International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-10, Issue-09, Dec 2024

Dynamics Explicit Study of Solid RCC Versus Hollow FRC Columns

Shilpa S, Assistant Professor, SUIET Mukka, India, shilpa.s8416@gmail.com

Fathima Nima, PG student, SUIET Mukka, India, fathimanima7@gmail.com

Abstract: Finite Element Analysis (FEA) is a numerical technique employed to solve complex engineering problems by discretizing a large system into smaller, simpler components called finite elements. ANSYS is a leading software platform for performing FEA, offering sophisticated tools for modelling, simulating, and analysing a wide range of engineering applications. This study investigates the dynamic behavior of hollow Fiber Reinforced Concrete (FRC) and solid Reinforced Cement Concrete (RCC) columns under high-velocity impact using ANSYS Explicit Dynamics. The analysis compares the columns' responses to varying velocities (1000 mm/sec to 8000 mm/sec) with fixed-fixed boundary conditions. The hollow FRC and solid RCC columns are modelled with different cross-sectional dimensions and material properties, including Young's Modulus and Poisson's ratio. The results highlight the deformation, stress distribution, and stability of both column types under dynamic loading. This research provides insights into the performance of these columns, aiding in the design of safer structures for dynamic load environments.

Keywords* —*ANSYS Software, Explicit Dynamics, Finite Element Analysis, FRC Columns, Hollow columns, Solid RCC columns, Structural Analysis, Total Deformation.

1. INTRODUCTION

1.1 FINITE ELEMENT ANALYSIS (FEA)

Finite Element Analysis (FEA) in structural engineering is a numerical method used to analyse and predict how structures respond to various physical forces and conditions [14]. By breaking down complex structures into smaller, manageable elements [12]. FEA allows engineers to evaluate the performance, safety, and durability of materials and designs. FEA has become essential in modern structural engineering, enabling the design of safer, more efficient structures and ensuring compliance with industry standards and regulations [25]. With the current development of science and technology, the method of structural simulations by finite element-based software is quite popular and highly effective [13].

FEA consists of the following steps:

- Establishing the geometry of the system
- Defining material properties
- Setting boundary conditions
- Solving the equations generated

Finite Element Analysis (FEA) is essential for structural engineers, allowing them to simulate and analyze how structures behave under various conditions [6]. Here's a brief overview of FEA software tools and their applications:

- **ANSYS**: Performs structural, thermal, and fluid analysis; outputs stress and deformation results.
- **ABAQUS**: Handles advanced non-linear and dynamic analysis; provides insights into material behavior.
- **COMSOL MULTIPHYSICS**: Allows coupled simulations across different physics; offers customizable results.
- **SOLID WORKS SIMULATION**: Features built-in structural and thermal analysis; provides quick design feedback.
- **MSC NASTRAN**: Supports static and dynamic analysis; delivers reliable performance data for various industries.
- **SAP2000**: Used for structural design of buildings and bridges; generates detailed design reports.
- **RFEM/DLUBAL**: Conducts structural analysis for various projects; visualizes forces and behavior.
- **ALTAIR HYPER MESH**: Focuses on mesh generation for FEA; ensures high-quality meshes for accurate analysis.
- **OPEN FOAM**: Combines CFD and FEA for fluid-structure interactions; offers custom simulation data.
- **CATIA**: Conducts structural and thermal analysis in aerospace/automotive; integrates design insights.
- **SIEMENS NX**: Provides comprehensive FEA tools; supports optimization and lifecycle management.

These software tools empower structural engineers to perform detailed simulations, providing critical insights

DOI : 10.35291/2454-9150.2024.0422

into safety, performance, and compliance, ultimately enhancing design efficiency and reliability [25].

1.2 ANSYS

ANSYS is versatile engineering simulation software used for structural, thermal, fluid dynamics, and electromagnetic analysis. It helps evaluate stress, deformation, and performance, making it essential for various industries. Ansys is used to study construction structures [13]. Ansys Workbench system is commonly used for simulation purposes [12].

1.3 UTILIZING ANSYS SOFTWARE IN STRUCTURAL ENGINEERING

- **Structural Assessment**: ANSYS helps engineers analyze how structures deform and endure stress under different load conditions, employing both linear and nonlinear static methods.
- **Dynamic Response Analysis**: The software evaluates how structures react to dynamic forces, such as vibrations and seismic events, using techniques like modal and harmonic response assessments. Concrete is one of the most widely used construction materials for many civil engineering applications subjected to impact load, accidental load, blast load and explosion load [7].
- **Fatigue Life Prediction**: ANSYS determines the fatigue life of components by examining their performance under cyclic loading, allowing for the identification of potential failure risks.
- **Thermal Impact Evaluation**: It analyzes the effects of temperature changes on structures, accounting for factors such as thermal expansion and heat transfer.
- **Design Optimization**: ANSYS provides tools for adjusting design parameters to improve structural efficiency, minimize material use, and reduce overall costs.
- **Impact and Crash Testing**: The software aids in assessing structural integrity under crash scenarios, particularly for composite materials.
- **Analysis of Composite Structures**: ANSYS evaluates how advanced composite materials behave, focusing on their strength and performance characteristics.
- **Stability and Buckling Analysis**: It examines stability under compressive forces to determine critical buckling loads and potential failure modes.
- **Results Visualization and Interpretation**: ANSYS offers tools for visualizing and interpreting analysis results, making it easier to understand structural behavior.

1.4 EXPLICIT DYNAMICS IN ANSYS

The Explicit Dynamics module in ANSYS is specifically designed for simulating high-speed, transient events. It is ideal for situations involving rapid changes and significant deformations, such as impacts, explosions, and crashes, where traditional analysis methods may fall short. ANSYS Explicit Dynamics is essential for accurately analyzing dynamic events, ultimately improving the performance and safety of engineered systems. Deformation and stress response has been obtained from the velocity impact simulations in ANSYS Explicit dynamics module [6].

KEY FEATURES:

- **Time Integration**: This module employs explicit time integration to solve motion equations at each time step, allowing for accurate capture of fast dynamic responses and intricate behaviors.
- **Advanced Interaction Modeling**: It provides tools to model interactions and contacts between various bodies, including critical elements like friction and contact mechanics for realistic impact assessments.
- **Accurate Predictions**: Engineers can effectively forecast how structures will respond under extreme conditions, enhancing safety and design reliability across different applications.

1.5 FIBER-REINFORCED CONCRETE

Fiber Reinforced Concrete (FRC) is a composite material that improves structural characteristics by incorporating fibers made from steel, glass, synthetic, or natural materials [19]. This enhancement leads to increased durability and resistance to cracking, improved flexural strength for better bending capacity, and greater impact resistance for dynamic loads. FRC also reduces shrinkage cracking, prolonging the lifespan of structures while minimizing maintenance. Its versatility allows for customization in applications like pavements, slabs, precast components, and harsh environments, making FRC a valuable choice for various construction projects [19].

1.6 STEEL FIBER-REINFORCED CONCRETE

Steel Fiber Reinforced Concrete (SFRC) is commonly utilized for both structural and non-structural applications, making it one of the most prevalent forms of fiber-reinforced concrete [25]. In SFRC, steel fibers are evenly distributed in small quantities, typically between 0.3% and 2.5% by volume. These fibers significantly improve the concrete’s structural properties, especially its tensile and

DOI : 10.35291/2454-9150.2024.0422

flexural strength. The fibers themselves vary in size, with lengths ranging from 12 mm to 60 mm and diameters between 0.25 mm and 1 mm.

1.7 ADVANTAGES OF STEEL FIBER-REINFORCED CONCRETE

- **Increased Strength**: Enhances tensile and flexural strength for better load support.
- **Improved Toughness**: Absorbs energy and resists impacts effectively.
- **Crack Control**: Reduces cracking by redistributing stress evenly.
- **Enhanced Durability**: Offers greater resistance to wear and environmental factors.
- **Lower Maintenance**: Results in reduced maintenance costs over time.
- **Versatility:** Suitable for various applications, including pavements and industrial flooring.
- **Dynamic Load Performance**: Performs well under dynamic and impact loads.
- **Ease of Use**: Integrates easily into standard concrete mixes.

1.8 APPLICATIONS OF STEEL FIBER-REINFORCED CONCRETE

- **Pavements**: Ideal for highways and airfields, offering durability and crack resistance.
- **Industrial Flooring**: Perfect for warehouses and factories due to its toughness.
- **Precast Elements**: Common in beams, slabs, and panels for added strength.
- **Shotcrete**: Used in tunnels and slope stabilization for improved dynamic load performance.
- **Bridges**: Enhances structural integrity in bridge decks and supports.
- **Parking Garages**: Provides durability and resistance to cracking.
- **Silos and Tanks**: Suitable for storage structures for liquids and bulk materials.
- **Retaining Walls**: Effective in withstanding lateral forces in retaining wall applications.

2. LITERATURE REVIEW

"Dynamic Analysis of Pine Flat Concrete Dam: Acoustic Fluid-Structural Interaction with ANSYS Workbench" (2020), T. Menouillard et.al- In this paper, a numerical analysis of the Pine Flat Dam, a 120-meter-high concrete gravity dam is provided. It deals with dynamic finite element analysis under various loading conditions and water levels. The study incudes fluid-structure interaction by explicitly portraying the reservoir as an acoustic domain within ANSYS. The main objective is to investigate how water levels affect the dynamic behavior of dam. Additionally, it emphasizes the differences in dynamic responses between 2D and 3D modelling results.

"Experimental research on torsional strength of synthetic/steel fiber-reinforced hollow concrete beam" (2020), Rafea F. Hassan et.al- The aim of this paper is to analyse the torsional performance of hollow concrete beams reinforced with different types of fibre. 1% of the fibre content with three varying lengths of SY. F, 19, 38 and 57mm along with 13mm of ST. F, was used. Four hollow beams were casted with ST. F and SY. F and two control beams were cast without using fibers, which acted as control beams. At first crack load and failure load and also at every interval of the load the twisting angle of the tested beams were calculated. The results show that use of SY. F and ST. F improved the overall performance in the reinforced concrete beams when compared to the behaviour of control beam under the application of torsional load. As the fibre length of SYF increased, the ultimate load capacity of the beams increased. For the beams cast with fibre length of 19mm, 37mm and 55mm, the ultimate load capacity of the beam was increased by 4.7, 9.4, and 21.9% respectively. And the ultimate load capacity was increased by 5.5% for the concrete beam reinforced with ST. F. Hence this paper recommends the use of SY. F with the normal concrete, because of its significant influence on the torsional performance.

