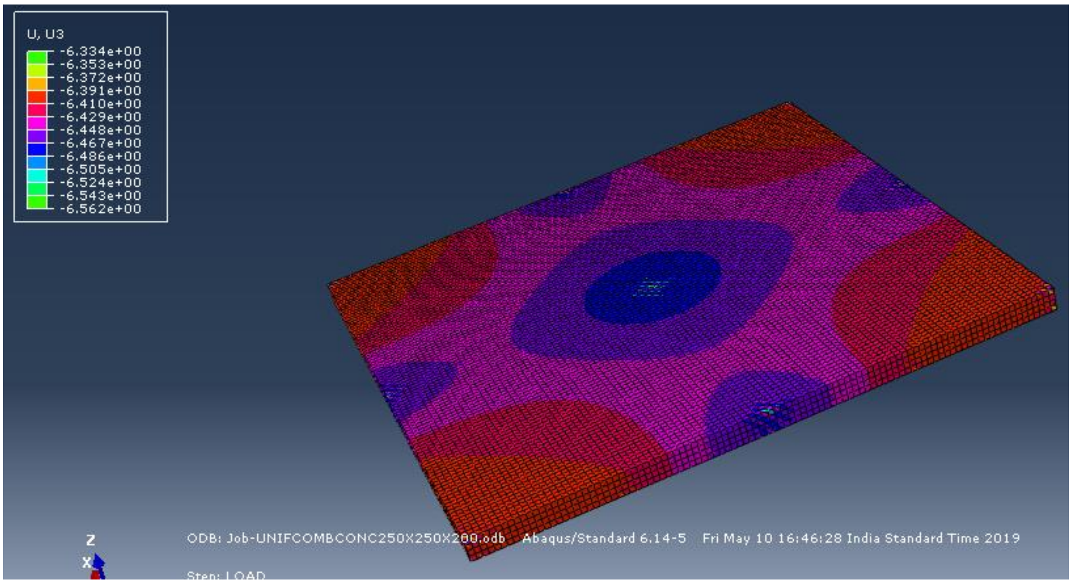

Figure 0.37 Displacement contour for Model 2

5.3.1.2 SET 2 – 400 X 400 MESH

The dimension of the slab and reinforcement, load details and spring details are same as SET 1. The only difference is in the size of the mesh element.

Model	Mesh element size
Model 1	400mm x 400mm
Model 2	400mm x 400mm

Table 0-38 Mesh details for SET 2

Model	Central displacement (mm)	Mid-point of edge displacement (mm)	Corner displacement (mm)
Model 1	4.465	4.33	4.235
Model 2	7.307	6.514	6.117

Table 0-39 Displacement results for SET 2 from ABAQUS

Model	Central displacement (mm)	Mid-point of edge displacement (mm)	Corner displacement (mm)
Model 1	4.933	4.589	4.029
Model 2	7.000	6.767	6.203

Table 0-40 Displacement results for SET 2 from STAAD Pro

Percentage change w.r.t ABAQUS	Central displacement	Mid-point of edge displacement	Corner displacement
Model 1	10.5%	5.98%	4.86%
Model 2	4.2%	3.88%	1.406%

Table 0-41 Variation in results from ABAQUS and STAAD Pro at different locations for SET 2

