

A
DISSERTATION REPORT
ON
FINITE ELEMENT MODELLING OF FIBER REINFORCED CONCRETE
Submitted in partial fulfillment of the requirements for the award of degree of
MASTER OF TECHNOLOGY
IN
STRUCTURAL ENGINEERING



Submitted by

SAURABH SHARMA

(2014PCS5094)

Under the esteemed guidance of

Dr. RAJESH GUPTA

Associate Professor

Dr. VINAY AGRAWAL

Associate Professor

DEPARTMENT OF CIVIL ENGINEERING
MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY
JAIPUR
JUNE 2016



CERTIFICATE

This is to certify that the dissertation work entitled “**FINITE ELEMENT MODELLING OF FIBER REINFORCED CONCRETE**” which is being submitted by **SAURABH SHARMA** (2014PCS5094) in partial fulfillment for the award of the degree of Master of Technology in Structural Engineering, MNIT, JAIPUR is a bonafide work done by him under our guidance and supervision.

Dr. RAJESH GUPTA

(Associate Professor)

Dr. VINAY AGRAWAL

(Assistant Professor)

Place: JAIPUR

Date: JUNE 2016



DECLARATION

I hereby certify that the work which is being presented in the Dissertation report '**FINITE ELEMENT MODELLING OF FIBER REINFORCED CONCRETE**', in partial fulfillment of the requirements for the award of the Degree of Master of Technology and submitted in the Department of Civil Engineering of the Malaviya National Institute of Technology Jaipur is an authentic record of my own work carried out during a period from July 2015 to June 2016 under the supervision of my guide **Dr. RAJESH GUPTA**, Associate Professor and my co-guide **Dr. VINAY AGRAWAL**, Assistant Professor, Department of Civil Engineering, Malaviya National Institute of Technology Jaipur, India.

SAURABH SHARMA

(2014PCS5094)

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

Dr. RAJESH GUPTA

Associate Professor

Dr. VINAY AGRAWAL

Assistant Professor

**© MALAVIYA NATIONAL INSTITUTE OF TECHNOLOGY
JAIPUR
ALL RIGHTS RESERVED**

ACKNOWLEDGEMENT

Praise to Almighty who has decided whatever is best for us!

This work would not have been what it is if it hadn't been for the people whose constant comments and valuable guidance have made it successful. As I write this acknowledge, I must clarify that this is not just a formal acknowledgement but also a sincere note of thanks and regard from my side.

Without proper combination of cooperation and perspiration it is not easy to achieve anything. There is always a sense of gratitude which we express to others for the help and needy services they render during different spheres of our lives.

I wish to express my heartiest thanks and a deep sense of gratitude to my guides, **Dr. Rajesh Gupta** , Associate Professor and **Dr. Vinay Agrawal**, Assistant Professor , Department of Civil Engineering, Malaviya National Institute of Technology, Jaipur for their valuable guidance, constant encouragement and creative suggestions offered while doing this dissertation work. It is my privilege to have worked under their supervision.

I am also thankful to Mr. Sanjeev Kumar K.S. , Mr. Syed Ahmed Kabeer and Mr. Arnav Kasar for their timely help and support.

My deepest gratitude goes to my family and friends for their unflagging love and support throughout my work. Thanks for those blessings which always protect me from all the dangers of life.

Thank You All.

SAURABH SHARMA

(2014PCS5094)

ABSTRACT

Fiber reinforced concrete can offer an easy, practical and economical method for overcoming micro-cracks and similar type of deficiencies. As we know concrete is weak in tension hence some measures must be adopted to overcome this deficiency. Fibers are strong in tension hence can be used for reinforcing concrete. The present study deals with the finite element modelling of fiber reinforced concrete . Two models were prepared – one for compressive behavior and other for flexural strength. Damaged plasticity approach is used to model the failure of concrete. Failure is characterized by softening which is defined as decrease in stress on increasing deformation. Number of trials were run using various parameters to model the concrete. The objective was to find out the parameters which give results in alignment with the experimental results available. Compressive parameters were found out using the test for compression and tensile parameters were found using test for flexural tensile strength.

TABLE OF CONTENTS

CERTIFICATE	i
DECLARATION	ii
ACKNOWLEDGEMENT	iv
ABSTRACT	v
LIST OF FIGURES	viii
LIST OF TABLE	ix
1. INTRODUCTION	1
1.1 ADVANTAGES OF FEM	1
2. LITERATURE REVIEW	3
3. FINITE ELEMENT MODELLING	6
3.1 VARIOUS CONSTITUTIVE MODELS	6
3.1.1 Empirical models.....	6
3.1.2 Linear Elastic Models.....	6
3.1.3 Nonlinear Elastic Models	7
3.1.4 Plasticity Based Models	7
3.1.5 Continuum Damage Models.....	7
3.2 CONCRETE DAMAGED PLASTICITY	7
Uniaxial Cyclic Behavior	10
3.3 SMEARED CRACKING MODEL	10
4. FEM MODELLING OF COMPRESSION TEST	13
4.1 PART MODULE.....	13
4.2 PROPERTY MODULE.....	14
4.2.1 Defining Material	14
4.2.2 Defining Section.....	15
4.3 ASSEMBLY MODULE.....	16
4.4 STEP MODULE.....	16
4.5 MESH MODULE.....	18
4.6 LOAD MODULE.....	19
5. FEM MODELLING OF FLEXURAL TENSILE TEST	21
5.1 PART MODULE.....	21
5.2 PROPERTY MODULE.....	22
5.3 LOAD MODULE.....	22
5.4 MESH MODULE.....	23
6. RESULTS AND DISCUSSION	24
6.1 COMPRESSION TEST.....	24

6.2 TEST FOR FLEXURAL TENSILE STRENGTH.....	27
6.3 CONCLUSION	31
REFERENCES	32

LIST OF FIGURES

Figure No	Description	Page No
Figure 3.1	Response of concrete under uniaxial tension	8
Figure 3.2	Assumed Behavior of concrete under uniaxial compression	9
Figure 3.3	Uniaxial Behavior of Concrete	11
Figure 4.1	Cube Created in Part Module	13
Figure 4.2	Property Module in Abaqus	14
Figure 4.3	Material Behavior used for defining concrete	15
Figure 4.4	Creating solid homogenous section	16
Figure 4.5	Step Manager Showing Two Steps Used	17
Figure 4.6	Seeding Definition	18
Figure 4.7	Meshing of Part Instance	18
Figure 4.8	Boundary Conditions at the bottom face	19
Figure 4.9	Boundary Conditions at the top face	19
Figure 5.1	Flexural Tensile Strength Test Arrangement	21
Figure 5.2	Beam Created in Part module	22
Figure 5.3	Pinned Boundary Conditions and Loading Applied on the beam	23
Figure 5.4	Meshing of the Beam Created	23
Figure 6.1	Deformed Shape in Compression Test	27
Figure 6.2	Deformed Shape in Test for Flexural Tensile Strength	30

LIST OF TABLE

Table No	Description	Page No
Table 6.1	Compression Parameters for 0% fiber	25
Table 6.2	Compression Parameters for 0% fiber	26
Table 6.3	Compression Parameters for 0% fiber	26
Table 6.4	Compression Parameters for 0% fiber	27
Table 6.5	Compression Parameters for 0% fiber	27
Table 6.6	Tensile Parameters for 0% fiber	29
Table 6.7	Tensile Parameters for 0% fiber	29
Table 6.8	Tensile Parameters for 0% fiber	30
Table 6.9	Tensile Parameters for 0% fiber	30
Table 6.10	Tensile Parameters for 0% fiber	31

CHAPTER 1

INTRODUCTION

Many reinforced concrete structures usually suffer from corrosion failure. Therefore these structures require constant maintenance and repairing to improve their life cycle. Reinforcing the concrete with fiber polymers is one of the ways of minimizing the failure of steel reinforced concrete. Adding fibers improves the hardness and tensile strength of concrete. It also helps to improve the characteristics of deformation and cracking of resulting composite. Sometimes steel bars in concrete are replaced to form fiber reinforced concrete..

The boundary conditions are imposed and the equations of equilibrium are solved for the nodal displacement. From the values of the nodal displacement for each element, the stresses and strains are evaluated using the element properties. Thus instead of solving the problem for the entire structure in one operation, in this Finite Element Method attention is mainly devoted to the formulation of properties of the constituent elements.

1.1 ADVANTAGES OF FEM

The Finite Element Method has many advantages of its own. Some of them are given below.

- Various types of boundary conditions are automatically handled in the formulation. They are systematically enforced just before the solution, for nodal values of the field variables are obtained.
- Higher order elements may be implemented with relative ease.
- The method can be used to account for irregular geometry
- Spacing of nodes need not follow a pattern or rule

1.1.1 DISADVANTAGES OF FEM

The Finite Element Method has its own limitations. They are listed below.

- There are areas such as “Aspect Ratio”, which may affect the final results.
- The cost involved in the solution of the problem is high.