"An Approach to Finite Element Modeling of Liquid Storage Tanks in ANSYS: A Review" (2021), M. Z. Kangda- This review addresses seismic safety in liquid storage tanks, focusing on the sloshing phenomenon in various tank shapes and the role of obstructions. It discusses modelling methods in ANSYS, optimal mesh sizing, and key parameters like sloshing frequency and hydrodynamic pressure, providing insights for designing earthquake-resistant storage structures.

"Relative Study on Concrete-Filled Square and Circular Steel Tubular Columns-Using ANSYS for Mathematical Analysis" (2021), S. Vinoth et.al- This paper investigates concrete-filled steel tubular (CFST) columns, valued for their high capacity, economic benefits, fire resistance, and improved seismic performance over traditional steel columns. Using ANSYS Workbench, the study compares square and circular CFST sections made from various concrete types (nominal, recycled aggregate, and high-performance). It examines stress-strain, load-deflection, and collapse patterns, and incorporates anti-corrosion measures to mitigate environmental degradation.

"Jacketing with Steel Angle Sections and Wide Battens of RC Column and its Influence on Blast Performance"

DOI : 10.35291/2454-9150.2024.0422

(2022), K. Menon et.al- This paper presents a new method to improve the blast performance of square reinforced concrete (RC) columns without increasing their size, addressing concerns about building functionality. The technique involves using four structural steel angle sections along the column's vertical edges, connected by battens or plates. Numerical analyses, conducted with ABAQUS and the Concrete Damage Plasticity (CDP) model, include strain rate effects per CEB-FIB MODEL CODE 2010. Results show significant improvements in the blast resistance of RC columns, reducing damage vulnerability. This method helps mitigate structural failure from extreme loads like explosions, preventing severe damage and loss of life.

"Analysis of Blended Concrete Cubes under Impact loading using ANSYS" (2022), Janhavi Singh and Shilpa Pal- This study aims to perform a finite element method (FEM) analysis of blended concrete cubes under impact loads. Deformation and stress responses are derived from velocity impact simulations conducted using the ANSYS Explicit Dynamics module. A parametric analysis is conducted by varying the height and shape of the impactor, boundary conditions, and the degree of cement replacement in the blended concrete with various supplementary cementitious materials. The results indicate that the strength of the blended concrete cubes decreases the height of the impact increases.

"Dynamic Response of Concrete Subjected to High Rate of Loading: A Parametric Study" (2023), A. Patel et.al- This paper investigates the dynamic behavior of concrete under high loading rates, which is found in blasts or impacts. Numerical simulations made on strain rate and material properties indicate the impact of these parameters on the performance of concrete. The study might be used as a means of providing insight to enhance structural designs to withstand extreme loading conditions.

"Analysis of Steel–Concrete Composite Beam Using ANSYS 18.1 Workbench" (2023), by A. Asif et.al- This study examines the structural behavior of steel–concrete composite beams under monotonic loading, focusing on beams that combine steel and concrete for enhanced stiffness and cost efficiency. Using ANSYS 18.1 Workbench for non-linear static analysis, the paper analyzes the effects of varying stud numbers and spacing on load-deformation and ductility. Results were compared to experimental data, showing less than 10% error, with findings that excessive studs may lead to cracks and impact structural serviceability.

"Predicting the Critical Load of Rectangular Concrete-Filled Steel Tube Columns with Ultra-High Strength Concrete Using ANSYS" (2023), P. Van-Phuc et.al- This research develops a method for predicting the critical load of short, rectangular concrete-filled steel tube columns using ultra-high strength concrete. Using ANSYS Workbench and the Drucker-Prager model, the study simulates confined compression and evaluates the influence of model parameters. Results align closely with experimental data, confirming that this method provides a simplified approach to accurately model high-strength concrete behavior in structural applications.

"Modelling and Simulation of Flexural Behaviour for Reinforced Concrete Beams Using ANSYS" (2023), Mohamed et-al- This study develops a numerical model using ANSYS 2022 R2 software to simulate the flexural behavior of reinforced concrete (RC) beams. The model is tested under a four-point loading configuration to assess the influence of the reinforcement steel ratio and the compressive strength of concrete on flexural strength. Results indicate a significant improvement in load-deflection capacity with increased tensile steel reinforcement. Additionally, while the compressive strength of concrete greatly influences load-deflection behavior, the structural stiffness of the model is marginally affected. This research provides key insights into designing RC beams with enhanced flexural performance.

"Evaluating dynamic behaviour of a concrete dam using modal analysis" (2023), Surabhi Saxena and Mahesh Patel- This study analyzes the dynamic behavior of the Shahpurkandi Dam, a 55.5-meter-high concrete gravity dam, using ANSYS finite element software. The investigation includes modeling the dam under full reservoir conditions to assess its modal response and accurately capture the interaction between the structure and the surrounding water. Using a direct solver with the block Lanczos method, the first six mode shapes are identified, and the total displacement is calculated through numerical simulations. The findings show that the maximum displacement of 0.0039 mm occurs at the dam's crest at the fundamental natural frequency. A comparison of the natural frequencies is also presented for both the full and empty reservoir conditions.

"Soil–pipe interaction and structural response under static and seismic loading for geopolymer concrete pipes" (2024), Kong Fah Tee and Sayedali Mostofizadeh- This study employs ANSYS Solid 185 and Pipe 288 elements to investigate the nonlinear behavior of soil and geopolymer concrete (GPC) pipes. A total of 43 static and dynamic analyses are performed to assess the impact of SPI parameters, with a primary focus on the optimal vertical displacement at the midpoint of GPC pipes. The analysis reveals that soil density is the most critical factor influencing both static and dynamic behavior, with its effect under seismic loading being approximately 46% greater than under static conditions. These findings contribute to a deeper understanding of how underground concrete pipes perform during seismic events, providing a solid foundation for improving the resilience and efficiency of underground pipe systems.

DOI : 10.35291/2454-9150.2024.0422

"Numerical simulation of projectile impact on reinforced concrete structures: a study of slab performance under varying projectile velocities using ANSYS" (2024), Abhishek Minhas and Seema- This paper presents an in-depth analysis of the behavior of reinforced concrete slabs under varying projectile impact velocities. Using numerical simulations in ANSYS, the study investigates the dynamics of projectile impacts on a standardized 200 mm thick concrete slab. The research explores different projectile velocities and geometries, focusing on their effects on internal stresses and deformations. A critical velocity of 313 m/s is identified for full penetration, and the study also examines how factors such as reinforcement spacing, diameter, and layering influence the slab's performance. The findings provide valuable insights for optimizing design practices and improving protective measures for structures subjected to projectile impacts.

"Dynamic Behavior of Ultra-High Performance Concrete Beams with Rectangular Openings Subjected to Impact Loads" (2024), Jian Liu et al- This study investigates the dynamic response of ultra-high-performance concrete (UHPC) beams with rectangular openings under low-velocity impacts, utilizing numerical simulations in ANSYS/LS-DYNA. The Continuous Surface Cap (CSC) model is validated through quasi-static four-point loading tests on UHPC beams with openings. Drop hammer impact simulations are then performed to analyze the effect of varying opening sizes, positions, stirrup quantities, and UHPC strengths on impact response. Findings reveal that opening configurations significantly affect the structural integrity, providing a basis for designing UHPC beams optimized for impact resistance.

"Three dimensional simulations of FRC beams and panels with explicit definition of fibres-concrete interaction" (2024), I. Marzec et.al- This paper introduces a novel and effective mesoscale modeling technique for steel fiber-reinforced high-performance concrete (HPC). The approach utilizes comprehensive 3D modeling, explicitly incorporating the distribution and orientation of steel fibers. To validate and refine the finite element model, numerical simulation results are compared with experimental data obtained from beams and panels made of high-performance concrete, varying in size and steel fiber dosage.

3. METHODOLOGY

3.1 GENERAL

Fixed-fixed hollow steel fiber-reinforced columns and solid RCC columns, each with lengths of 1000 mm, 3000 mm, and 5000 mm, as well as varying cross-sections and thicknesses, were modeled and analyzed using ANSYS. The analysis involved applying different velocities. Initially, at a velocity of 1000 mm/sec, no significant deformation was observed. However, as the velocity was gradually increased, significant deformation was noticed at 3000 mm/sec.

Table 1: Specifications of Hollow FRC Columns Analyzed in the Study

Sl. No	Dimensions (mm)	Thickness (mm)	Velocity (mm/sec)
1	D1:200x300x1000	2	1000
			3000
			5000
			8000
		4	1000
			3000
			5000
			8000
		8	1000
			3000
			5000
			8000
2	D2:200x300x3000	2	1000
			3000
			5000
			8000
		4	1000
			3000
			5000
			8000
		8	1000
			3000
			5000
			8000
3	D3:200x300x5000	2	1000
			3000
			5000
			8000
		4	1000
			3000
			5000
			8000
			1000
			3000
			5000

DOI : 10.35291/2454-9150.2024.0422

		8	**8000**
4	**D4:300x350x1000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000
5	**D5:300x350x3000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000
6	**D6:300x350x5000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000
7	**D7:400x400x1000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000
8	**D8:400x400x3000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000
9	**D9:400x400x5000**	**2**	**1000**
			3000
			5000
			8000
		4	**1000**
			3000
			5000
			8000
		8	**1000**
			3000
			5000
			8000

Table 2: Specifications of Solid RCC Columns Analyzed in the Study

Sl. No	Dimensions (mm)	Length (mm)	Velocity (mm/sec)
1	D1:200x300	1000	1000
			3000
			5000
			8000
		3000	1000
			3000

DOI : 10.35291/2454-9150.2024.0422

2	**D2:200x300**		**5000**
			8000
3	**D3:200x300**	**5000**	**1000**
			3000
			5000
			8000
4	**D4:300x350**	**1000**	**1000**
			3000
			5000
			8000
5	**D5:300x350**	**3000**	**1000**
			3000
			5000
			8000
6	**D6:300x350**	**5000**	**1000**
			3000
			5000
			8000
7	**D7:400x400**	**1000**	**1000**
			3000
			5000
			8000
8	**D8:400x400**	**3000**	**1000**
			3000
			5000
			8000
9	**D9:400x400**	**5000**	**1000**
			3000
			5000
			8000

3.2 ANSYS EXPLICIT DYNAMICS ANALYSIS OF COLUMNS

The columns are modeled and analyzed using ANSYS Explicit Dynamics to study their dynamic behavior under varying loading conditions, such as impact, stress wave propagation, and rapidly fluctuating time-dependent forces. This analysis method focuses on the exchange of momentum between interacting bodies and the effects of inertia on the system. It is particularly useful for simulating events that occur within a very short time frame, typically less than one second, often in the millisecond range. The key steps in performing the analysis are outlined below:

3.2.1: Setting Up the Analysis System

Launch the ANSYS software and open a new project. From the toolbox, drag and drop the Explicit Dynamics module into the Project Schematic to begin setting up your analysis.

