Analysis Of Reinforced Concrete Structures Using Ansys

Analysis of Multistory Building with shear wall by using ansys
April 8th, 2019 - models in the analysis of reinforced concrete structures by Antonio F Barbosa etal 2000 The results of some analyses performed using the reinforced concrete model of the general purpose finite element code ANSYS are presented and discussed The differences observed in the response of the same reinforced concrete beam as some variations

Finite Element Modeling and Analysis of Reinforced
April 11th, 2019 - practicing engineers to use the polymer composites in the field of rehabilitation of structures This paper in deals with the finite element analysis of beam retrofitted with different fiber reinforced polymer FRP composite sheets carried out using ANSYS 15 software RC beams with different FRP composite

DOWNLOAD ANALYSIS OF REINFORCED CONCRETE STRUCTURES USING

Finite Element analysis of reinforced concrete beams with
April 3rd, 2019 - Transverse openings are often provided through reinforced concrete beams to accommodate utility ducts and pipes Finite element analysis has been used in order to study this problem Fifty seven beams analyzed using finite element program ANSYS V12 The analysis results compared with fifteen experimental beams had been done by Ibrahim

Finite Element Analysis of Composite Material Using ANSYS
April 18th, 2019 - accomplished using ANSYS The analysis done will be useful for determining deformation of concrete column reinforced with FRP bar The dynamic analysis in terms of fundamental linear analysis is illustrated by using ANSYS these overall material constants as of analysis Key words Fibre reinforced polymer FRP linear analysis ANSYS I
Reinforced Concrete Frame Structure Based on ANSYS
April 17th, 2019 - This article mainly focus on the research in two way horizontal earthquake under the action of 5 layer reinforced concrete frame structure optimization analysis. This research utilizes ANSYS software to realize parametric modeling. By using beam and column section sizes as design variables and structural strength stiffness, the maximal displacement between the layers as well as reinforcement.

ANSYS Tutorial Reinforced Concrete Beam RC BEAM Static Structural
April 7th, 2019 - ANSYS Workbench Tutorial using Static Structural to model a RC Beam. Reinforced Concrete Beam. Failed elements or cracked and crushed elements are shown using the PLCRACK command. Deformation of

Nonlinear Analysis of Reinforced Concrete Column with
April 7th, 2019 - Nonlinear finite element analysis results were compared to experimental data. Characteristic used to analyze reinforced concrete columns up to failure with points on the load deformation curve predicted using FEA FEM software. ANSYS Reinforced concrete column subjected to were then compared to theoretical results.

Analysis of Flanged Shear Wall Using Ansys Concrete Model
April 12th, 2019 - nonlinear models in the analysis of reinforced concrete structures. The results of some analyses performed using the reinforced concrete model of the general purpose finite element code ANSYS are presented and discussed. The differences observed in the response of the

NONLINEAR PROBABILISTIC ANALYSIS OF THE REINFORCED
April 18th, 2019 - 22nd SVSFEM ANSYS Users Group Meeting and Conference 2014 SVSFEM s r o NONLINEAR PROBABILISTIC ANALYSIS OF THE REINFORCED CONCRETE STRUCTURES USING ANSYS CRACK SOFTWARE JURAJ KRÁLIK Abstract The paper presents an application of the nonlinear analysis of reinforced concrete structures under extreme static loads.
Analysis of Flanged Shear Wall Using Ansys Concrete Model
April 16th, 2019 - Beams and post tensioned concrete beams developed in ANSYS using the concrete element Solid 65 have accurately captured the nonlinear flexural response of these systems up to failure. Anthony J. Wolanski B.S. 2004 did research on the flexural behavior of reinforced and prestressed concrete beams using finite element analysis.

How to Model Concrete using Finite Elements CAE Associates
April 18th, 2019 - Yet more than 50 years later there is still no consensus on the best method to model concrete using finite elements. The current best method depends on the purpose of the calculation. Concrete is one of the most complex materials to model.

Finite Element Analysis of Reinforced Concrete Beam Using
April 13th, 2019 - Finite element analysis of reinforced concrete beam retrofitted with different fibre composites. They use Pro E software for modelling and ANSYS for analysis. The modelled geometry is used carbon fibre reinforced polymer CFRP sheet for first layer of RC beam glass fibre reinforced polymer GFRP sheet.