With the increase in modelling capabilities of the various software packages, it is beneficial to have a finite element model of any test. This reduces the cost involved in actual testing in laboratories. But one thing that needs to be noted is that modelling can never be used to completely eliminate testing. Some amount of testing is always

required. Though finite element modelling is a relatively new phenomenon in civil engineering but large amount of work is going on in this field

CHAPTER 2

LITERATURE REVIEW

The importance of finite element modelling is increasing day by day. With the increase in modelling capabilities of the various software packages, it is beneficial to have a finite element model of any test. This reduces the cost involved in actual testing in laboratories. But one thing that needs to be noted is that modelling can never be used to completely eliminate testing. Some amount of testing is always required. Though finite element modelling is a relatively new phenomenon in civil engineering but large amount of work is going on in this field. Many research papers have been published which use various finite element packages to find a reasonably good model that conforms to experimental results. Various finite element softwares like ADINA, ANSYS, ABAQUS, FEA Tool, and RFEM are available which provide wide range of modelling capabilities.

The software package used in the present work is ABAQUS. Many research papers have been published which use ABAQUS for modelling various types of concrete. The advantage of using ABAQUS is that it provides many inbuilt models related to concrete which can cater to different kinds of needs.

A 3D model of a concrete cube can be used to test various types of concrete for compression behavior (M.A.Chakrabarti 2012). The cube represents the standard cube of size 150mm×150mm×150mm. The material of the cube is concrete and various material properties of the cube like elasticity, poisson's ratio, yield stresses, inelastic strains etc. can be entered. Many models like the concrete damaged plasticity model, hashin damage model also provide us the option of inserting damage parameters. They are used to model the damage that the concrete undergoes with increase in inelastic strain. Now various types of loading can be applied to this cube and the behavior of the cube can be checked. ABAQUS provides us the option of determining various kinds of stresses in post processing like Maximum principle stress, Mises stress, Tresca , direct stresses in any direction like S11, S22, S33 or shear stresses like S12, S23, S13. Similarly such strains can also be found. Deformed shapes of the model can also be found.

But there are some limitations also to it. All these softwares are not tailor made for concrete where we can just feed the data and get the results. The accuracy of your model will depend on the input parameters regarding material , the size of meshing and the

computational effort provided. Generally there is a trade off between accuracy and computational effort.

Concrete is a very non homogenous material. Hence one of the major challenges encountered in finite element modelling of concrete is how to add the material properties of concrete. Hence identifying the material parameters is of utmost importance. A class of material models called elastic –damage models are also generally used while trying to describe the behavior of non – homogenous material like concrete. In these models, parameters related to damage mechanics were used along with the linear elastic response of the material. The elastic-damage models were successful in capturing the behavior of concrete in uniaxial compression and tension tests but were unable to capture plastic irreversible deformations and the effects of more complicated type of loadings. To overcome such shortcomings continuum damage mechanics was used along with the elastic plastic constitutive equations. One way to do it is to define the damage parameters with respect to plastic strains. Some laboratory tests are also required to identify the material parameters. The identification of material parameters has been presented in (JANKOWAIK and LODYGOWSKI 2005) for Concrete Damaged Plasticity model in B50 concrete. In CDP model with increase in plastic strain there is decrease in the stiffness of the material and thus the concept of degraded stiffness is used. The softening part of the stress strain curve is obtained by entering the values of scalar damage parameter. The value of damage parameter varies from 0 to 1, 0 representing no damage and 1 representing 100 % damage. Damage can also be based on energy where total fracture energy is specified at failure.

Another way of modelling damage is using smeared cracking approach (L, et al. n.d.). This approach is generally used when the crack pattern in the concrete is not known. Hence in smeared cracking approach it is assumed that the cracks are randomly distributed all over the surface. Unlike the discrete cracking approach, the smeared cracking approach does not require specialized software and most software packages now –a –days support smeared cracking approach. This approach is generally beneficial to use when there are large number of small cracks.

Another way of defining the damage in fibre reinforced composites in ABAQUS is by using Hashin Damage model. (Barbero, et al. 2013). Damage is divided into Damage evolution and Damage stabilization. Compressive and tensile strengths in the direction of fibers and transverse to the direction of fibers are required to be entered. Also there is an option to use temperature dependent data.

After deciding on which material model to use it is also important to calibrate your model with the available experimental results. Various works have been published in which certain parameters are taken up for calibration. (Michal and Andrzej 2015) have calibrated the viscosity parameter in tension test and dilation angle in compression test. Also the influence of mesh size is taken into consideration while performing tension tests in two different specimens.

CHAPTER 3

FINITE ELEMENT MODELLING

3.1 VARIOUS CONSTITUTIVE MODELS

Out of the many finite element packages available, the software using in the present study is Abaqus CAE. It is one of the most commonly used softwares in the field of engineering due to its wide range of applicability. There are many material models available in Abaqus to model various types of materials depending of the suitability of various tests to be simulated. Various constitutive models are described below:

3.1.1 Empirical models

The constitutive laws governing the behavior of material are obtained on the basis of observation on experiments. Functions defining various aspects of the behavior are proposed based on experimental data. But the most commonly encountered difficulty while using empirical models is the lack of sufficient experimental information. One of the reasons for insufficient experimental data is the difficulty associated with testing. Most of the standard compression testing machines apply loads rather than deformations. Thus uncontrolled sudden failure occurs after peak loads. Several techniques have been developed to overcome this difficulty but some of them are costly and are not available in a normal testing lab. Another reason for the scarcity of test data is scatter of test data associated with machine precession, testing technique and statistical variation of material properties from sample to sample.

3.1.2 Linear Elastic Models

These are the simplest constitutive models available in the literature. In linear elastic models the behavior of concrete is treated as linearly elastic until it reaches ultimate strength and subsequently it fails in brittle manner. In case of tension in concrete , because the load at which failure occurs is small, linear elastic model is quite accurate and sufficient to predict the behavior of concrete till failure. But these models are often inappropriate as concrete is a material sensitive to a lot of external things whose general response under imposed load is highly non linear and inelastic. These models also fail to accurately predict the behavior of concrete where there is reversal of loads

3.1.3 Nonlinear Elastic Models

When the concrete is under multiaxial state of stress, the linear elastic models fail to depict the behavior of concrete. Under such states of stresses it is better to use the nonlinear material models. In such methods incremental stress –strain relationships are used. Thus in place of just defining one set of data as in case of linear elastic models, many set of data are provided which are able to give better results than linear elastic models.

3.1.4 Plasticity Based Models

In the recent past many plasticity based models have been developed. These models take into consideration the inelastic deformations undergone by the concrete and the resulting damage that occurs in the concrete matrix. All the models have some advantages and disadvantages. The suitability of a particular model depends upon the type of application of the model. In plasticity based models, total strain is divided into two parts elastic strain and inelastic strain. To define the behavior of concrete in plastic region a flow rule is required. Also a yield criteria is entered based on the data available from experiments.

3.1.5 Continuum Damage Models

Damage based models are generally used to describe the mechanical behavior of concrete. A set of damage variables are used to describe the degradation in stiffness. . Damage parameters can be scalar or tensor depending upon the level of complexity. Continuum Damage models are also used to predict the growth of damage in laminated composites. Thus it is very important to choose an appropriate material models based upon the type of material used, types of loading and taking other factors into consideration.

In Abaqus CAE mainly two types of material models are used to describe the behavior of concrete: smeared cracking model, damaged plasticity model.

3.2 CONCRETE DAMAGED PLASTICTY

Concrete Damaged Plasticity model, generally referred as CDP model is used to enter the material properties of concrete. The inelastic behavior of concrete is represented by using isotropic damaged elasticity along with isotropic damaged plasticity both in tensile and compressive states. It is applicable to both plain and reinforced concrete. It also finds its application in various types of loading like monotonic, cyclic and dynamic. CDP model can also be defined to be sensitive to the rate of straining.

The CDP model is a plasticity –based, continuum, damage model for concrete. Basically two failure modes are considered – tensile cracking and compressive crushing. Two hardening variables, ϵ_t^{pl} and ϵ_c^{pl} control the evolution of yield surface. They represent the tensile and compressive plastic strains. The behavior of concrete in uniaxial tension and compression is assumed to be dependent on the damaged plasticity of the concrete. CDP model also allows one to define tension stiffening. Tension stiffening can either be specified by means of a relation between post failure stress – strain or by using the criterion based on fracture energy.