3.2.2: Set Up Engineering Data

Begin by accessing the Engineering Data section to define the material properties for the column being modelled. These properties can be customized for each specific analysis, and you can also store them in a material library for future use through the Engineering Data tab. For this particular analysis, Concrete is chosen as the material. The software offers the flexibility to combine various material properties to obtain precise simulation results. In this case, the primary properties utilized are Young's Modulus and Poisson’s Ratio.

Table 3: Material Property Definitions in Engineering Data

Sl. No.	Material Property	value
1	Density	2500kg/m^3
2	Youngs modulus	30GPa
3	Poisson’s Ratio	0.25

3.2.3: Defining the Geometry

The geometry of the model is created using ANSYS Design Modeler. First, a sketch is drawn by selecting a plane and defining the appropriate dimensions. This sketch is then extruded to form a solid, 3-dimensional parametric model.

3.2.4: Defining and Assigning Material Properties

After the geometry is modelled, proceed to the Model section, which opens the ANSYS Mechanical interface. In this interface, you can modify the material properties assigned to different parts and bodies of the model.

3.2.5: Establishing Model Connections

Once the materials are assigned, the next step is to define the connections between the components of the model. In ANSYS Explicit Dynamics, the software automatically generates the necessary contact interactions and body connections. For this analysis, the default settings are used, keeping the contacts program-controlled.

3.2.6: Mesh Generation

Meshing is the process of discretizing the geometry into smaller elements and nodes for analysis. The mesh is essential for mathematically representing the stiffness and mass distribution of the structure. To generate the mesh, select 'Mesh' in the Outline. In the detail view, set the physics preference to 'Explicit,' the element order to 'Linear,' and the element size to 15 mm. A linear hexahedral mesh is generated, where each hexahedral

DOI : 10.35291/2454-9150.2024.0422

element consists of 8 nodes. For a column of size 200 mm x 300 mm x 5000 mm, the generated mesh consists of 45816 elements and 91720 nodes.

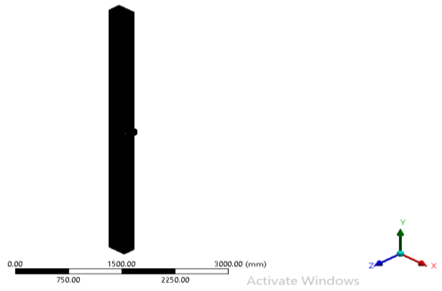


Fig 1: Mesh Generation

3.2.7: Setting Initial Conditions and Parameters

The definition of initial conditions is determined by the selected type of analysis. In the case of an Explicit Dynamics analysis, the Initial Conditions object is used to specify parameters such as Velocity, Angular Velocity, and Drop Height, which can be applied to specific sections of the column geometry. For this particular analysis, velocity is the primary parameter applied.

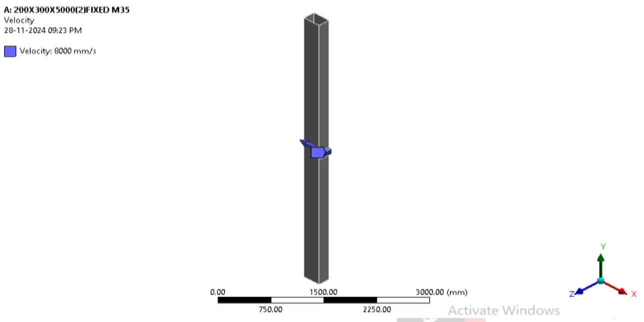


Fig 2: Initial Conditions – Application of Velocity

3.2.8: Specifying Boundary Conditions

For this particular analysis, velocity is the primary parameter applied. For this particular analysis, a 'fixed support' boundary condition is applied. This is implemented at both ends of the column, where the bottom faces of the supports are constrained, creating a fixed-to-fixed column setup. This ensures the column is securely fixed at both ends, providing the necessary conditions for accurate load analysis.

3.2.9: Run the Analysis

The parameters to be calculated need to be chosen for the analysis. In this case, the total deformation of the column is being calculated. To do this, right-click on the Solution in the outline, navigate to Insert, and select Total Deformation. After selecting the desired parameter, right-click on Solution again and choose Solve. The progress of the solution is tracked through a 'Solution Status' window within the mechanical application. This window displays conventional progress bars and includes options to 'Stop Solution' or 'Interrupt Solution' for managing the solution process.

4.RESULTS AND DISCUSSION

The analysis of various Solid RCC and hollow fiber reinforced concrete Columns of Different cross- sections (D1, D2, D3, D4, D5, D6, D7, D8, D9), under different velocities and different thickness for column is carried out.

4.1 Total deformation of Solid RC Fixed-Fixed Column of dimension 200X 300X1000mm (D1).

4.1.1: Velocity=8000mm/s, 126000mm/s (D1).

The total deformation versus time graph is generated for the solid RC column (D1), under a velocity of 8000 mm/sec, 126000 mm/sec.

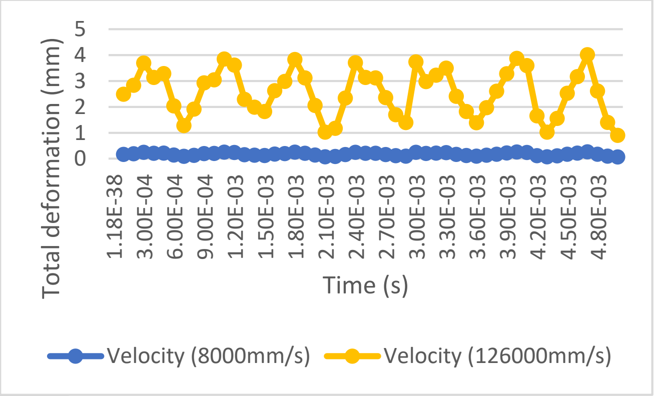


Fig 3: Total deformation v/s time graph, Column (D1)

Based on the analysis of the graph, the maximum observed deformation is 0.2551 mm, which is considered minimal and significantly below the permissible limit of 4 mm for a column with a length of 1000 mm, as outlined in IS 1343:1980. This clearly indicates that the column remains structurally stable and safe when exposed to a velocity of 8000 mm/sec, as the deformation is well within acceptable limits. However, in contrast, the graph shows a maximum observed deformation of 4.0046 mm, which slightly exceeds the permissible limit of 4 mm for the same column size. As a result, this deformation suggests that the column would be structurally unsafe when subjected to a much higher velocity of 126000 mm/sec, exceeding the safety threshold.

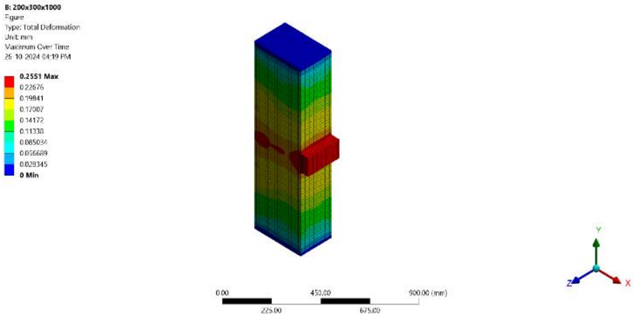


Fig 4: Total deformation of Column (D1), v=8000mm/s

DOI : 10.35291/2454-9150.2024.0422

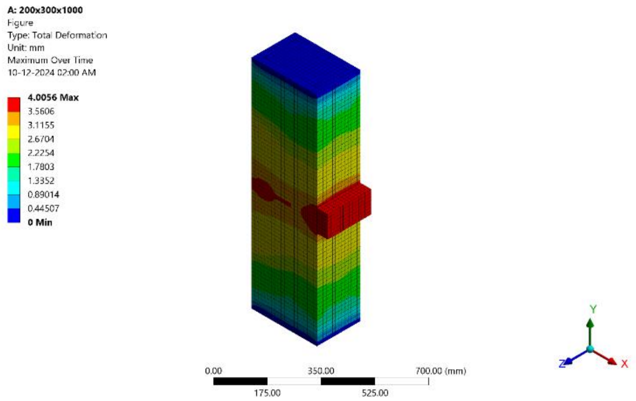


Fig 5: Total deformation of Column (D1), v=126000mm/s

4.2 Total deformation of Solid RC Fixed-Fixed Column of dimension 200X 300X3000mm (D2).

4.2.1: Velocity=8000mm/s, 180000mm/s (D2).

The total deformation versus time graph is generated for the solid RC column (D2), under a velocity of 8000 mm/sec, 180000 mm/sec.

