EXPLOSION ANALYSIS ON FRAMED STRUCTURE BY USING FINITE ELEMENT METHOD

Submitted in partial fulfillment of the requirements of degree of
Bachelor Of Civil Engineering

by

Mohd Riyaz Mohd Usman Gani 15DCE68

Mohd Hasnat Zafir Faroodui 15DCE67

Zaidh Ali Muhamed Ali Jinna 15DCE62

Afreen Ameerkhan Mulla 13CE01

Guide:

Prof.Wasim Shaikh

Department of Civil Engineering
School of Engineering and Technology
Anjuman-I-Islam's Kalsekar Technical Campus
New Panvel New Mumbai 410206

2017-2018

CERTIFICATE

This is to certify that the project entitled **Explosion analysis of framed structure by using finite element method** is a bonafide work of **Mr. Mohd Riyaz Mohd Usman Gani(15DCE68), Mr. Mohd Hasnat Zafir Farooqui(15DCE67), Mr. Zaidh Ali Muhamed Ali Jinna(15DCE62), Ms. Afreen Ameerkhan Mulla(13CE01),** submitted to the University of Mumbai in partial fulfilment of the requirement for the award of the degree of Bachelor of Engineering in Civil Engineering

Prof. Wasim Shaikh
(Guide)

Dr. Rajendar B. Magar
(Head of Department)

Dr. Abdul Razak Honnutagi
(Director, AIKTC)

PROJECT REPORT APPROVAL FOR B.E

This project report entitled **"Explosion analysis of framed structure by using finite element method"** by **Mr. Mohd Riyaz Mohd Usman Gani(15DCE68), Mr. Mohd Hasnat Zafir Farooqui(15DCE67), Mr. Zaidh Ali Muhamed Ali Jinna(15DCE62), Ms.Afreen Ameerkhan Mulla(13CE01),** is approved for the Bachelor's degree in **Civil Engineering.**

Examiners:

1) ____________________

2) ____________________

Date:

Place:Panvel

DECLARATION

We declare that this written submission represents Our ideas is Our own words and where other ideas or words have been included, we have adequately cited and referenced the original sources ,we also declare that we have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea data fact source in we submission, we understand that any violation of the above will be cause for disciplinary action by the institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Mr. Mohd Riyaz Mohd Usman Gani, (15DCE68)

Mr. Mohd Hasnat Zafir Farooqui, (15DCE67)

Mr. Zaidh Ali Muhamed Ali Jinna, (15DCE62)

Ms. Afreen Ameerkhan Mulla, (13CE01)

May 2, 2018

ACKNOWLEGEMENT

This is to express our sincerest regards to our project Guide, Prof. Wasim Shaikh for their valuable inputs, valuable, guidance, encouragement, whole-hearted cooperation throughout the duration of our project. We deeply express our sincere thanks to our Head of Department Dr. Rajendra Magar and our Director Dr. Abdul Razak Hunnutagi for encouraging and allowing us to present the project on the topic Effect of Rate of Loading On the Compressive Strength of Concrete in partial fulfillment of the requirement leading to the award of Bachelor of Civil Engineering degree. We take this opportunity to thank all our professors and non-teaching staff who have directly or indirectly helped our project, especially Prof. Dada Patil. We pay our respects and love to our parents and all our family members and friends for their love and encouragement throughout our career.

ABSTRACT

Blast is an explosion which is a rapid release of potential energy characterized by eruption enormous energy to the atmosphere. A part of energy is converted to thermal energy radiation (flash) and a part is coupled as air blast and shock waves which expand radially

The Finite Element Method (FEM) is a numerical technique to find approximate solutions of partial differential equations. It was originated from the need of solving complex elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering. In a structural simulation, FEM helps in producing stiffness and strength visualizations. The solution to the numerical model equations are, in turn, an approximation of the real solution to the PDEs. The finite element method (FEM) is used to compute such approximations

The terrorist activities and threats have become a growing problem all over the world and protection of the citizens against terrorist acts involves prediction, prevention and mitigation of such events. Explosive is widely used for demolition purposes in military applications, construction or development works, demolitions, etc.

It is, also, a very common terrorist weapon as it is available, easy to produce, compact and with a great power to cause structural damage and injuries. In this report we have explored the available literature on blast loads based on standard structural analysis software with limited linear and non-linear capabilities. Apart from this we have performed mesh sensitivity analysis by taking into account various mesh sizes and corresponding deflection is computed. The analytical results are obtained based on the load vs deflection curve for linear and non linear properties of concrete. It is shown that with the present knowledge and common software, it is possible to perform the analysis of structures exposed to blast loads and to evaluate their response

Keywords:-

Blast, Finite element method, explosives, radiation, non-linear analysis

Contents

List of Figures

List of Tables

NOTATION AND NOMENCLATURE

- FEM Finite Element Method
- CAD Computer Aided Drafting
- FDM Finite Difference Method
- FVM Finite Volume Method
- FEM Finite Element Method
- BEM Boundary Element Method
- PDE Partial Differential Equations
- FEA Finite Element Analysis
- SAP Structural Analysis Programme
- TNT Trinitrotoluene
- ANFO Ammonium Nitrate Fuel Oil
- SDOF Single Degree of Freedom

1 Introduction

1.1 General

Concrete is the main constituent material in many structures.The behavior of concrete is assumed to be linear but it was proven by the experimental results that concrete is a non linear material. The stress-strain behavior of concrete was demonstrated to be highly non-linear, even at very low stress levels. Due to these reasons understanding exact behavior of concrete structure is very difficult .But due to introduction of advanced computing techniques equipped with finite element methods (FEM) it is possible to make non linear model of concrete. However, the complex behavior of concrete creates limitation in implementing FEM.

The complexity is mainly due to non linear stress-strain relationship under multi axial stress conditions, strain softening and anisotropic stiffness reduction, progressive cracking and bond between concrete and reinforcement. Overcoming all these limitation is key in predicting non linear behavior of concrete. Non linear analysis can be done in many software such as ANSYS, ABAQUS, NASTRAN and ADINA. All these are not ready made applications which can work automatically on just giving commands and input data. To work in any of this software required knowledge of finite element method and firm understanding of concrete behavior.

Among these software ABAQUS is capable of simulating complex geometrically non linear and material model. In this study a 3D reinforced concrete frame will be modeled in ABAQUS by using concrete damage plasticity model. Concrete damaged plasticity model works on Druker-Prager model which has to be totally understood and implemented for creating material model of concrete. The frame which is to be modeled will be subjected to lateral and gravity loading so as to understand local and global mechanisms of structural response.

1.2 Finite Element Method

The Finite Element Method (FEM) is a numerical technique to find approximate solutions of partial differential equations. It was originated from the need of solving complex elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering. In a structural simulation, FEM helps in producing stiffness and strength visualizations.

It also helps to minimize material weight and its cost of the structures. FEM allows for detailed visualization and indicates the distribution of stresses and strains inside the body of a structure. Many of FE software are powerful yet complex tool meant for professional engineers with the training and education necessary to properly interpret the results.

Several modern FEM packages include specific components such as fluid, thermal, electromagnetic and structural working environments. FEM allows entire designs to be constructed, refined and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications.

The use of FEM has significantly decreased the time to take products from concept to the production line. One must take the advantage of the advent of faster generation of personal computers for the analysis and design of engineering product with precision level of accuracy.

1.2.1 Background of finite element analysis

- The finite element analysis can be traced back to the work by ALEXANDER HRENNIKOFF (1941) and RICHARD COURANT (1942).
- ALEXANDER HRENNIKOFF introduced the framework method, in which a plane elastic medium was represented as collections of bars and beams. These pioneers share one essential characteristic. mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

- In 1950s, solution of large number of simultaneous equations became possible because of the digital computer.
- In 1960, RAY W. CLOUGH first published a paper using term Finite Element Method.
- In 1965, First conference on finite elements was held.
- In 1967, the first book on the Finite Element Method was published by ZIENKIECZ and CHUNG.
- In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.
- In the 1970s, most commercial FEM software packages (ABAQUS, NASTRAN, ANSYS, etc.) originated. Interactive FE programs on supercomputer lead to rapid growth of CAD systems.
- In the 1980s, algorithm on electromagnetic applications, fluid flow and thermal analysis were developed with the use of FE program.
- Engineers can evaluate ways to control the vibrations and extend the use of flexible, deployable structures in space using FE and other methods in the 1990s.
- Trends to solve fully coupled solution of fluid flows with structural interactions, biomechanics related problems with a higher level of accuracy were observed in this decade.
- With the development of finite element method, together with tremendous increases in computing power and convenience, today it is possible to understand structural behaviour with levels of accuracy. This was in fact the beyond of imagination before the computer age.

1.2.2 Numerical methods

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature.

For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques.
Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below.

- Finite Difference Method (FDM)
- Finite Volume Method (FVM)
- Finite Element Method (FEM)
- Boundary Element Method (BEM)
- Mesh less Method

1.2.3 Basic concept

The subdivision of a whole domain into simpler parts has several advantages:

- Accurate representation of complex geometry
- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects.

A typical work out of the method involves (1) dividing the domain of the problem into a collection of subdomains, with each subdomain represented by a set of element equations to the original problem, followed by (2) systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

In the first step above, the element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). To explain the approximation in this process, FEM is commonly introduced as a special case of Galerkin method. The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero.

In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

- A set of algebraic equations for steady state problems,
- A set of ordinary differential equations for transient problems

These equation sets are the element equations. They are linear if the underlying PDE is linear, and vice versa. Algebraic equation sets that arise in the steady state problems are solved using numerical linear algebra methods, while ordinary differential equation sets that arise in the transient problems are solved by numerical integration using standard techniques such as Euler's method or the Runge-Kutta method.

In step (2) above, a global system of equations is generated from the element equations through a transformation of coordinates from the subdomains' local nodes to the domain's global nodes. This spatial transformation includes appropriate orientation adjustments as applied in relation to the reference coordinate system. The process is often carried out by FEM software using coordinate data generated from the subdomains.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analysing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high fidelity situations.

For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

FEM mesh created by an analyst prior to finding a solution to a magnetic problem using FEM software. Colours indicate that the analyst has set material properties for each zone, in this case a conducting wire coil in orange; a ferromagnetic component (perhaps iron) in light blue; and air in grey. Although the geometry may seem simple, it would be very challenging to calculate the magnetic field for this setup without FEM software, using equations alone.

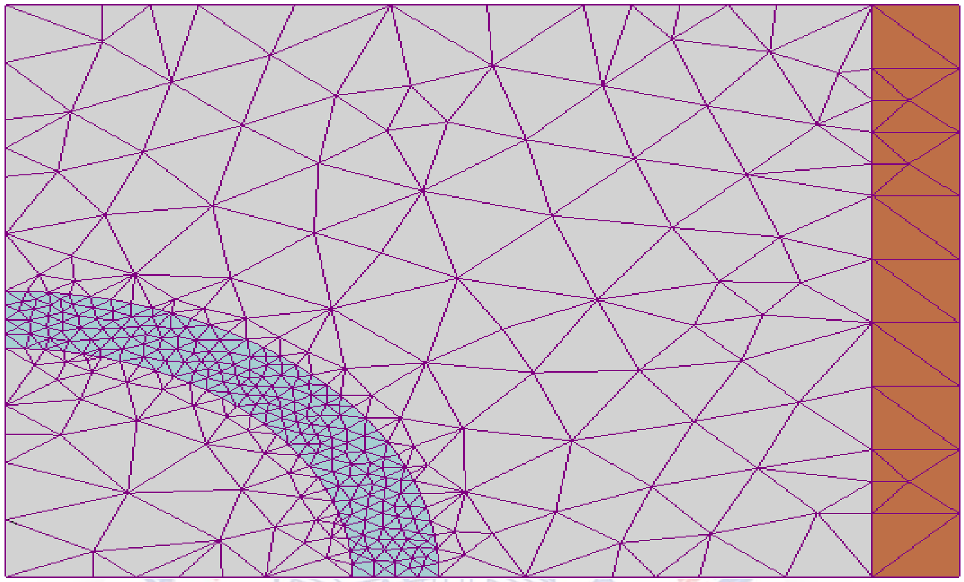

Figure 1: Representation of an element model

1.2.4 Structure of finite element methods

Finite element methods are numerical methods for approximating the solutions of mathematical problems that are usually formulated so as to precisely state an idea of some aspect of physical reality. A finite element method is characterized by a variational formulation, a discretization strategy, one or more solution algorithms and post-processing procedures. Examples of variational formulation are the Galerkin method, the discontinuous Galerkin method, mixed methods, etc.

A discretization strategy is understood to mean a clearly defined set of procedures that cover (a) the creation of finite element meshes, (b) the definition of basis function on reference elements (also called shape functions) and (c) the mapping of reference elements onto the elements of the mesh. Examples of discretization strategies are the h-version, p-version, hp-version, X-FEM, isogeometric analysis, etc. Each discretization strategy has certain advantages and disadvantages. A reasonable criterion in selecting a discretization strategy is to realize nearly optimal performance for the broadest set of mathematical models in a particular model class.

There are various numerical solution algorithms that can be classified into two broad categories; direct and iterative solvers. These algorithms are designed to exploit the sparsity of matrices that depend on the choices of variable formulation and discretization strategy.

Post-processing procedures are designed for the extraction of the data of interest from a finite element solution. In order to meet the requirements of solution verification, post processors need to provide for a posteriori error estimation in terms of the quantities of interest. When the errors of approximation are larger than what is considered acceptable then the discretization has to be changed either by an automated adaptive process or by action of the analyst. There are some very efficient post processors that provide for the realization of super convergence.

1.3 Blast loading

This article specifically addresses the affects of shock loading from air blast. This type of load is applied to the perimeter structural elements of a building due to a high explosive blast event external to the building. The pressure wave applied to the building is characterized by short duration and high intensity.

The blast wave duration, td , is typically in the range of 0.1 to 0.001 seconds. This is often much shorter than, or at most on the order of, the natural period, Tn , of typical structural elements. For situations where td ¡ 0.4Tn (some sources advise td ¡(0.1Tn)), the blast wave effectively imparts an initial velocity to a structural element and the element then continues to respond at its natural frequency. The magnitude of that initial velocity, for a single-degree-of-freedom (SDOF) is v = 0td/2m , where 0 and td are shown in Figure 1 and m is the mass.

Thus, in this response regime, the mass of the structural element is the only system parameter that controls the magnitude of the initial motion of the system the more massive the structural element, the less it will be excited by the impulse from the blast wave. In this regard, the greater mass of concrete structures can be used to great advantage. This load response to a blast is significantly different from the load response to a seismic event, for which the natural frequency of the structure, rather than the mass, is the primary factor in the response.