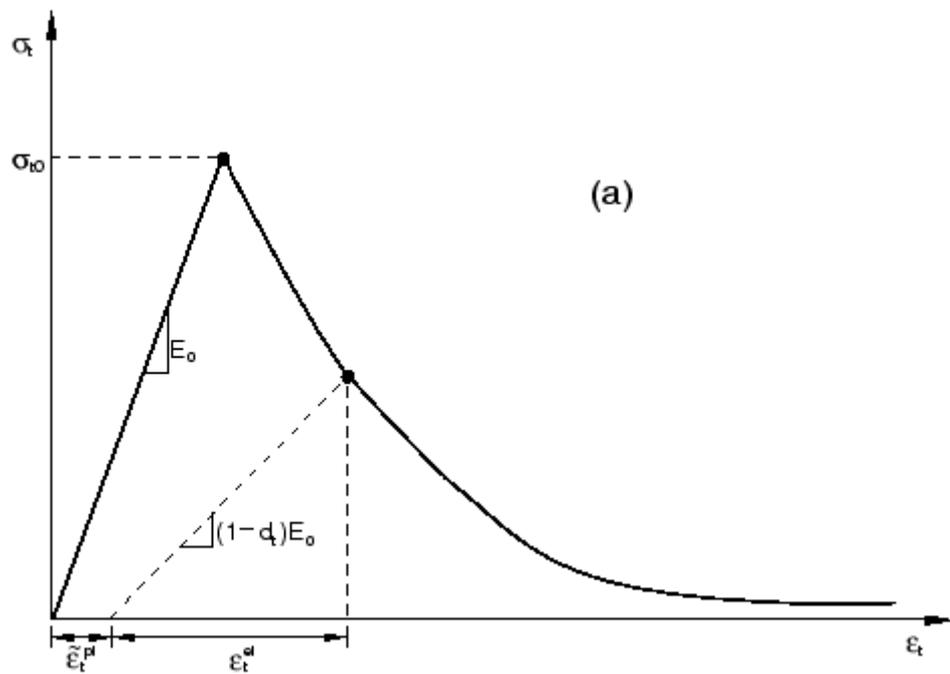


Figure 3.1: Response of concrete under uniaxial tension (Abaqus Analysis User's Manual n.d.)

Figure 3.1 shows the response of concrete under uniaxial tension. A linear elastic behavior is observed till the failure stress is σ_{10} is reached. Failure stress denotes that the micro-cracking in the concrete material has started. After this a softening response is observed. This shows that micro-cracks have been formed.

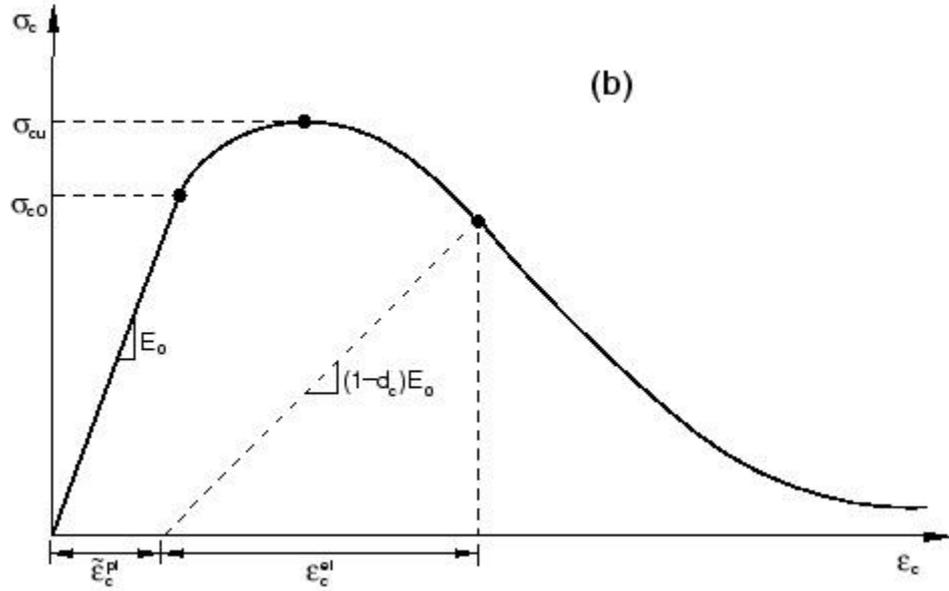


Figure 3.2: Assumed behavior of concrete under uniaxial compression

In figure 3.2 the behavior of concrete under uniaxial compression is shown. A linear response is observed till the stress reaches the value of initial yield, σ_{c0} . After this plastic response starts. After the value of first yield stress hardening is observed till the value of ultimate stress σ_{cu} is reached. Stress hardening is then followed by strain softening. The CDP model assumes that the above uniaxial stress-strain curves can be used to find the relationship between stress and inelastic strain. As is shown in above figures, on the strain softening region of the curve, if the specimen is unloaded, the response of concrete is a weakened one. Hence it is concluded that the elastic stiffness of concrete appears to be damaged. This damage of stiffness is represented by two variables (called damage variables), d_t and d_c . These damage variables are considered to be dependent on plastic strains, temperature and field variables:

$$d_t = d_t(\varepsilon^{pl}, \theta, fi); 0 \leq d_t \leq 1$$

$$d_c = d_c(\varepsilon^{pl}, \theta, fi); 0 \leq d_c \leq 1$$

These damage variables can have values between 0 and 1. Zero value represents the undamaged state of concrete material while one represents the 100 percent damage or the total loss of strength. If E_o is the undamaged or initial elastic stiffness of concrete, the stress-strain relations under uniaxial tension and compression loading are, respectively:

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

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

Uniaxial Cyclic Behavior

Experimental observations indicate that there is some recovery of elastic stiffness when the load reverses its nature (or sign) during a uniaxial cyclic test.”. This is an important aspect of the concrete behavior under cyclic loading. This effect is usually noticed when the load changes from tension to compression. Due to closing of tensile cracks there is some recovery in the compression stiffness.

In CDP model , the degraded stiffness or the reduction of the elastic modulus is given in terms of a scalar degradation variable d as

$$E = (1 - d)E_o$$

Where E_o is the initial (undamaged) modulus of the material. This expression holds both in the tensile and compressive sides of the cycle.

3.3 SMEARED CRACKING MODEL

This is another one of the material model available in Abaqus for modelling the behavior of concrete besides the damaged plasticity model. Similar to the damaged plasticity model this can also be used for both plain and reinforced concrete. This is mainly designed to be used when there is monotonic straining being applied on the concrete. Thus crack detection , representing cracks and behavior after cracking assumes the major challenge while modelling. This model is named as ‘smeared crack model’ because it does not trace individual “macro” cracks. At each integration point of the model , constitutive calculations are performed. The influence of cracks in the calculations comes into play by the way they change the stress and stiffness of the material which is associated with the integration point.

Once the stress reaches the failure value, the behavior of the concrete after failure is characterized with tension stiffening. Tension stiffening can allow one to define the strain softening part of the stress- strain curve of the cracked concrete. Tension stiffening is essential in the smeared cracking model. It can be entered by using a stress-strain relation which defines the behavior of concrete after cracking. It can also be applied by using a fracture energy cracking criterion.

When the stresses are mainly compressive, the behavior of concrete can be modelled using elastic- plastic theory.

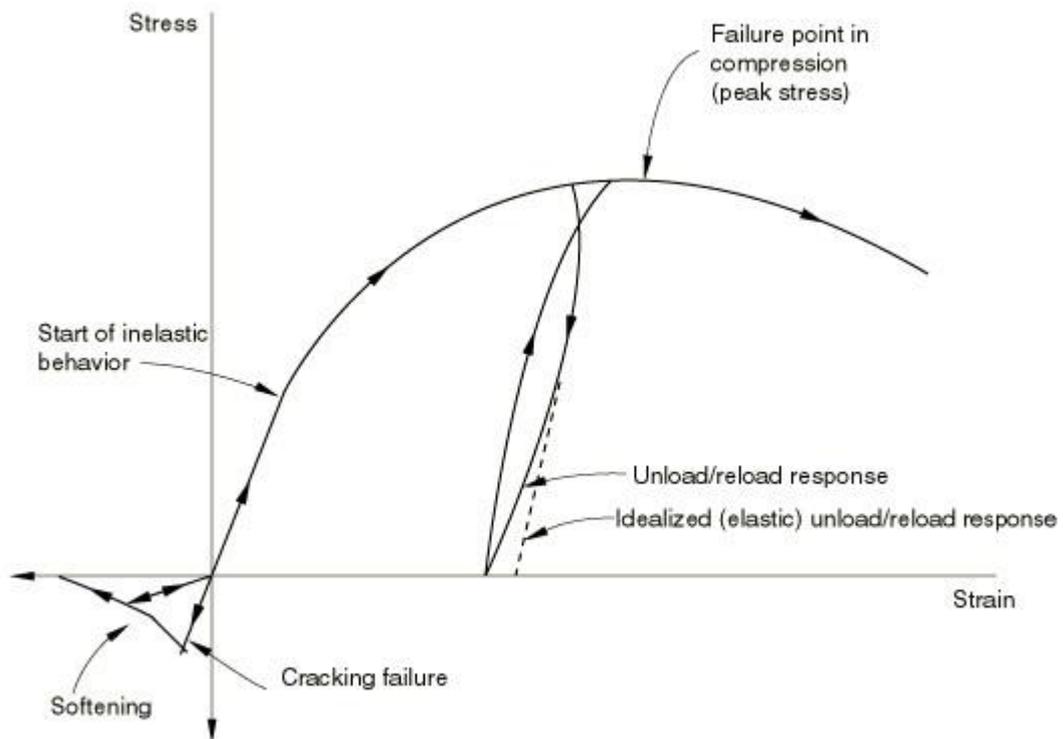


Figure 3.3: Uniaxial Behavior of concrete

The response of concrete under uniaxial cracking and compression that are assumed in the smeared cracking model is illustrated in figure 3. Under compressive loading, the initial response of concrete is mainly an elastic one. As we go on increasing the stress, some inelastic (non recoverable) straining is occurs which leads to the softening of the response of the concrete. On further increasing the stress an ultimate stress is reached after which the stress begins to fall till the time concrete can no longer take any stress. If we remove the load after the onset of inelastic straining, the unloading response of concrete is not similar to initial elastic response. It is somewhat softer than the initial elastic response. This denotes that the elasticity has been damaged. Since the smeared cracking model primarily deals with monotonic straining, this change in the unloading response is neglected.