Chapter 4 Finite Element Analysis of Reinforced Concrete Beams
April 7th, 2019 - Finite Element Analysis of Reinforced Concrete Beams 4.1 General Reinforced concrete structures are largely employed in engineering practice in a variety of situations and applications. In most cases these using ANSYS. Using these models the load deformation behaviour of

Nonlinear Analysis of Shear Dominant Prestressed Concrete
April 18th, 2019 - FE Analysis of the T beams using the ‘ANSYS’. The partially prestressed concrete T beams have been analyzed using the ANSYS. The ‘ANSYS’ model accounts for the nonlinearity such as bond slip of longitudinal reinforcement post cracking tensile stiffness of the concrete stress transfer across the cracked blocks of the concrete load.
Analysis Of Reinforced Concrete Structures Using Ansys
April 18th, 2019 - PDF This paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures. The results of some analyses performed using the reinforced concrete analysis of Reinforced Concrete Columns with ANSYS software.

Nonlinear analysis of Reinforced Concrete Columns with ANSYS software
April 17th, 2019 - finite element analysis on 10 column specimens was achieved by using ANSYS software. The nonlinear finite element analysis program ANSYS is utilised owing to its capabilities to predict either the response of reinforced concrete columns in the post elastic range or the ultimate strength of a reinforced concrete columns reinforced by FRP bars.

Analysis of Flexural Behaviour of Singly and Doubly Reinforced Concrete Beams using ANSYS software
April 11th, 2019 - reinforced concrete beams using ANSYS software. The goal of this study was to know the different phases of the FE model behaviour from initial cracking yielding of steel until failure of the concrete beam and to know the applicability of ANSYS software for analyzing and predicting of crack patterns in the reinforced concrete beam.

Nonlinear Analysis of Reinforced Concrete Column with ANSYS
April 16th, 2019 - reinforced concrete columns subjected to axial symmetric and eccentric loading are used. Nonlinear finite element analysis is used to analyze reinforced concrete columns up to failure with FEM software ANSYS. Reinforced concrete column subjected to the axial symmetric loading are modeled considering the frequent use in the laboratory.

Review of non linear analysis of reinforced concrete
April 16th, 2019 - RC columns are fundamental load bearing individuals in a structure which contribute lateral stiffness moreover. Finite Element
Analysis of Reinforced Concrete Column under Lateral Load International Journal of Engineering Research and Applications 3 4 228 231 2010 Crack identification in reinforced concrete beams using ANSYS

Analysis of Reinforced Concrete Column using FRP composites
April 9th, 2019 - upgrade the stiffness and ductility of the reinforced concrete column using FRP composite material The finite element method ANSYS is used to model and analyze the reinforced concrete column first without FRP composite material and subsequently with different FRP composite materials The

Crack Identification in Reinforced Concrete Beams Using
April 16th, 2019 - UDC 539 4 Crack Identification in Reinforced Concrete Beams Using ANSYS Software L Dahmani a A Khennane b and S Kacia a Mouloud Mammeri University Tizi Ouzou Algeria b University of Southern Queensland Toowoomba Australia OAE 539 4 Aliažheq õcõõõõõõ õõõõõõ åõåõõåõõãõõåõõ åõõõõõ åõõõõõ

Analysis Of Polymer Fibre Reinforced Concrete Pavements By
April 7th, 2019 - Analysis Polymer Fibre Reinforced Concrete Pavements Using ANSYS 1 INTRODUCTION Concrete is weak in tension and has a brittle character The concept of using fibres to improve the characteristics of construction materials is very old Early applications include addition of straw to mud bricks horse hair to reinforce

Seismic Structural Analysis of NPP Reinforced Concrete
April 11th, 2019 - reinforced concrete structures is a classical dynamic problem of solid mechanics 11 12 Problem statement for the propagation of seismic waves in the soil does not reach beyond the widely known mathematical models and methods of solid mechanics Applied methods for computational analysis of structure seismic
ANALYSIS OF REINFORCED CONCRETE STRUCTURES USING ANSYS
April 11th, 2019 - Key words Reinforced Concrete Nonlinear Analysis Finite Element Analysis Abstract This paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures The results of some analyses performed using the reinforced concrete model of the general purpose finite element code Ansys are presented and

Nonlinear Analysis of Reinforced Concrete Beam by ANSYS
March 11th, 2019 - The model simulating the test process was established the calculation results of ANSYS are compared with the experimental results The comparison shows that ANSYS analysis results are similar to experimental results which indicates ANSYS analysis software can be used to simulate the mechanical property of reinforced concrete structures

