Non-Linear Analysis of Reinforced Concrete Column using Ansys

Dr J Rex[1], Wilfred Rohit Peters[2]
[1] Assosiate Professor, Department of Civil Engineering, Malla Reddy Engineering College (Autonomous), Hyderabad, Telangana, India
[2] Student, Department of Civil Engineering, Malla Reddy Engineering College (Autonomous), Hyderabad, Telangana, India

Abstract

Numerous techniques have been used to evaluate behaviour then occurrence of typical failures of reinforced concrete structures, such as flexural, shear, torsion, buckling, etc. The behaviour of reinforced concrete is usually investigated through large-scale experimental experiments. The complicated behaviour of reinforced concrete members may now be modelled using finite element modelling thanks to the development of powerful numerical analytical tools like the finite element method (FEM). Used in this work are models of reinforced concrete columns that have been loaded axially, symmetrically, and eccentrically. With the help of the FEM programme ANSYS, reinforced concrete columns are analysed using nonlinear finite element analysis up to failure. Simulations about reinforced concrete supports that are imperiled to axially symmetric filling take into account how frequently they are used in laboratories.

Index Terms Flexural, Torsion, Shear, Buckling, Nonlinear Finite Element Analysis, ANSYS

DOI Number: 10.48047/NQ.2022.20.16.NQ880585 **NeuroQuantology 2022; 20(16): 5756-5764**

INTRODUCTION

It is common practise to conduct experimental analysis to investigate the strength of concrete under different loading conditions and for individual component members. This approach reveals the structure's actual behaviour. However, it is costly and time intensive. These structural elements are also examined using finite element theory. In order to evaluate structures, a technique known as finite element analysis (FEA) is performed. This technique accurately predicts how a component will react to different structural loads. Due to its speed and cost-effectiveness, FEA has traditionally been chosen as the method of choice for studying concrete behaviour. The complex behaviour of reinforced concrete columns can now be modelled using finite element modelling thanks to growth about cultured arithmetical investigation gears similar FEM.

Objectives:

This initiative aims to accomplish three primary things:

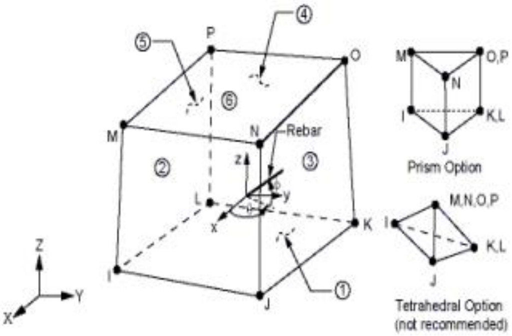
- To lessen the quantity of input data necessary for large-scale structural analysis.
- To minimise the procedures required for structure analysis.
- To create a reliable MATLAB tool that can analyse a big arrangement utilizing the aforementioned techniques.

MODELLING AND MATERIAL PROPERTIES

2.1 FINITE ELEMENT MODELLING

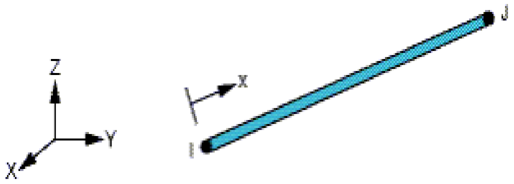
2.1.1 Element Types

Concrete: Concrete be modelled using Solid65 element. Eight knobs make up the nodes of this element, and each node has three degrees of freedom, allowing for translations in the nodal x, y, and z directions. This substance has the ability to deform plastically, fracture in 3 orthogonal instructions, plus crush. A diagram about component is exposed in Fig.



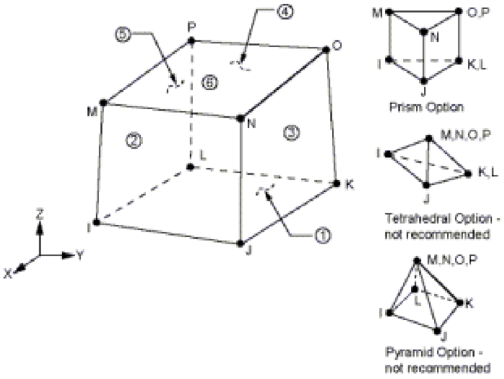
Solid 65 element

Steel Reinforcement: Steel reinforcement is modelled using a Link180 element. This element has two nodes with three degrees of freedom each, which allow for translations in the nodal x, y, and z dimensions. It is a 3D spar element. This substance can also distort plastically. This component is depicted in Figure 2.