Similarly if the specimen is loaded in tension, elastic response is observed till the load remains low in the range of 7% to 10% of the ultimate stress in compression. After this cracks start to initiate. After the initiation of cracks, it becomes very difficult to observe the behavior of concrete even in the stiffest testing machines. It is assumed that the

damage caused by cracking can be associated with the loss of elastic stiffness. The non-recoverable(permanent) strain associated with cracking are ignored in this model. This means that if the stress changes its nature and becomes compressive, all the cracks will close completely. These observations can also be generalized to incorporate the multi axial states of stress. For the calibration of the simplest version of the model two experiments are required – uniaxial tension and uniaxial compression

CHAPTER 4

FEM MODELLING OF COMPRESSION TEST

Modelling in Abaqus is a step by step procedure. It basically involves working in various modules where every module corresponds to a specific portion of modelling. Therefore each module contains only those tools which are relevant with only that part of modelling. The various aspects of the model that was used in the present study are shown below:

4.1 PART MODULE

In this module individual parts are created by sketching or importing their geometry. The part that is created in the present study is a cube of 150mm × 150mm × 150mm. The cube has following features:

Modelling space – 3D

Type – Deformable

Shape- Solid

Type- Extrusion

Approximate size – 200

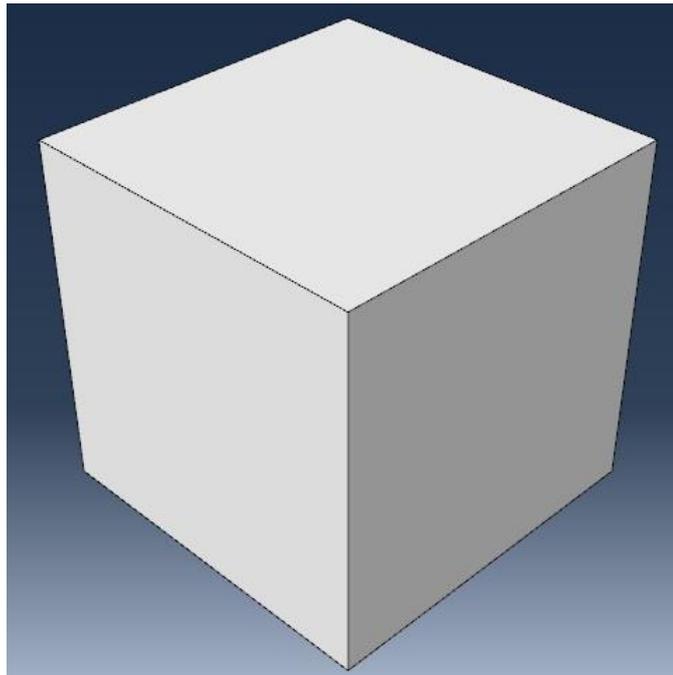


Figure 4.1: Cube created in Part module

4.2 PROPERTY MODULE

In property module, generally three tasks are performed – creating material, creating section and assigning section. Various material definitions are created and assigned to various parts. Figure 5 shows the property module in Abaqus.

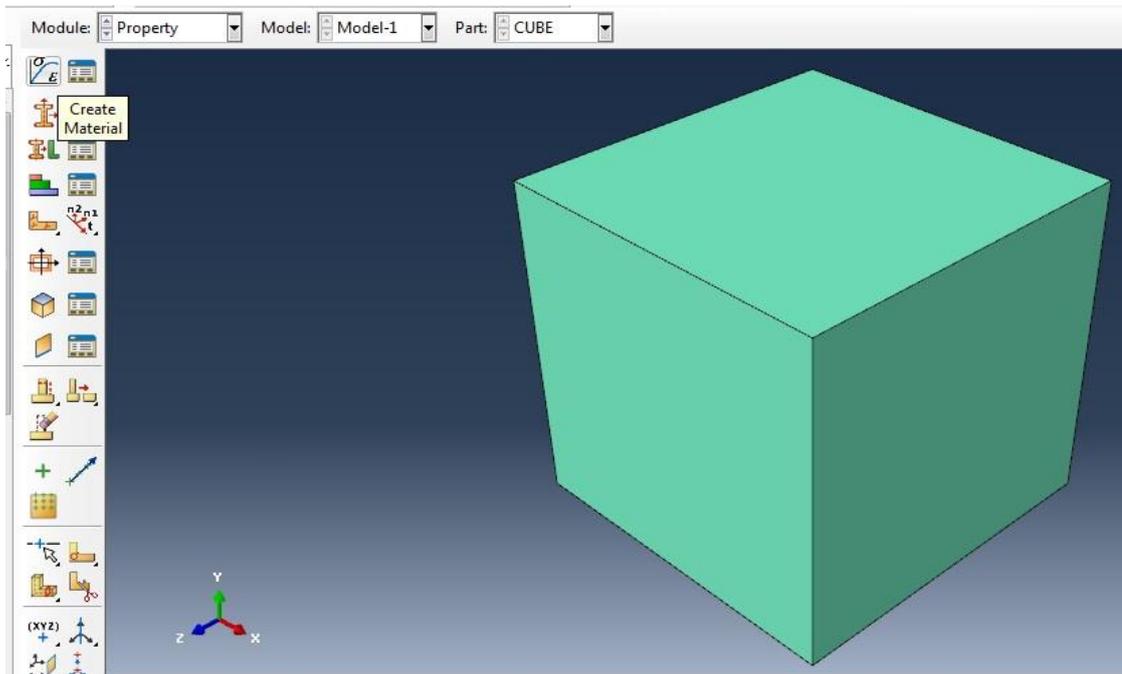


Figure 4.2: Property module in Abaqus

4.2.1 Defining Material

In a material definition all the property data relevant to a material are specified. While specifying a material definition, many types of material behaviors can be included in a single material definition like elastic, hyper elastic, hypo elastic, porous elastic, plastic, cap plasticity etc. In each material behavior property data are to be supplied. The material behaviors used for specifying concrete in the present case are:-

- Density
- Elastic
 - Young's Modulus
 - Poisson's Ratio
- Concrete Damaged Plasticity
 - Dilation Angle
 - Eccentricity
 - f_{b0}/f_{c0} – Ratio of strength in biaxial state to strength in uniaxial state
 - Viscosity Parameter