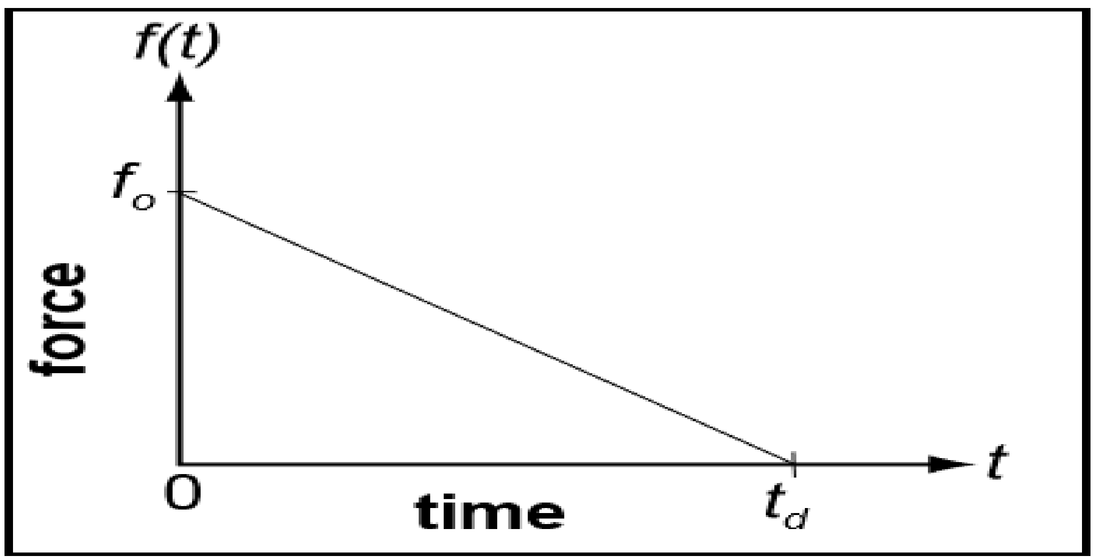

Figure 1: Idealized blast pulse with a peak intensity, f_0 and duration, t_d

Figure 2: Idealized blast pulse with a peak intensity, f0 and duration, td

1.3.1 Blast resistance and progressive collapse

Progressive collapse-resistant design mitigates disproportionately large failures following the loss of one or more structural elements. Progressive collapse-resistant design is system-focused, and is often divided into two approaches, direct and indirect. The direct method designs the structural system to respond to a specific threat either by providing an alternate load path in the event of failure of one or more members, or by specific local-resistance improvements of key elements. This method is similar to blast-resistant design. The indirect method provides general systemic improvements to toughness, continuity and redundancy; tension ties are an example of an indirect detailing technique. Blast-resistant design is element-focused.

It enhances toughness, ductility, strength and dynamic characteristics of individual structural elements for resistance to air-blast induced loading. This article is devoted to blast-resistant design, though there is overlap with progressive collapse-resistant design.

2 Review of literature

In order to carry out the project work various literatures were studied and findings obtained by them were used to identify the research area, summarizations of literatures are as follows:-

2.1 Effect of in-plane forces in beam column junction of RC substitutive frame in the linear regime (2009)(1)

This paper simulates the elastic response of the substitute frame.3D modeling of frame is done in ABAQUS to achieve more realistic solution. In this paper 3D linear hexahedral lower order liner element C3D8I which uses incompatible modes is used. For purpose of analysis modeled is prepared Considering (1)concrete as a solid isotropic homogeneous medium which uses linear elasticity based constitutive model (2) lower order solid element to represent concrete medium, which reduces time and associated. Further a comparative study is done with standard analytical methods e.g. Moment distribution method and STAAD Pro analysis (matrix method of analysis)

problem layout

A single bay substitute frame with column section 125 mm x 250 mm and beam dimension 125 mm x 200 mm is modeled in ABAQUS. The beam is 2.0 m long (clear span) and is subjected to two point loads (40 KN each) at quarter span of the beam alone with self-weight

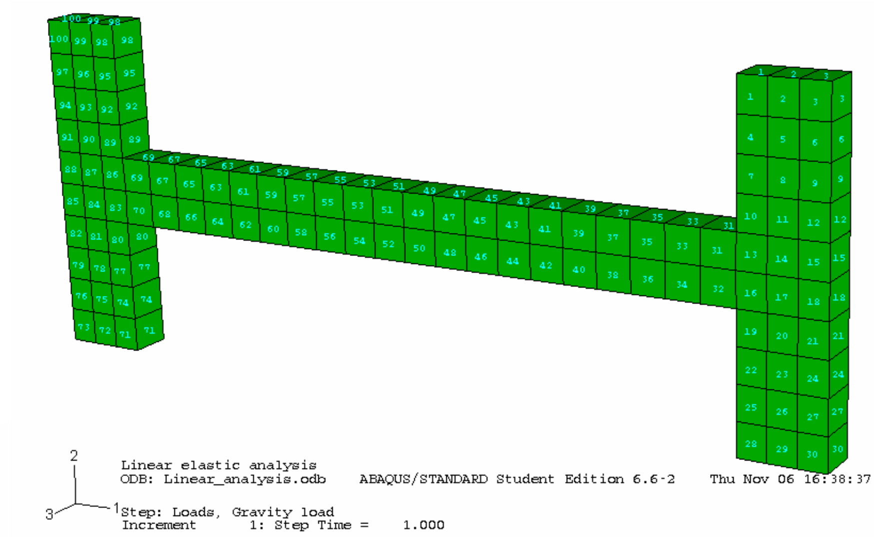

Figure 3: Model before loading

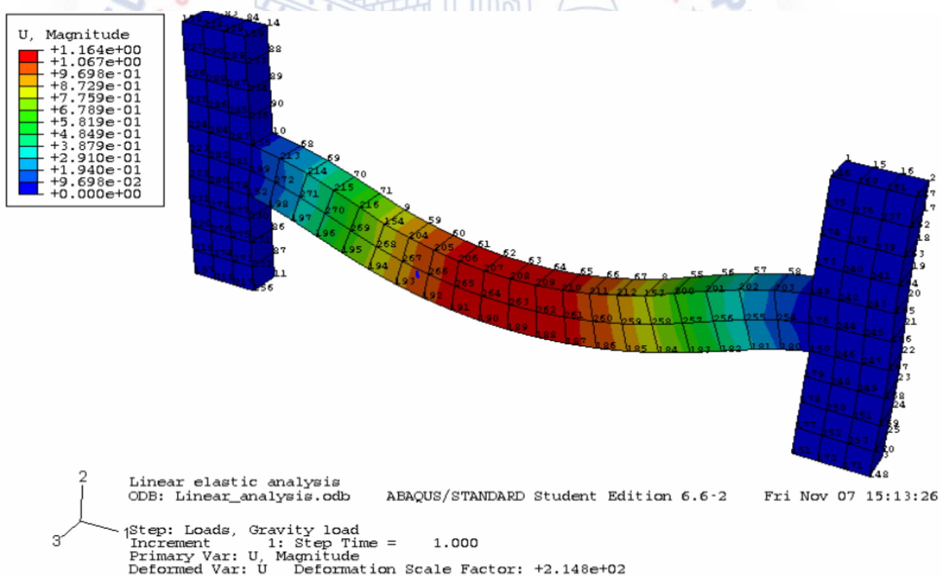

Figure 4: ABAQUS model after loading

Result and conclusions

This software uses finite element method of analysis so we can obtain the stress coming on even small element of the beam-column system. More over this software uses the 3 D model of the structure, which helps in getting more precise result.

Table 1: Comparison of stress

method	At support				At mid span	
	M (KNm)	Stress(Mpa)	BM	B stress	M(KNm)	Stress(Mpa)
1	8.55	10.26	6.05	7.26	3.75	4.5
2	8.60	10.32	6.1	7.32	3.9	4.68
3	–	–	–	8.49	–	6.37

- Method 1-Moment distribution method
- Method 2-STAAD Pro (Matrix method)
- Method 3-ABAQUS analysis

In table 1 the stress values for method 1 and method 2 are very close to each other whereas for the method 3 it grossly varies. The values as obtained from moment distribution method are very close to values obtained by STAAD Pro. But on the other hand values obtained from ABAQUS at beam column junction are very different its happening because Abaqus uses 3 D stress analysis, it represent true load deformation behaviour close to the actual.

2.2 Modelling of concrete for nonlinear analysis Using Finite Element Code ABAQUS (2012)(2)

A three dimensional concrete cube modelled using smeared crack model and concrete damaged plasticity approach. The validation of the model to the desired behaviour under monotonic loading is then discussed. Smeared crack and concrete damage plasticity approach

A concrete cube of size 150 mm is modelled in ABAQUS using C3D8 (continuum element) element. A steel plate of thickness 25 mm is placed on top and at the bottom of the cube to ensure the uniform distribution of the compressive load applied. The plate is also modelled with C3D8 elements. The plates are secured in place by applying Constraint type tie, available in the ABAQUS. The material properties are used for M30 grade concrete as per IS 456-2000 for concrete. i.e. The average compressive stress, Xu= 30Ma , ultimate strain, Yu= 0.0035 , and the strain at pick stress, Y0= 0.002. They have used following expression to develop stress-strain graph.

$$X/Xcu = 2Y/y0(1 - Y/2y0)$$

$$For 0 < Y < y0$$

$$X/Xcu = 0.15Y/y0(1 - Y/2y0)$$

For the Concrete Damaged Plasticity model following stress strain relation are used by authors.

$$Y = Ye + Yp And X = (Y - Yp)Del$$

Effect of Tension Softening and shear retention are taken during analysis of concrete blocks.

Conclusion:

1) In both the cases the concrete shows a perfectly non-linear behaviour.

2) Using smeared crack modelling at mesh size 25 the obtained stress-strain curve gives max stress about 29.39MPa at 0.00190 strains and then after the curve shows the descending nature.

3) In case of Concrete damaged plasticity model at mesh size 25, the stress obtained is 32.33 MPa at 0.00195 strains and then after it starts descending.

4) The smeared crack mo of concrete is found suitable as it gives desired results at coarser mesh size in comparison with concrete damage plasticity model as it doesnt over estimate.

2.3 Identification of parameters of concrete damage plasticity constitutive model (2005)(3)

In this paper the method and requirement of the material parameters for identification of concrete damage plasticity are studied. For that purpose laboratory test are conducted. They have used two standard applications. The first is analysis of three points bending single edge notched concrete beam specimen. The second presents the four points bending single edge notched concrete beam specimen under static loading. The parameters which were identified are p ,q, m and f. The parameter p and m were used to describe the shape of potential function, while q and f responsible for shape of potential function. All these function were found out by using laboratory tests such as The Uniaxial Compression, The Uniaxial tension, Hydrostatic state of stress.

Table 2: parameters of concrete damage

Parameter name	value
Dilatation angle	36
Eccentricity	0.1
fbo/fco	1.16
K	0.667

The example shows that using CDP model enables a proper definition of the failure mechanism in concrete element. The CDP can be used to model the behaviour of the concrete and reinforced concrete structures too. They have proved that the point of initiation and evolution of was correctly estimated.

2.4 Evaluation of reinforced concrete beam behaviour using finite element analysis by ABAQUS (2012)(4)

A damaged plasticity approach is used to model reinforced concrete beam in ABAQUS. The model was checked to flexural loading and result from ABAQUS and experimental results were compared. Displacement, tensile strain for main reinforcement compressive strain for concrete and crack pattern was compared. The load Vs tensile graph were plotted for tensile steel reinforcement at mid span are plotted by experiment and results were compared with finite element analysis. The tensile steel reinforcement bars at mid span did not yield at failure in either experimental result or finite element analysis. Compressive strain in observed in concrete Deflection was measured at mid span at center and bottom face of beam.

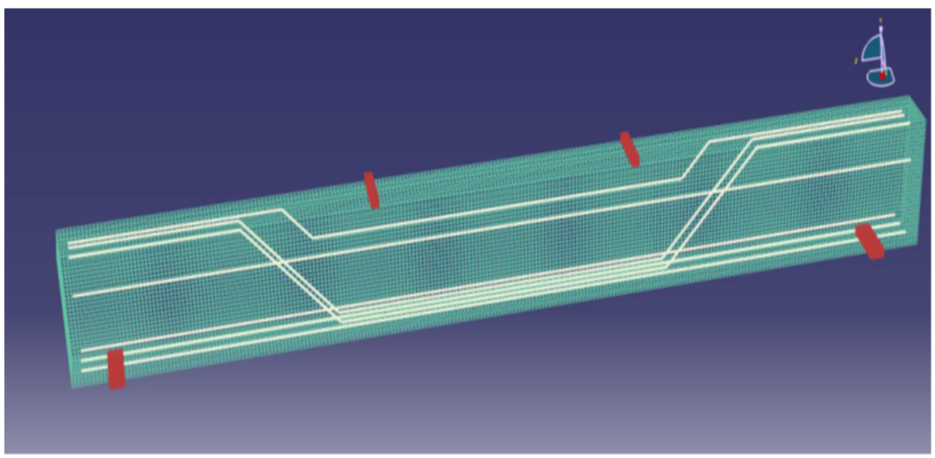

Figure 5: Finite element modeling parts in ABAQUS.

2.5 Modelling of reinforced concrete structures and composite with concrete strength (2011)(5)

The CDP (Concrete Damaged Plasticity) model used in the ABAQUS software is a modification of the DruckerPrager strength hypothesis. According to this hypothesis, failure is determined by non-dilatational strain energy and the boundary surface itself in the stress space assumes the shape of energy and the boundary surface itself in the stress space assumes the shape of a cone. The advantage of the use of this criterion is surface smoothness and thereby no complications in numerical applications. The drawback is that it is not fully consistent with the actual behavior of concrete.

This hypothesis is again modified, according to modification the failure surface in the deviatoric (study it properly) cross section needs not to be a circle and it is governed by parameter Kc. parameter Kc is interpreted as a ratio of the distances between the hydrostatic axis and respectively the compression meridian and the tension meridian in the deviatoric cross section. This ratio is always higher than 0.5 and when it assumes the value of 1, the deviatoric cross section of the failure surface becomes a circle (as in the classic DruckerPrager strength hypothesis).According to experimental results this value for mean normal stress equal tozero amounts to 0.6 and slowly increases with decreasing mean stress. The CDP model recommends to assume Kc= 2/3.It is a theoretical-experimental criterion based on triaxial stress test results.

Table 3: Default parameters of CDP model under compound stress

Parameter name	value
Dilatation angle	36
Eccentricity	0.1
fbo/fco	1.16
K	0.667

2.6 Non linear analysis of cracked reinforced concrete (2011) (6)

Smeared crack model consider initiation of cracking process at any location happens when concrete stresses reach one of failure surface either in biaxial tension region or combined tension compression region. This model is useful for creation of model in which the concrete is subjected to essentially monotonic loading and material shows either compressive crushing or tensile cracking. The plastic straining compressive in nature and controlled compression yield surface. The cracking is assumed to be most important aspect of behavior and representation of cracking and post cracking dominates modeling process. The concrete model is said to be smeared crack model is because it does not track individual macro crack. Constitutive relations are to be found out at independently for finite element model.