Link 180 element

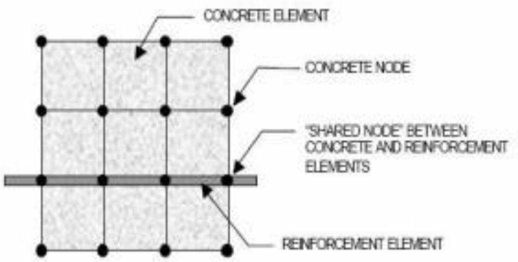
Steel Plates and Supports: A modelling element called Solid185 is used to represent steel plates and supports. Eight nodes that have 3 degrees about freedom for conversions in the nodal x, y, and z instructions make up component's definition. Allure element has the ability to be plastic, hyper elastic, stress stiffen, creep, have a huge refraction, have a unlimited strain. Solid185 comes now 2 different shapes:



Solid185 Element (Homogeneous Structural Solid) in ANSYS

2.2 Finite Element Modelling of Steel Reinforcement

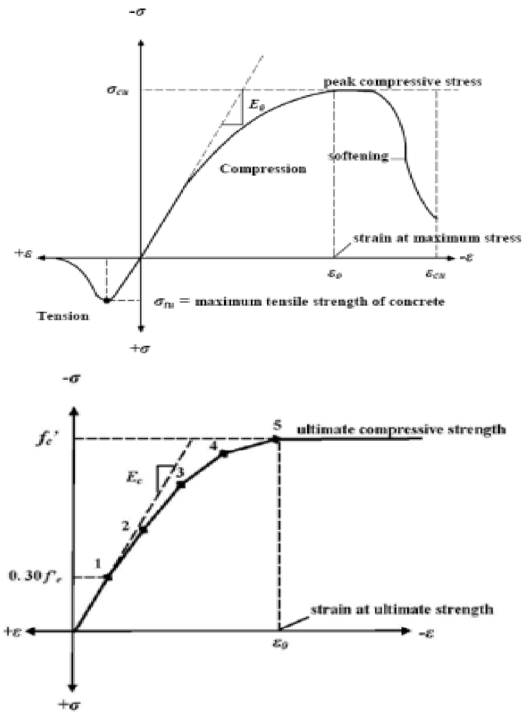
FEM aimed at protected concrete can simulate steel reinforcement using one of three methods: discrete, embedded, or smeared modelling. The reinforcement is modelled in the work provided the research using a separate modelling method. The bar or beam elements used for reinforcement in the discrete model (Fig) are attached to concrete mesh nodes using bar or beam elements. Because of this, concrete with support net segment the same nodes, and the concrete takes up similar space as support does.



Discrete Models for Reinforcement

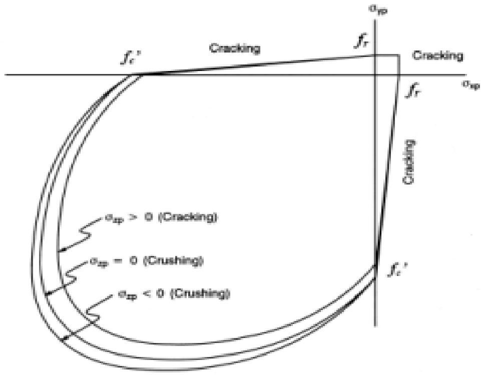
2.3 Material Properties

Concrete: Concrete has a stress-strain relationship that is both extremely nonlinear and ductile and is a quasi-brittle material. the slow development of microcracks under loading, which is responsible for the nonlinear behaviour. Concrete's tensile strength is normally between 8 and 15 percent of its compressive strength. A typical stress-strain curvature amid normal weight concrete is shown trendy Figure. Concrete's stress-strain bend in compression is linearly elastic up to around 30% of its extreme compressive strength. Outside here opinion, pressure gradually rises until it reaches the highest compressive strength.