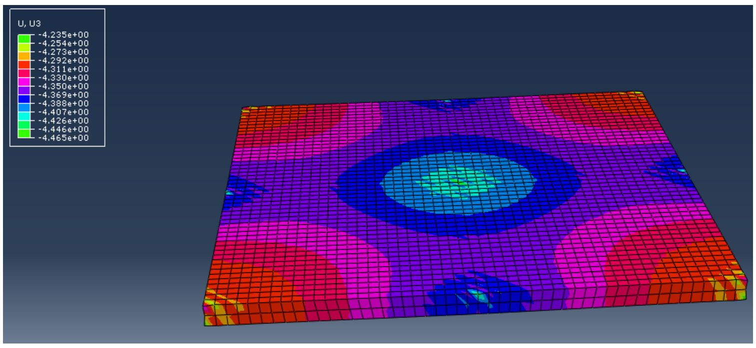

Figure 0.38 Displacement contour Model 1

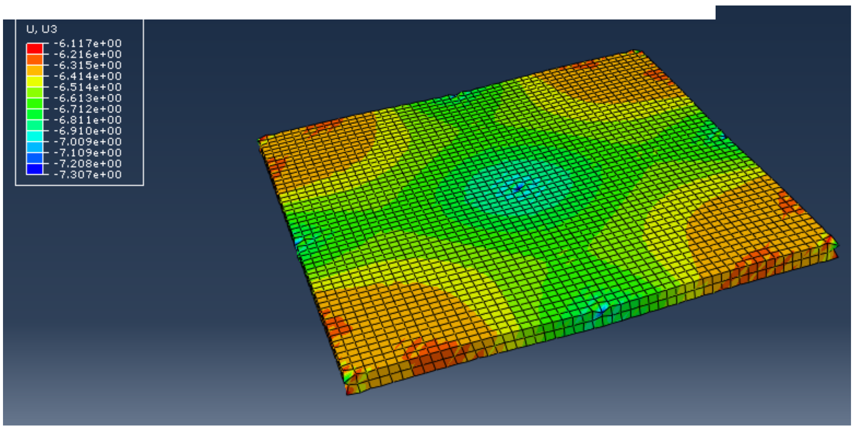

Figure 0.39 Displacement contour for Model 2

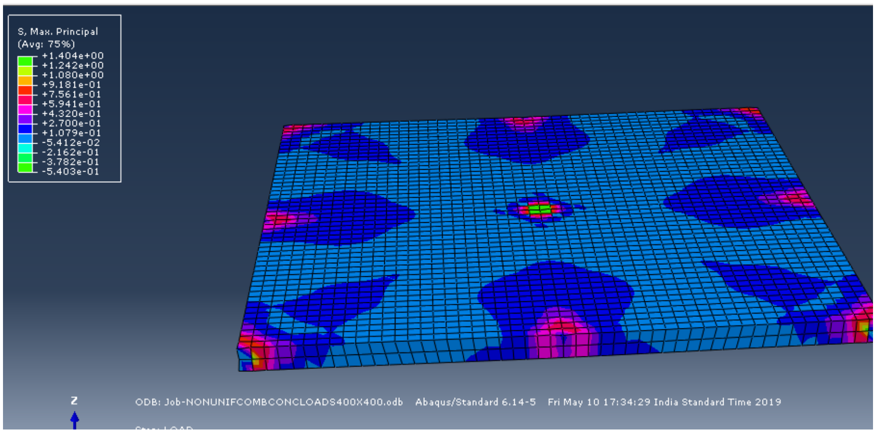

Figure 0.40 Maximum Principal stress contour Model 1

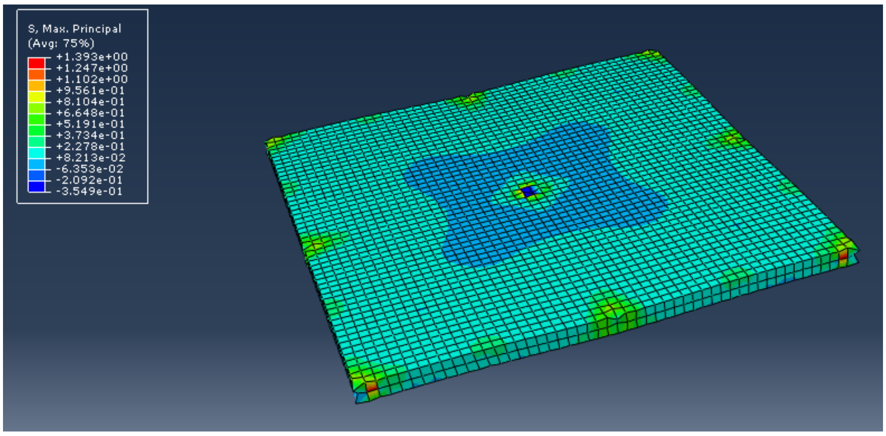

Figure 0.41 Maximum Principal stress contour Model 2

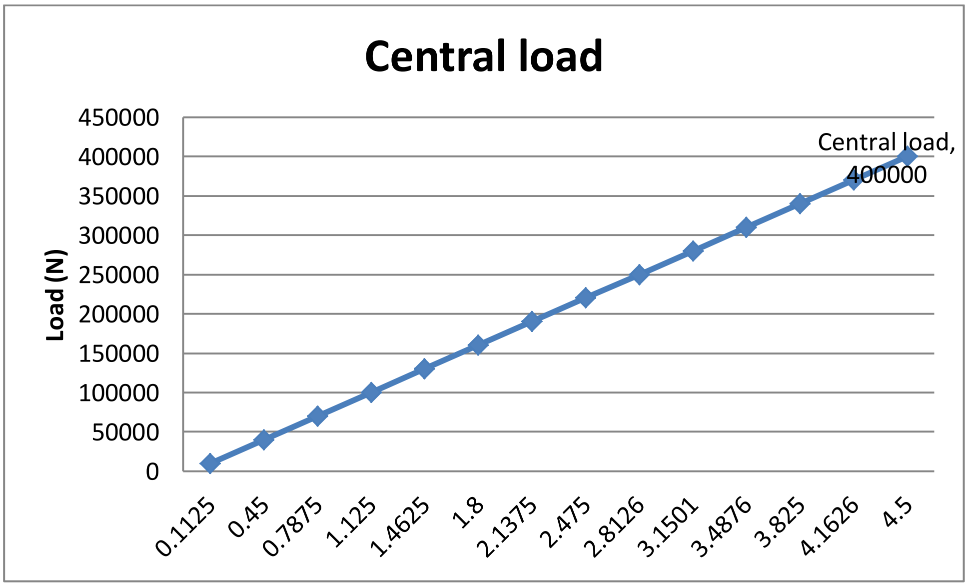

Figure 0.42 Load vs Displacement for Model 1

5.4 CRACK PROPAGATION

In this section, it has been attempted to see how an initial crack progresses through the bulk concrete. Modelling a crack can be done using Contour Integral, XFEM and Virtual Crack Closure Technique (VCCT) in ABAQUS. In this work the XFEM method is relied upon. Here the crack has been modelled as a separate part using the 3D Shell element and superposed onto the main model. Concrete damage plasticity model data mentioned in Tables 4-2, 4-3 and 4-4 were used for all the models. In addition to that Damage for Traction-Separation law using Maximum Principal stress criterion was also used. The models 5.4.1 (cantilever beam), 5.4.2.1 (slab with uniform spring constants over uniform locations), 5.4.2.2 (slab with rigid pin support along the edges only) and 5.4.3 (slab with rigid pin support along the edges only) displayed convergence errors during analysis. On removing the Damage Plasticity parameters, it showed the convergence in the results. Hence the results tabulated here are based on Maximum Principal stress criterion only. Model 5.4.1 shows prominent crack propagation path from the top surface towards the bottom surface of the beam. In Models 5.4.2.1 and 5.4.2.2 the initial crack location has been given at the middle of one edge. In both the models, it has been observed that the propagation of that crack was towards the centre of the slab, but in case of spring supported slab, the length of propagation is more compared to the pinned supported slab under same external load. In Model 5.4.3 the initial crack location has been given near one corner and it is observed that the zone of cracked concrete is shown to be extended towards the centre of the slab along the diagonal direction. It is also observed that along with the above crack zone, the cracks are also observed at the other three corners representing a realistic cracking profile for this slab model. The displacement profiles along the central line are also plotted for the above mentioned slabs and showing reasonably accurate variation.

5.4.1 CANTILEVER BEAM

Dimension	0.2m x 0.3m x 1m

Table 0-42 Dimension of Cantilever beam

Parameter	Density (kg/m^3)	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	2400	29100	0.2

Table 0-43 Material parameters for Concrete

Crack	3D Shell Extrusion element	200mm x 50mm

Table 0-44 Crack details

Load/Boundary condition	Load/BC type	Magnitude
Load 1	Pressure load	2000 (N/mm^2)
Boundary condition 1	Encastre	U1=U2=U3=UR1=UR2=UR3=0