The presence of crack enters into this calculation by the way in which the crack affects the stress and material stiffness associated with integration point. The smeared crack model concrete model provides general capability for modeling concrete in all types of structures, including beams, trusses shells and solids. It can be used for plain concrete. It is most suitable for concrete which is subjected to monotonic loading. For the calculation of strain the formula used by these them is Y=Ye+Yp, where Ye is elastic strain and Yp is plastic strain. This paper aims to present the model including, smeared crack representation, rotating cracked approach, tension stiffening, stress degradation, and shear retention of concrete. The material model is then tested against experimental dat and nonlinear analysis of concrete model is done.

2.7 Non linear properties of reinforced concrete structures (2002)(7)

In this paper of various nonlinearities involved in the static and dynamic analyses of Reinforced Concrete structures is investigated. The nonlinearities studied here are geometric as well as material (due to the nonlinear stress-strain relationship of concrete and steel). In the first part of the paper, the nonlinear moment-curvature relationship of arbitrary Reinforced Concrete cross-sections is developed numerically using nonlinear stress-strain relationships for concrete and steel. The relative importance of geometric and material nonlinearity is studied for a simple 2-storied frame under static vertical load.

The effect of axial load on the moment-curvature relationship is studied, and the effect of typical axial loads on the flexural behavior of column is found to be significant. The shear strength of the beams and columns (obtained from empirical equation suggested in the ACI Code) prove to be very important here. Using the nonlinear sectional properties thus obtained, the nonlinear structural dynamic analyses of the building are performed subjecting the structure to seismic vibrations using nonlinear structural dynamics. Recorded ground motion data from two major earthquakes of the past; e.g., the El Centro earthquake in USA (1940) and the Kobe earthquake in Japan (1995) are used in the dynamic analyses. The results show the difference between the linear and the nonlinear structural response.Structural shape due to lagged formations. While this requires complicated formulation, reasonable accuracy can be achieved by suitable approximation of the problem.

2.8 Analysis of Blast Loading Effect on Buildings (2014)(8)

This paper presents the dynamic response of a High Rise Structure subjected to blast load. The fundamentals of blast hazards and the interaction of blast waves with structures are examined in this study it is about the lateral stability of a high rise building modeled using SAP2000. The model building was subjected to two different charge weights of 800lbs and 1600lbs TNT at a two different standoff distances of 5m and 10m.The blast loads are calculated using the methods outlined in section 5 of TM5-1300 and a nonlinear modal analysis is used for the analysis of the dynamic load of the blast. The primary performance parameters that will be used to evaluate the behavior of the building from a global perspective are the total drift and the inter-storey drift. They are good indicators of nonstructural damage, collapse and ability of the structure to resist P-delta effect. Behavior of R.C frame and concrete infill frame will be computed in dynamic condition.

2.9 Protective design of concrete buildings under blast loading (2002)(9)

Designing buildings to resist failure due to blast loads is an extremely complex procedure. It is a process that has been investigated for many years, yet it warrants tluther research. Several issues related to the design of concrete structures to survive blast loads are discussed in this paper. General design issues of terrorist-proof buildings show how the threat of harrnfil blasts is affecting the thought process in designing government and public buildings as well as international and high-visibility organizations. Understanding the loads produced by explosions is an integral part of dealing with blast-resistant design. Case studies of buildings subjected to blasts reveal how actual structures have handled the dynamic loads. Current research on the subject is also reviewed.

2.10 Study on the Response of RC Frames Subjected to Blast Loading (2016)(10)

The use of explosives by terrorist groups is becoming a great threat to the society. The analysis and design of structures subjected to blast loads require a detailed understanding of the blast phenomena and the dynamic response of various structural elements. This paper aims on an overview of the effects of blast loading on reinforced concrete frames. The negative phase of the blast wave is usually not taken into consideration as the main structural damage is associated with the positive phase. But the negative phase of the blast wave should be taken into account if the overall structural performance of the structure is assessed and not only its structural integrity. The negative phase can either increase or decrease the deflections. In this paper a comparison and assessment was done to present the differences between the standard blast load model, neglecting the negative phase, and the blast load model, which takes both the positive phase and the negative phase into account. The responses of the frames were studied for two charge weights. The finite element package ANSYS was used to model the RC frames.

2.11 Study On The Effect Of Blast Load On Industrial Structure (2015)(11)

In this paper we studied the effect of surface burst explosion on steel structure behavior under three different weight charges at the same stand-off distance. A several points were selected at the model front to calculate the reflected pressure and the duration time, and then the pressure -time history functions defined for each member by using SAP2000 software. Based on the maximum displacement the moment connection joint frame was a bit better than the pin connection joint frame to resist blast load due to the moment connections.

2.12 Analysis of the Response of a One-Storey One-Bay Steel Frame to Blast (2016)(12)

This paper compared the performances of a seismically designed SDOF frame structure at blast loads at various standoff distances. For the charge weight used in this study, the main beam entered the plastic zone at the critical standoff distance. This indicates that once seismic codes are followed and a building is detailed accordingly, the probability of withstanding a blast load of the magnitude studied is uncertain and should be further investigated. This study is purely comparative since the positive effect for the design due to earthquakes and the behavior of the frame is different from the behavior of the frame under blast loading.

2.13 Review Paper On Blast Loading And Blast Resistant Structures(2017)(13)

The objective of this paper is to review the works on effects of blast loading on structures that as already been done till now. Due to the recent increase in various terrorist activities all over the world, the safety of the structures should be designed to resist bomb blasts. However, these designs could be and will be economicallyinefficient, but they can be used for designing commercial office buildings, shopping malls, government buildings and even 5-star hotels. Blast loads are basically dynamic loads of a type that needs to be attentively calculated like wind and seismic loads and the structures should be designed by considering them to make it blast resistant. But this philosophy will cause the cost of construction to increase by a big amount. So special care must be taken if the structure is located in a sensitive place where bombarding, explosions or war are the chiefs. Also, if it is located in the region of the high-intensity earthquake. The objectives of this study are to elucidate on blast resistant building design theories, the improvement of building security against the effects of explosives in the process of structural design and the techniques in design that should be carried out. The paper also includes information about explosives, blast loading parameters and enhancements of blast resistant building design Behavior of R.C frame and concrete infill frame will be computed in Dynamic condition.

2.14 Structural Analysis of Blast Resistant Structures (2016)(14)

The effect of blast load on buildings is, a serious matter that should be taken into consideration in the design Even though designing the structures to be fully blast resistant is not a realistic and economical option we can improve the new and existing buildings to ease the effects of a blast. In this study I have analyzed the effects caused by blast loads and to find how to reduce the effects using ETAB 2015 and from my studies. I conclude that the shear wall is resisting the blast loads than any other alternatives.

2.15 Simulation Of The Response Of Concrete Structure(2013)(15)

The response of concrete structures subjected to blast load can be studied numerically using commercial finite element programs. The complexity of reinforced concrete makes modeling and simulation of large structures time and computer power consuming. Consequently, The Swedish Defense Research Agency FOI, has an interest in finding a fast and numerically efficient model to study this response to a reasonable cost. The initial step is to find a reliable but simple and numerically efficient finite element model of reinforced concrete slabs subjected to quasi static load and blast load using shell element formulation.

2.16 Prediction Of Blast Loading(2009)(16)

The paper describes the process of determining the blast load on structures and provides a numerical example of a fictive structure exposed to this load. The aim was to become familiar with the issue of blast load because of ever growing terrorist threat and the lack of guidelines from national and European regulations on the verification of structures exposed to explosions. The blast load was analytically determined as a pressure-time history and numerical model of the structure was created in SAP2000. The results confirm the initial assumption that it is possible with conventional software to simulate an explosion effects and give a preliminary assessment of the structure.

2.17 Blast Loading and Blast Effects on Structures (2007)(17)

The use of vehicle bombs to attack city centers has been a feature of campaigns by terrorist organizations around the world. A bomb explosion within or immediately nearby a building can cause catastrophic damage on the building's external and internal structural frames, collapsing of walls, blowing out of large expanses of windows, and shutting down of critical life-safety systems. Loss of life and injuries to occupants can result from many causes, including direct blast-effects, structural collapse, debris impact, fire, and smoke. The indirect effects can combine to inhibit or prevent timely evacuation, thereby contributing to additional casualties. In addition, major catastrophes resulting from gas-chemical explosions result in large dynamic loads, greater than the original design loads, of many structures. Due to the threat from such extreme loading conditions, efforts have been made during the past three decades to develop methods of structural analysis and design to resist blast loads. The analysis and design of structures subjected to blast loads require a detailed understanding of blast phenomena and the dynamic response of various structural elements. This paper presents a comprehensive overview of the effects of explosion on structures. An explanation of the nature of explosions and the mechanism of blast waves in free air is given. This paper also introduces different methods to estimate blast loads and structural response.

2.18 Finite element analysis of reinforced concrete columns under different range of blast loads (2014)(18)

Columns are the significant load-bearing elements in any structure and exterior columns are probably the most vulnerable structural elements to accidental explosions. Blast load close to or nearby a building might cause sudden failure on the building's external and internal structural skeleton. Abaqus version 6.19 is used in this study to perform numerical simulations of the dynamic responses and residual strength of reinforced concrete columns subjected to different blast charge weights. The finite element model is verified through correlated experimental studies listed on the literature. A parametric study was chosen to cover wide range of factors on the global columns response; the effects of transverse and longitudinal reinforcement ratios, the charge weight, and the column aspect ratio. The model is validated to be used in the analysis of structures, and can closely predict the exact response behaviour of reinforced concrete columns under different detonation Scenarios. The study concluded that after a certain charge weight, the residual displacement became more significant. Providing more lateral reinforcement improved the blast resistant design by decreasing the residual displacement. The results of this study are valid only for a specific standoff distance, and if the charge weight comes in contact with the structure, the results will be significantly different.

2.19 Numerical Study of Reinforced Concrete beam subjected to blast loading using FE package ABAQUS(2017)(19)

The threat of terrorism rising all over the world has increased the awareness among people. Efforts have been made to design structures which offers better resistance against blast explosion. Studies were conducted on the behaviour of structural members subjected to blast loads. In this study, the finite element package ABAQUS/Explicit was used to model a reinforced concrete beam, which was previously tested and reported in an experimental research paper the concrete damage plasticity approach was used to define the non-linearity of concrete. The effect of blast loading on the RC beam was analytically observed and deflection at mid-span of the beam was compared with the experimental results.

2.20 Concluding remark on literature review

In this literature review I went through past work done in ABAQUS and formulation required for carrying out non-linear analysis of RCC frame. One thing I found noteworthy that, Behaviour of concrete is non-linear due to non-linear stress strain relation hence its very difficult to extract exact solution of any problem. The effective method available for understanding non-linear behaviour of concrete is experimental and more recent the Finite Element Methods (FEM). Substantial amount of research has already been done in past for modeling behaviour of concrete using FEM. Many researchers emphasize use of Concrete Damaged Plasticity (CDP) approach for modelling of concrete. In past decade advanced computation techniques has led better study of behaviour of concrete. Software like ABAQUS witch can perform better analysis in less time and hence very effective in modelling of non-linear concrete behaviour. The complexity in concrete such as non-linear stress strain relation, strain softening, anisotropy, progressive cracking and bond stress can be taken care by using ABAQUS.

3 Direction of further study

Till now most of work done on non linear behaviour of concrete is either experiment or purely theoretical, but very few amount of work done in numerical modelling of non linear concrete. ANSYS is mostly used in last one decade which gives fairly good results but now as much better software ABAQUS is available which can be used as a replacement to ANSYS to study non linear behaviour of reinforced concrete. Defining material properties in ABAQUS is very important and it can only be done by firm knowledge of studying compression and tension behaviour of concrete.

ABAQUS need proper definition of constrain between two regions of model or between concrete and reinforcement. Simply keeping reinforcement inside the concrete does not mean bond between reinforcement and concrete. Constrains has to be properly defined so as to get desired results.

4 Validation of software

4.1 Introduction to ABAQUS

ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method that can solve problems ranging from relatively simple linear analyses to the most challenging non-linear simulations. ABAQUS contains an extensive library of elements that can model virtually any geometry. It has an equally extensive list of material models that can simulate the behaviour of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock. Designed as a general-purpose simulation tool, ABAQUS can be used to study more than just structural (stress/displacement) problems.

It can simulate problems in such diverse areas as heat transfer, mass diffusion, thermal management of electrical components (coupled thermal-electrical analyses), acoustics, soil mechanics (coupled pore fluid-stress analyses), and piezoelectric analysis. ABAQUS offers a wide range of capabilities for simulation of linear and nonlinear applications. Problems with multiple components are modelled by associating the geometry defining each component with the appropriate material models and specifying component interactions. In a nonlinear analysis ABAQUS automatically chooses appropriate load increments and convergence tolerances and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently.
For validation purpose above beam is chosen from Joshua s Tayu [9]. In this paper a concrete beam is analysed by simple three point bending. Following are details of beam in [9].

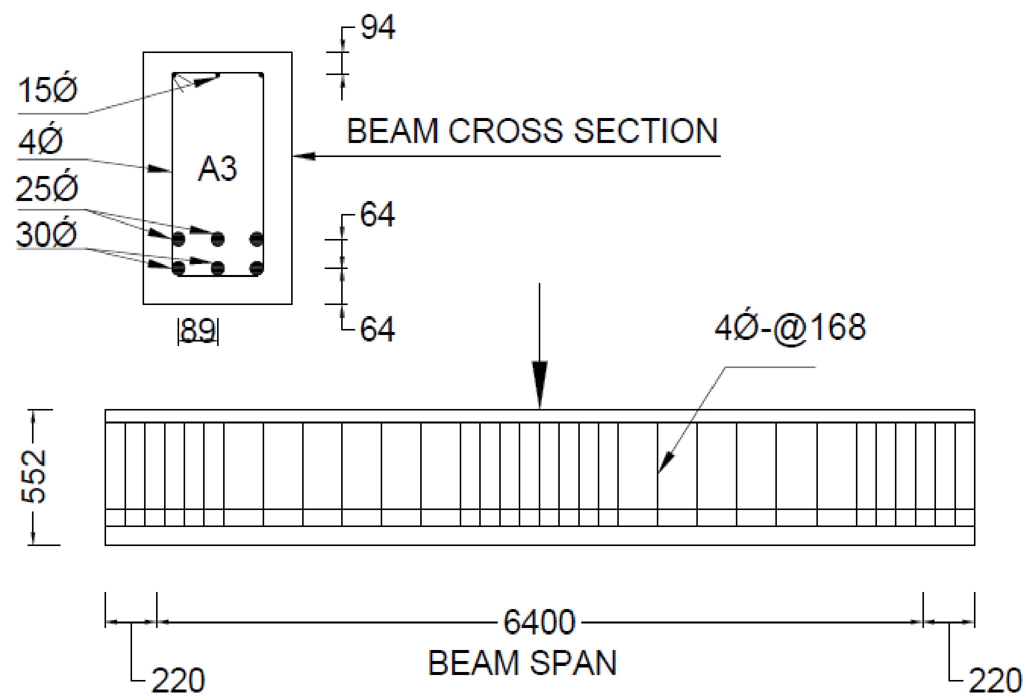

Figure 6: Beam details

- Modulus of elasticity E = 3430 Mpa
- Compressive strength fck = 43.5 Mpa
- Rebar size 30 mm 25 mm 10 mm and D4(6mm) bars are used
- Yield stress of rebars is $415 N/mm^2$
- Span of beam 6400 mm
- Cross section of beam is 305 x552 (mm)