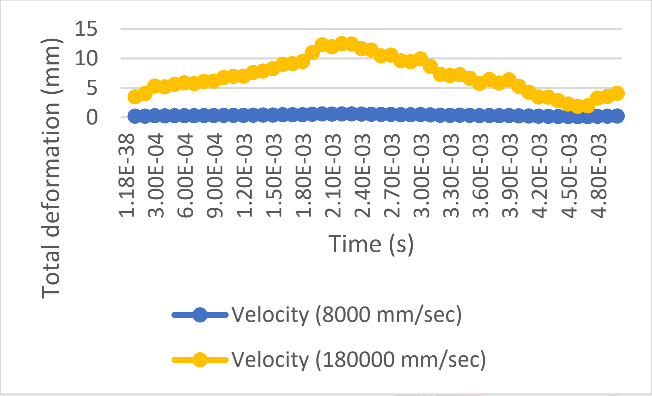


Fig 6: Total deformation v/s time graph, Column (D2)

According to the graph, the maximum observed deformation is 0.55501 mm, which is negligible and well within the permissible limit of 12 mm for a column with a length of 3000 mm, as specified in IS 1343:1980. This indicates that the column is structurally safe when exposed to a velocity of 8000 mm/sec. In contrast, the graph shows a maximum observed deformation of 12.472 mm, which slightly exceeds the permissible limit of 12 mm for the same column size. However, this deformation remains within an acceptable range, suggesting that the column is still structurally safe when subjected to a velocity of 180,000 mm/sec.

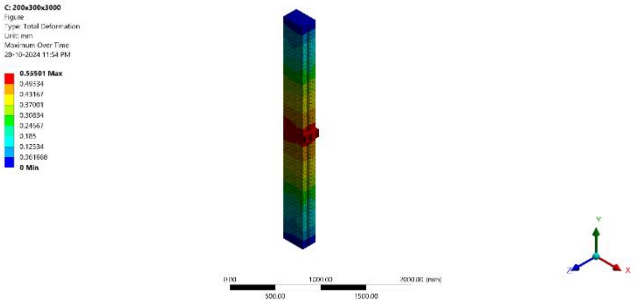


Fig 7: Total deformation of Column (D2), v=8000mm/s

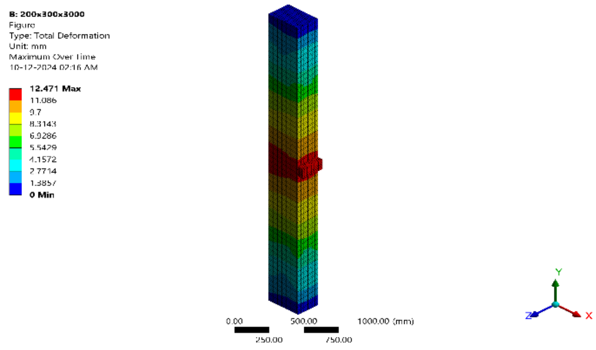


Fig 8: Total deformation of Column (D2), v=180000mm/s

4.3 Total deformation of Solid RC Fixed-Fixed Column of dimension 200X 300X5000mm (D3).

4.3.1: Velocity=8000mm/s, 210000mm/s (D3).

The total deformation versus time graph is generated for the solid RC column (D3), under a velocity of 8000 mm/sec, 210000 mm/sec.

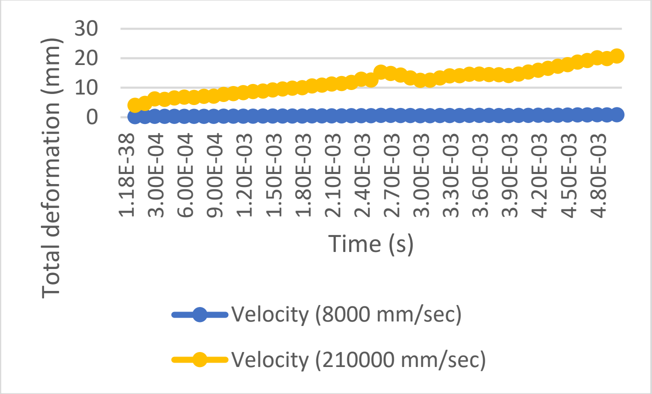


Fig 9: Total deformation v/s time graph, Column (D3)

The graph indicates that the maximum observed deformation is 0.78994 mm, which is minimal and well within the permissible limit of 20 mm for a 5000 mm column, as per IS 1343:1980. Therefore, the column can be considered structurally safe when subjected to a velocity of 8000 mm/sec. In contrast, the graph shows that the maximum observed deformation is 20.729 mm, which exceeds the permissible limit of 20 mm for the same column size. However, this deformation remains just slightly beyond the limit, suggesting that the column is still structurally safe when exposed to a velocity of 210,000 mm/sec.

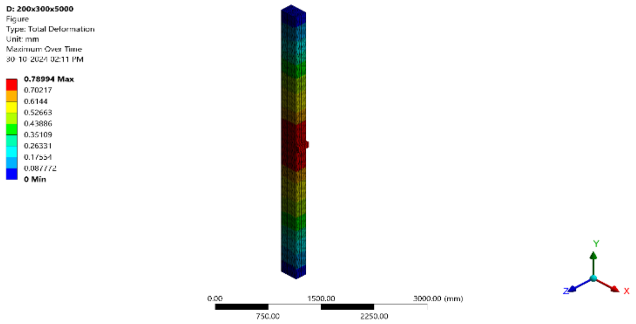


Fig 10: Total deformation of Column (D3), v=8000mm/s

DOI : 10.35291/2454-9150.2024.0422

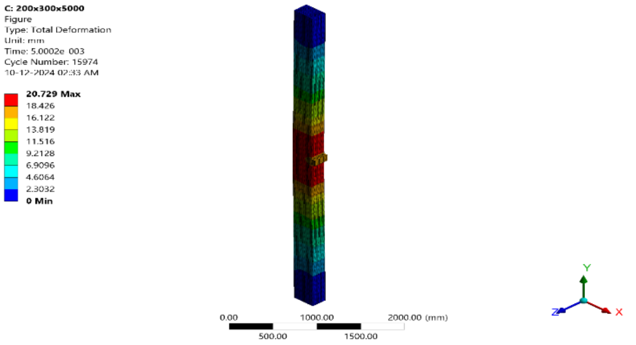


Fig 11: Total deformation of Column (D3), v=210000mm/s

Similarly, deformation is analyzed for different velocities applied to cross-sections D4, D5, D6, D7, D8 and D9 until at which velocity column remains unsafe.

4.4 Total deformation of hollow FRC Fixed-Fixed Column of dimension 200X 300X1000mm (D1).

4.4.1: Thickness =2mm, Velocity=1000mm/s ,2000mm/s, 3000mm/s (D1).

The total deformation versus time graph is generated for the hollow column (D1) with thickness 2mm, under a velocity of 1000 mm/sec, 2000mm/s, 3000mm/s.

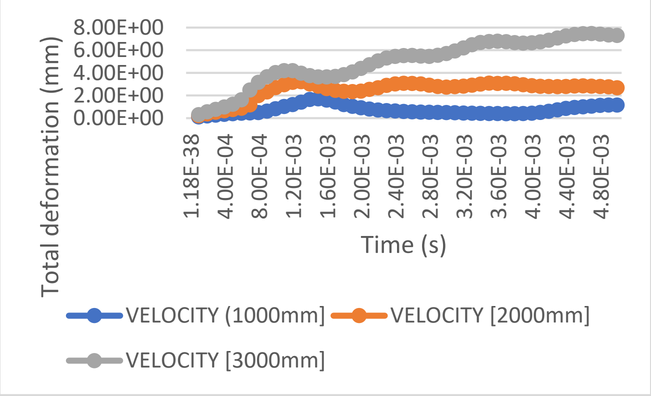


Fig 12: Total deformation v/s time graph, Column (D1), Thickness 2mm

Based on the analysis of the graph, it was observed that for a column with a thickness of 2 mm, the maximum deformation at a velocity of 1000 mm/s is 1.6925 mm, and at 2000 mm/**s**, it is 3.2057 mm, both of which are within the permissible limit of 4 mm for a column with a length of 1000 mm, as specified in IS 1343:1980. This confirms that the column is structurally **s**afe under these conditions. However, at a velocity of 3000 mm/**s**, the graph shows a maximum deformation of 7.4551 mm, which exceeds the permissible limit, indicating that the column is structurally unsafe at this higher velocity.

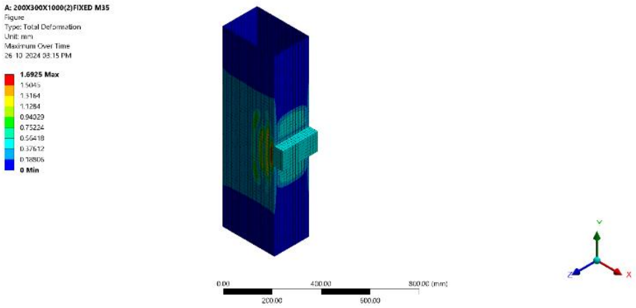


Fig 13: Total deformation of Column (D1), 2mm thickness, v=1000mm/s

Based on the analysis of the graph, it was observed that for a column with a thickness of 2 mm, the maximum deformation at a velocity of 1000 mm/s is 1.6925 mm, and at 2000 mm/**s**, it is 3.2057 mm, both of which are within the permissible limit of 4 mm for a column with a length of 1000 mm, as specified in IS 1343:1980. This confirms that the column is structurally **s**afe under these conditions. However, at a velocity of 3000 mm/**s**, the graph shows a maximum deformation of 7.4551 mm, which exceeds the permissible limit, indicating that the column is structurally unsafe at this higher velocity.

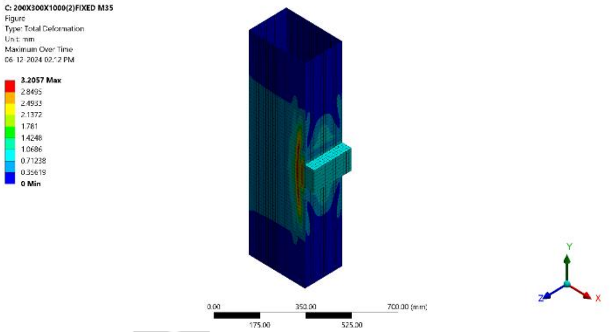


Fig 14: Total deformation of Column (D1), 2mm thickness, v=2000mm/s

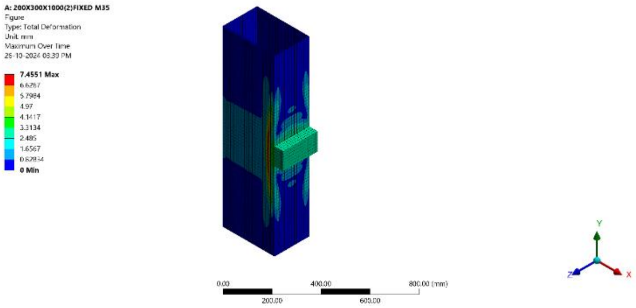


Fig 15: Total deformation of Column (D1), 2mm thickness, v=3000mm/s

4.4.2: Thickness =4mm, Velocity=1000mm/s ,2000mm/s, 3000mm/s, 5000mm/s (D1).