- Concrete compression damage
- Concrete tensile damage

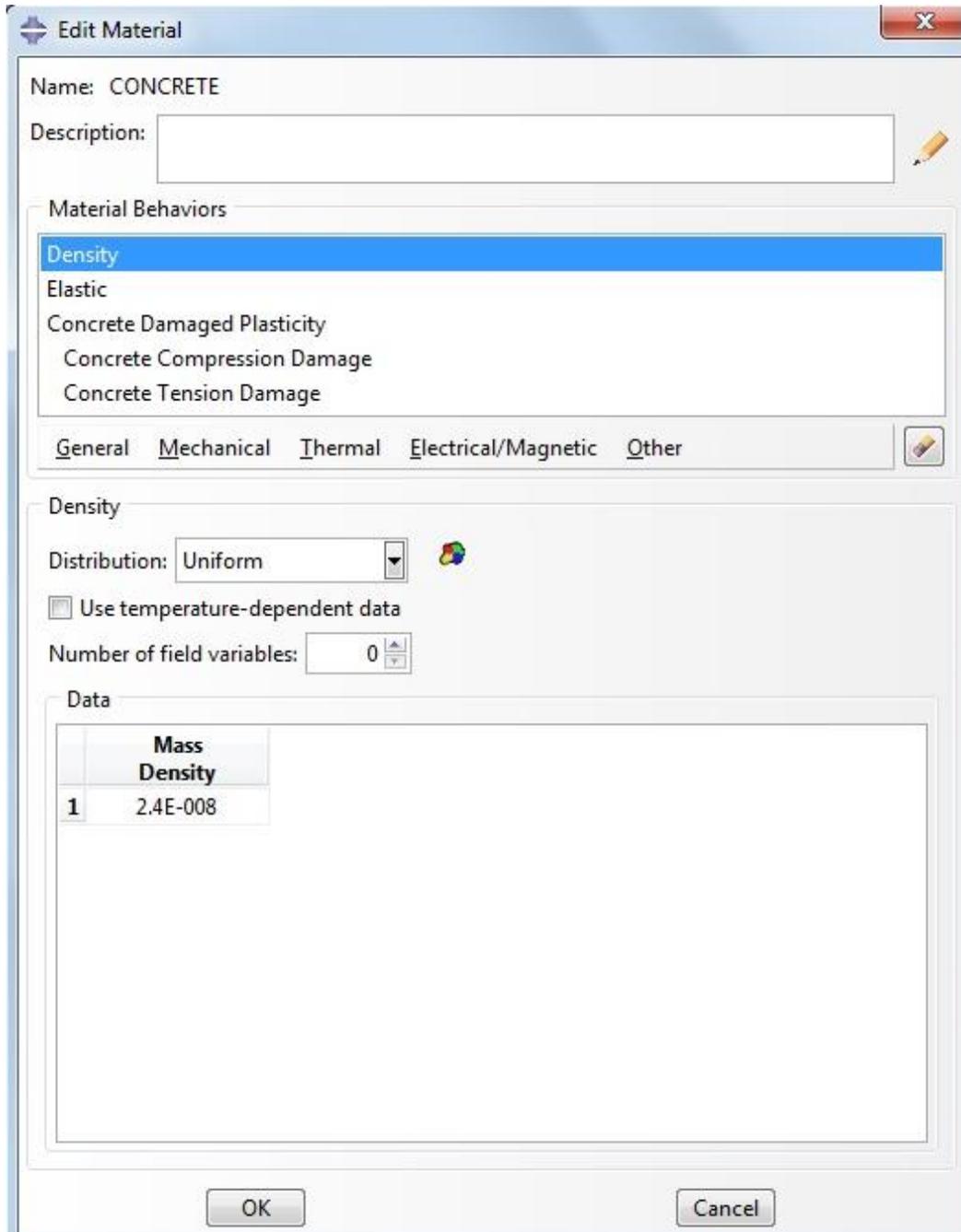


Figure 4.3: Material behaviors used for defining concrete

4.2.2 Defining Section

A section contains information about the properties of a part or a region of a part. The section used here is solid and homogenous.

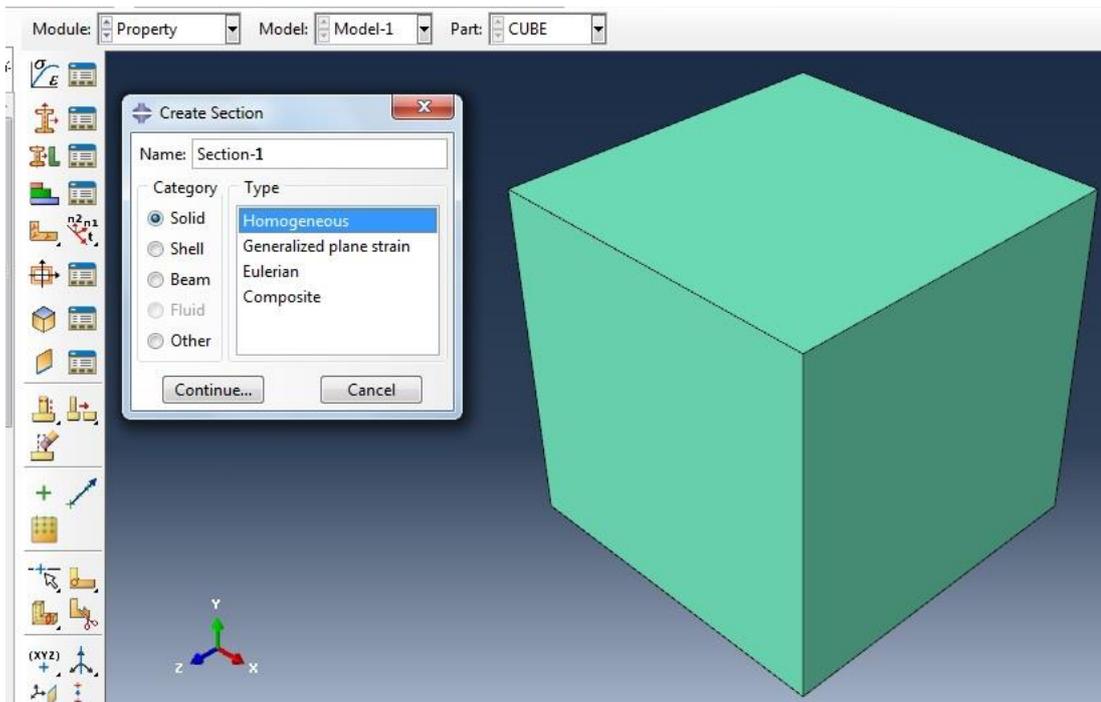


Figure 4.4: Creating solid homogenous section

4.3 ASSEMBLY MODULE

Assembly module is used to create and modify the assembly. When we create more than one parts, each part has its own coordinate system. To put each part into a global coordinate system, assembly module is used. In assembly module we work on part instances. A part instance is basically a representation of the original part. There can be both dependent or independent part instances. An independent instance can be thought of as a copy of the part. A dependent instance is only a pointer to a part. Hence a dependent instance can not be meshed. Therefore the original part is meshed when dependent part instance is used. Hence it is advised to use dependent instances of the part if the assembly contains identical part instances. Dependent instances also consume fewer memory resources and result in smaller input file. Here we have only one part and therefore the assembly will consist of only the instance of the part created earlier.

4.4 STEP MODULE

Initial step is already created at the beginning. This initial step can not be edited, replaced, copied or deleted. The initial step allows us to define conditions that are applicable at the very beginning of the analysis like some boundary conditions, predefined fields and interactions. For example, if a boundary condition is applied

throughout the analysis, it is beneficial to apply this boundary condition in the initial step.

Also there are two types of steps static nonlinear steps and linear perturbation steps. In static nonlinear steps , the state at which the model is after the end of one step forms the initial state for the next general step. Linear perturbation analysis step gives the linear response about the state of the last general nonlinear step. In the present study besides the initial step, one more analysis step is created.

The description of various steps used in the analysis is available in step manager in the step module



Figure 4.5: Step Manager showing two steps used

4.5 INTERACTION MODULE

Interaction module is use to define various types of interations like contact interaction, elastic foundation, cavity radiation, tie constraints, rigid body constraints , cracks, inertia etc. While defining interactions it is required to indicate the analysis steps in which they are active. Many types of interaction can be defines like General contact, Surface-to-surface contact, self contact, fluid cavity, fluid exchange, Model change etc. Also there are many interaction properties like contact , film condition , Cavity radiation, Fluid cavity, fluid exchange, acoustic impedance, incident wave etc.

Also there are many types of constraints in the interaction module which can be applied to restrain the degrees of freedom of the part instances. For example , a tie constraint can be used to fuse two regions or two surfaces, by using a rigid body constraint the motion of regions of the assembly can be constrained to a reference point. Similarly there are many other constraints like display body, coupling, adjust points, shell-to-solid coupling, embedded region, equation etc.

As there is only one part in the present study, there was no need to define interactions in our case.

4.5 MESH MODULE

In mesh module, meshes can be created on parts and assemblies which are created . Seeds can be placed uniformly over all the parts of the instance or along the edges . Or else they can also be placed such that the mesh density increases towards one end of the edge. After specifying the seed size, meshing is done. There are two ways of meshing in Abaqus – top down meshing and bottom up meshing.

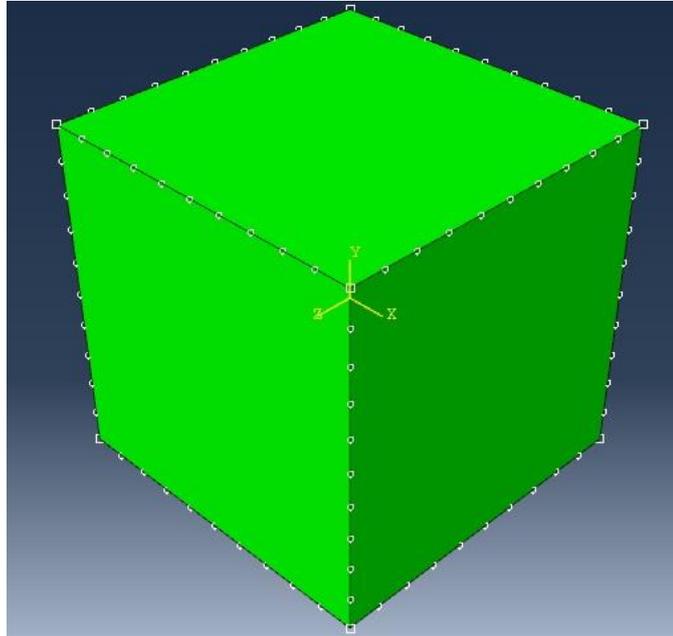


Figure 4.6: Seeding definition)

With each element type there is an element shape which can be used in the region. For example- by default a triangular and quadrilateral element is associated with shell mesh region.