Above beam is modeled in ABAQUS as following

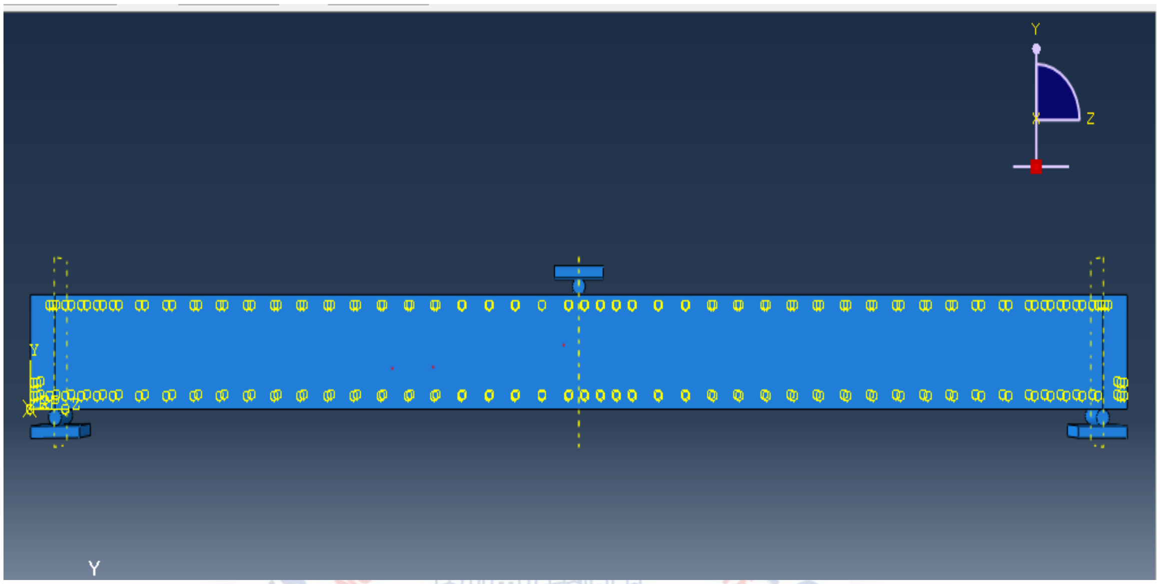

Figure 7: Beam model in ABAQUS

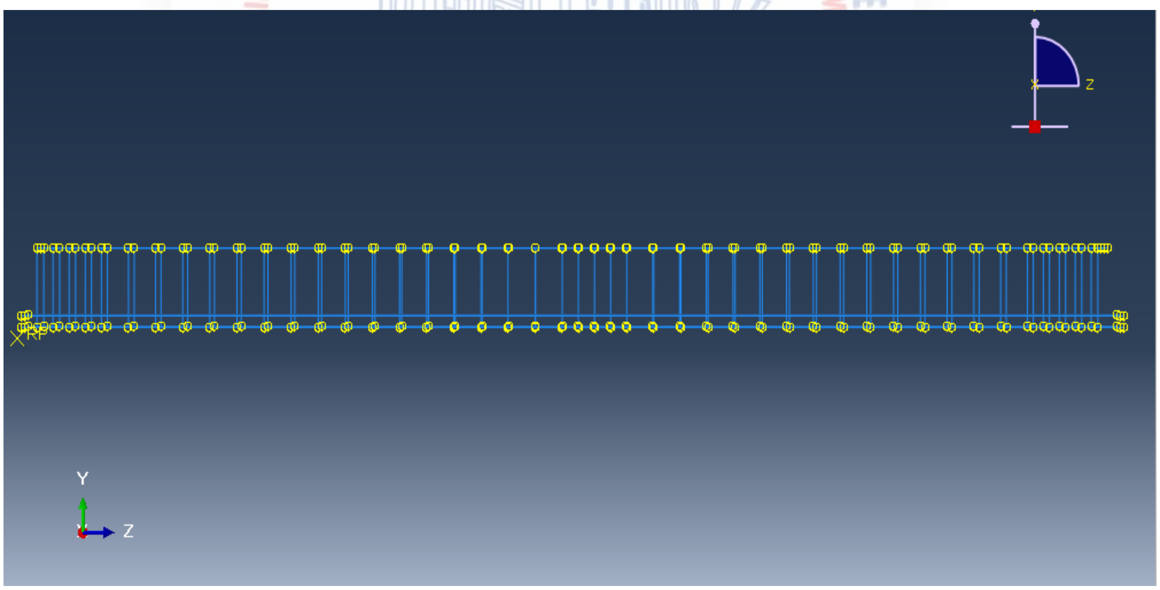

Figure 8: Reinforcement modelling

4.2 Analysis of beam

Loads condition and modeling is kept as [9] and analysis is done. load is increased at rate of 50kN and results of three point bending are plotted as bellow.In addition to that a conversion study is done by taking various mesh size i.e 20 mm 25 mm and 30 mm mesh size. From observation it can be concluded that, the load vs deflection comes close to experimental results at mesh size 25 mm. hence here after modeling of frame is done by using 25 mm mesh size.

Loads are applied at constant rate of 50 kN and load vs. deflection curves are plotted. Table showing deflection at midpoint of beam.

Table 4: Deflection at midpoint of beam

Exp deflection(m)	30 mm mesh size (m)	25 mm mesh size (m)	20 mm mesh size (m)
0	0	0	0
0.0082	0.0050	0.0080	0.0090
0.1203	0.0180	0.0100	0.0140
0.0155	0.0160	0.0140	0.0210
0.0190	0.0200	0.0180	0.0260
0.0230	0.0205	0.0220	0.0310
0.0283	0.0320	0.0270	0.0340
0.0341	0.0320	0.0360	0.0380

After getting deflection a load Vs defection curve is plotted along with various mesh size.

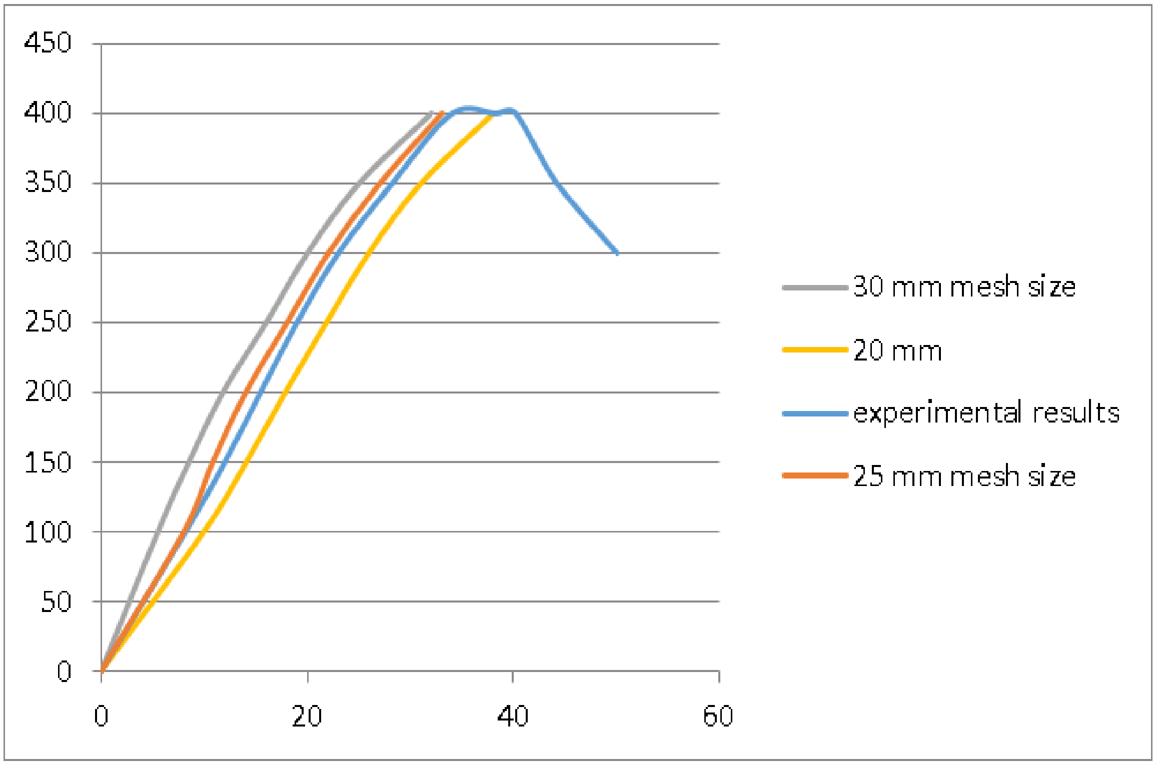

Figure 9: Plot of load Vs deflection

From above plot it can be concluded that mesh size 25mm should be used for for conversion and compatibility of FEM solution of problem dealing with CDP model of concrete.

5 Methodology

5.1 Finite Element Modelling

A methodology for analyzing a G+1 RCC framed structure under blast load has been discussed below. Three dimensional mathematical modelling of a RCC frame has been carried out in Finite Element analysis package considering surface to surface interactions at the interface. The beam element has been modelled using C3D8R element i.e 8 noded brick elements where as the reinforcement has been modelled using truss element i.e. C2DR elements. The behavior of concrete is considered to be linear thus no nonlinear analysis is done. Nonlinear analysis can be done in many software such as ANSYS, ABAQUS, NASTRAN and ADINA.

All these are not ready made applications which can work automatically on just giving commands and input data. To work in any of this software required knowledge of finite element method and firm understanding of concrete behavior. The concrete damage plasticity has also been neglected. The deflection for different loads have been observed and accordingly a graph has been plotted showing load on X axis and deflection on Y axis. Thus with the help of this graph the analytical results can be compared with the experimental results. The nearest value will be helpful in determining the right size of mesh for that grade of concrete.

Table 5: Dimension and property of members

Member	Size	Depth	E	Poissons ratio	Density
Beam	300x300	4200	25000	0.2	2.4E-008
Column	300x300	4200	25000	0.2	2.4E-008
Steel	16mm	4150	200000	0.3	7.85E-008
Steel	12mm	2920	200000	0.3	7.85E-008
Stirrups	8mm	-	200000	0.3	7.85E-008

5.1.1 Mesh And Boundary Conditions.

The mesh size is kept varying till determination of the correct mesh size for the given grade of concrete. After the determination of the proper mesh size there is a variation in loading patterns in the incremental order. Depending upon the results a graph is plotted stating to the corresponding load trials and based on this graph the concrete damage parameter can be obtained. The boundary conditions are defined as fixed from bottom. The experimental results for the given grade can prove more efficiency in the results as well as it becomes easy for comparison of these results.

5.1.2 The loading pattern

Various load in the incremental order are considered without the consideration of their amplitudes and time variations. As discussed above blast load varies from 0.001 to 0.01 seconds. Thus these amplitudes can be considered for various incrementation of loads. The application of load is among any of the direction basically on one face of the member or either it can be applied on one face of the frame. This frame is analyzed for lateral loading exclusive of its self weight. After that a critical T joint having maximum lateral load is chosen for analysis purpose. Detailing of frame is done as per IS 456.

5.1.3 The Part Module

Parts are building blocks of an Abaqus CAE model. The module has been used for creating each part. The parts can be created by clicking on create part and then creating the parts which are required for the structure. The sizes for each part is defined in the part module. The part is named i.e. it may be a beam or a column or reinforcement. The type is defined i.e it is deformable or rigid or discrete. Then the base feature is defined i.e. whether it is a shell or a wire. For beam elements 3D element is selected and solid base with deformable type is selected whereas for reinforcement wire base feature with deformable type is selected.