Table 0-45 Load and Boundary condition details

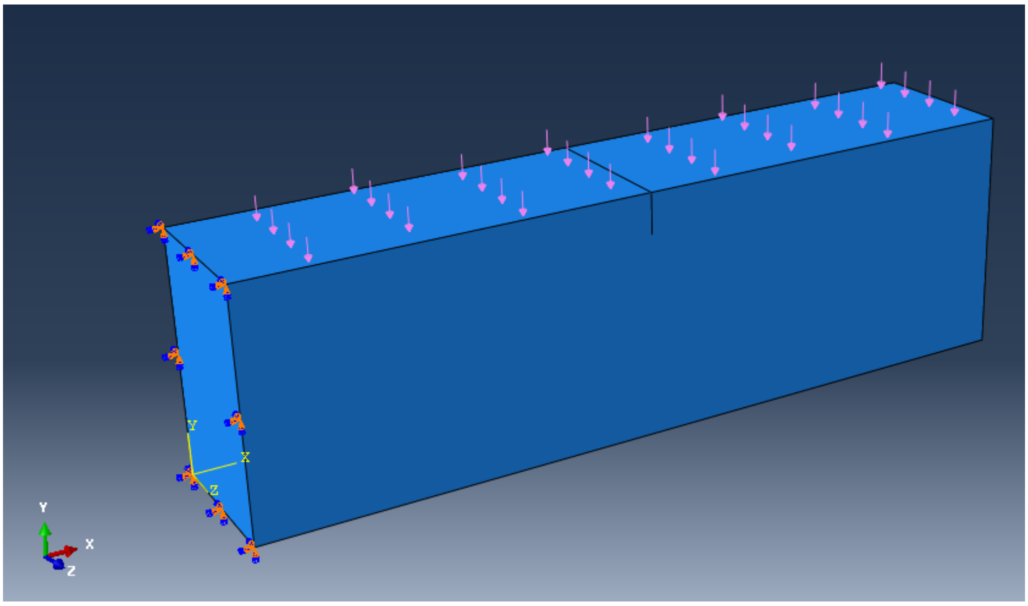

Figure 0.43 Fixed end and Pressure load along with Crack element

Mesh element size	50mm x 50mm x 50mm

Table 0-46 Mesh size

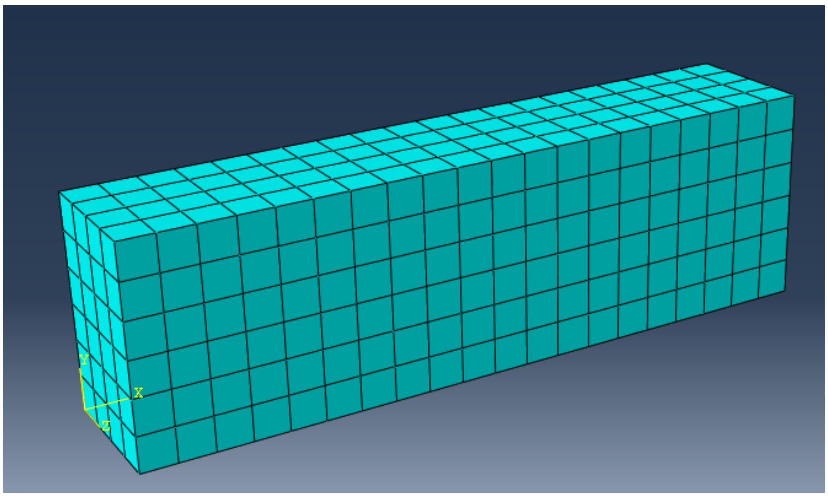

Figure 0.44 Meshed beam

Displacement	Magnitude (mm)
Edge displacement	1.205
Middle displacement	0.7049

Table 0-47 Displacement results

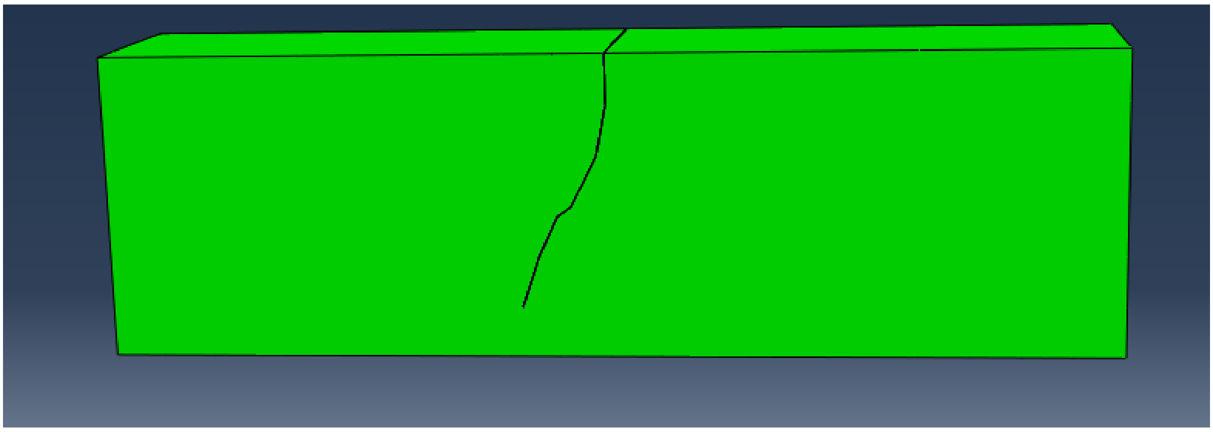

Figure 0.45 Fully propagated crack

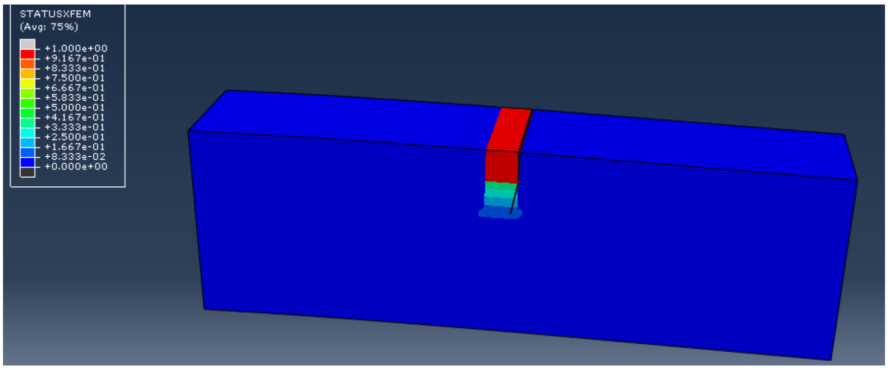

Figure 0.46 XFEM Status 1

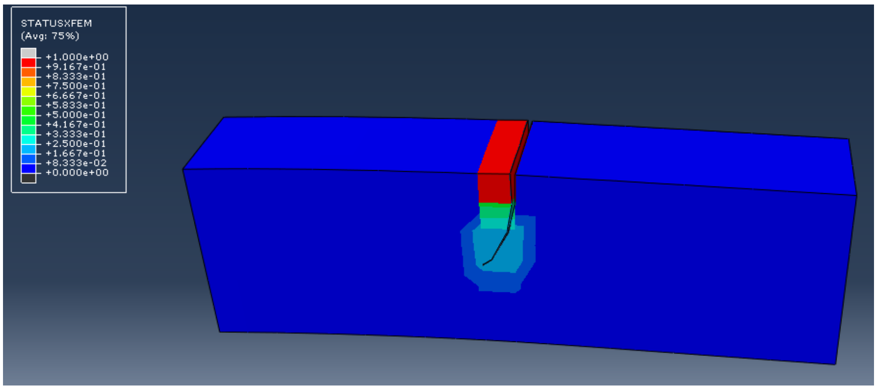

Figure 0.47 XFEM Status 2

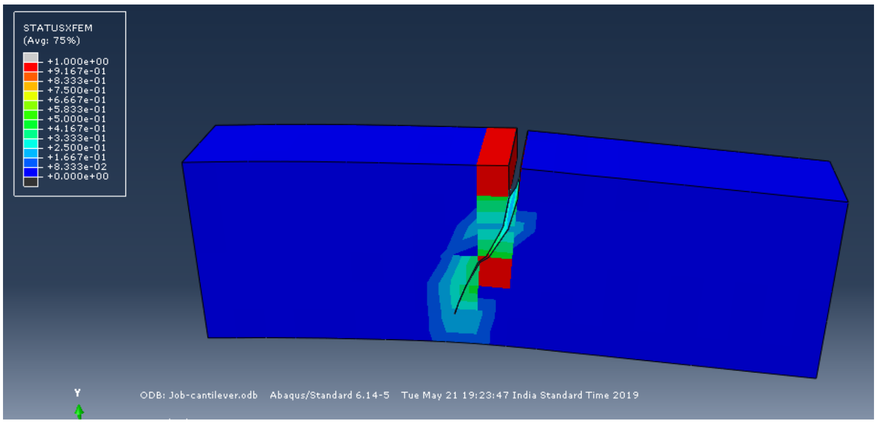

Figure 0.48 XFEM Status 3

5.4.2 SLAB SUBJECTED TO CONCENTRATED LOADS

5.4.2.1 SUPPORTED OVER SPRINGS

Dimension	3m x 3m x 0.3m
Reinforcement	12ϕ HYSD

Table 0-48 Dimension details

Parameter	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	27386.13	0.2
Steel	210000	0.3

Table 0-49 Material parameters

Maxps damage criterion under Mechanical Traction Separation law was opted for while defining the material parameters. Here the maximum principal stress was kept as 5 MPa and displacement at failure was 50mm.

Load	1500 kN	750mm spacing (total 7nos.)
Spring	10000 N/mm	750mm spacing (total 25nos.)