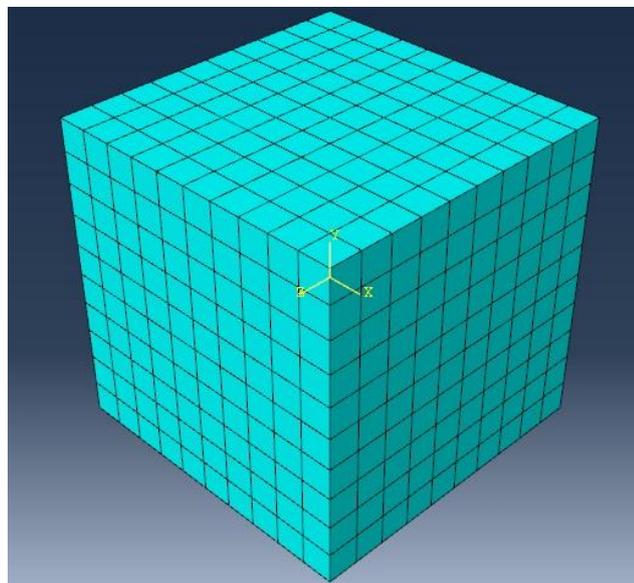


Figure 4.7: Meshing of part instance

4.6 LOAD MODULE

Load module is used to define and manage following prescribed conditions:

- Loads
- Boundary conditions
- Predefined fields
- Load cases

For all the above prescribed conditions, it is mandatory to specify the analysis steps in which they are activated. These prescribed conditions can be viewed and manipulated through the use of load manager, boundary conditions manager and predefined field manager.

In the present case there are two boundary conditions used – one at the top face and other at the bottom face. The boundary condition used at the bottom face is encastre. This boundary condition is equivalent to a fixed support. It neither allows translation in any direction x , y or z , nor it allows rotation about any axis ($U1=U2=U3=UR1=UR2=UR3=0$)

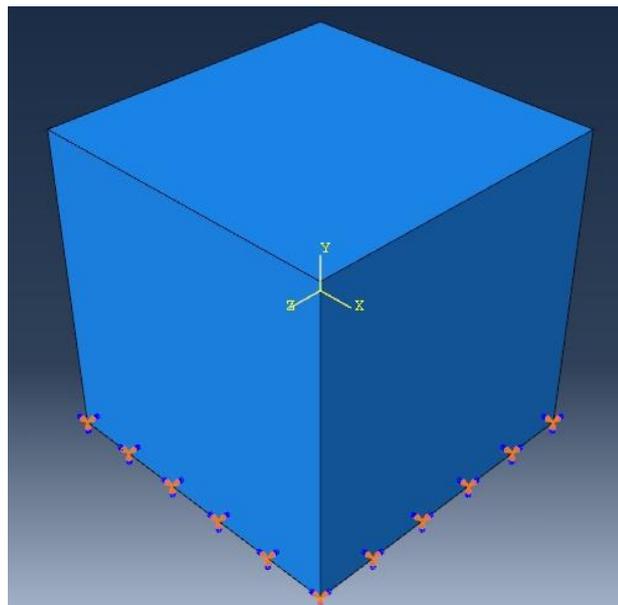


Figure 4.8: Boundary conditions at the bottom face

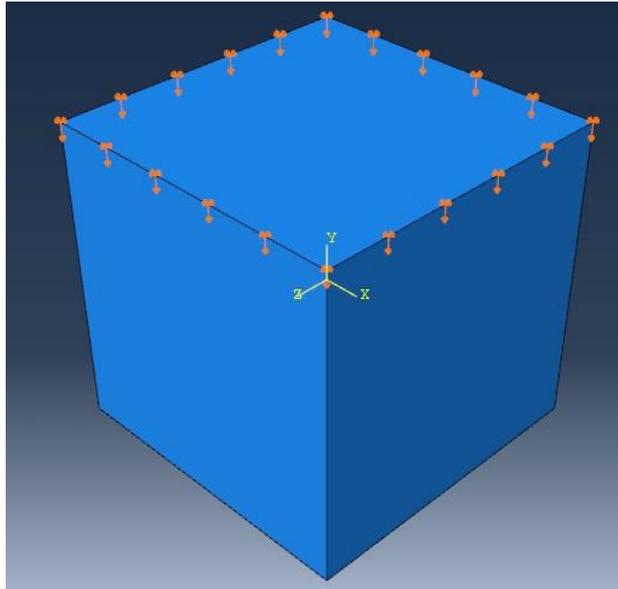


Figure 4.9: Boundary conditions at the bottom face

CHAPTER 5

FEM MODELLING OF FLEXURAL TENSILE TEST

Similar to the compression test, modelling the test for flexure will also involve working with various modules. While a cube was used in the compression test, the test specimen in the flexure test is a beam of standard size 10cm×10cm×50cm.. The arrangement for flexural strength test is shown in figure 5.1.

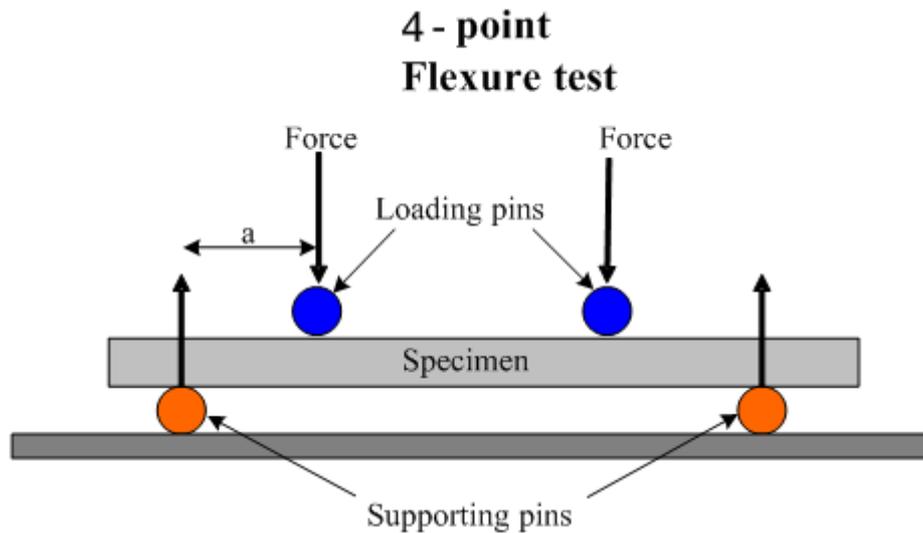


Figure 5.1: Flexural Tensile strength test arrangement

5.1 PART MODULE

For modelling the beam in the part module, a square of size 100mm × 100mm is sketched first and then it is extruded to a length of 400mm using the extrude option. On the top surface of the beam two thin strips are drawn using the partition option. This is done for the purpose of applying loading. These strips divide the length of the beam in three equal parts as is required for the test.

The part is 3D deformable and the base shape is solid and base type is extrusion. The approximate size is 200.

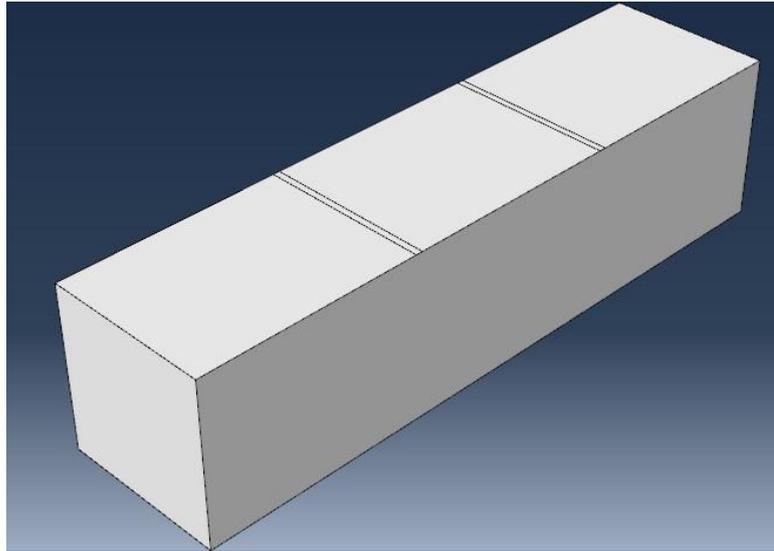


Figure 5.2: Beam created in Part module

5.2 PROPERTY MODULE

In the property module, similar to the compression test properties of concrete are added. Concrete Damaged Plasticity model is used to define the behavior of concrete. In the Concrete Damaged Plasticity model used for the flexural tensile test, our focus will be on finding out the tensile behavior and tensile damage parameters, which lead us to the results that are closest to the experimental results. As our focus is not on the compressive behavior of concrete hence the compression data used in this test will be the same as that in the compression test.