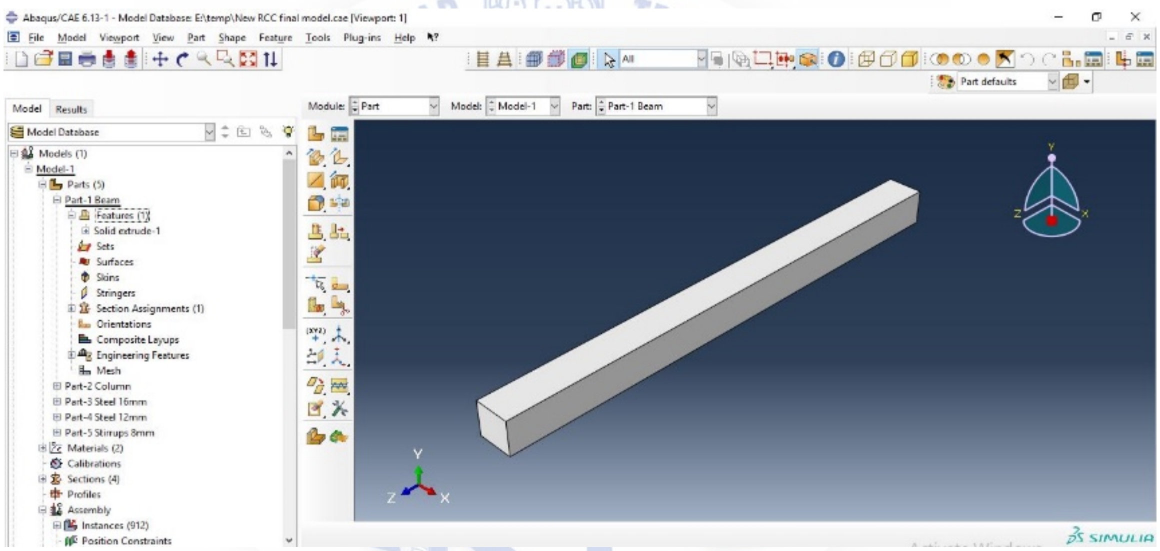

Figure 10: Creating beam of size 200x200mm and 4.2m length

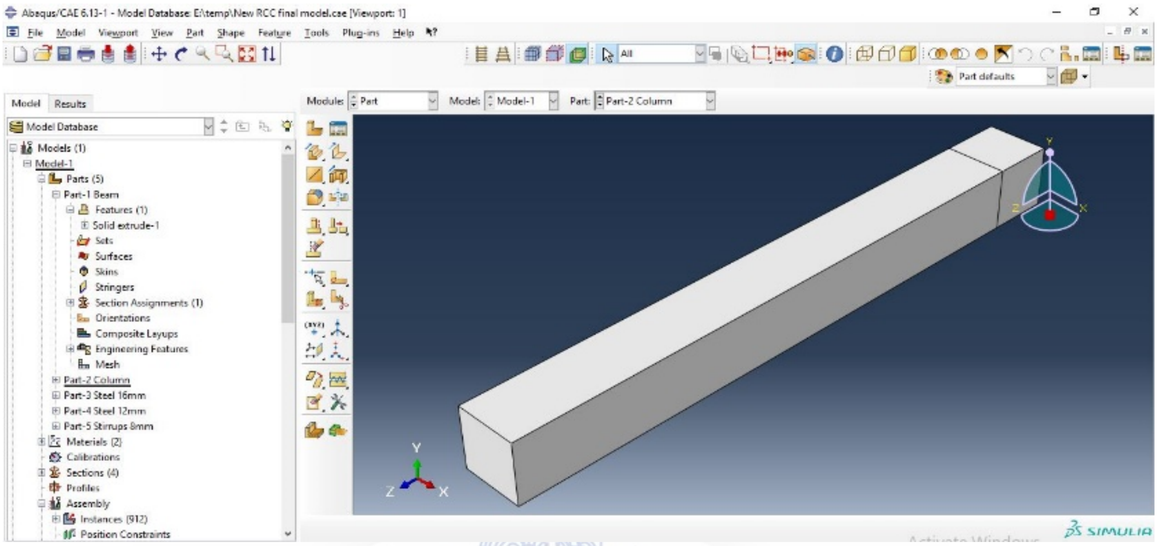

Figure 11: Creating column of size 200x200mm and 4.2m length

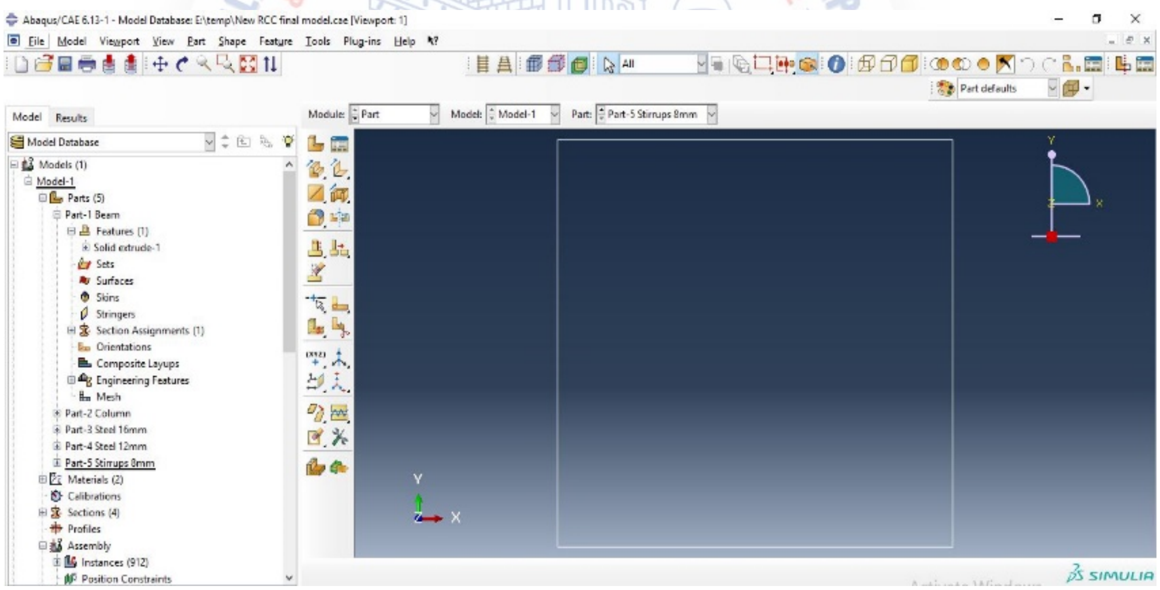

Figure 12: Creating reinforcement of size 16mm in diameter

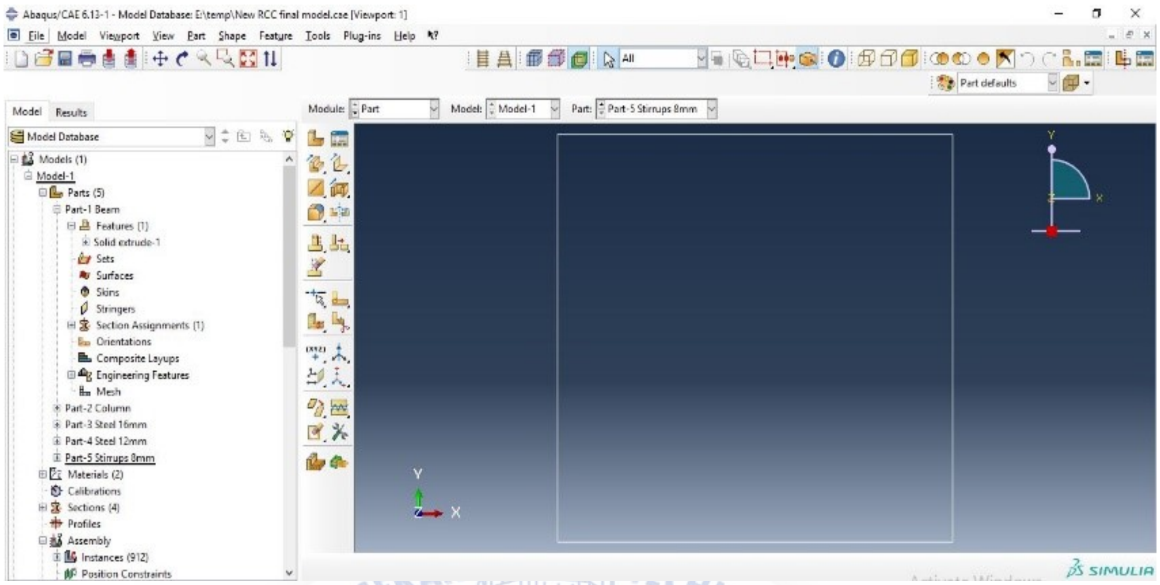

Figure 13: Creating stirrup for beam and column of size 8mm in diameter

5.1.4 The Property Module

The property module has been used for defining the properties of the materials used by the parts. The definition of sections and the profiles is also done in the property module. As discussed earlier the behavior of the material is considered as elastic so the elastic properties of the materials have been assigned but even the plastic properties and damage properties can be defined in this module. The section here is considered to be a solid section but the software provides various options such as eulerian and beam sections too. The properties of the materials are given below.

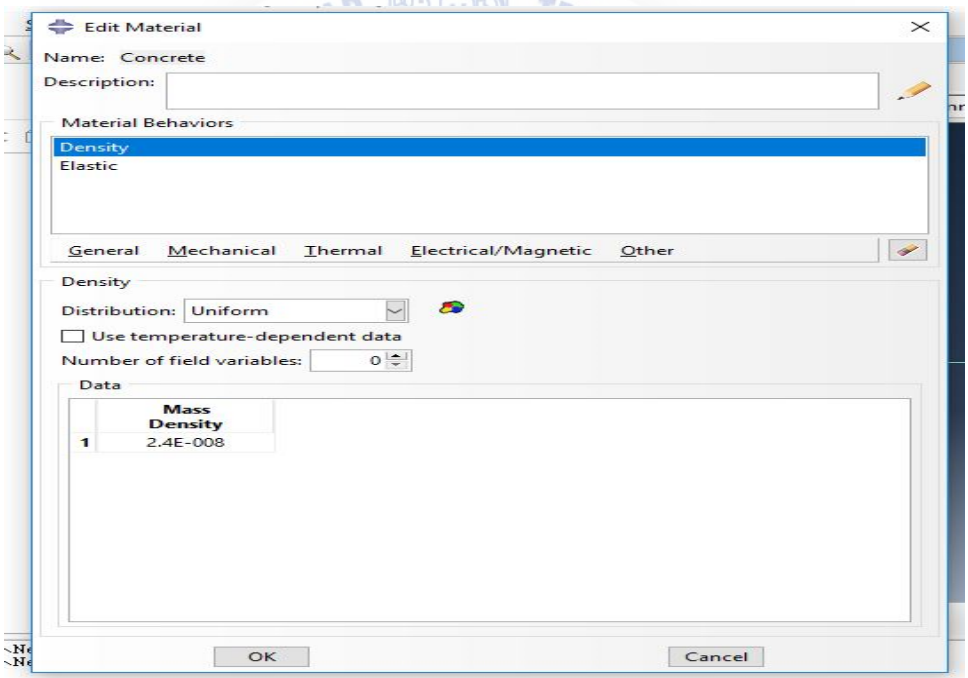

Figure 14: selecting of Mass density of concrete

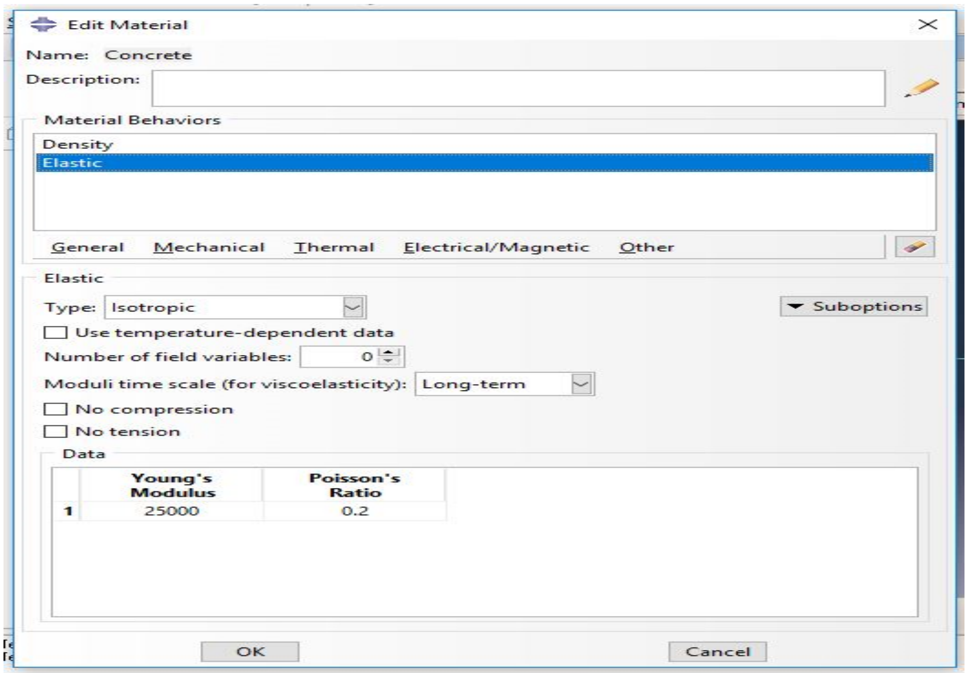

Figure 15: The values of youngs modulus and poissons ratio

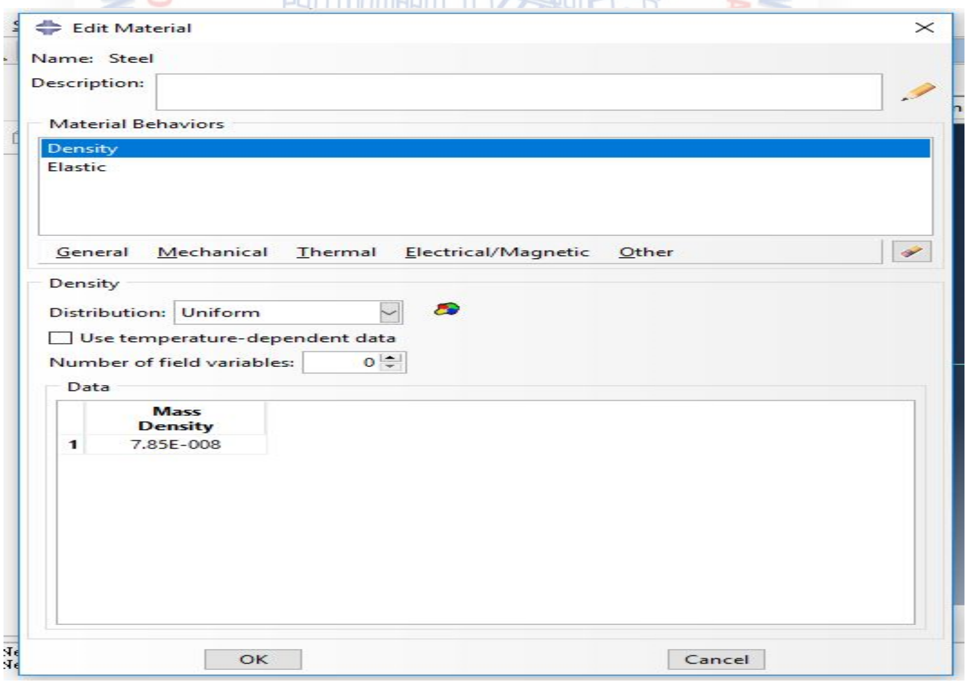

Figure 16: Mass Density of Second Material Steel

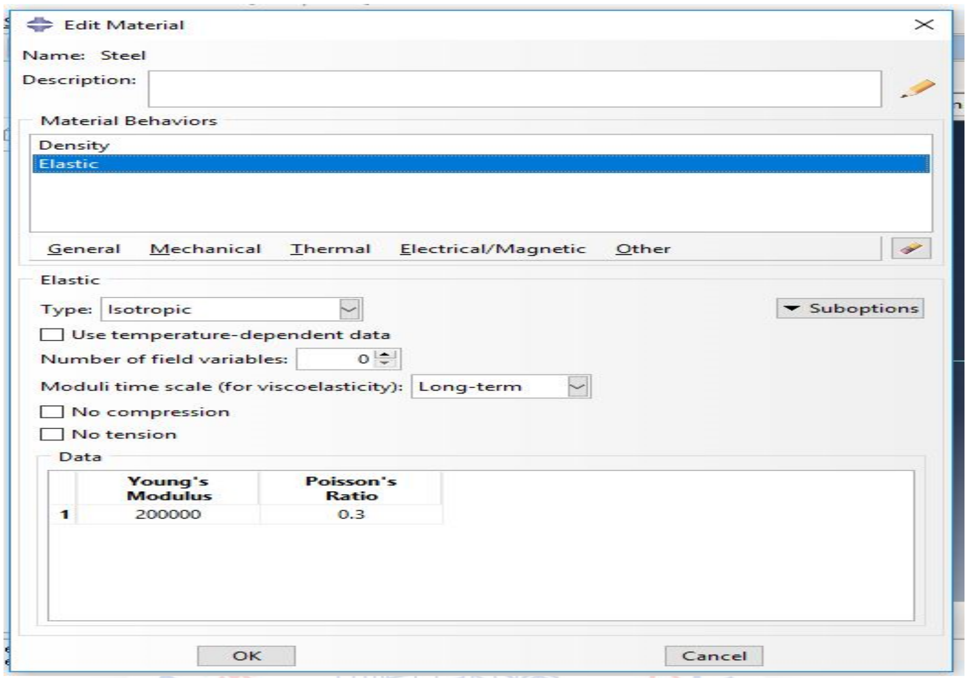

Figure 17: The values of youngs modulus and poissons ration for steel

5.1.5 The Assembly module

In the assembly module the parts which are created are assembled by making number of instances. Each part acts as a single instance and then this instance is analysed. The instances are located at their position by using rotate and translate options and they are multiplied by using linear pattern. The instances are assembled by creating surfaces and the they act as a single unit when these surface are commanded as tied or in case of reinforcement they act as embedded region.