Table 0-50 Load and Spring details

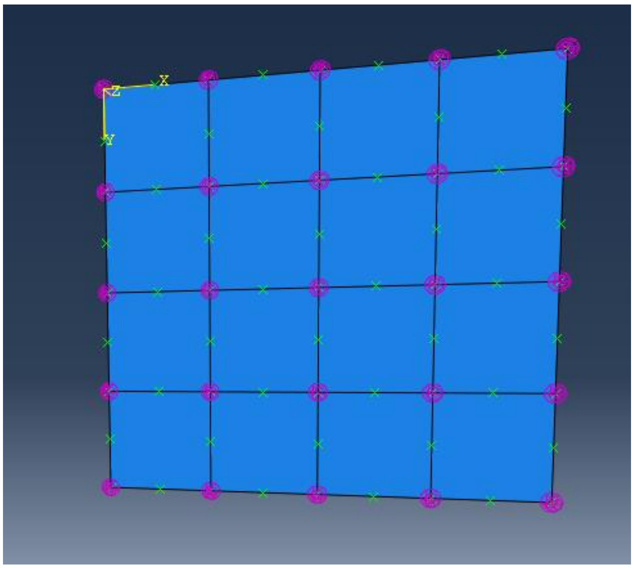

Figure 0.50Spring locations

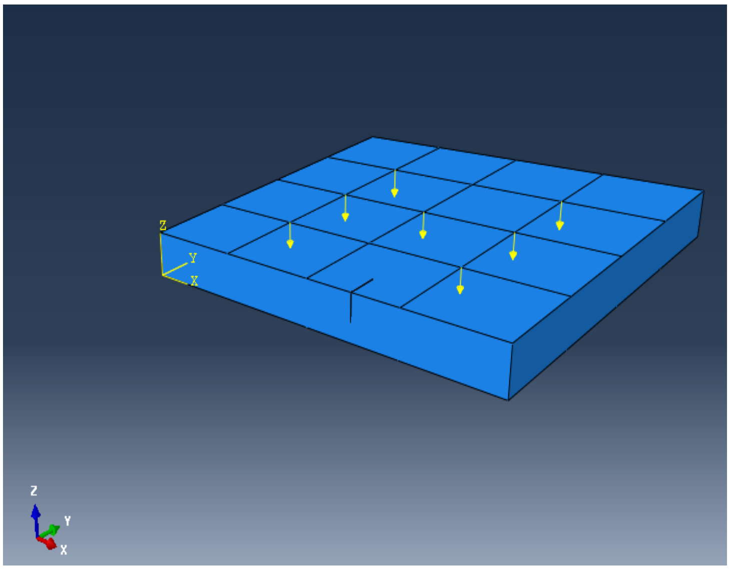

Figure 0.49 Load locations and Initial crack location

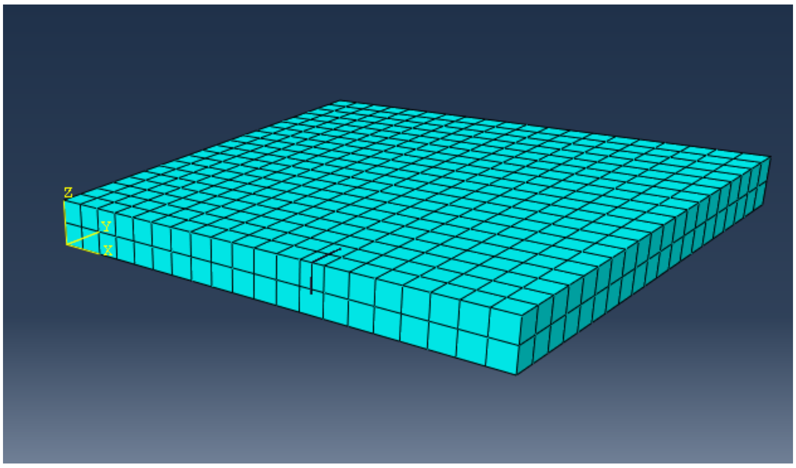

Figure 0.51 150mm x 150mm Mesh

Location	Centre (mm)	Mid-point of edge (mm)	Corner (mm)
Displacement	9.768	5.498	2.448