The total deformation versus time graph is generated for the hollow column (D1) with thickness 4mm, under a velocity of 1000 mm/s, 2000mm/s, 3000mm/s, 5000mm/s.

DOI : 10.35291/2454-9150.2024.0422

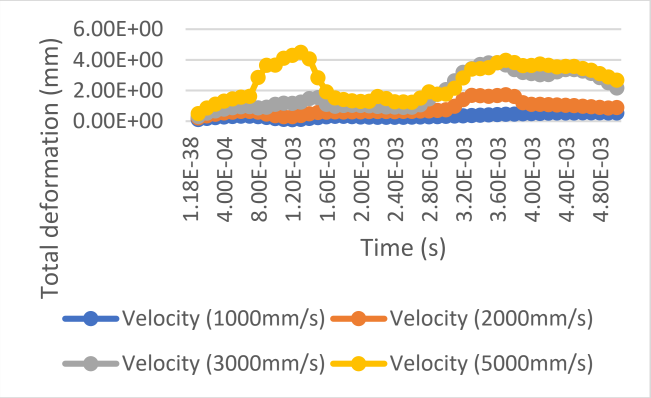


Fig 16: Total deformation v/s time graph, Column (D1), thickness 4mm

Due to the change in thickness, the deformation of the column varies, with thicker columns generally exhibiting lower deformation under the same velocity conditions. Based on the analysis of the graph, for a column with a thickness of 4 mm, the maximum deformation at a velocity of 1000 mm/s is 0.52694 mm, and at 2000 mm/s, it is 1.7104 mm, both of which are within the permissible limit of 4 mm for a column with a length of 1000 mm, as specified in IS 1343:1980. This confirms that the column is structurally safe under these conditions. However, at a velocity of 3000 mm/s, the graph shows a maximum deformation of 3.805 mm, which is still within the permissible limit. But when exposed to a velocity of 5000 mm/s, the maximum observed deformation increases to 4.483 mm, exceeding the permissible limit of 4 mm for a column with a length of 1000 mm, as specified in IS 1343:1980. Therefore, the column becomes structurally unsafe at this higher velocity.

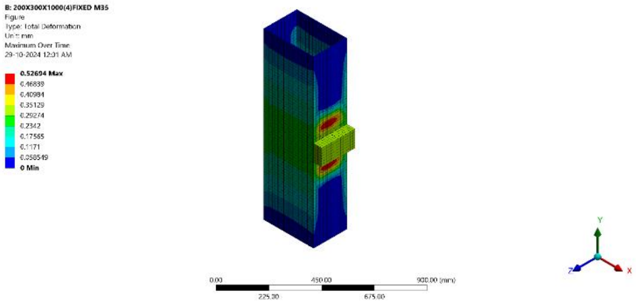


Fig 17: Total deformation of Column (D1), 4mm thickness, v=1000mm/s

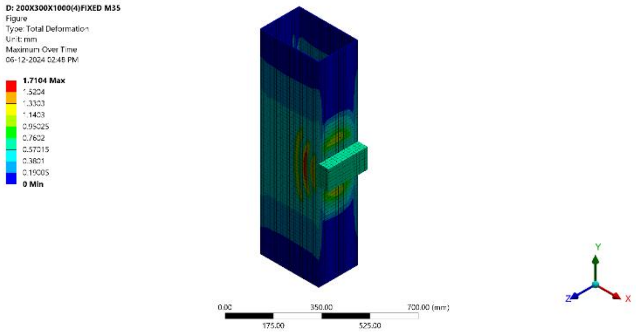


Fig 18: Total deformation of Column (D1), 4mm thickness, v=2000mm/s

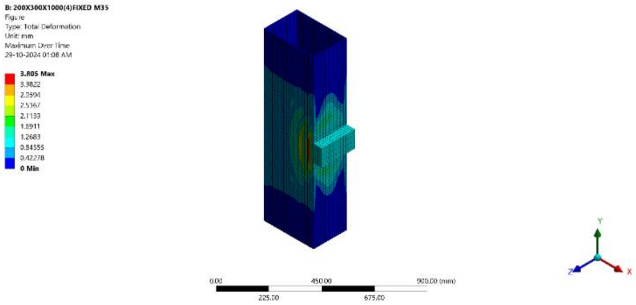


Fig 19: Total deformation of Column (D1), 4mm thickness, v=3000mm/s

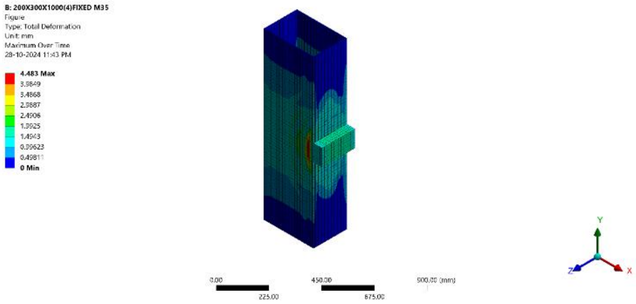


Fig 20: Total deformation of Column (D1), 4mm thickness, v=5000mm/s

4.4.3: Thickness =2mm, Velocity=1000mm/s ,2000mm/s, 3000mm/s (D1).

The total deformation versus time graph is generated for the hollow column (D1) with thickness 8mm, under a velocity of 1000 mm/s, 2000mm/s, 3000mm/s, 5000mm/s.

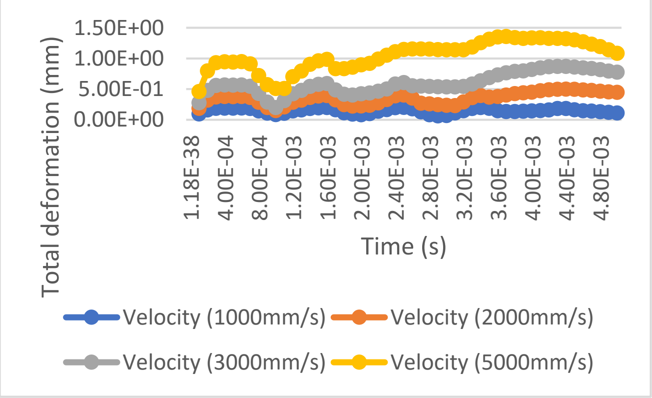


Fig 21: Total deformation v/s time graph, Column (D1), thickness 8mm

Due to the variation in thickness, the deformation of the column changes, with thicker columns generally experiencing lower deformation at the same velocity. Based on the analysis of the graph, for a column with a thickness of 8 mm, the maximum deformation at 1000 mm/s is 0.20043 mm, and at 2000 mm/s, it is 0.49799 mm both well within the permissible limit of 4 mm for a column with a length of 1000 mm, as per IS 1343:1980. Similarly, at velocities of 3000 mm/s and 5000 mm/s, the maximum deformations are 0.86927 mm and 1.3564 mm, respectively, which are still within the permissible limit. In fact, the column remains structurally safe for all velocities up to 8000 mm/s. However, at velocities exceeding 9000 mm/s, the maximum

DOI : 10.35291/2454-9150.2024.0422

deformation surpasses the permissible limit of 4 mm, making the column structurally unsafe beyond this point.

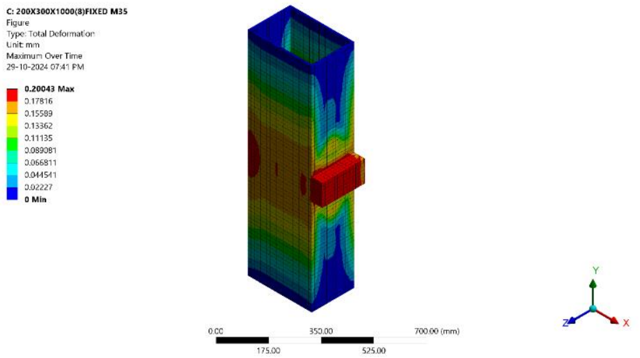


Fig 22: Total deformation of Column (D1), 8mm thickness, v=1000mm/s

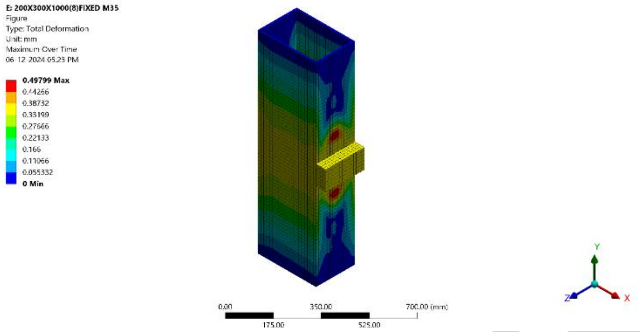


Fig 23: Total deformation of Column (D1), 8mm thickness, v=2000mm/s

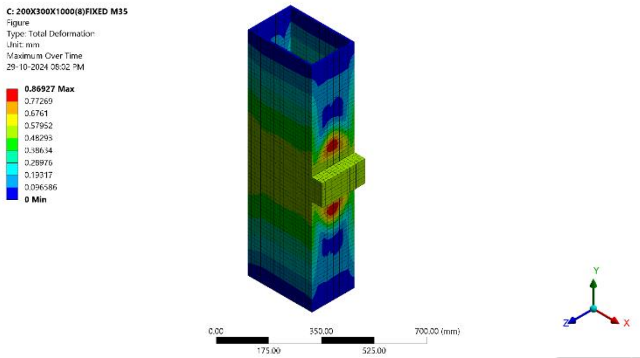


Fig 24: Total deformation of Column (D1), 8mm thickness, v=3000mm/s

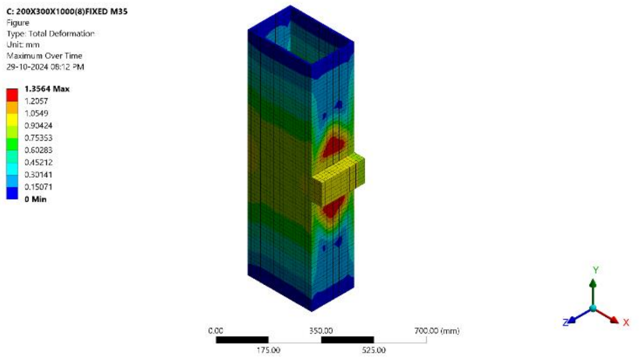


Fig 25: Total deformation of Column (D1), 8mm thickness, v=5000mm/s

Similarly, deformation is analyzed for different velocities and thickness applied to cross-sections D2, D3, D4, D5, D6, D7, D8 and D9 until at which velocity column remains unsafe.