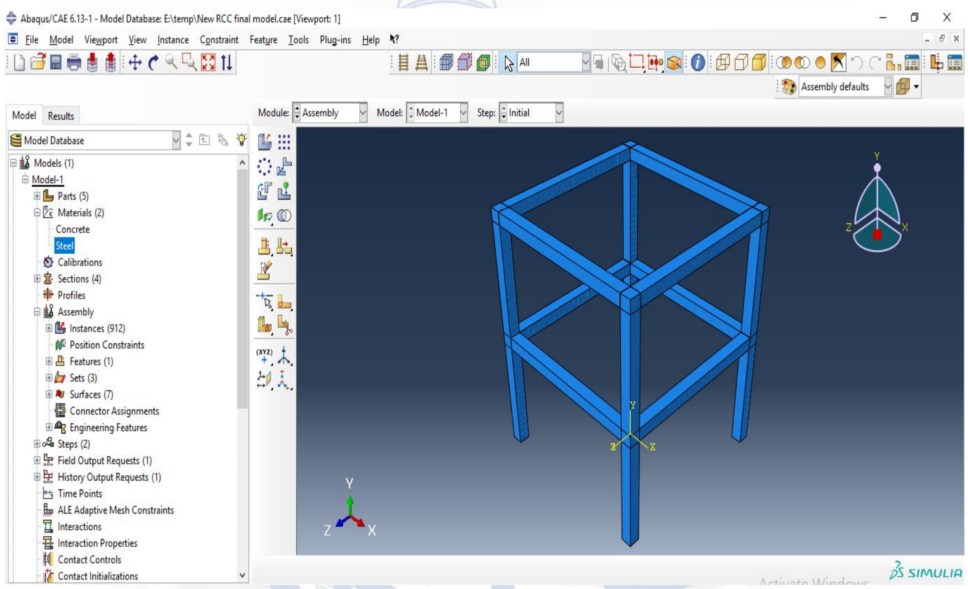

Figure 18: Assembling the parts created by creating surfaces on the faces

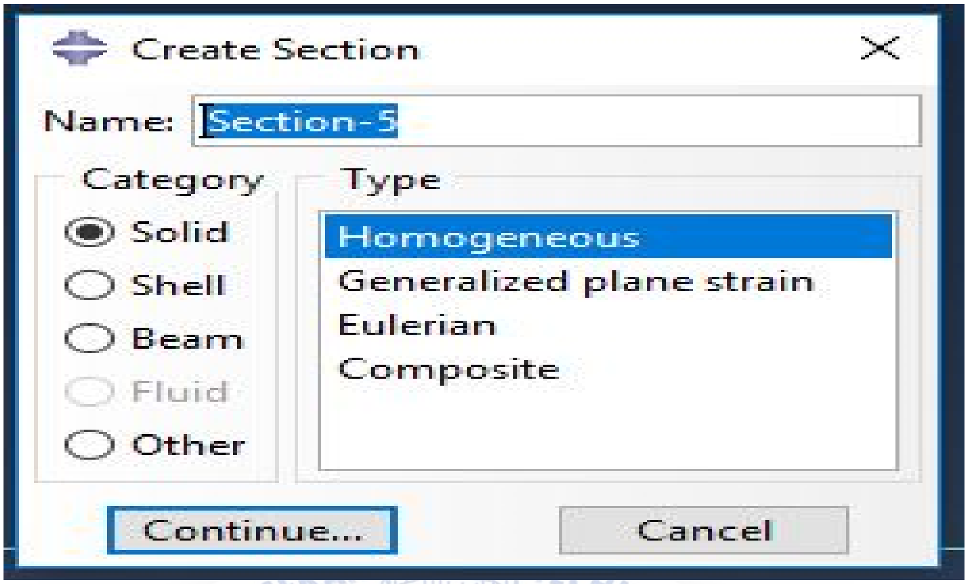

Figure 19: Assigning the section properties to the parts created.

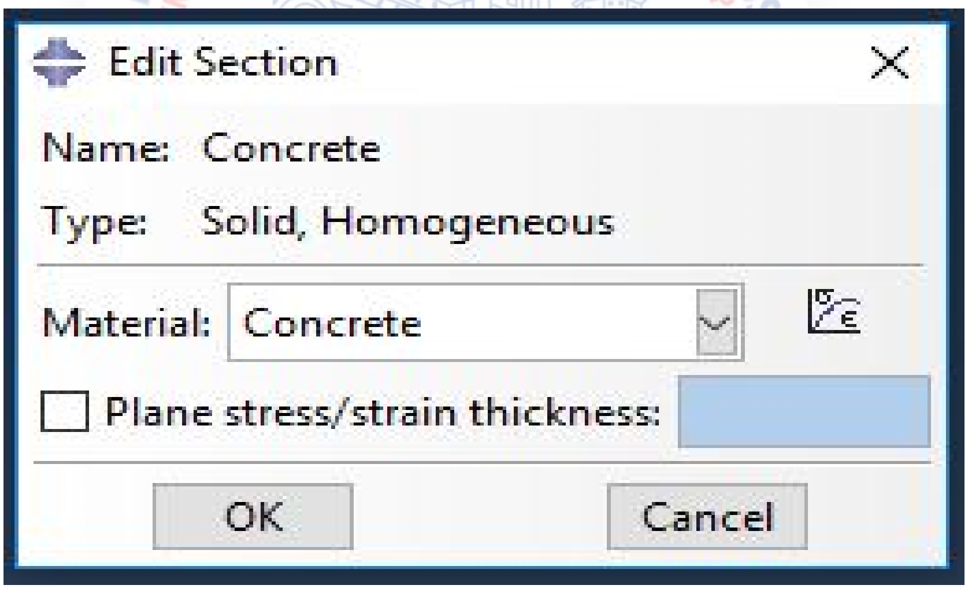

Figure 20: Assigning the materials to the parts such as for beams and colummns

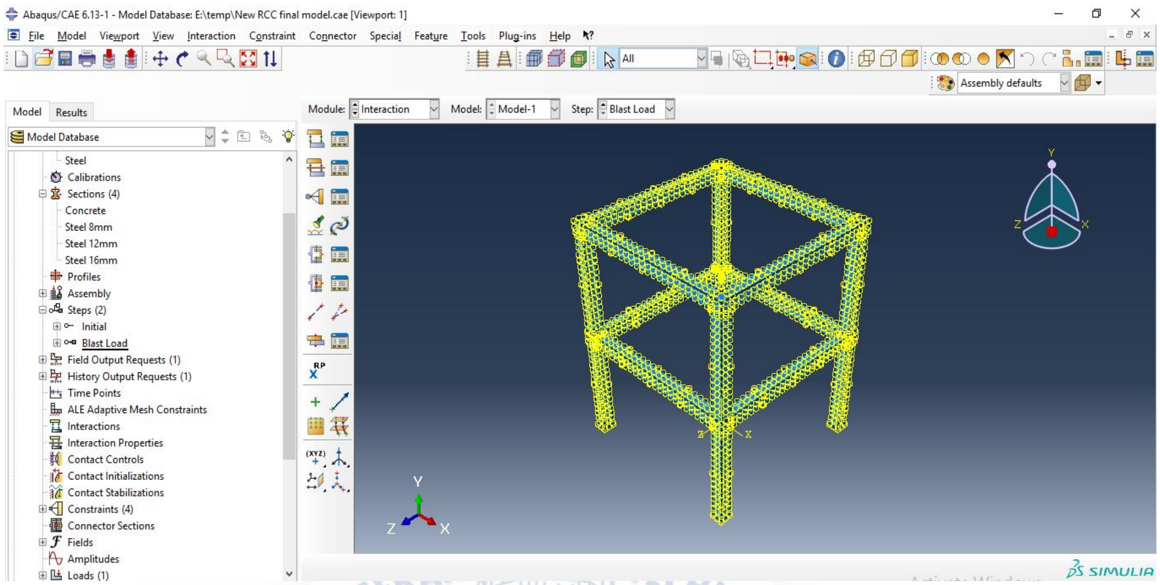

Figure 21: Interactions are created between the parts i.e the beams and columns

5.1.6 The Step Module

The step module is used for creating the steps in analysis, specifying output requests and specifying adaptive meshing and analysis control. The first step is initial which the default step is and then the steps are created based on the loading patterns.

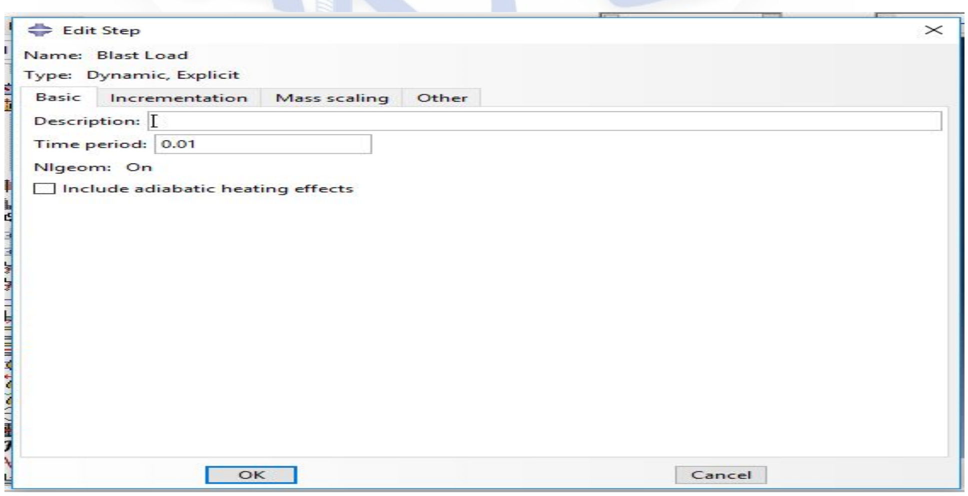

Figure 22: The loading as Blast is assigned.

5.1.7 The load module

In the load module the loads are defined. In case of blast load their amplitudes can be defined. The boundary conditions can also be managed in this module. Mostly the software allows to define the boundary condition depending upon us but we have chosen to encastre inn all directions i.e. the boundary conditions are fixed at bottom. It is restrained from displacement and rotation from its position. The application of blast load in incrementation pattern is applied on one face of the structure.

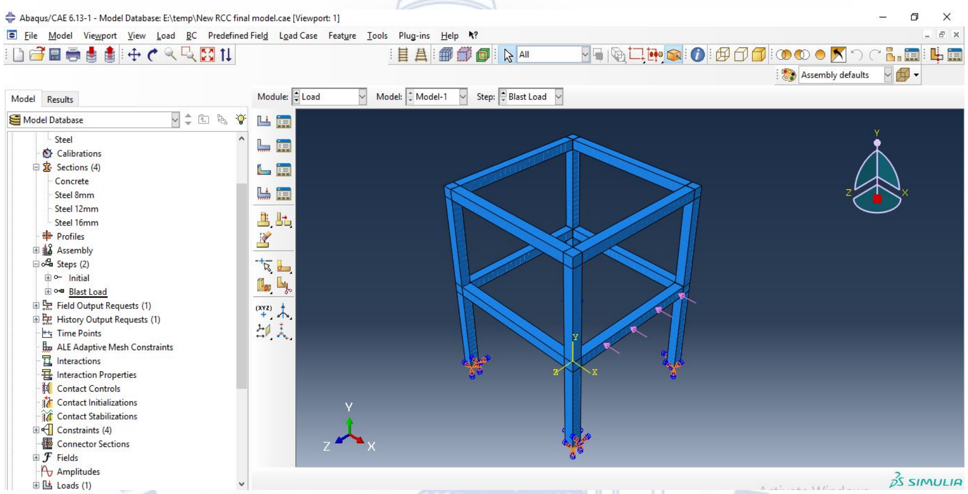

Figure 23: The load is applied as blast with variation in magnitude but in constant direction on one face of the part.

5.1.8 The mesh module

The mesh module contains tools that allow us to generate mesh of various size. This is the key part of the software where finite element method actually works and is depended entirely upon this module. The mesh of different sizes have been selected i.e. 25mm,30mm,35mm,40mm etc and the correct size of mesh for the concrete is determined. Mostly 20mm or 25mm is suitable for concrete by the past experimental results. Initially use the seed part to seed the structure and then go to mesh the part or mesh the assembly. A dialogue box will appear asking the size of mesh then the mesh size should be defined appropriately.

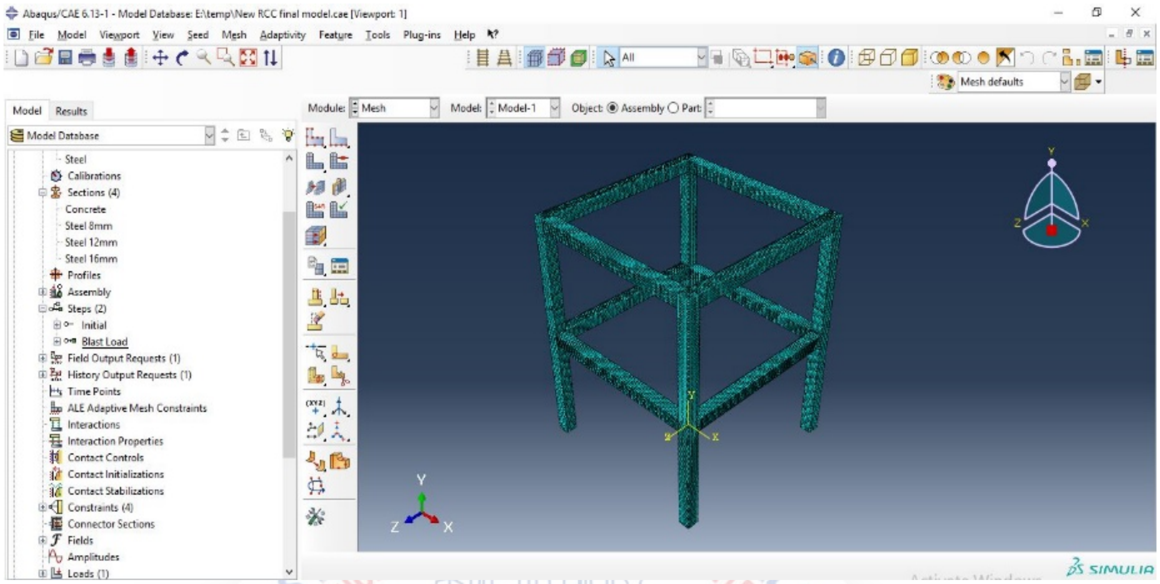

Figure 24: Initially the parts or instances as an assembly is seeded and they are mesh to part or assembly. The mesh size was kept varying.