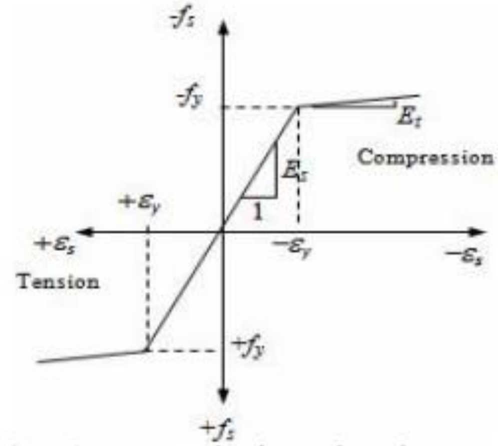
Typical uniaxial compressive and tensile stress-strain curve for concrete

The perfect be able to forecast material failure in concrete (Fig). Failure modes related to both crushing and cracking are taken into account. To specify a disappointment external aimed at the concrete, the 2 input power parameters—ultimate uniaxial tensile and compressive strengths—are required. As a result, a threshold pro material disappointment brought on by a multiaxial stress national can stand resolute.



Failure Surface for concrete

Steel Reinforcement: It is believed that the mechanical behaviour of steel reinforcement bars will be elastic bilinear under monotonic tension. A linear elastic phase is first present in the steel bar, charted via a yield phase, strain hardening, and finally a stress decline till break happens. According to Fig., steel bar behaviour under compression and tension loads is same. The yield strength, tangent modulus, and modulus of elasticity are the three most important inputs for steel solid model. Here solidity of the FE perfect of the beam, work-hardening component is introduced here to the steel characteristics.

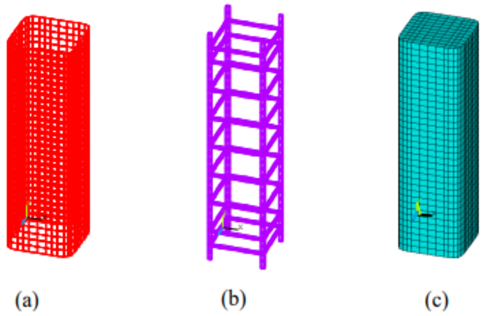


Strain curve for the steel reinforcement

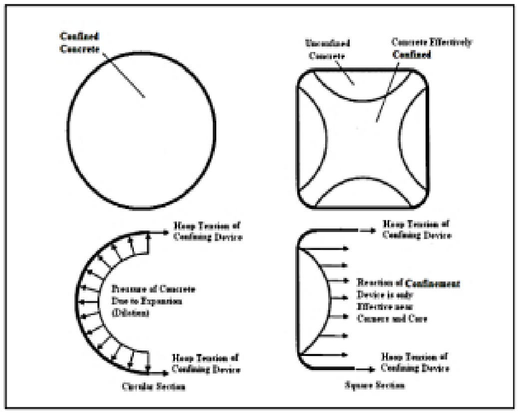
METHODOLOGY AND MATERIAL INVESTIGATION

3.1 Methodology of the Study

Based on the experimental data that is currently available, essential behaviour of protected concrete columns reinforced by carbon fibre reinforced polymers be modelled in the current study. Four series were created from the analysis of 37 column specimens using FEM. Here confirmation revision be conducted to ensure that the theoretic consequences obtained through experimental testing are valid. Following this, a parametric study was conducted to look into impact about maximum significant limitation scheduled the behaviour of RC supports reinforced by CFRP mixtures.



(a) Finite element model column (b) steel elements (c) Mesh of CFRP.



3.2 Material Investigation

Composites are made of two materials combined together, with one of the materials, the reinforcing phase, being embedded in the matrix phase and taking the form of fibers, sheets, or particles. Metal, ceramic, or polymer are all acceptable choices for the matrix and reinforcing materials. Because the overall qualities of composites are better than those of the separate components, they are utilised. For instance, composites made of a polymer and a ceramic material have a higher modulus than the polymer alone yet are less brittle than ceramics. High strength-to-weight ratio (low density, high tensile strength), high creep resistance, high tensile strength at high temperatures, and high toughness are only a few of the factors that influence the choice of composites for certain applications.

3.4 Mechanism of Concrete Column by Confinement

When FRP jackets or any other confining device (steel plates, transverse reinforcing steel, etc.) is fitted to the concrete column, the concrete is not contained since no initial stresses are introduced in the confining device at low levels of stresses in the concrete. According to tests, a concrete column's circular section provides substantially more effective confinement than its square or rectangular counterpart. Figure 1 explains the explanation of this difference in efficacy by demonstrating how a circular section will put quarantine expedient here girdle under pressure with force to continuously apply restricting gravity everywhere boundary, subsequent here total imprisonment. Contrarily forces confinement mechanism to apply the confining response solitary close to the section's angles with in its centre while leaving the sides unconstrained, resulting in the provision of only incomplete detention aimed at pillar.