4.5: Comparison of Deformation in RC Solid and Hollow FRC Columns (D1 to D9)

4.5.1 Solid RCC Columns

The comparison of deformation in RC Solid Column D1 to D9 is graphically represented by plotting the total deformation versus velocity graph, when subjected to varying velocities.

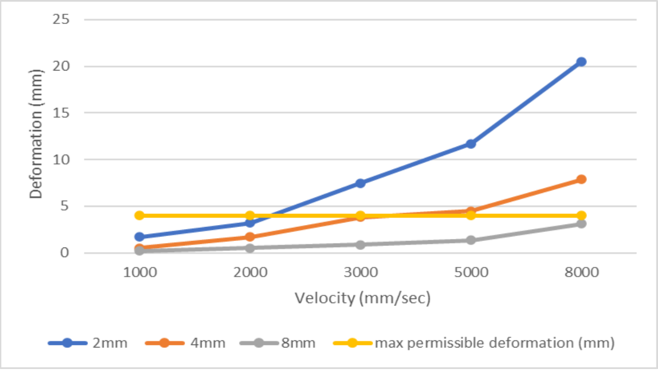


Fig 26: Comparison of deformation in Solid Column D1

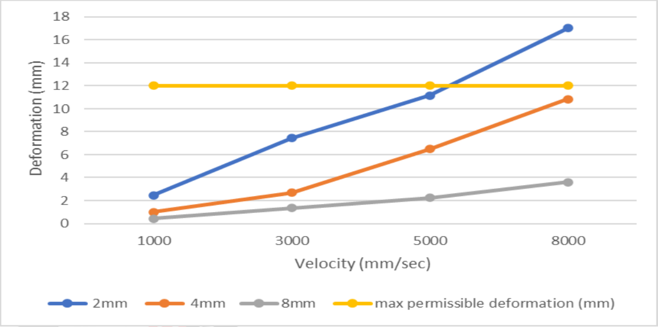


Fig 27: Comparison of deformation in Solid Column D2

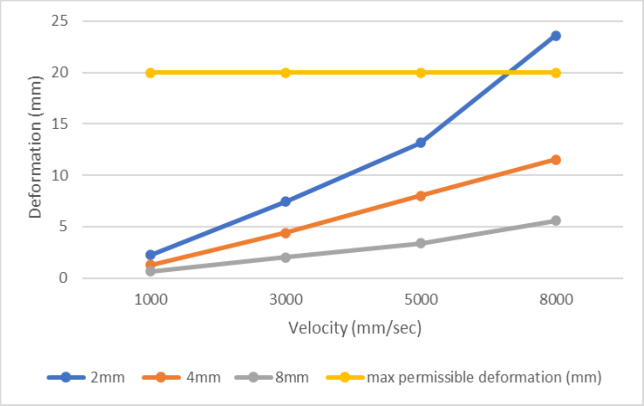
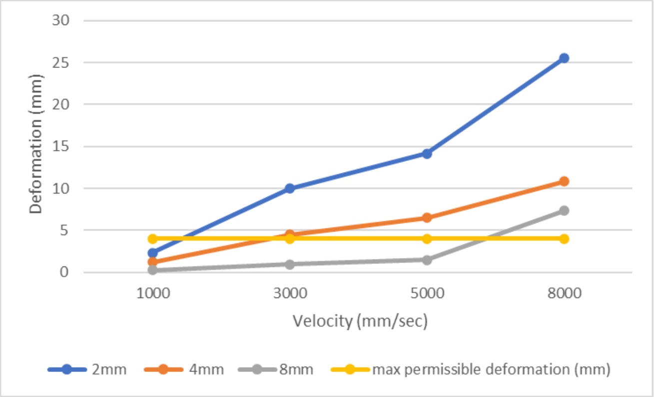


Fig 28: Comparison of deformation in Solid Column D3



DOI : 10.35291/2454-9150.2024.0422

Fig 29: Comparison of deformation in Solid Column D4

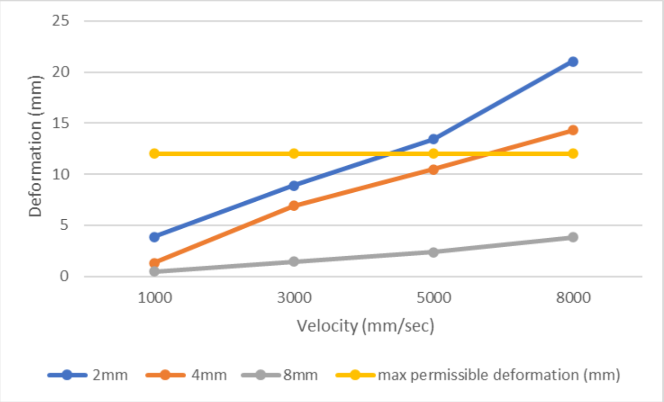


Fig 30: Comparison of deformation in Solid Column D5

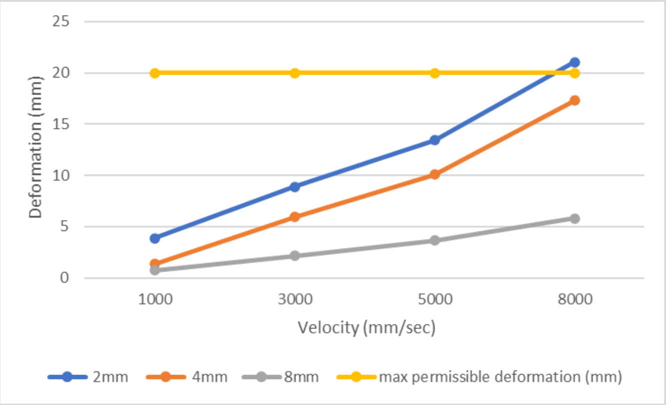


Fig 31: Comparison of deformation in Solid Column D6

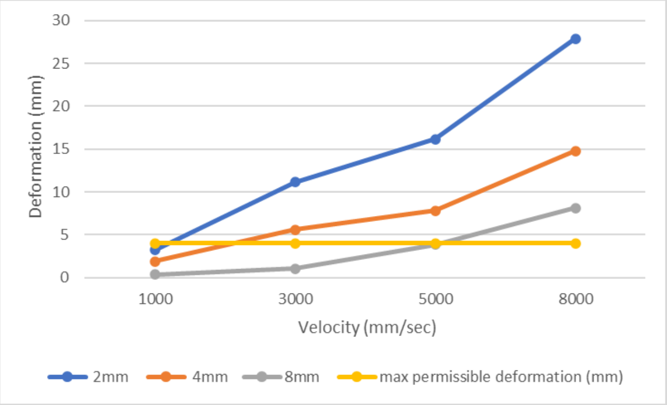


Fig 32: Comparison of deformation in Solid Column D7

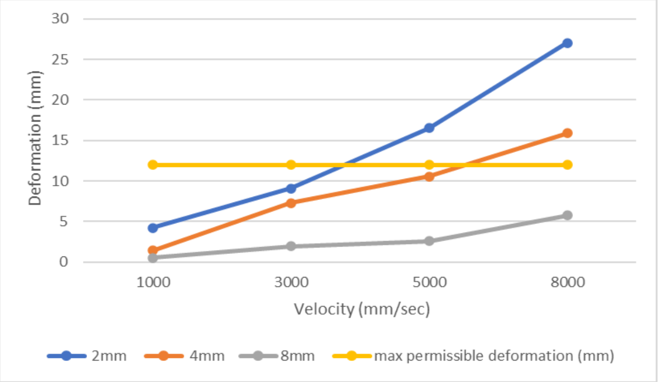


Fig 33: Comparison of deformation in Solid Column D8

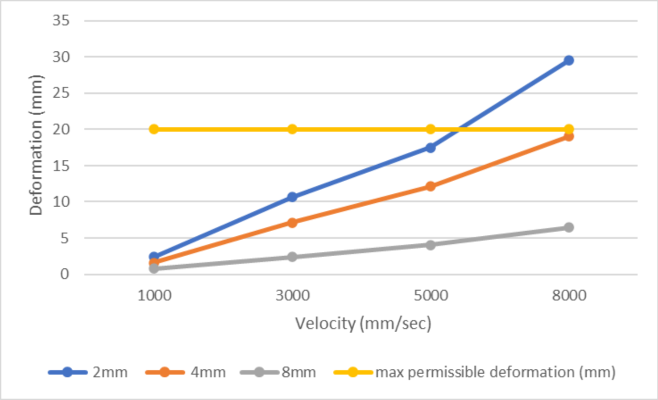


Fig 34: Comparison of deformation in Solid Column D9

According to IS 1343:1980, the maximum permissible deformation for a column is determined by its size: 4mm for a 1000mm column, 12mm for a 3000mm column, and 20mm for a 5000mm column. The graph shows how the structural safety of various column sizes and lengths behaves at different velocities. For columns with dimensions of 200x300mm, 300x350mm, and 400x400mm, the columns remain structurally safe at lower velocities, but as the velocity increases, they eventually exceed the permissible deformation limits and become unsafe. Specifically, for the 200x300x1000mm column, it stays safe up to a velocity of 126,000mm/sec, while the 200x300x3000mm column remains safe up to 180,000mm/sec, and the 200x300x5000mm column stays safe up to 210,000mm/sec. Similarly, for the 300x350mm columns, the 1000mm long column stays safe up to 150,000mm/sec, the 3000mm long column can handle up to 240,000mm/sec, and the 5000mm long column is safe up to 270,000mm/sec. For the 400x400mm columns, the 1000mm column stays safe up to 150,000mm/sec, the 3000mm column is safe up to 280,000mm/sec, and the 5000mm column can withstand up to an impressive 301,000mm/sec. This data suggests that as the column's size and thickness increase, its ability to withstand higher velocities improves, allowing it to remain structurally safe even under greater stress. The trend indicates that larger and thicker columns can handle higher deformation levels, making them more suitable for applications involving higher velocities and potential for greater forces.