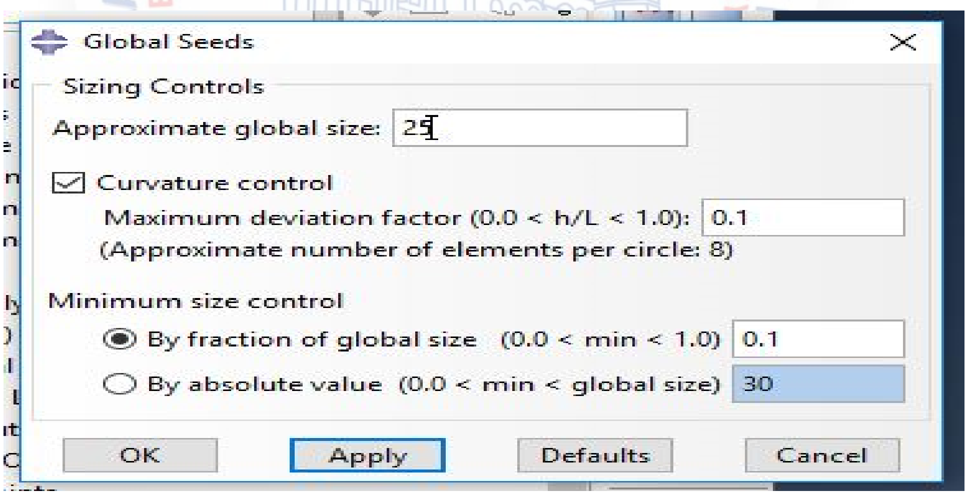

Figure 25: The mesh size of 25mm is selected for the seeded part or instance.

5.1.9 The job module

In the job module the job is created and the name of job is written taking care that the file is not overwritten as it may lead to many errors in analysis thereby aborting the job file. In the job file full analysis needs to be selected and the job file has to be submitted for the computation of results.

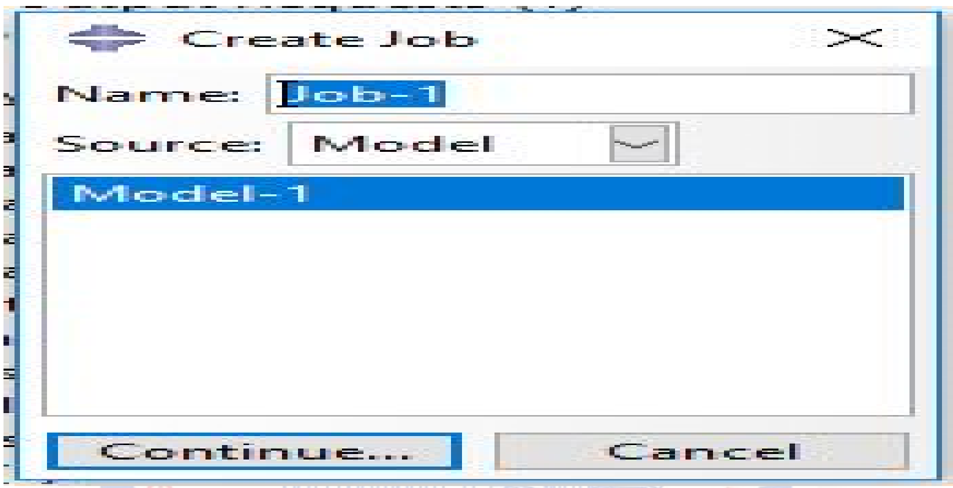

Figure 26: The job is created for various mesh sizes.

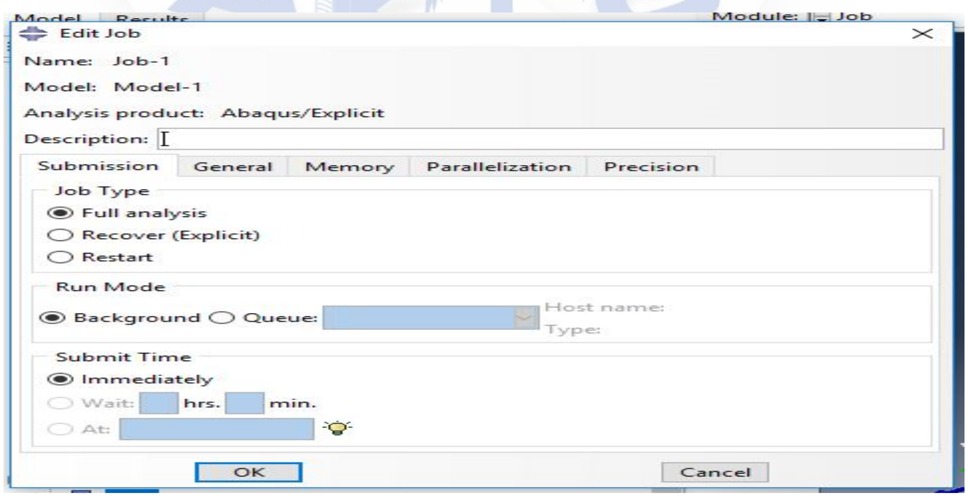

Figure 27: Results and Data Can be Chacked And Monitored

5.1.10 The Optimization module

This module basically deals with the optimization of results i.e the stresses, shear force, axial force, deflection is determined based on the given loading. If the mesh size is small the computation of results take longer time and vice versa. Thus based on the current loading the reults are computed and are as follows

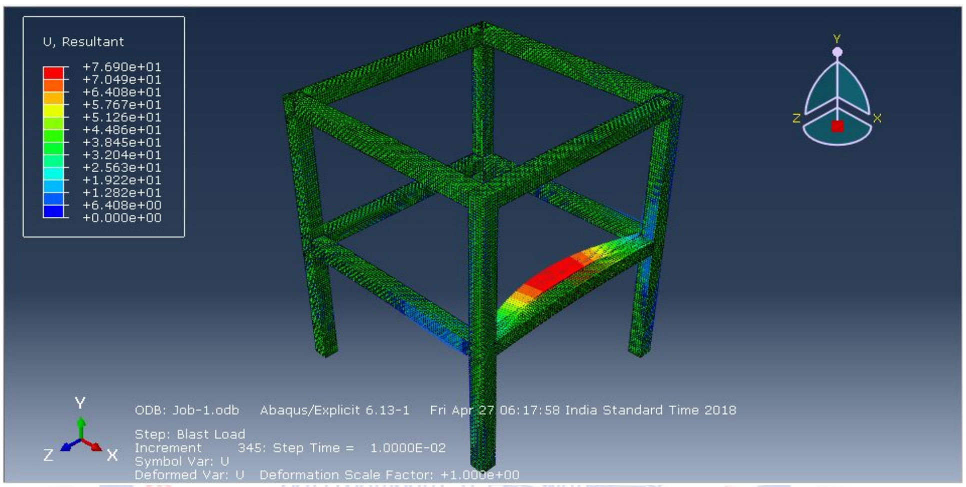

Figure 28: The deflection of beam under a blast load.

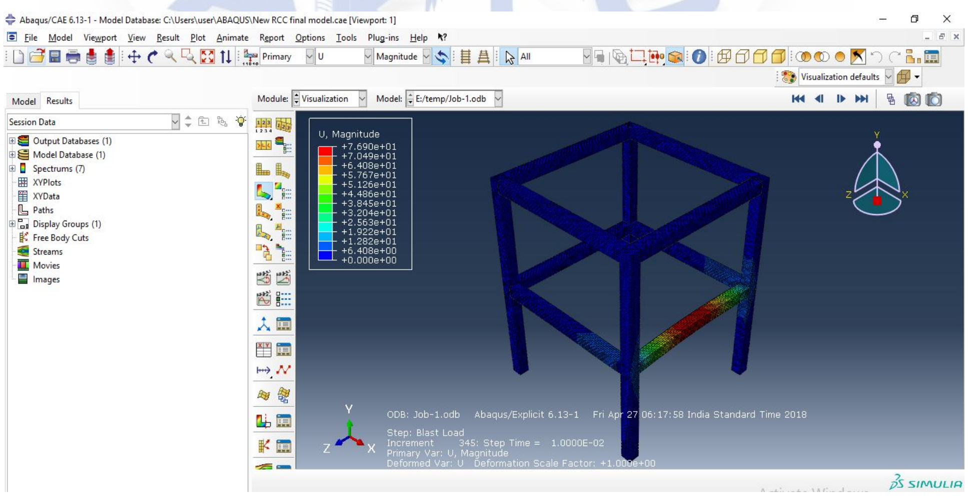

Figure 29: output file is received by submitting the job

Computation of results

The results are then computed after the submission of job file. If the job file is aborted due to errors, the errors need to be rectified and corrected for post processing. Mostly the errors are in mesh module and the element module.

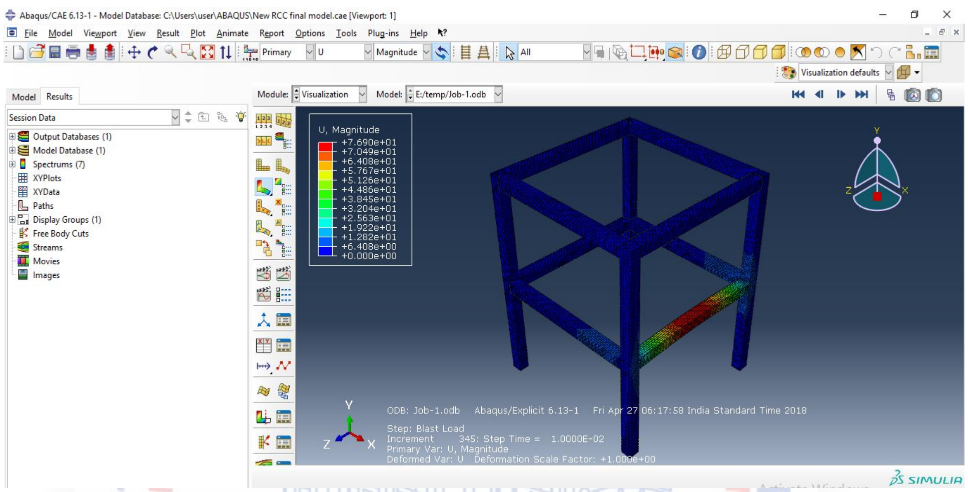

Figure 30: output file is received by submitting the job

6 Analysis

Analysis of the frame by considering different mesh sizes The analysis of the frame is done by varying the mesh sizes and load vs deflection curve is obtained with a load interval of 10KN. Load condition and modeling is kept as per the modelling and analysis is done. Load is increased at a rate of 10KN in addition to this a conversion study has been carried out by considering various mesh size of 20mm, 25mm, 30mm, 35mm, 40mm, 45mm and 50mm and from this a load vs deflection curve is plotted. From the results obtained it is noted that all the mesh size have almost the same deflection value. Also in addition to this it is also concluded that the deflection is continuous as in case of finite element method there is no breaking point or failure point for nonlinear analysis.

Table 6: Different load on RCC structure

load	20	25	30	35	40	45	50
0	0	0	0	0	0	0	0
10	76.94	76.9	76.94	76.9	76.92	76.9	76.9
20	153.7	153.7	153.8	153.7	153.8	153.7	153.7
30	230.3	230.3	230.5	230.3	230.2	230.4	230.3
40	306.7	306.7	306.8	306.7	306.8	306.7	306.7
50	382.6	382.6	382.7	382.6	382.7	382.6	382.6
60	457.9	457.9	458.1	457.9	458	457.9	457.9
70	532.6	532.6	532.9	532.6	532.8	532.6	532.6
80	606.5	606.5	606.8	606.5	606.7	606.5	606.5
90	679.6	679.6	679.9	679.6	679.8	679.6	696.6
100	751.8	751.8	752.1	751.8	752	751.7	751.8

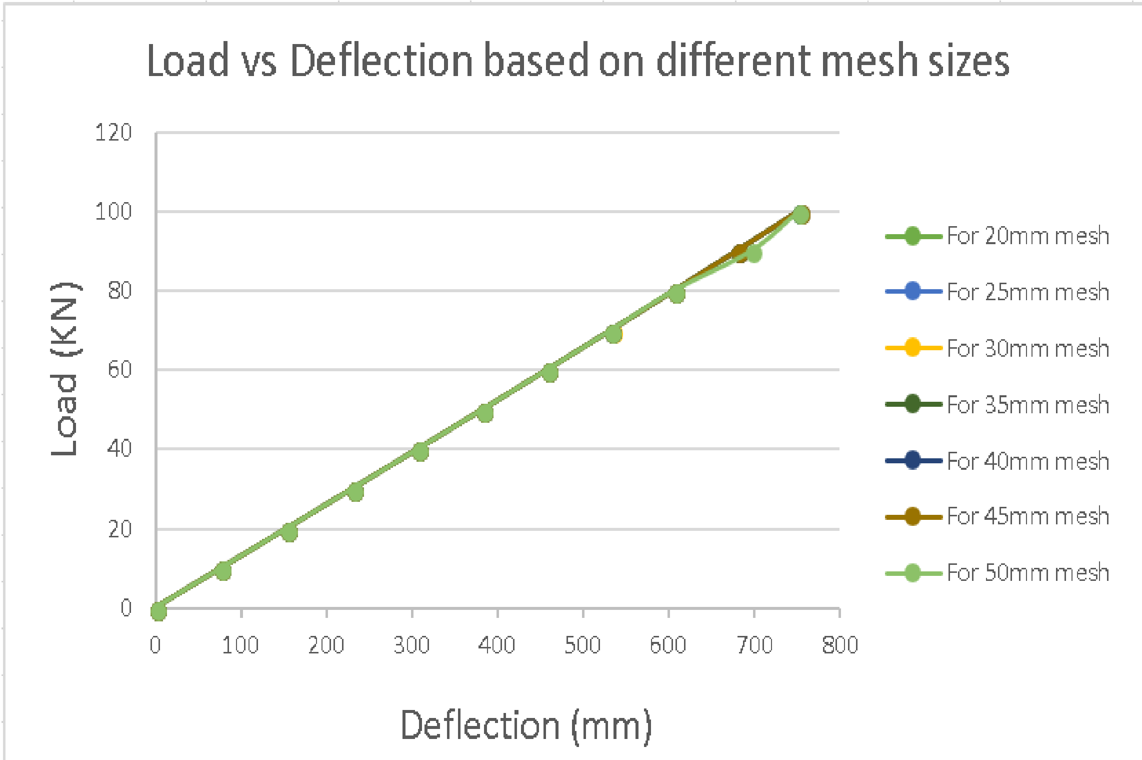

Figure 31: loads vs deflection base on different mesh size

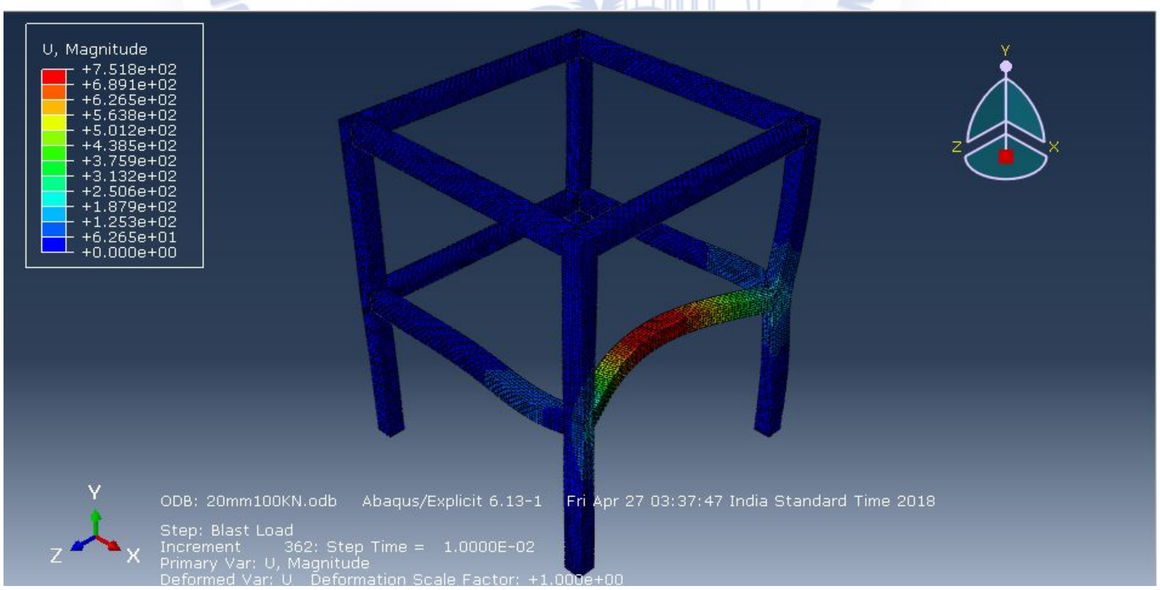

Figure 32: Max deflection at 20mm mesh at 100KN

6.1 Analysis of RCC framed structure under blast loading

For analysis 4.2m x 4.2m frame is chosen as shown below (Fig 01) This frame is analysed for blast loading which is applied laterally to one face of the member. After that a critical condition for different load conditions having maximum lateral deflection is chosen for analysis purpose.