Table 0-51 Displacement results

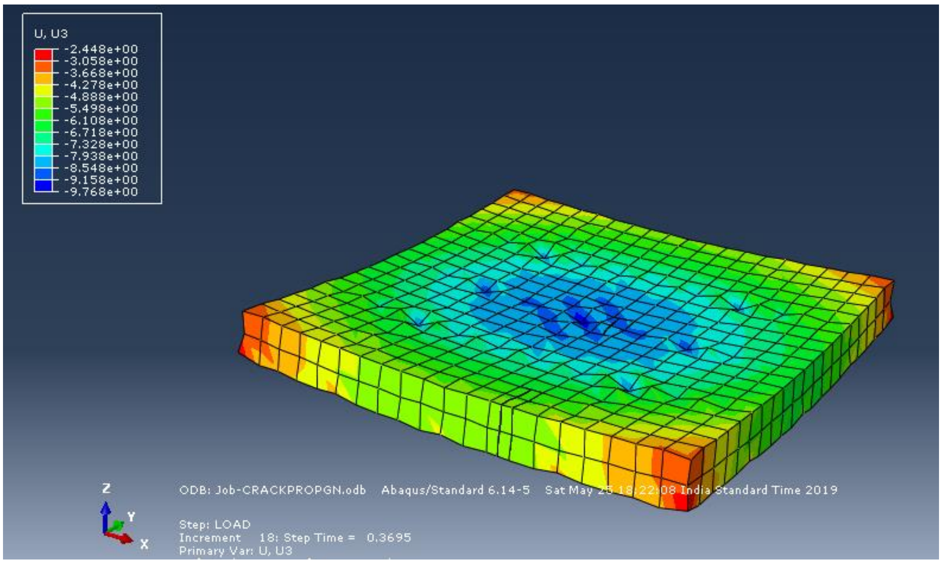

Figure 0.52 Displacement contour

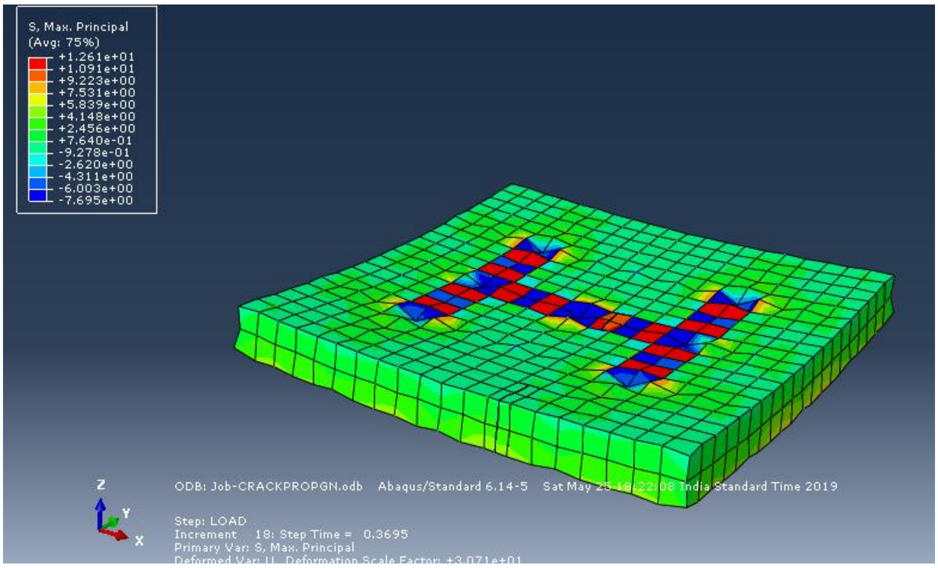

Figure 0.53 Maximum Principal stress contour

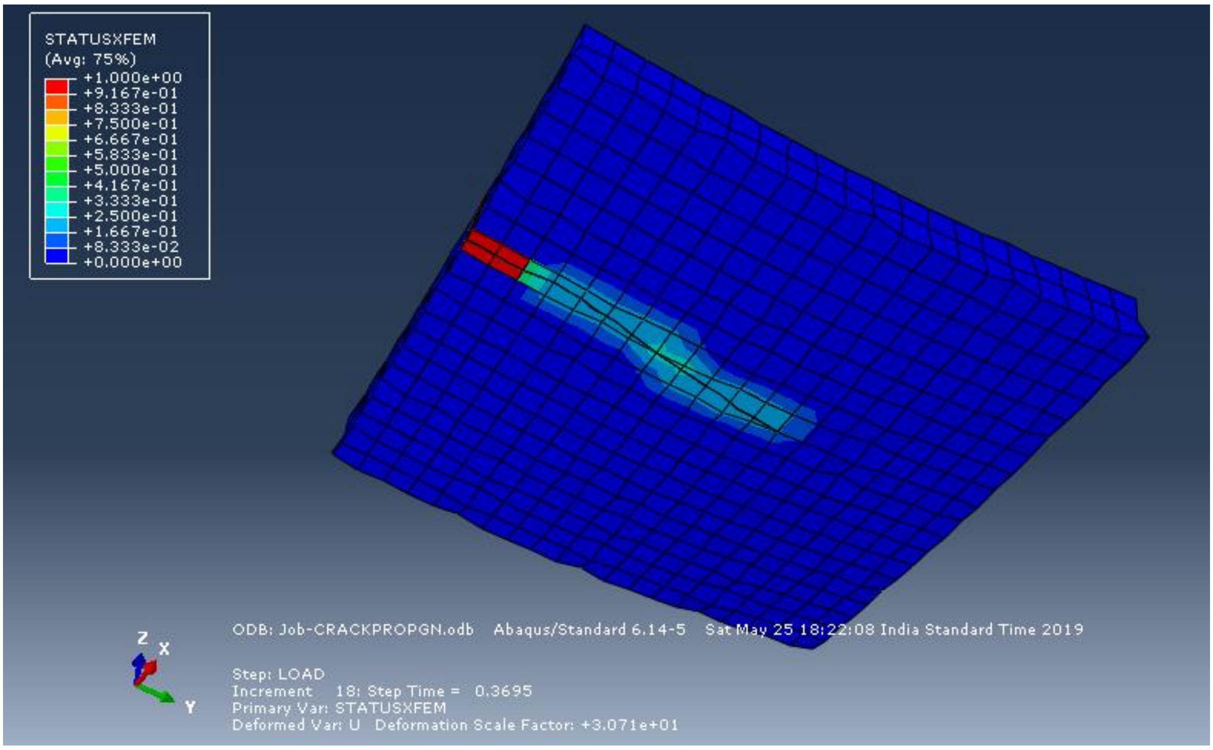

Figure 0.54 Full propagation of the crack at the base of the slab

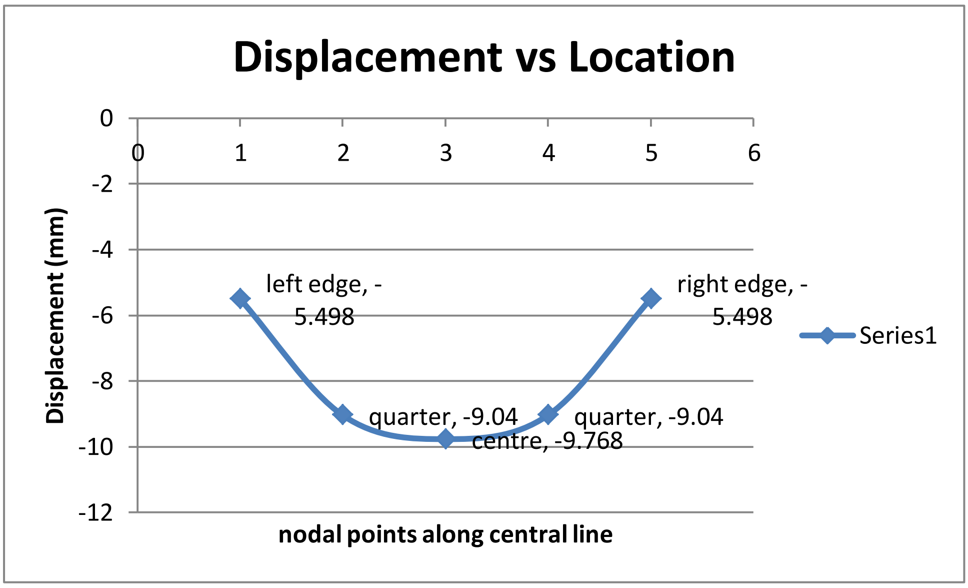

Figure 0.55 Displacement vs location plot

5.4.2.2 SUPPORTED OVER PIN SUPPORT

Dimension, material, load and mesh conditions are same as the slab in 5.4.2.1. The only difference lies in the support conditions, wherein this slab lies on pin supports at various nodes.

Boundary condition	Pinned support U1=U2=U3=0	750mm spacing (total 16nos.)

Table 0-52 Support condition

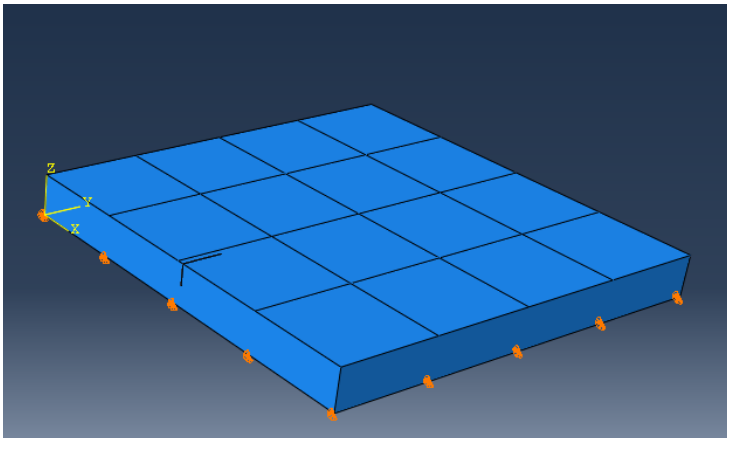

Figure 0.56 Pin supports

Location	Centre (mm)	Mid-point of edge (mm)	Corner (mm)
Displacement	8.151	2.894	-0.861