Finite Element Analysis Of Composite Element For FRP
April 8th, 2019 - possibilities of different reinforced concrete models in practical use It reports the results of some analyses performed using the reinforced concrete model of the general purpose finite element code Ansys A series of analysis of the same structure has been performed exploring different aspects of material modeling In recent years numbers

Analysis Of Reinforced Concrete Structures Using Ansys
March 6th, 2019 - CiteSeerX Document Details Isaac Councill Lee Giles Pradeep Teregowda This paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures The results of some analyses performed using the reinforced concrete model of the general purpose finite element code Ansys are presented and discussed

NONLINEAR FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE
April 18th, 2019 - 3 Structural Walls Considered for Analysis Six reinforced concrete structural walls were proposed to investigate the influence of vertical slits in the wall structure Comparative analysis were conducted on slitted walls and solid walls There are considered three types of structural walls with one level 3 30 m with two levels 6 60 m and
Global Safety Assessment of Concrete Structures using Nonlinear Finite Element Analysis Master of Science Thesis in the Master’s Programme Structural Engineering and Building Technology MATTIAS BLOMFORS Department of Civil and Environmental Engineering Division of Structural Engineering Concrete Structures

FIRE ANALYSIS OF A SIMPLY SUPPORTED REINFORCED CONCRETE
April 18th, 2019 - FIRE ANALYSIS OF A SIMPLY SUPPORTED REINFORCED CONCRETE BEAM USING ANSYS WORKBENCH thermal and structural analysis ANSYS Workbench is a finite element software capable of conducting thermal

Study the Behavior of Reinforced Concrete Beam Using
April 12th, 2019 - Study the Behavior of Reinforced Concrete Beam Using Finite Element Analysis Abdulsamee Halahla Fahad Bin Sultan University P O Box 15700 Tabuk Saudi Arabia halahla4@fso edu sa Abstract – Several methods have been utilized to study the response of concrete structural components Experimental based testing

A Comparative Analysis of Slab with Different Shape of
April 6th, 2019 - reinforced concrete slab with cut out and ways of strengthening by using overlay concrete and Carbon Fiber Reinforced Polymer CFRP sheet The nonlinear finite element analysis use to model different CFRP strengthening arrangement of one way slab by using ANSYS Sheetal Gawas and S V Itti 2014 Studied Two way RC slab using

Analytical Analysis on Reinforced Concrete Beam
April 10th, 2019 - Analytical Analysis on Reinforced Concrete Beam Strengthened by FRP Lamine using ANSYS Shweta S Shetty1 Kiran M Malipatil2 1Post graduate Student 2Assistant Professor 1 2Department of Civil Engineering 1 2KLE M S Sheshgiri College of Engineering and Technology Belgaum 590008 India Abstract- In this paper analytical analysis has been carried

Course documents on Finite Element Analysis of Concrete
April 11th, 2019 - Course documents Finite element analysis for reinforced concrete 0 Textbook Finite element analysis of concrete structures HTM
Analysis of reinforced concrete structures with Ansys 14.5 PART 1 & 2
March 31st, 2019 - Analysis of reinforced concrete structures simple beam Analysis of reinforced concrete structures with Ansys 14.5 PART 1 & 2
Axis Symmetrical Closed cylinder Analysis in ANSYS APDL 16.0

ANSYS IMPLEMENTATION OF A REINFORCED CONCRETE MATERIAL MODEL
April 8th, 2019 - ANSYS is also used in modeling of reinforced concrete members and structures However at the present time ANSYS provides only one dedicated material model for reinforced concrete - the constitutive model for triaxial behavior of concrete after

Analysis of Reinforced Concrete Structures Using ANSYS
March 15th, 2019 - Analysis of Reinforced Concrete Structures Using ANSYS Nonlinear Concrete Model Download as PDF File pdf Text File txt or read online Analysis of Concrete structures

Structural Analysis Software ANSYS Structural
April 15th, 2019 - Structures Structural Analysis ANSYS structural analysis software enables you to solve complex structural engineering problems and make better faster design decisions With the finite element analysis FEA tools available in the suite you can customize and automate solutions for your structural mechanics problems and parameterize them to