5.3 LOAD MODULE

In a four point bending test, the load is applied in such a way that the lines along which the load is applied divide the length of the specimen in three equal parts. In the load module a displacement along the two strips created on the top surface of the beam is applied. Also to simulate the effect of the two roller supports, two pinned boundary conditions are provided along the two bottom edges of the beam. Figure 15 shows the pinned boundary conditions used and the loading applied on the beam.

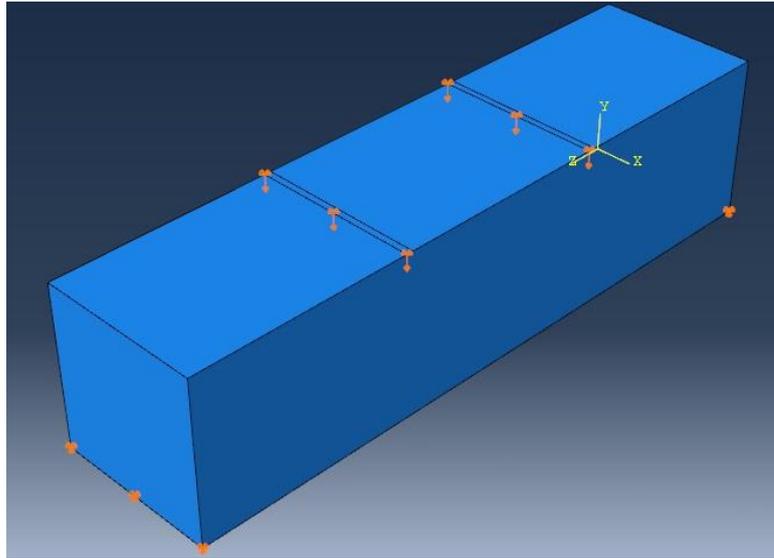


Figure5.3: Pinned boundary conditions and loading applied on the beam

5.4 MESH MODULE

Meshing is done using the global seed size of 20.

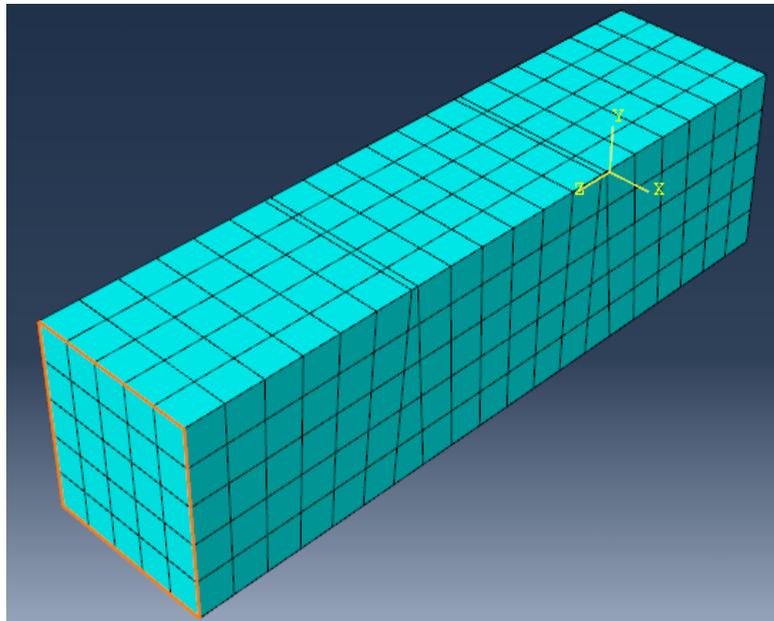


Figure5.4: Meshing of beam creates

CHAPTER 6

RESULTS AND DISCUSSION

After modelling the tests for compression strength and flexural tensile strength, analysis was run for various magnitude of loadings and various parameters given in the Concrete Damaged Plasticity model for compressive behavior, tensile behavior, compression damage and tensile damage and the maximum stress generated in the finite elements was recorded. Analysis was also run for varying mesh sizes and the attempt was made to get the maximum stress level as closer to the actual experimental value as possible

6.1 COMPRESSION TEST

In the Concrete Damaged Plasticity model, the compression behavior of the concrete is governed by the compressive behavior and compression damage parameters. Hence on various trials the maximum stress generated in the finite elements was recorded. The results obtained are compared with the 28 day compressive strength of the fiber reinforced concrete. The material parameters which gave satisfactory results are presented below:

Table 6.1: Compression Parameters for 0% Fiber Reinforcement

Compressive Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
15	0	0	36.594	36	1.65%
20	7.5E-05	0			
20	9.00E-05	0			
20	0.00015	0.05			
15	0.00076	0.48			
15	0.0025	0.6			
10	0.0056	0.7			
5	0.011	0.95			

Table 6.2: Compression Parameters for 0.25% Fiber Reinforcement

Compressive Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
15	0	0	39.7804	39.56	0.557%
20	7.5E-05	0			
20	9.00E-05	0			
22	0.00015	0.05			
15	0.00076	0.45			
15	0.0025	0.5			
10	0.0056	0.6			
5	0.011	0.9			

Table 6.3: Compression Parameters for 0.5% Fiber Reinforcement

Compressive Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
15	0	0	41.0985	41.11	0.27%
20	7.5E-05	0			
20	9.00E-05	0			
25	0.00015	0			
20	0.00076	0.28			
15	0.0025	0.5			
10	0.0056	0.6			
5	0.011	0.9			

Table 6.4: Compression Parameters for 0.75% Fiber Reinforcement

Compressive Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
15	0	0	41.3832	41.56	0.425%
20	7.5E-05	0			
20	9.00E-05	0			
25	0.00015	0			
20	0.00076	0.19			
15	0.0025	0.5			
10	0.0056	0.6			
5	0.011	0.9			

Table 6.5: Compression Parameters for 1% Fiber Reinforcement

Compressive Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
15	0	0	42.4161	42.22	0.464%
20	7.5E-05	0			
20	9.00E-05	0			
25	0.00015	0.14			
20	0.00076	0.47			
15	0.0025	0.5			
10	0.0056	0.6			
5	0.011	0.9			

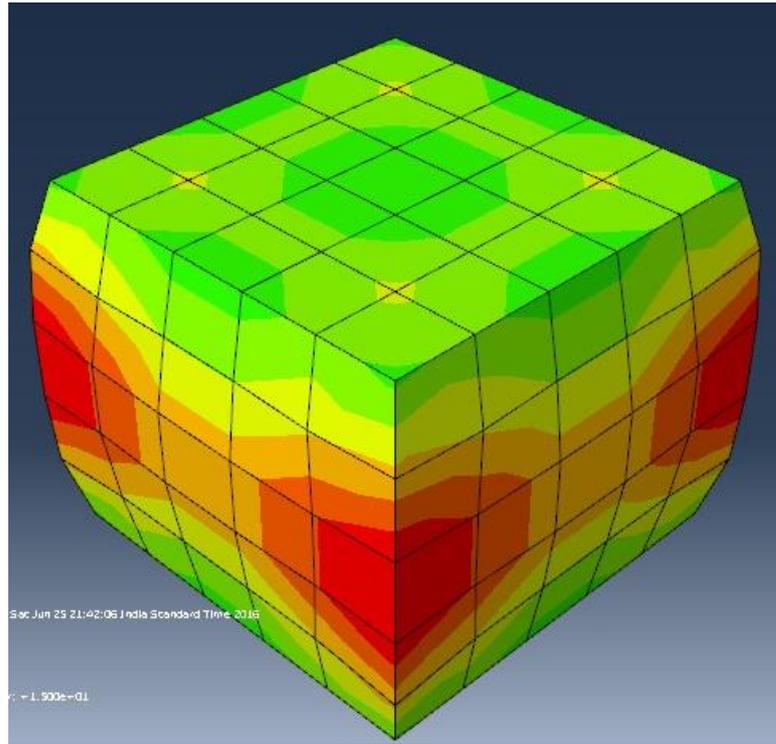
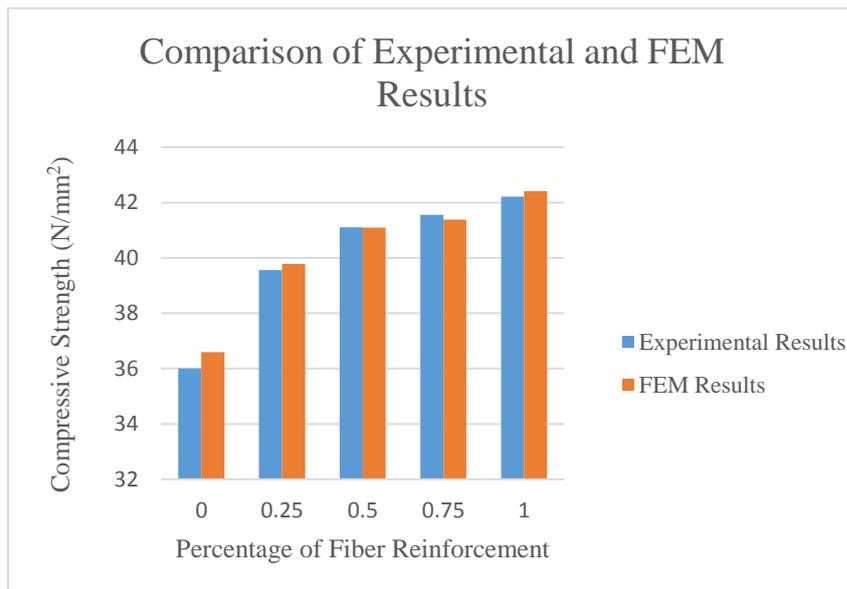