Table 0-53 Displacement results

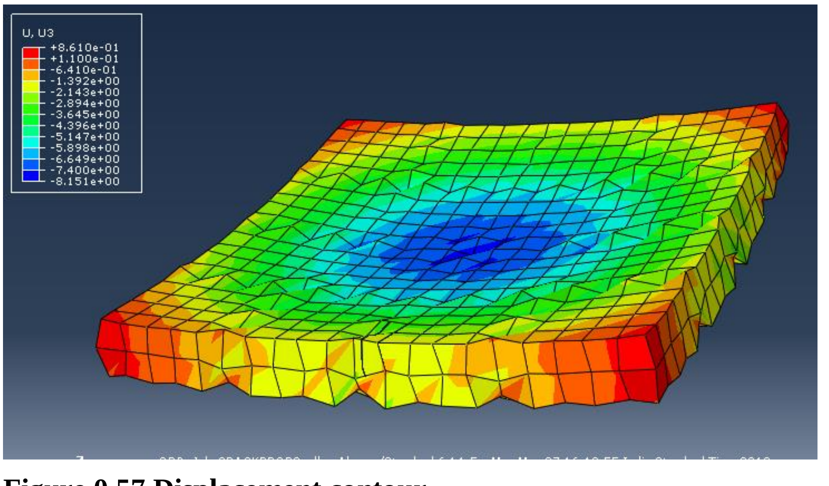

Figure 0.57 Displacement contour

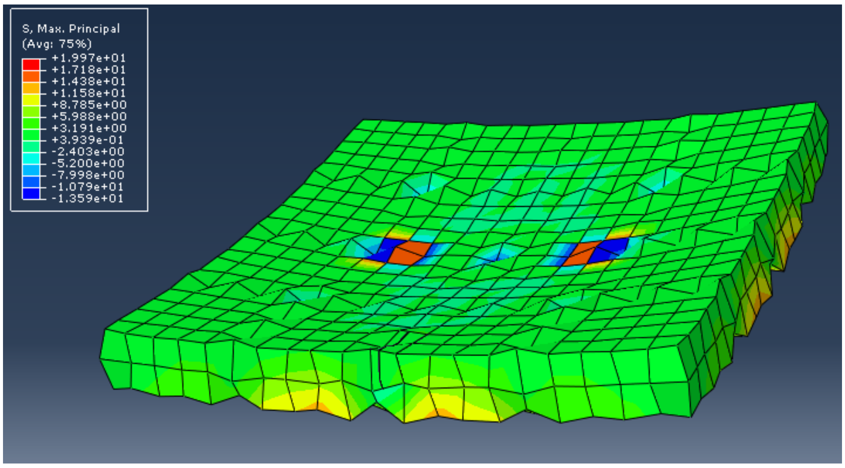

Figure 0.58 Maximum Principal stress contour

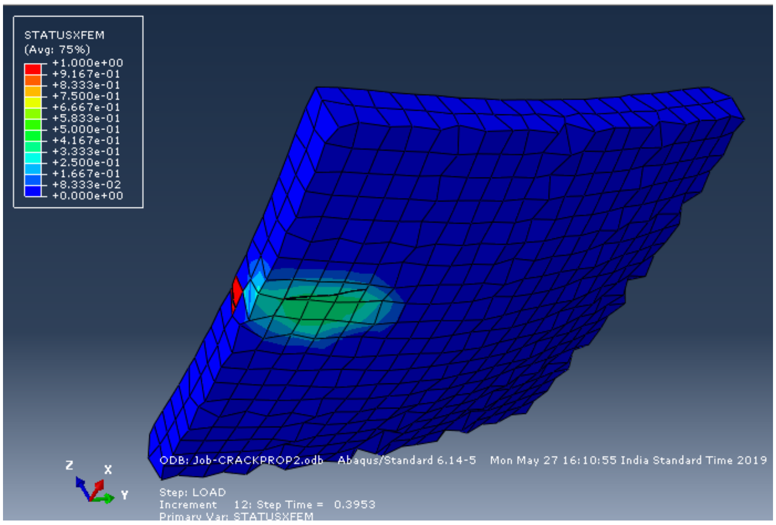

Figure 0.59 Fully propagated crack at the base of the slab

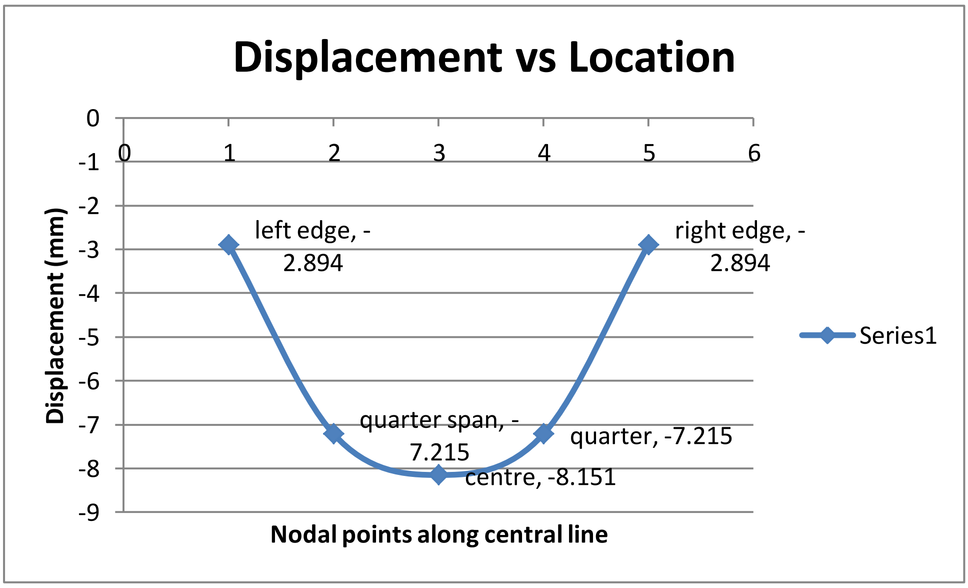

Figure 0.60 Displacement vs Location plot

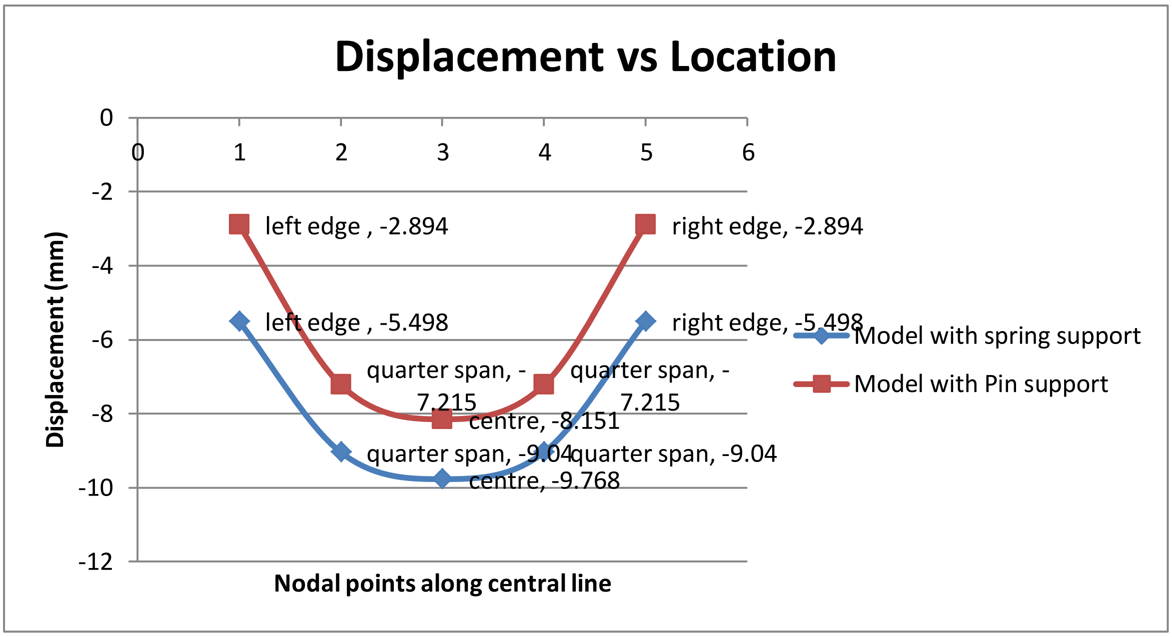

Figure 0.61 Comparison of nodal displacements for Model 1 and 2

Location	Centre	Mid-point of edge	Corner
Percentage change w.r.t Model 5.4.2.1	-16.55%	-47.36%	+135.17%

Table 0-54 Variation in displacement with location

5.4.3 PINNED SLAB WITH CRACK

Dimension	1.83m x 1.83m x 0.051m
Reinforcement	12ϕ HYSD

Table 0-55 Dimension details

Parameter	Young's Modulus of Elasticity (MPa)	Poisson's ratio
Concrete	27386.13	0.2
Steel	210000	0.3

Table 0-56 Material properties

Load	160 kN/m^2	Uniformly distributed load
Boundary condition	Pinned support U1=U2=U3=0	Supported on all the 4 edges