NONLINEAR FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE
April 15th, 2019 - NONLINEAR FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE STRUCTURES SUBJECTED TO IMPACT LOADS Design of reinforced concrete structures against extreme loads such as impact and blast loads is increasingly gaining importance However due to the problem’s complicated nature there exists no commonly accepted methodology or a design code
3D ANSYS Numerical Modeling of Reinforced Concrete Beam
April 15th, 2019 - This paper discusses about 3D ANSYS FE modeling of the failure behavior of structural reinforced concrete beam element. The capacity of the bending moment deformation, stress strain, and fracture patterns is determined that occurs on a single reinforced concrete beams with different types of collapsed mechanisms. The RC beam specimens of normal strength is modeled by rectangular section with

Crack identification in reinforced concrete beams using
April 11th, 2019 - The general purpose finite element package ANSYS 8.0 is employed for the numerical analyses. Using SOLID65 solid elements, the compressive crushing of concrete is facilitated using plasticity algorithm while the concrete cracking in tension zone is accommodated by the nonlinear material model.

Effects of Concrete Nonlinear Modeling on the Analysis of
April 10th, 2019 - This study considers the practical application of nonlinear models in the analysis of reinforced concrete. The results of some analyses performed using the reinforced concrete model of the general purpose finite element code ANSYS are presented and discussed.