4.5.2 Hollow FRC Columns

The comparison of deformation in Column D1 to D9 is graphically represented by plotting the total deformation versus velocity graph for thicknesses of 2mm, 4mm, and 8mm, when subjected to varying velocities.

DOI : 10.35291/2454-9150.2024.0422

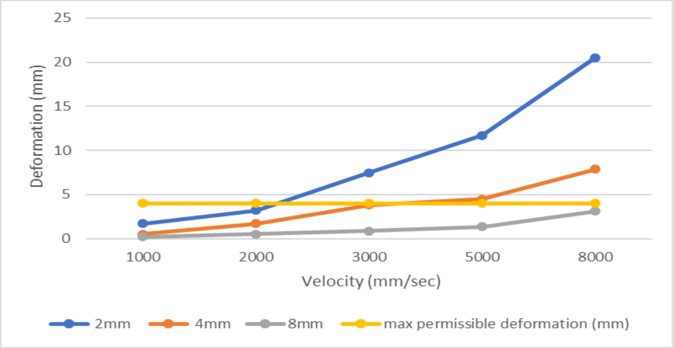


Fig 35: Comparison of deformation in Hollow Column D1

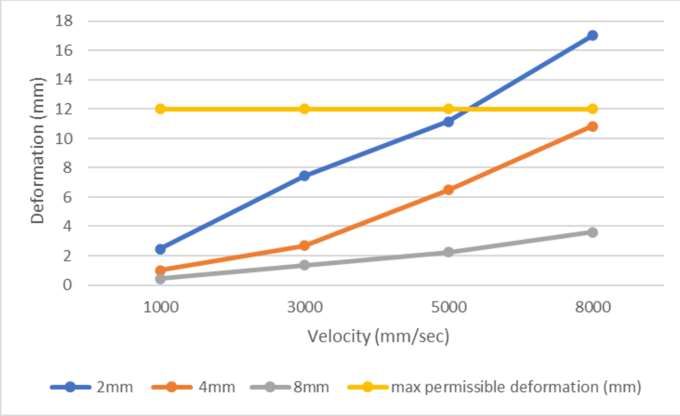


Fig 36: Comparison of deformation in Hollow Column D2

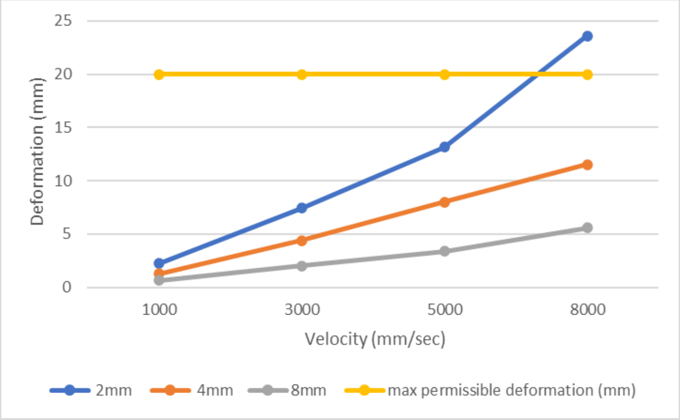


Fig 37: Comparison of deformation in Hollow Column D3

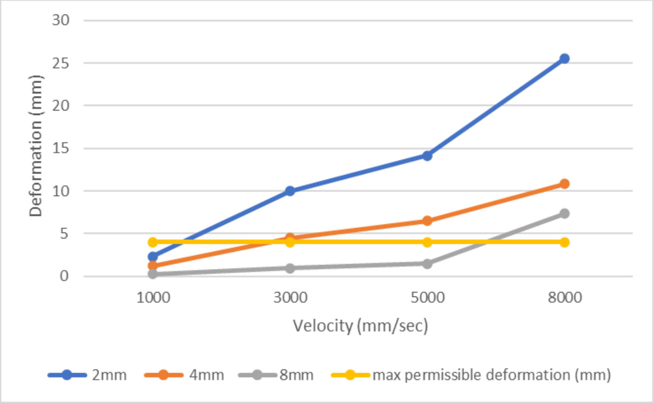


Fig 38: Comparison of deformation in Hollow Column D4

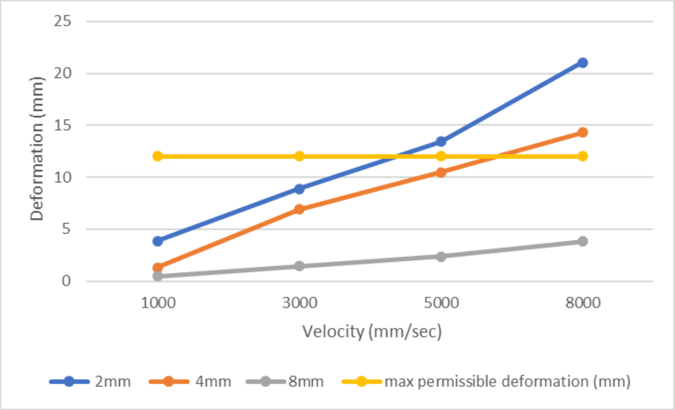


Fig 39: Comparison of deformation in Hollow Column D5

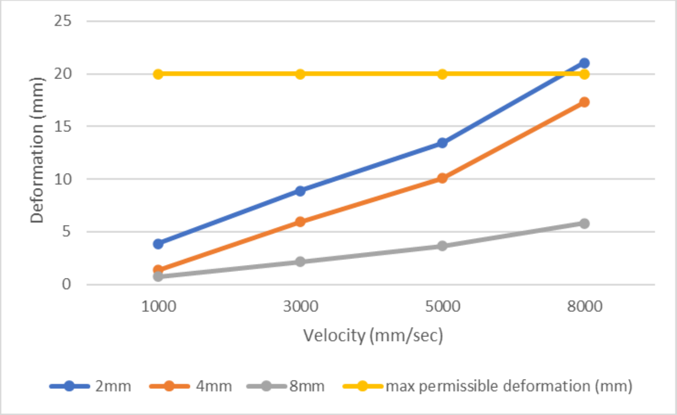


Fig 40: Comparison of deformation in Hollow Column D6

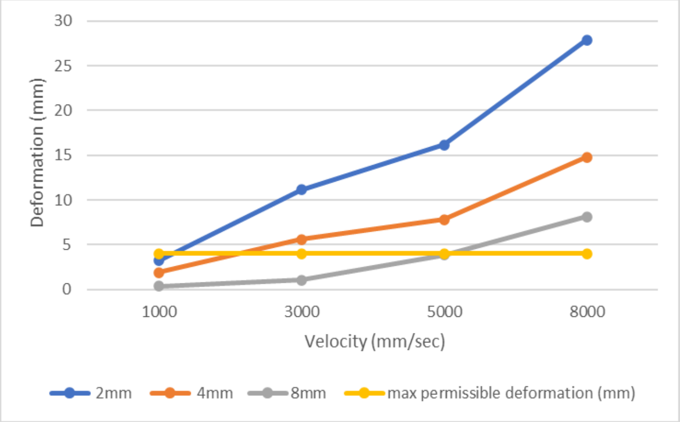


Fig 41: Comparison of deformation in Hollow Column D7

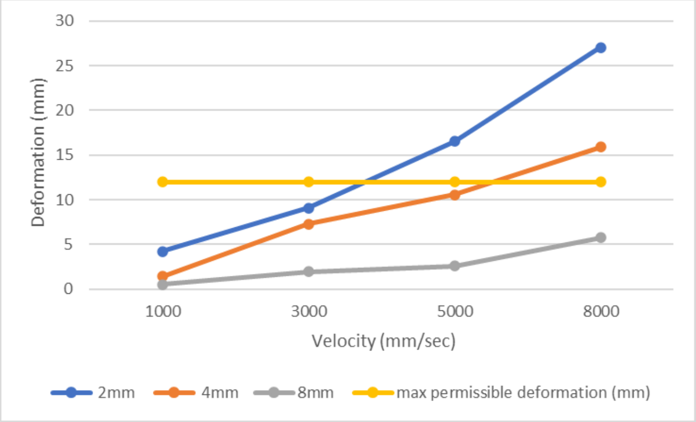


Fig 42: Comparison of deformation in Hollow Column D8

DOI : 10.35291/2454-9150.2024.0422

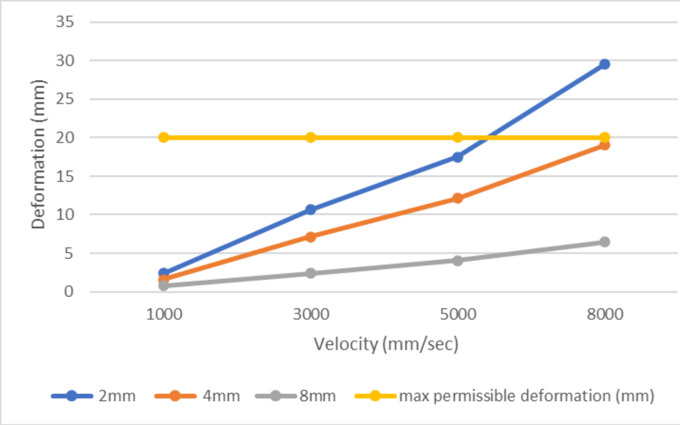


Fig 43: Comparison of deformation in Hollow Column D9

According to IS 1343:1980, the maximum permissible deformation for a column varies based on its size. For a column with a length of 1000mm, the allowable deformation is limited to 4mm; for a column of 3000mm in length, the limit increases to 12mm; and for a column of 5000mm in length, the permissible deformation is 20mm. The graph shows that for columns of different sizes, such as 200x300mm, 300x350mm, and 400x400mm, the structural safety is maintained at lower velocities when the column thickness is 2mm or 4mm. However, as the velocity increases, the deformation surpasses the permissible limits, causing these columns to become unsafe. This suggests that at higher velocities, thinner columns cannot handle the strain and exceed the deformation limits set by IS 1343:1980. Interestingly, when the column thickness is increased to 8mm, the columns are able to withstand much higher velocities, staying structurally safe even at velocities up to 8000mm/sec. This indicates that increasing the thickness of the column significantly enhances its ability to resist deformation, ensuring its structural safety at higher velocities. The data highlights the importance of selecting an appropriate column thickness for specific conditions to prevent unsafe deformations and ensure the longevity and safety of the structure.