Table 0-57 Load and Boundary condition details

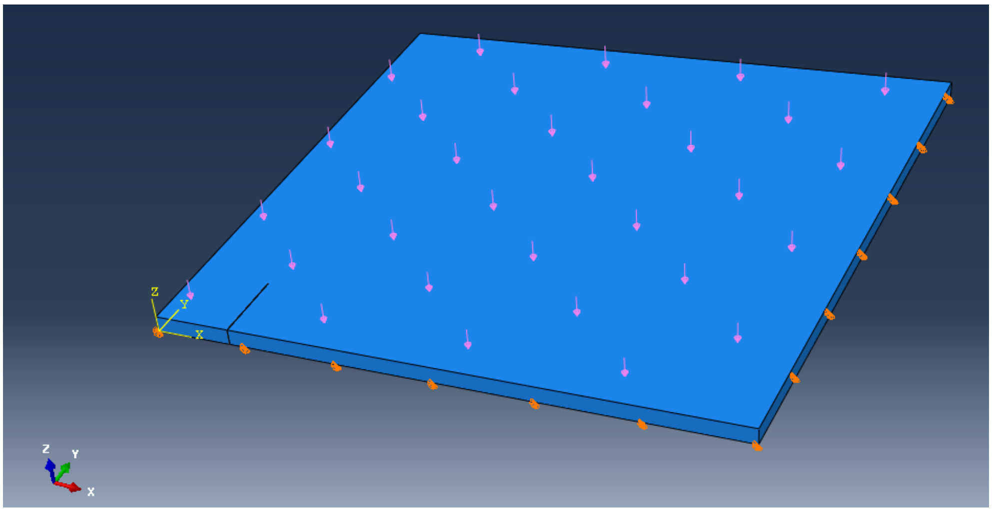

Figure 0.62 Load and support location

Mesh element size	152.5mm x 152.5mm x 51mm

Table 0-58 Mesh details

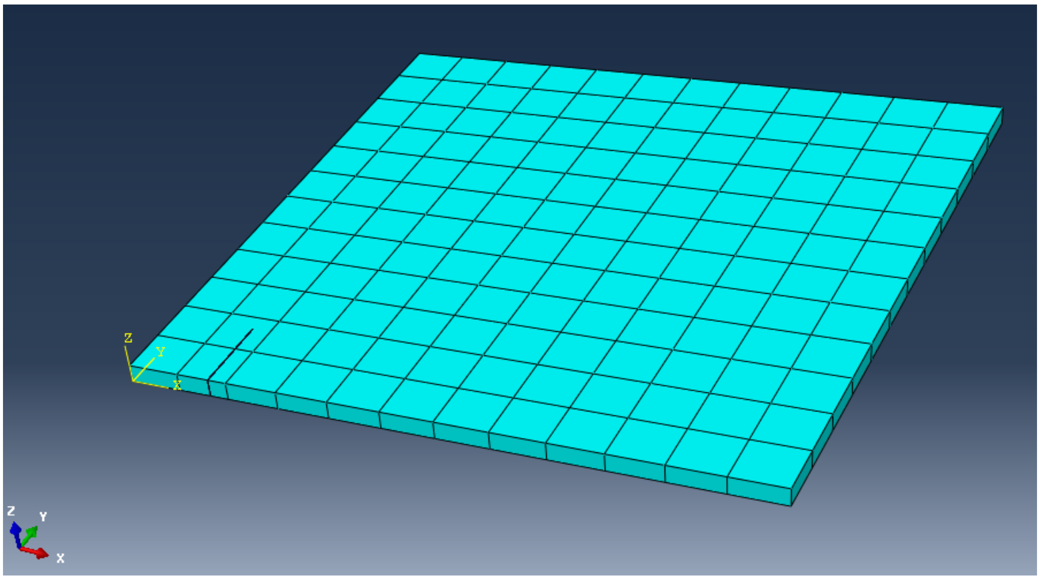

Figure 0.63 Meshed slab with location of initial crack

Location	Centre	Mid-point of edge	Corner
Magnitude (mm)	67.81	33.74	5.349

Table 0-59 Displacement results

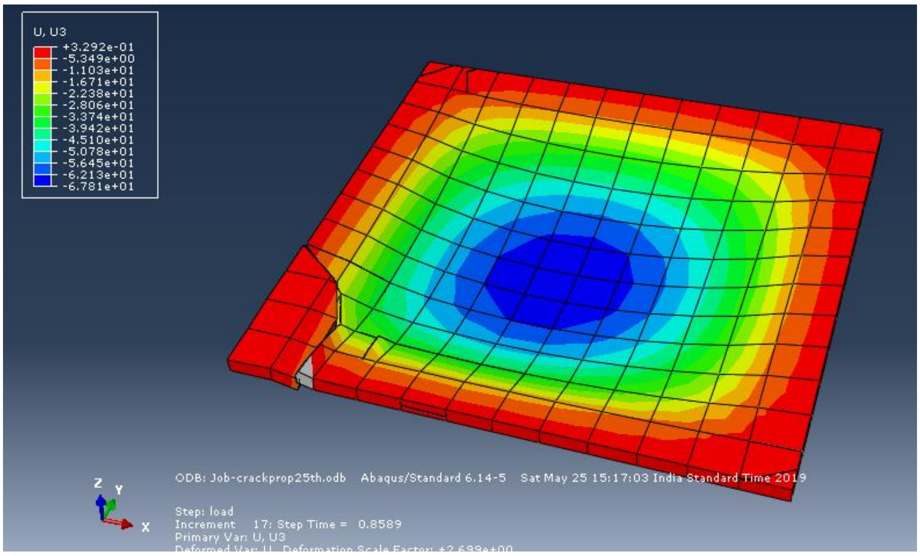

Figure 0.64 Displacement contour and location of cracks

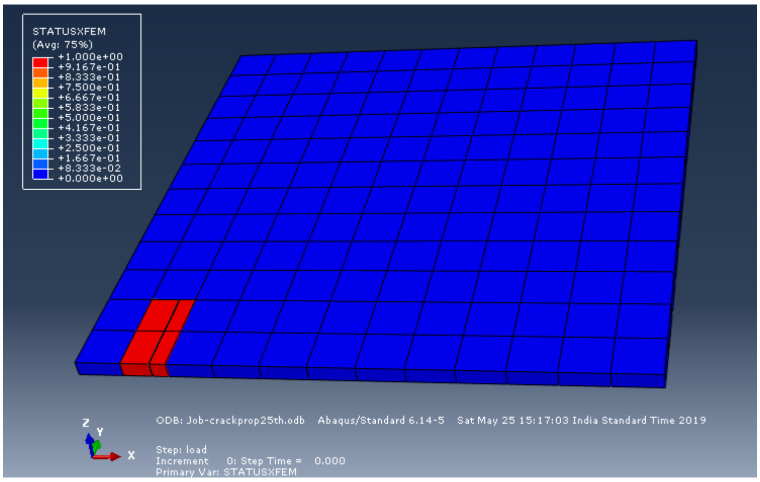

Figure 0.65 XFEM status 1

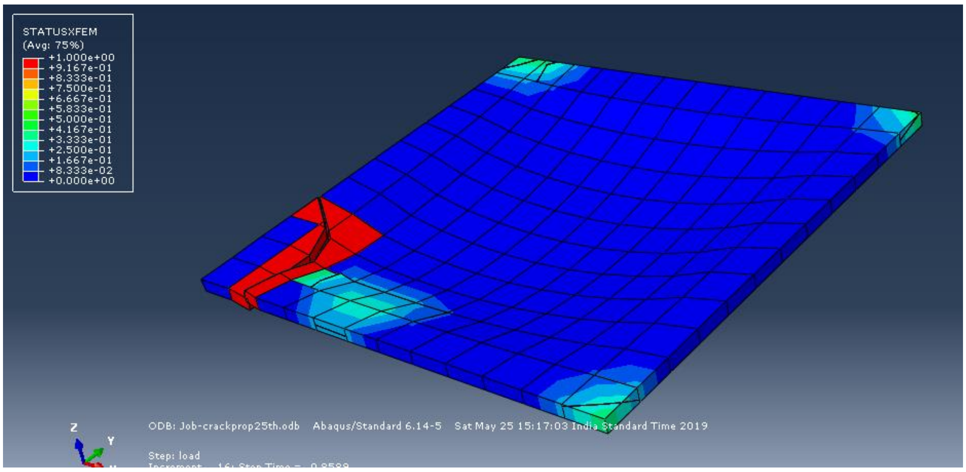

Figure 0.66 Final XFEM status

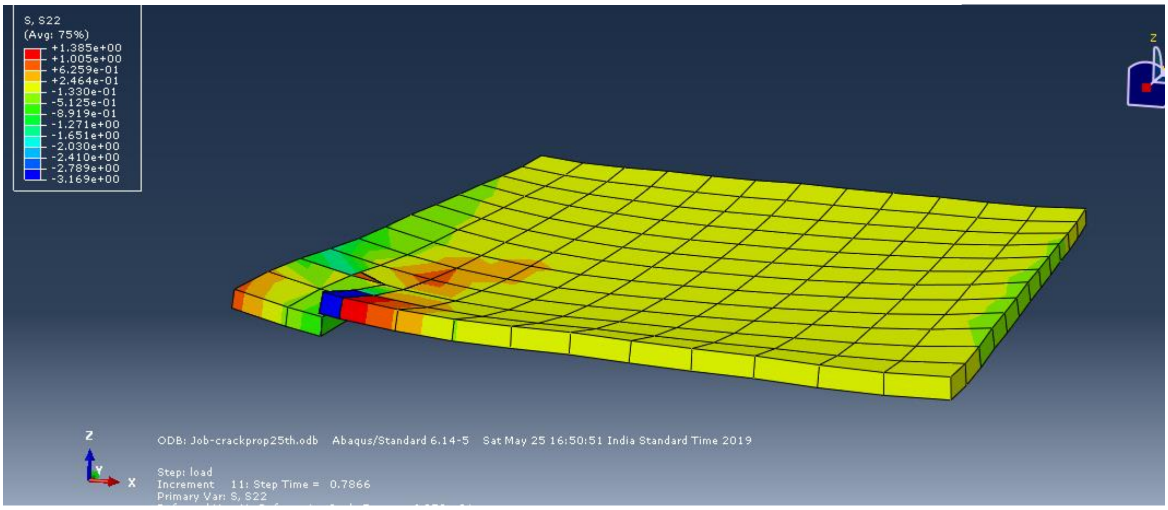

Figure 0.67 Sigma x contour

6 CONCLUSION

In the present research work it has been tried to analyse reinforced concrete slab supported over linear independent springs representing the subgrade modulus of the soil using Finite Element Method (FEM). For developing the FEM models and analysis, ABAQUS, a reputed FEM software has been used. 8-noded linear brick elements available in ABAQUS are used to prepare all slab models. Reinforcement bars are also modelled using the same element. The linear elastic properties of concrete and steel are used along with the parameters representing post-yield behaviour of both the materials. The slabs are considered to be subjected to uniformly distributed loads along with concentrated loads in different case studies. The spring supports are implemented at the selective nodal points to represent the linear elastic nature of the supporting medium in the transverse direction. The spring constants have not been chosen based on realistic soil parameter values. For the sake of this study, they have been chosen arbitrarily. As one of the main objectives was to study the crack propagation paths under the application of external loads, maximum principal stress criterion has been considered to implement the cracking failure of concrete.