How to Model Concrete Reinforcement Using Finite Elements
April 14th, 2019 - Alternately reinforcement can be modeled in a discrete manner using link spar or beam elements found in all finite element software. These reinforcing spars can either be merged to the solid concrete elements shared nodes or may be tied to the concrete elements using either point to point or surface to surface contact with the added advantage of providing the ability to model bond slip.
structures using finite element analysis of reinforced concrete beams with ansys, finite element analysis of composite material using ansys, reinforced concrete frame structure based on ansys, nonlinear probabilistic analysis of the reinforced concrete model, finite element analysis of reinforced concrete column with ansys tutorial reinforced concrete beam rc beam static structural, analysis of flanged shear wall using ansys concrete model, how to model concrete using finite elements in ansys, nonlinear analysis of reinforced concrete columns with ansys, finite element analysis of reinforced concrete beams based on ansys, analysis of composite material using ansys, nonlinear analysis of shear dominant prestressed concrete, analysis of reinforced concrete structures using ansys, nonlinear analysis of reinforced concrete column with ansys, analysis of flexural behaviour of singly and doubly, nonlinear analysis of reinforced concrete column with ansys, review of nonlinear analysis of reinforced concrete, analysis of reinforced concrete structures using ansys, ansys tutorial reinforced concrete beam, static structural analysis software ansys structural, nonlinear finite element analysis of reinforced concrete, analysis of polymer fibre reinforced concrete pavements using ansys, nonlinear finite element analysis of reinforced concrete, analysis of reinforced concrete structures using ansys, ansys implementation of a reinforced concrete material model, analysis of reinforced concrete structures using ansys, 3d ansys numerical modeling of reinforced concrete beam, crack identification in reinforced concrete beams using ansys, effects of concrete nonlinear modeling on the analysis of reinforced concrete structures by antonio f barbarosetal 2000 the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ansys are presented and discussed the differences observed in the response of the same reinforced concrete beam as some variations in the field of rehabilitation of structures this paper in deals with the finite element analysis of beam retrofitted with different fiber reinforced polymer frp composite sheets carried out using ansys 15 software rb beams with different frp composite analysis of reinforced concrete structures using ansys www bikemonash com analysis of reinforced concrete pdf version aug 10 2017 reinforced concrete shear wall analysis and design a structural reinforced concrete shear wall in a 5 story building provides lateral and gravity load resistance for the reinforced concrete shear wall analysis and design, transverse openings are often provided through reinforced concrete beams to accommodate utility ducts and pipes finite element analysis has been used in order to study this problem fifty seven beams analyzed using finite element program ansys v12 the analysis results compared with fifteen experimental beams had been done by ibrahim 2010 accomplished using ansys the analysis done will be useful for determining deformation of concrete column reinforced with frp bar the dynamic analysis in terms of fundamental linear analysis is illustrated by using ansys these overall material constants as of analysis key words fibre reinforced polymer frp linear analysis ansys i, this article mainly focus on the research in two way horizontal earthquake under the action of 5 layer seismic structural analysis of npp reinforced concrete frame structures, analysis of reinforced concrete structures using ansys, ansys tutorial reinforced concrete beam, static structural analysis software ansys structural, nonlinear finite element analysis of reinforced concrete, analysis of polymer fibre reinforced concrete pavements using ansys, nonlinear finite element analysis of reinforced concrete, analysis of reinforced concrete structures using ansys, ansys implementation of a reinforced concrete material model, analysis of reinforced concrete structures using ansys, 3d ansys numerical modeling of reinforced concrete beam, crack identification in reinforced concrete beams using ansys, effects of concrete nonlinear modeling on the analysis of reinforced concrete structures by antonio f barbarosetal 2000 the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ansys are presented and discussed the differences observed in the response of the same reinforced concrete beam as some variations in the field of rehabilitation of structures this paper in deals with the finite element analysis of beam retrofitted with different fiber reinforced polymer frp composite sheets carried out using ansys 15 software rb beams with different frp composite
reinforced concrete frame structure optimization analysis this research utilizes ansys software to realize parametric modeling by using beam and column section sizes as design variables and structural strength stiffness the maximal displacement between the layers as well as reinforcement, ansys workbench tutorial using static structural to model a rc beam reinforced concrete beam failed elements or cracked and crushed elements are shown using the plcrack command deformation of nonlinear finite element analysis is results were compared to experimental data characteristic used to analyze reinforced concrete columns up to failure points on the load deformation curve predicted using fea fem software ansys reinforced concrete column subjected to were then compared to theoretical results, nonlinear models in the analysis of reinforced concrete structures the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ansys are presented and discussed the differences ob served in the response of the, beams and post-tensioned concrete beams developed in ansys using the concrete element solid 65 have accurately captured the nonlinear flexural response of these systems up to failure anthony j wolanski b s 2004 did research on the flexural behavior of reinforced and prestressed concrete beams using finite element analysis, yet more than 50 years later there is still no consensus on the best method to model concrete using finite elements the current best method depends on the purpose of the calculation concrete is one of the most complex materials to model, finite element analysis of reinforced concrete beam retrofitted with different fibre composites they have use pro e software for modelling and ansys for analysis the used carbon fibre reinforced polymer cfrp sheet for first layer of rc beam glass fibre reinforcedpolymer gfrp sheet, finite element analysis of reinforced concrete beams 1 general reinforced concrete structures are largely employed in engineering practice in a variety of situations and applications in most cases these using ansys using these models the load deformation behaviour of, analyzed using the ansys the ansys model accounts for the nonlinearity such as bond slip of longitudinal reinforcement post cracking tensile stiffness of the concrete stress transfer across the cracked blocks of the concrete load, pdf this paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures the results of some analyses performed using the reinforced concrete, finite element analysis on 10 column specimens was achieved by using ansys software the nonlinear finite element analysis program ansys is utilised owing to its capabilities to predict either the response of reinforced concrete columns in the post elastic range or the ultimate strength of a reinforced concrete columns reinforced by frp bars, reinforced concrete beams using ansys software the goal of this study was to know the different phases of the fe model behaviour from initial
cracking yielding of steel until failure of the concrete beam and to know the applicability of ansys software for analyzing and predicting of crack patterns in the reinforced concrete beam, reinforced concrete columns subjected to axial symmetric and eccentric loading are used nonlinear finite element analysis is used to analyze reinforced concrete columns up to failure with fem software ansys reinforced concrete column subjected to the axial symmetric loading are modeled considering the frequent use in the laboratory, rc-columns are fundamental load bearing individuals in a structure which contribute lateral stiffness moreover finite element analysis of reinforced concrete column under lateral load international journal of engineering research and applications 3 4 228 231 2010 crack identification in reinforced concrete beams using ansys, upgrade the stiffness and ductility of the reinforced concrete column using frp composite material the finite element method ansys is used to model and analyze the reinforced concrete column first without frp composite material and subsequently with different frp composite materials the, udc 539 4 crack identification in reinforced concrete beams using ansys software l dahmani a a khennane b and s kacia a mouloud mameri university tizi ouzou algeria b university of southern queensland toowoomba australia 539 4, analysis polymer fibre reinforced concrete pavements using ansys 1 introduction concrete is weak in tension and has a brittle character the concept of using fibres to improve the characteristics of construction materials is very old early applications include addition of straw to mud bricks horse hair to reinforce, reinforced concrete structures is a classical dynamic problem of solid mechanics 11 12 problem statement for the propagation of seismic waves in the soil does not reach beyond the widely known mathematical models and methods of solid mechanics applied methods for computational analysis of structure seismic, key words reinforced concrete nonlinear analysis finite element analysis abstract this paper considers the practical application of nonlinear models in the analysis of reinforced concrete structures the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ansys are presented and the model simulating the test process was established the calculation results of ansys are compared with the experimental results the comparison shows that ansys analysis results are similar to experimental results which indicates ansys analysis software can be used to simulate the mechanical property of reinforced concrete structures, possibilities of different reinforced concrete models in practical use it reports the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ansys a series of analysis of the same structure has been performed exploring different aspects of material modeling in recent years numbers of the general purpose finite element code ansys are presented and discussed, 3 structural walls considered for analysis six reinforced concrete structural walls were
proposed to investigate the influence of vertical slits in the wall structure comparative analysis were conducted on slitted walls and solid walls there are considered three types of structural walls with one level 3 30 m with two levels 6 60 m and, global safety assessment of concrete structures using nonlinear finite element analysis master of science thesis in the masters programme structural engineering and building technology mattias blomfors department of civil and environmental engineering division of structural engineering concrete structures,