The influence of confinement on circular and square sections. (Chaallal et.al.,)

3.8 About Ansys

Since the tested parts have thin walls and are perforated along their whole length, finite element analysis is mostly employed to validate them because of how complex their behaviour is under axial stresses. The shell elements found in the ANSYS [1] software offer a useful way to validate the experimental findings. An engineering simulation programme is called ANSYS (computer aided engineering, or CAE). In 1996, ANSYS became a publicly traded company on the NASDAQ. ANSYS received the highest score possible on Investor's Business Daily's Smart Select Composite Ratings in late 2011, making it one of only six technology companies in the world.

- **About ANSYS**

Numerous other publications have also acknowledged ANSYS as a top performance. In order to continuously improve the software, the company dedicates 15% of its annual revenues to research. When it comes to engineering simulation, ANSYS offers a wide variety of solution sets that give users access to almost every area that is needed for the design process. Software from ANSYS is used by businesses across numerous productions. Before becoming an actual entity, implements subject a virtual product to extensive analysis (Here being driven hooked on a brick partition before successively for some years on an asphalt way).

3.9 ANSYS Products

Simulation Technology: Fluid dynamics, explicit dynamics, electromagnetism, structural mechanics, and multiphysics

Workflow Technology: High-Performance Computing, Geometry Interfaces, Simulation Process, and Data Management are some of the technologies used by ANSYS Workbench. The incorporation of computer-based engineering simulation early in the development process, which enables engineers to revise and test ideas at a stage where the cost of making modifications is modest, is now widely acknowledged as a fundamental strategy for success in virtually every industry. The most difficult design problems are solved by ANSYS by using quick, exact, with trustworthy model. With the help of our technology, businesses can confidently forecast how well their products will perform in the real world. They rely on our software to support maintaining the quality of the products and fostering corporate success through innovation. Every product makes a commitment to meet and exceed expectations. By using ANSYS software to simulate frequently and early, our customers can develop their own products more quickly, more efficiently, and with greater innovation.

ANSYS

3.10 ANALYSIS OF RC COLOUMN USING ANSYS

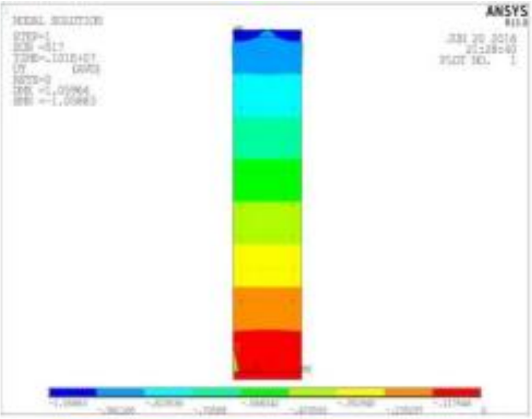
3.10.1 Modeling and Meshing

To model Column, volumes are made. Rectangular mesh is employed to achieve satisfactory results from the Solid65 element. The modelling process uses the nodes produced by the concrete volume's mesh to build the individual reinforcing components. Here "merge items" command combines several things bes located in similar place. They will subsequently be combined into a single entity. In order to limit ideal with obtain a singular resolution, movement limit conditions are required. Applying boundary constraints supports and loadings will guarantee that the model behaves in the same way as the experimental Column. The support has been designed with a hinge and roller in mind. All nodal lines receive the same amount of force.

DESIGN AND DETAILS OF THE RC FRAME

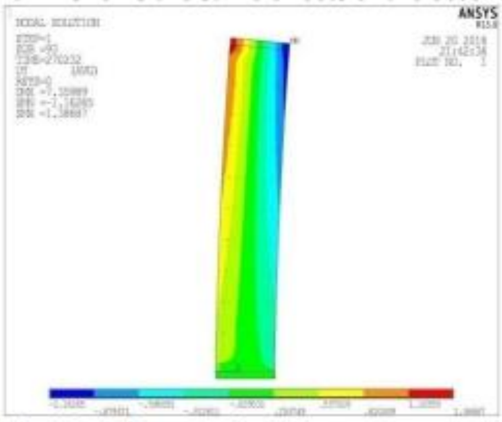
Vertical deflection in columns

As seen in Fig., the free end of the axial column experiences greater vertical deflection when a load is placed there.