CONCLUSIONS

The study investigated the structural performance of hollow fiber reinforced concrete (FRC) and solid reinforced concrete (RC) columns with varying cross-sections (D1 to D9), with different lengths (1000mm, 3000mm, and 5000mm) and thicknesses (for hollow sections), under dynamic loading conditions. The results highlight significant differences in their ability to withstand deformation. Solid RC columns consistently performed better, maintaining structural integrity and staying within the permissible deformation limits specified by IS 1343:1980 at all tested velocities. These columns demonstrated superior strength and stability, making them suitable for high-velocity or seismic applications where durability and resistance to deformation are critical.

In contrast, hollow FRC columns, while lighter and more cost-effective, exhibited reduced structural performance due to internal voids, which led to increased deformation, particularly at higher velocities. Thicker hollow FRC columns (8 mm) were able to withstand higher dynamic loads more effectively, whereas thinner sections (2 mm and 4 mm) failed under dynamic loading. The study further showed that smaller hollow FRC columns required thicker sections to remain within safe deformation limits under high-velocity conditions. Larger hollow FRC columns experienced greater deformation but could still remain within the permissible limits when appropriately designed with thicker walls.

Overall, the findings suggest that solid RC columns are the more reliable choice for applications requiring high strength and minimal deformation under dynamic or seismic loading. Their ability to maintain structural stability across a range of conditions makes them suitable for critical structural applications. On the other hand, hollow FRC columns offer advantages in terms of cost efficiency, material savings, and weight reduction, but their performance may be compromised under extreme dynamic loads. Therefore, hollow FRC columns may be appropriate for less demanding applications where the dynamic load is moderate and material efficiency is a priority. This study underscores the importance of selecting the right column type based on specific performance requirements and loading conditions.

REFERENCES

[1] T. Menouillard, A. D. Tzenkov & M.V. Schwager (2020) "Dynamic Analysis of Pine Flat Concrete Dam: Acoustic Fluid-Structural Interaction with ANSYS Workbench", Lecture Notes in Civil Engineering, vol. 91, pp 161–174, DOI: 10.1007/978-3-030-51085-5_6.

[2] Rafea F. Hassan, Muna H. Jaber, Nabeel H. Al-Salim, Husam H. Hussein (2020), "Experimental research on torsional strength of synthetic/steel fiber-reinforced hollow concrete beam", Engineering Structures, vol. 220, Doi.org/10.1016/j.engstruct.2020.110948.

[3] Muhammed zain kangda, (2021), "An Approach to Finite Element Modeling of Liquid Storage Tanks in ANSYS: A Review", Innovative Infrastructure Solutions, Volume 6, https://doi.org/10.1007/s41062-021-00589-8.

[4] S. Vinoth, S. Nandhini & K. Pavithra (2021)"Relative Study on Concrete-Filled Square and Circular Steel Tubular Columns Using ANSYS for Mathematical Analysis," International Journal of Civil Structures, vol. 42, pp. 72–88. https://doi.org/10.1016/j.matpr.2021.05.007.

[5] K. Menon et al. (2022)"Jacketing with Steel Angle Sections and Wide Battens of RC Column and its Influence on Blast Performance," Advanced Structural Design Journal, vol. 78, pp. 45–62. https://doi.org/10.1007/s42107-022-00437-9.

[6] Janhavi Singh and Shilpa Pal, (2022), "Analysis of Blended Concrete Cubes under Impact loading using ANSYS", IOP Conference Series: Earth and Environmental Science, Doi:10.1088/1755-1315/1084/1/012067.

DOI : 10.35291/2454-9150.2024.0422

[7] V. Anuradha, T. Ch. Madhavi (2022), “Behaviour of self-compacting concrete hybrid fiber reinforced hollow beams”. Structures, vol. 35, Pages 990-1001, doi.org/10.1016/j.istruc.2021.11.058.

[8] Arya Sajith & Shilpa Pal, (2022), “Finite Element Analysis and Parametric Study of Concrete Beams Under Impact Loading”, Lecture Notes in Civil Engineering, vol.284, 2366-2565, Doi.org/10.1007/978-3-031-12011-4_20.

[9] Mohamed Khalifa Bneni and Samir Hasuna Ashour, (2023), “Modelling and simulation of flexural behavior for reinforced concrete beams using ANSYS”, Al-Mukhtar Journal of Engineering Research, 07 (1): 43-56, DOI:10.54172/mx8n6k70.

[10] Liwen Zhang, Junjie Zhang, Guangming Chen, Guan Lin, (2023), “Flexural behavior of hybrid FRP-recycled aggregate concrete-steel hollow beams”, Journal of Constructional Steel Research, vol 200, doi.org/10.1016/j.jcsr.2022.107650.

[11] A. Patel et al. (2023)"Dynamic Response of Concrete Subjected to High Rate of Loading: A Parametric Study," Concrete Research Bulletin, vol. 85, pp. 24–38.

[12] A. Asif, M. Dhanapal, M. U.S., S. Nazar & S. R. Jose (2023)"Analysis of Steel–Concrete Composite Beam Using ANSYS 18.1 Workbench," Structural Mechanics Quarterly, vol. 58, pp. 33–49. https://doi.org/10.1016/j.matpr.2023.05.285.

[13] P. Van-Phuc, P. Ngoc-Minh, N. T. Dieu-Thuy & C. T.-Hao (2023)"Predicting the Critical Load of Rectangular Concrete-Filled Steel Tube Columns with Ultra-High Strength Concrete Using ANSYS," Finite Elements in Structural Design, vol. 92, pp. 198–212. https://doi.org/10.1016/B978-0-323-95682-6.00005-X.

[14] Mohamed et al. (2023)" Modelling and Simulation of Flexural Behaviour for Reinforced Concrete Beams Using ANSYS," International Journal of Civil Engineering Research, vol. 64, pp. 112–126.

[15 Surabhi Saxena, Mahesh Patel (2023), “Evaluating dynamic behaviour of a concrete dam using modal analysis”, Materials Today: Proceedings, 93(Part 3), pages 296-301, Doi.org/10.1016/j.matpr.2023.07.259

[16] Kong Fah Tee & Sayedali Mostofizadeh (2024), “Soil–pipe interaction and structural response under static and seismic loading for geopolymer concrete pipes”, Innovative Infrastructure Solutions, Volume 9, article number 49, https://doi.org/10.1007/s41062-023-01357-6.

[17] Abhishek Minhas and Seema (2024), “Numerical simulation of projectile impact on reinforced concrete structures: a study of slab performance under varying projectile velocities using ANSYS”, Innovative Infrastructure Solutions, 9, 372. https://doi.org/10.1007/s41062-024-01671-7.

[18] Jian Liu et al. (2024)"Dynamic Behavior of Ultra-High Performance Concrete Beams with Rectangular Openings Subjected to Impact Loads," Journal of Impact Engineering, vol. 72, pp. 56–72. https://doi.org/10.1016/j.istruc.2024.107362.

[19] Marzec, J. Suchorzewski and J. Bobiński (2024), “Three dimensional simulations of FRC beams and panels with explicit definition of fibres-concrete interaction, Engineering Structures 319 (2024), https://doi.org/10.1016/j.engstruct.2024.118856

[20] S. M. Mubin, Agusril Syamsir, Daud Mohamad, A.B.M Supian, Muhammad Najeeb, Z. Itam, and Vivi Anggraini (2024), “The Numerical and Experimental Analysis of the Utilization of Used Glass Fibre Reinforced Polymer (Gfrp) Strip on Reinforced Concrete (Rc) Slab Exposed to Impact Force”, http://dx.doi.org/10.2139/ssrn.4764134.

[21] Sibo Su, Guangda Zhang, Qiang Han, Jiexin Yu, Xianzhuo Jia and Xiuli Du (2024), “Numerical analysis of high-strength centrifugal precast RC hollow pipe columns using grouted corrugated duct connection: Confinement effect and ductility evaluation”, Engineering Structures, Volume 301, 15 February 2024, https://doi.org/10.1016/j.engstruct.2023.117326.

[22] Jian Liu, Jie Wei, Jun Li, Yu Su and Chengqing Wu (2024), “A comprehensive review of ultra-high-performance concrete (UHPC) behaviour under blast loads”, Cement and Concrete Composites, Volume 148, April 2024, 105449, https://doi.org/10.1016/j.cemconcomp.2024.105449.

[23] Ping Li, Yuhong Yan, Enming Liu, Shan Li, Yiyan Lu, and Caiwang Tai (2024), “Dynamic mechanical behavior of steel fiber reinforced-recycled aggregate concrete subjected to uniaxial loading”, Journal of Building Engineering, Volume 90, 1 August 2024, https://doi.org/10.1016/j.jobe.2024.109385.

[24] Liangming Sun, Shuguang Liu, Hanbing Zhao, Umar Muhammad, Da Chen and Wengui Li (2024), “Dynamic performance of fiber-reinforced ultra-high toughness cementitious composites: A comprehensive review from materials to structural applications”, Engineering Structures, Volume 317, 15 October 2024, https://doi.org/10.1016/j.engstruct.2024.118647.

[25] Rukiyath Amrha &Shilpha.S (2024) Explicit dynamics of hollow FRC beams “International Journal for Research in Engineering Application & Management”, Volume 10,0ctober 2024, DOI: 10.35291/2454-9150.2024.0356.