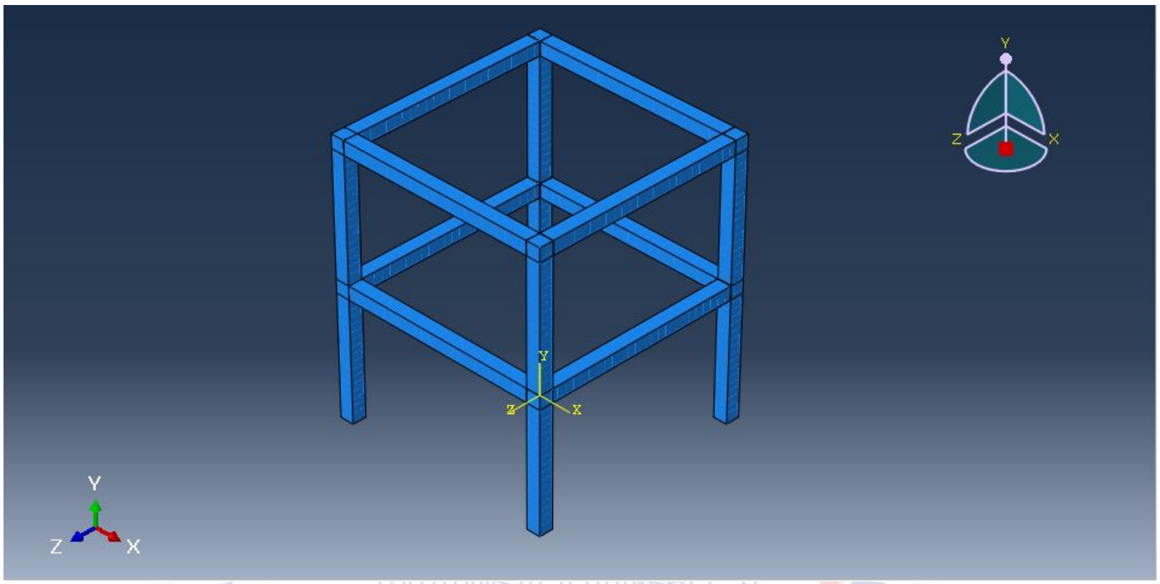

Figure 33: A model of frame

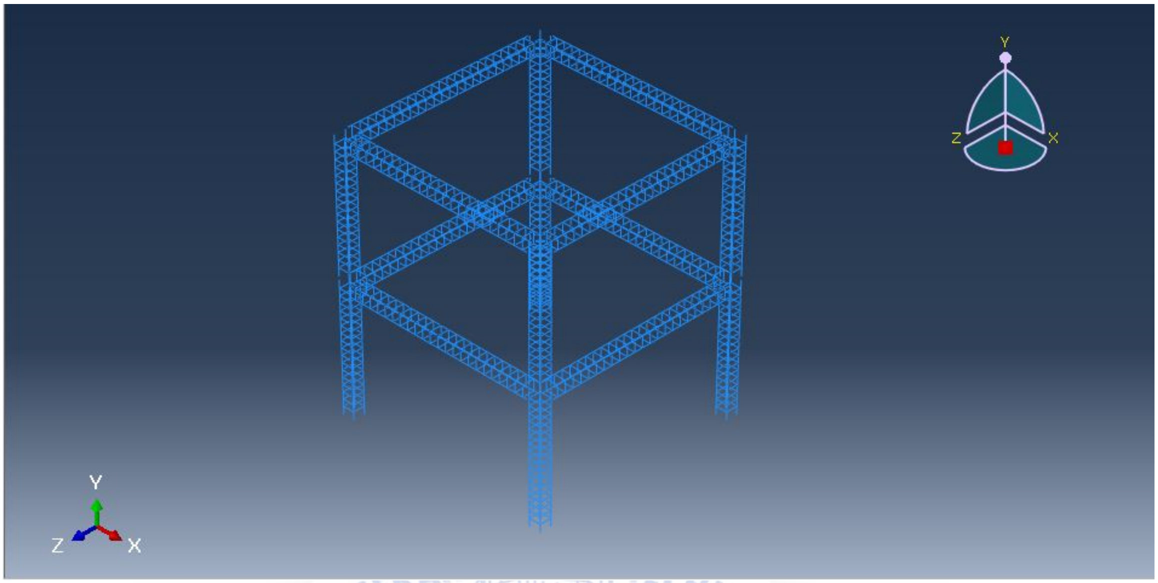

Figure 34: A framework of steel with a cover of 25mm. Reinforcement modelling in ABAQUS

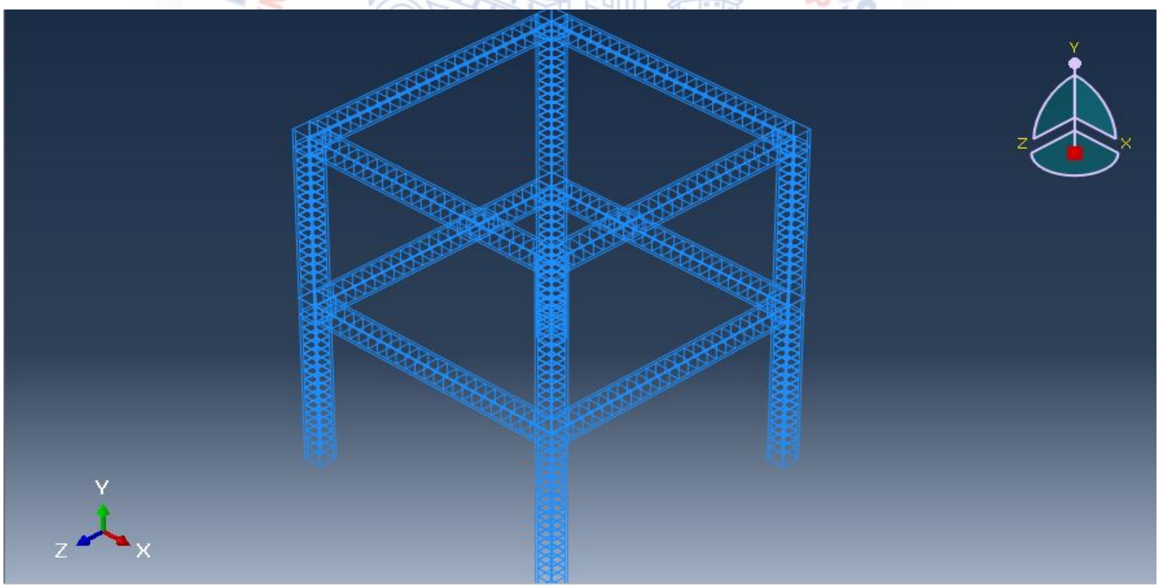

Figure 35: A frame showing the embedded region of reinforcement

The properties of concrete and steel are as per IS 456 and modeling of frame is done in ABAQUS as following. Reinforced modeling is done by using embedded region method

6.2 Analysis of Frame

Load is applied on one face of the beam in the form of pressure in various amplitudes with variable values of load and load vs deflection curves are plotted by considering fixed boundary conditions. The result obtained is nonlinear but concrete damage plasticity and plastic properties of concrete are neglected. Following values for different amplitude of load vs deflection is stated below with the graph showing linear vs nonlinear behavior.

Table 7: Load vs Deflection

Load	Deflection
0	0
1	5.257
2	21.03
3	47.32
4	84.13
5	105.35
6	189.1
7	257
8	376.2
9	422.3
10	518.6

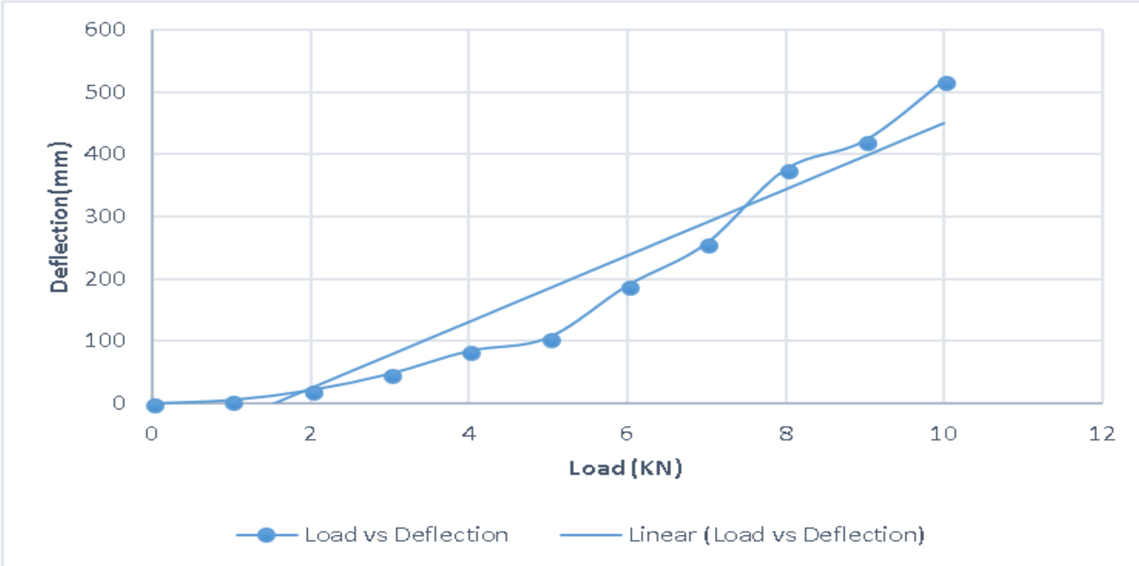

Figure 36: Graph showing deflection linear vs non-linear

7 conclusion

- The RCC framed structure is analysed and the values for various mesh sizes and for various loads are obtained. Reducing the mesh size may lead to tedious and time consuming results. Thus from the above analysis it is concluded that a mesh size in range of 20-30mm is satisfactory for non-linear analysis of concrete.
- The frame can also be analysed by considering various support conditions.
- Finite element method does not states a well-defined failure point in case of non-linear analysis and the deflection is continuous.
- For more accuracy concrete damage plasticity parameters can be considered.
- To minimize the deflections the grade of concrete can be increased.

8 References

BOOKS

(1) An introduction of finite element method by J N REDDY Oscar s. Wyatt chair in mechanical engineering

(2) Finite element method by S.S. BHAVIKATTI.

(3) Finite element method by C.S. Desai.

(4) Finite element method by (Y.M. Desai, T.I. ELDO and A.H. SHAH)

RESEARCH PAPERS

(1) Amiya Kr. Samantha and S. Tripathi, Effect of in plane forces in beam column junction of RC frame in linear regime. ARPN Journal of Engineering and Applied Sciences

(2) Tomasz jankowiak, Tomasz Odygowski, Identification of Concrete Damage Plasticity Constitutive Model. Foundation of Civil and Environmental Engineering. No. 6, 2005

(3) S. V. Chaudhari. M. A. Chakrabarti, Modeling of concrete for non linear analysis using finite Element Code ABAQUS. International journal of computer application (0975-8887) volume 44-No.07 April 2012

(4) Hamid Sinaei, Mahdi Shaiati, Amir HoseinAbna, Evolution of reinforced concrete beam using finite element analysis by ABAQUS. Science Research and Essays Vol. 7921), pp. 2002-2009 7 June, 2007

(5) P. Kmiecik, M. Kaminski P.Modeling of Reinforced Concrete Structures and Composite Structures with Strength Degradation taken in consideration. Archive of Civil and Mechanical Engineering.Vol XI No. 3, 2011

(6) HsuhanTeh Huand William C. Schnobrich, Nonlinear Analysis of Cracked Reinforced Concrete. ACI Structural journal title No. 399a, Vol. 49, No.22

(7) IftekharAnam and Zebun N. Shoma, Non linear properties of reinforced cement concrete

(8) Analysis of Blast Loading Effect on High Rise Building,A.V.Kulkarni ,Sambireddy (2014).

(9) Protective design of concrete buildings under blast loading, Mir M. Ali, (2002)

(10) Study on the Response of RC Frames Subjected to Blast Loading, Meenu Murali,Sujisha (2016)

(11) Study On The Effect Of Blast Load On Industrial Structure Paresh Tank,Dr. K.B.Parikh(2015)

(12) Analysis of the Response of a One-Storey One-Bay Steel Frame to Blast, Rupert G. Williams,(2016)

(13) Review Paper On Blast Loading And Blast Resistant Structures, Sourish Mukherjee, Rittik Bhowmik, Aparna Das(2017)

(14) Structural Analysis of Blast Resistant Structures, Demin George, Varnitha M.S (2016)

(15) Simulations Of The Response Of Concrete Structures Subjected To Air Blasts, Peter Svantesson(2013)

(16) Prediction Of Blast Loading And Its Impact On Buildings Nitesh N. Moon

(17) Blast Loading and Blast Effects on Structures, A.Gupta and J.Ramsay, (2007)

(18) Finite element analysis of reinforced concrete columns under different range of blast loads Ahmed Samir Eisa (2014)

(19) Numerical Study of Reinforced Concrete beam subjected to blast loading using Finite element package ABAQUS, Ann M Biju,(2017)