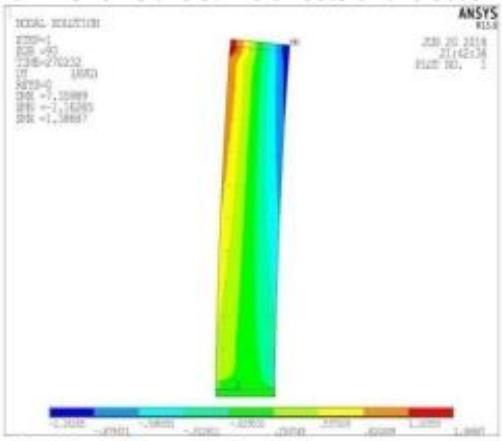


Axial column

As eccentricity rises, the maximum value shifts in favour of the eccentricity and also rises due to the bending effect.



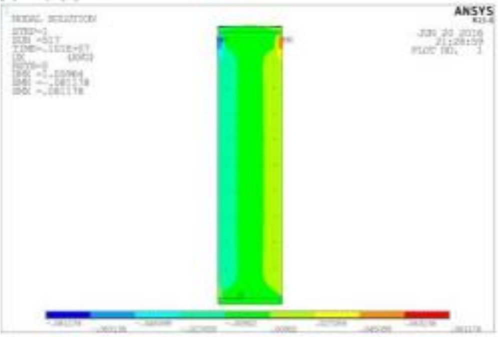
Column with 50mm eccentricity



Column with 100mm eccentricity

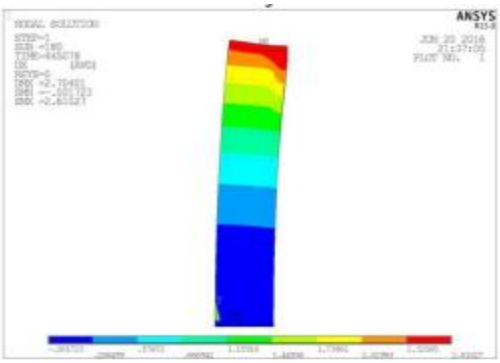
Horizontal Deflection in Columns

Axial force on a column results in relatively little horizontal displacement, which is caused by concrete deformation from shear failure at the top and great bottom interaction surfaces.



Axial column

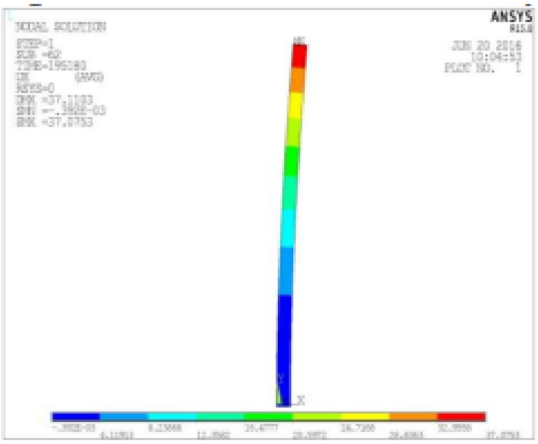
The horizontal deflection rises with increasing eccentricity, as depicted in Fig. On the eccentricity's opposing side, tension rises.



Column with 50mm eccentricity

Additionally, Figure 1 demonstrates that tall columns have higher horizontal refraction than short columns for a given eccentricity. This is because the long column began to buckle due to the slenderness effect, which caused a substantial horizontal deflection before disappointment.

Column with 100mm eccentricity



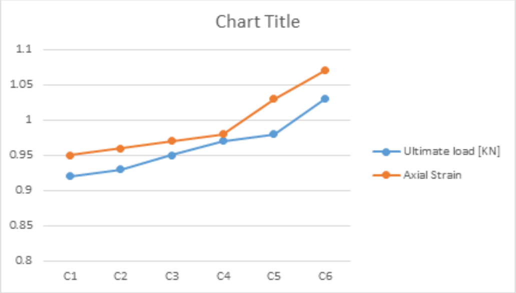
Long column with 50mm eccentricity

RESULTS

5.1 Ultimate load vs Axial Strain

Experimental and numerical results of ultimate load and axial strain

Column	Ultimate load [KN]	Axial Strain
C1	0.93	0.97
C2	0.96	0.95
C3	0.92	1.03
C4	0.98	0.98
C5	0.95	0.98
C6	0.97	1.07