fire analysis of a simply supported reinforced concrete beam using ansys workbench thermal and structural analysis ansys workbench is a finite element software capable of conducting thermal, study the behavior of reinforced concrete beam using finite element analysis abdulsamee halahla fahad bin sultan university p o box 15700 tabuk saudi arabia ahalahla fbsu edu sa abstract several methods have been utilized to study the response of concrete structural components experimental based testing,

concrete slab with cut out and ways of strengthening by using overlay concrete and carbon fiber reinforced polymer cfrp sheet the nonlinear finite element analysis use to model different cfrp strengthening arrangement of one way slab by using ansys sheetal gawas and s v itti 2014 studied two way rc slab using analytical analysis on reinforced concrete beam strengthened by frp laminate using ansys shweta s shetyi kiran m

malipatil2 1post graduate student 2assistant professor 1 2department of civil engineering 1 2kle m s sheshgiri college of engineering and technology belgaum 590008 india abstract in this paper analytical analysis has
1. Introduction

PDF Download 2. Stress and Strain Analysis

Analysis of concrete with ANSYS Solid 65 PDF Download 15

Analysis of reinforced concrete structures: Simple beam analysis of reinforced concrete structures with ANSYS 14.5 Part 1.2 Axisymmetric closed cylinder analysis in ANSYS APDL 16.0. ANSYS is also used in modeling of reinforced concrete members and structures, however, at the present time ANSYS provides only one dedicated material model for reinforced concrete: the constitutive model for triaxial behavior of concrete after analysis of reinforced concrete structures using ANSYS nonlinear concrete model download as pdf file pdf text file txt or read online analysis of concrete structures. Structures structural analysis ANSYS structural analysis software enables you to solve complex structural engineering problems and make faster, better design decisions. With the finite element analysis (FEA) tools available in the suite, you can customize and automate solutions for your structural mechanics problems and parameterize them. Nonlinear finite element analysis of reinforced concrete structures subjected to impact loads: Design of reinforced concrete structures against extreme loads such as impact and blast loads is increasingly gaining importance; however, due to the problems' complicated nature, there exists no commonly accepted methodology or a design code. This paper discusses about 3D ANSYS FE modeling of the failure behavior of structural reinforced concrete beam elements. The capacity of the bending moment deformation stress-strain and fracture patterns is determined that occurs on a single reinforced concrete beams with different types of collapsed mechanisms. The RC beam specimens of normal strength is modeled by rectangular section with the general purpose finite element package ANSYS 8.0 is employed for the numerical analyses using Solid65 solid elements, the compressive crushing of concrete is facilitated using plasticity algorithm, while the concrete cracking in tension zone is accommodated by the nonlinear material model. This study considers the practical application of nonlinear models in the analysis of reinforced concrete the results of some analyses performed using the reinforced concrete model of the general purpose finite element code ANSYS are presented and discussed. Alternately reinforcement can be modeled in a discrete manner using link spar or beam elements found in all finite element software. These reinforcing spars can either be merged to the solid concrete elements shared nodes or may be tied to the concrete elements using either point to point or surface to surface contact with the added advantage of providing the ability to model bond slip.