Mesh convergence study has been undertaken with different mesh sizes and it has been observed that for the models considered 200mm x 200mm mesh size has been found ideal. To validate the proposed FEM model, the results obtained from ABAQUS for four models are compared with the same obtained from another software, STAAD Pro. It is evident from the comparison that the differences in the results are negligibly small establishing the acceptance of the present results. As it has been reported in different published literatures that the uniform distribution of spring constants does not give realistic deformational characteristics of these kind of slab, suggested variations of the spring constants along with its location are also considered as the parametric study. Both the variation in spring locations and spring constants are taken into account for this study. It reveals the fact that the displacement values are changing considerably with the change of spring constants and location, thereby indicating the need for proper consideration of these parameters for getting a true deformational representation. Lastly, the crack propagation study has been done to identify the difference between rigid supported slab and spring supported slab in this regard. It has been shown from different case studies that the crack propagation length is more in case of spring supported as slab compared to the rigid supported one, under the same load. This should be more carefully treated for the structural design of this spring supported slab.

As the Finite Element analysis of spring supported RC slab involves the complex behaviour of concrete, steel along with the characteristics of the springs representing the behaviour of elastic medium like soil including their interactions, the conclusions made above requires more number of case studies with the variation in different parameters accounted for in order to get more definite and specific observations.

7 REFERENCES

1. Bulut, R., "Finite Element Method Analysis of Slabs on Elastic Half-space Expansive Soil Foundations", Dissertation submitted to the Office of Graduate studies of Texas A&M University, 2001.
2. Fattah Y. Mohammed, Hamood Md., Abbas A. S., "Behaviour of Plate on Elastic Foundation under impact load", Engineering & Tech Journal, Volume 32, Part A No.4, 2014.
3. Shakir H. Z., Abbas A. S., "Effect of Soil Stiffness on Impact Resistance of concrete plate." Dec 2016
4. Fattah Y. Mohammed, Hamood J. Mohammed, Abbas A. S., "Dynamic response of plates on elastic foundation under eccentric impact load", Matec Web of Conferences, 120, 06006, 2017.
5. Malekova .V, Jendzelovsky, "An analysis of contact elements of foundation structures."
6. Hussam K. Risan, "Modal analysis of RC slab on elastic foundation", International Journal of Applied Engineering Research ISSN 0973-4562, Volume 11, No. 5, Pg 3151-3154, 2016.
7. Paiva de B. J., Mendonca V. A, "A boundary element method for static analysis of raft foundations on piles", Engineering Analysis with Boundary Elements, 24, Pg 237-247 2000.
8. Rashed F. Y, "A boundary/domain element method for analysis of building raft foundations", Engineering Analysis with Boundary Elements, 29, Pg 859-877, 2005.
9. Ribeiro B. D., Paiva de B. J., "Mixed FEM/BEM formulations applied to Soil-Structure interactions", Proceedings of World Congress Engineering, Volume 2, 2014.
10. Vlasov Z. V., Leontev N.N, "Beams, Plates and Shells on Elastic Foundations", 1966.
11. Pavlou G. D., Bancila R., Lucaci G., Belc F., Dan D., Pavlou G. M., Tirtea A., Gruin A., Baera C., "An exact solution of the plate resting on elastic foundation under impact loading", 3rd WSEAS International Conference on Applied and Theoretical Mechanics, Pg 79-83, 2007.
12. Kukreti R. A., Ko M., "Analysis of a rectangular plate resting on elastic half space using energy approach", Applied Math Modelling, Volume 16, Pg 338-356, 1992.

13. Rui Li, Ming Li Yang Zhong, "Analytic bending solution of free rectangular thin plates resting on elastic foundations by a new symplectic superposition method", Proceedings of Royal Society, 2013.
14. Kazakov K., Karamanski T., "Modified model of a plate based on the Winkler foundation", Computational Methods and Experimental Measurement, Volume 12, Pg 417-423, 2005.
15. Musat V., Vrabie M., Teodoru B. I., "A Finite Element study of the bending behaviour of beams resting on a two-parameter elastic foundation", Pg 7-19, 2006.
16. Civalek O., Yavas A., "Large deflection static analysis of rectangular plates on two parameter elastic foundations", International Journal of Science, Volume 1, No 1, Pg 43-50, 2006.
17. Motaghian S., Mofid M., Akin E. J., "On the free vibration response of rectangular plates, partially supported on elastic foundation", Applied Mathematical Modelling, 36, Pg 4473-4482, 2011.
18. Rajpurohit K. V., Gore G. N., Sayagavi G. V., "Analysis of structure supported on elastic foundation", International Journal of Engineering and Advanced Technology, Volume 4 Issue 1, 2014.
19. Loukidis D., Tamiolakis G. P., "Spatial distribution of Winkler spring stiffness for rectangular mat foundation analysis", Engineering Structures, 153, Pg 443-459, 2017.
20. Daloglu A.T., VallabhanGirija C.V., "Values of k for slab on Winkler Foundation", Journal of Geotechnical and Geo-environmental Engineering, Vol. 126 No. 5, May, 2000.
21. Dey A., Basudhar P.K., Chandra S., "Distribution of Subgrade Modulus beneath Beams on Elastic Foundations", Indian Geotechnical Journal 41(2), 2011, Pg 54-63.
22. Bowles J.E., "Foundation Analysis and Design", 5th edition McGraw-Hill, 1997.
23. ACI Committee 336, "Suggested analysis and design procedures for combined footings and mats", American Concrete Institute (ACI), Pg 336.2R 1 – 336.2R 21, 1988.
24. Coduto D.P., "Foundation Design - principles and practices", 2nd edition Prentice Hall, 2000.
25. Wahalathantri B.L., Thambiratnam D.P., Chan T.H.T, Fawzia S., "A Material Model for Flexural crack simulation Reinforced Concrete

elements using ABAQUS”, Proceedings of the First International Conference on Engineering, Designing and Developing the Built Environment for Sustainable wellbeing, 2011, Pg 260-264.

26. Alfarah B., Lopez Almansa F., Oller S., “New Methodology for calculating damage variables evolution in Plastic Damage model for RC Structures”, Engineering Structures (132), February 2017, Pg 70-87.
27. Vallabhan C. V. Girija, and Turhan Daloglu A., “CONSISTENT FEM-VLASOV MODEL FOR PLATES ON LAYERED SOIL”, JOURNAL OF STRUCTURAL ENGINEERING, / JANUARY 1999, pp108
28. O¨zdemir, Y.I., “Development of a higher order finite element on a Winkler foundation”, Finite Elements in Analysis and Design. 2012, v.48,pp 1400-1408
29. Mohammed Y. Fattah, Mohammed J. Hamood, Sura A. Abbas, “Behavior of Plate on Elastic Foundation under Impact Load”,*Eng. & Tech. Journal , Vol.32,Part (A), No.4, 2014*
30. Joaquim Barros, ANALYSIS OF CONCRETE SLABS SUPPORTED ON SOIL, MÉTODOS NUMÉRICOS EN INGENIERÍA,R. Abascal, J. Domínguez y G. Bugeda (Eds.)© SEMNI, España 1999