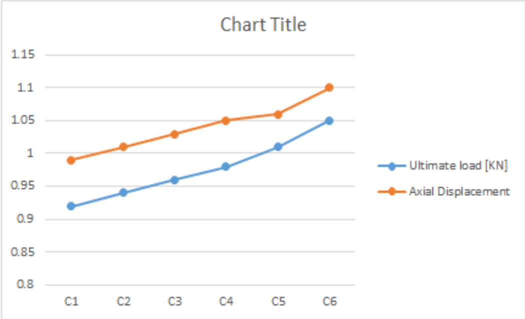


Experimental and numerical results of ultimate load and axial strain

5.2 Ultimate load vs Axial Displacement

Experimental and numerical results of ultimate load and axial Displacement

Column	Ultimate load [KN]	Axial Displacement
C1	0.92	0.99
C2	0.94	1.01
C3	0.96	1.03
C4	0.98	1.05
C5	1.01	1.06
C6	1.05	1.10

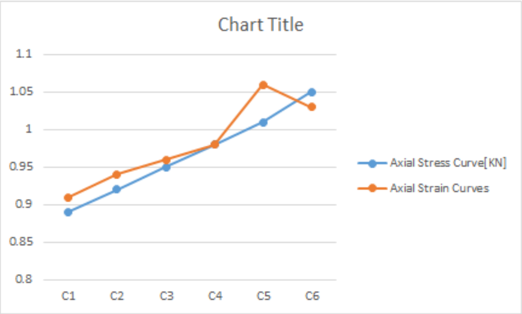


Experimental and numerical results of ultimate load and axial Displacement

5.3 Axial Stress Curves vs Axial Strain Curves

Experimental and numerical results of Axial stress-axial strain curves

Column	Axial Stress Curve[KN]	Axial Strain Curves
C1	0.89	0.91
C2	0.92	0.94
C3	0.95	0.96
C4	0.98	0.98
C5	1.01	1.06
C6	1.05	1.03



Experimental and numerical results of Axial stress-axial strain curves

CONCLUSIONS

The object about revision existed to explore whether nonlinear finite element investigation of reinforced concrete columns could be carried out using the ANSYS Package software. Thanks to accurate FEA modelling, Here disappointment processes about axial with unusual reinforced concrete supports be clearly identified and closely match predictions. here nonlinear mutual axial with flexible reaction of these schemes up to disappointment have been precisely represented by finite element models of reinforced concrete supports built in ANSYS 15 utilizing the specific concrete components. Future applications about finite element analysis pro non-linear investigation about RC supports will benefit from a fuller understanding of the analysis process employed in this paper, which was developed by previous researchers, and various output plots created by FEA. It has been originate the effects be extra subtle to elements like mesh size, material quality, load increases, etc. founded on ANSYS investigates performed on RC supports.

REFERENCE

[1] Antonio F. Barbosa and Gabriel O. Ribeiro, “Analysis of Reinforced Concrete Structures Using Ansys Nonlinear Concrete Model”, Computational Mechanics, New Trends and Applications, Barcelona, Spain (1998).

[2] L. Dahmani, A. Khennane, and S.Kaci, “Crack Identification in Rein-forced Concrete Beams using ANSYS Software,” Strength of Materials, Vol. 42, No. 2, pp. 232–240, 2010.

[3] A.J. Wolanski, “Flexural Behavior of reinforced and Prestressed Concrete Beams Using Finite Element Analysis”, Master of Science The-sis, Marquette University, Wisconsin, 2004.

[4] D. Kachlakev, T. Miller, S. Yim, “Finite Element Modeling of Rein-forced Concrete Structures Strengthened with FRP Laminates”, Final Report SPR 316, Oregon Department of Transportation, 2001.

[5] M.Y.H. Bangash, “Concrete and Concrete Structures: Numerical Modeling and Applications”, Elsevier Science Publishers Ltd., London, England. 1989.

[6] J.G. MacGregor, “Reinforced Concrete Mechanics and Design”, Prentice-Hall, Inc., Englewood Cliffs, NJ., 1992.

[7] K.J. Willam, and E. P. Warnke, “Constitutive Model for Triaxial Behavior of Concrete,” Seminar on Concrete Structures Subjected to Triaxial Stresses, International Association of Bridge and Structural Engineering Conference, Bergamo, Italy, 174. (1974)