Figure 6.1: Deformed shape in compression test



6.2 TEST FOR FLEXURAL TENSILE STRENGTH

The failure in a test for flexural strength is a tensile one. Hence, the flexural strength of concrete is governed by its behavior in tension. In Concrete Damaged Plasticity model, the failure stress depends on how the tensile behavior and tension damage parameters are defined. Hence various trials were taken adopting different values of

damage parameters and tension behavior. The maximum tensile stress obtained was recorded and compared with the experimental results available to check the validity of different parameters. Also it is to be noted that as the compressive behavior does not play a major role in the test for flexural strength, hence the compressive behavior same as that of the above test was used.

Table 6.6: Tensile Parameters for 0% Fiber Reinforcement

Tensile Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
5	0	0	6.02	5.82	3.43%
5	1E-05	0.00055			
5	2E-05	0.0007			
6	3E-05	0.00085			
6	3.7E-05	0.00095			
6.3	4E-05	0.001			
6.25	5E-05	0.002			

Table 7: Tensile Parameters for 0.25% Fiber Reinforcement

Tensile Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
5	0	0	6.464	6.54	1.162%
5	1E-05	0.0006			
5	2E-05	0.0007			
6	3E-05	0.00082			
6.1	3.7E-05	0.0009			
6.3	4E-05	0.001			
6.7	5E-05	0.002			

Table 8: Tensile Parameters for 0.5% Fiber Reinforcement

Tensile Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
5	0	0	6.7204	6.66	0.915%
5	1E-05	0.0006			
5	2E-05	0.0007			
6	3E-05	0.00079			
6.3	3.7E-05	0.0009			
6.8	4E-05	0.001			
7	5E-05	0.002			

Table 9: Tensile Parameters for 0.75% Fiber Reinforcement

Tensile Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
5	0	0	6.9039	6.9	0.056%
5.2	1E-05	0.0006			
5.61	2E-05	0.0007			
6.18	3E-05	0.00074			
6.25	3.7E-05	0.0008			
6.8	4E-05	0.001			
7.2	5E-05	0.002			

Table 10: Tensile Parameters for 1% Fiber Reinforcement

Tensile Behavior	Inelastic Strain	Damage Parameters	Strength obtained on Analysis	Actual Strength	% error
5	0	0	6.9445	6.96	0.227%
5.2	1E-05	0.0006			
5.69	2E-05	0.0007			
6.3	3E-05	0.00074			
6.44	3.7E-05	0.0008			
6.57	4E-05	0.0009			
7.24	5E-05	0.0015			

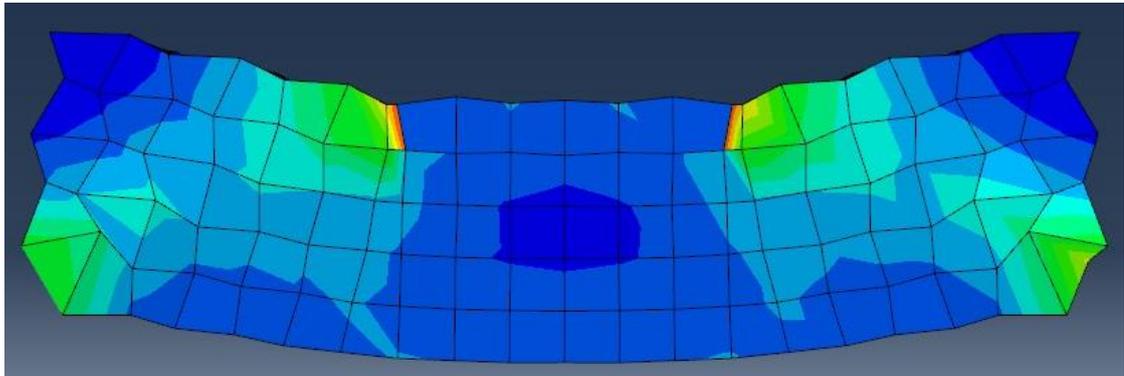
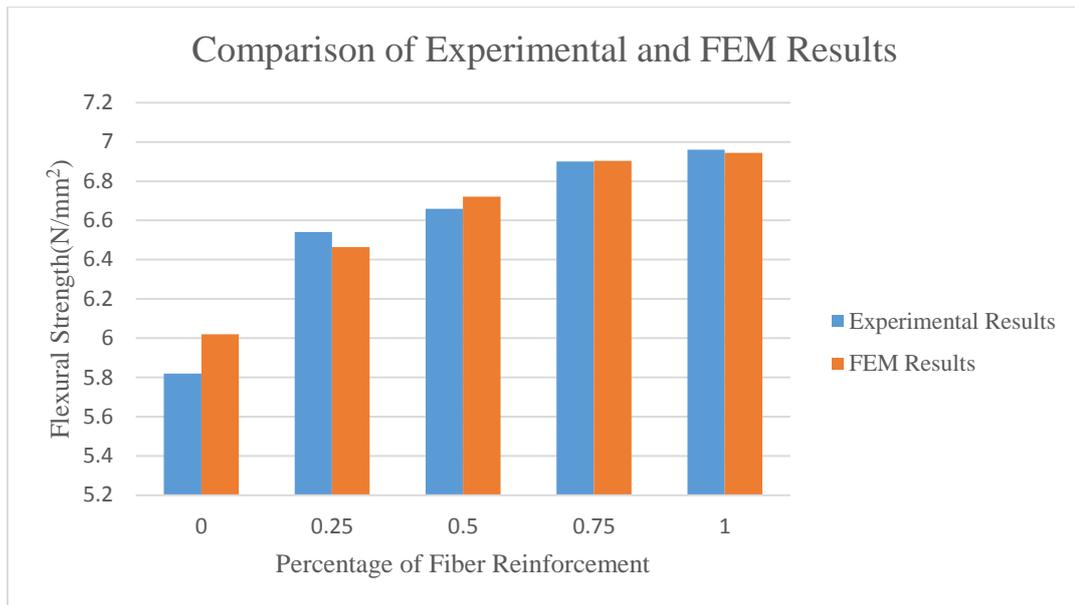


Figure 6.2: Deformed shape in the Test for Flexural Tensile Strength



6.3 CONCLUSION

- Concrete Damaged Plasticity Model can be used to model the behavior of Fiber Reinforced Concrete.
- With increase in fiber reinforcement , lesser damage parameters should be used to get correct results.
- With increase in fiber reinforcement, the values of yield stresses should be increased.
- Maximum value of yield stress entered in compression test is 25N/mm^2 which is entered at an inelastic strain of 0.00015.
- Maximum value of damage parameter entered for compression test was 0.9 at an inelastic strain of 0.011
- For tensile test , the maximum value of yield stress was 7.24 N/mm^2 at an inelastic strain of 0.00005.
- For tensile test, the maximum value of damage parameter was 0.002 at an inelastic strain of 0.00005.

REFERENCES

- "Abaqus Analysis User's Manual." *Abaqus Analysis User's Manual*. n.d.
https://things.maths.cam.ac.uk/computing/software/abaqus_docs/docs/v6.12/books/usb/default.htm.
- Barbero, E.J., F.A Cosso, R Roman, and T.L. Weadon. "Determination of Material Parameters for Abaqus Progressive Damage Analysis of E-Glass Epoxy Laminates." 2013.
- JANKOWAIK, Tomasz, and Tomasz LODYGOWSKI. "Identification of Parameters of concrete Damage Plasticity Constitutive Model." *Foundations of Civil and Environmental Engineering*, 2005.
- L, Jendele, J Cervenka, V Saouma, and R Pukl. *On the choice between discrete or smeared approach in practical structural FE analyses of concrete structures*. Prague , Czech Republic: CERVENKA CONSULTING , n.d.
- M.A.Chakrabarti, S.V.Chaudhari &. "Modelling of concrete for nonlinear analysis using Finite Element Code ABAQUS." *International Journal of Computer Applications* , 2012.
- Michal, Szczecina, and Winnicki Andrzej. "Calibration of the CDP model parameters in Abaqus." *World Congress on Advances in Structural Engineering and Mechanics*. Incheon, 2015.