International Journal on
"Technical and Physical Problems of Engineering"
(IJTPE)
Published by International Organization of IOTPE

ISSN 2077-3528
IJTPE Journal
www.iotpe.com
ijtpe@iotpe.com

June 2024 | Issue 59 | Volume 16 | Number 2 | Pages 224-230

FINITE ELEMENT ANALYSIS OF FLEXURAL BEHAVIOR IN GLASS FIBER-REINFORCED BEAMS

A.B. Sazairov A.I. Mehdiyev Z.M. Khalilov R.S. Akbarli

Azerbaijan University of Architecture and Construction, Baku, Azerbaijan
sazairov_emirxan@mail.ru, ayd13759@gmail.com, zohrab.xalilov@hotmail.com, akbarli89@mail.ru

Abstract- Steel corrosion poses a significant challenge in reinforced concrete structures, impacting both their serviceability and durability. This issue diminishes the overall performance of the elements. Fiber Reinforced Polymers (FRPs) emerge as a promising substitute for the steel reinforcement because of their remarkable strength to corrosion, enhanced tensile resistance, pure weight, and non-magnetizing qualities. Nevertheless, studies indicate that concrete elements reinforced with Fiber Reinforced Polymers materials may indicate substantial displacement and a linear stress-strain relationship, primarily owing to lower elasticity modulus. The ANSYS Finite Element Program was used in this study to examine the flexural behavior and ductility of reinforced concrete (RC) beams with glass fiber reinforced polymer (GFRP) bars. Scaled-down versions of rectangular reinforced concrete beams, measuring 150mm×300mm and length 3 m were modeled. Every example was subjected to four-point bending tests and had the same compression reinforcement. Quantity of glass fiber reinforced polymer material constituted the primary variable in the study. The goal was to ensure ductility and minimize displacement in order to effectively represent the real behavior of these hybrid beams. The load-displacement behavior and load-carrying capability of example from finite element models were satisfactorily matched in the investigation. It was noted that an increased ratio of GFRP bars resulted in higher ultimate loading capacity and deflection, while a higher ratio of steel bars led to enhanced ductility and flexural rigidity.

Keywords: Steel Corrosion, Fiber Reinforced Polymers (FRPs), Tensile Resistance, Glass Fiber Reinforced Polymer (GFRP) Bars.

1. INTRODUCTION

Reinforced concrete is the most widely used construction type, with concrete and steel being the traditional materials constituting reinforced concrete. With innovations in the construction sector, new materials have started to be used alongside traditional ones. Additionally, challenges encountered in building materials and efforts to address these issues have led to the emergence of new material types and applications [1]. In structures such as ports, piers, bridges, tunnels, parking lots, and multi-story buildings, reinforcement corrosion is a common problem. Due to factors such as insufficient protection against corrosion, spalling of reinforcement due to freeze-thaw effects, and exposure to environmental elements like seawater, corrosion of reinforcement occurs. Corrosion not only shortens the service life of reinforced concrete structures but also poses a threat to structural safety, often requiring frequent repairs.

To address these issues, there is a growing need to explore alternative materials for reinforced concrete, and Fiber Reinforced Polymers (FRPs) are being considered due to their enhanced tensile resistance, resistance, pure weight, and non-magnetizing qualities. In recent years, the use of FRPs in the construction sector has rapidly increased. Commonly used composite materials include glass, carbon, steel and aramid fibers, which are preferred for reinforcement, profiles, and strengthening materials. Despite the superior properties of FRP reinforcement, it is known to exhibit brittle fracture behavior and has a lower elastic modulus compared to steel.

In structures using FRP reinforcement, it is anticipated that larger crack widths, increased deformations, and higher flexural strength can be achieved compared to traditional reinforced concrete [7]. However, the brittle nature of FRP reinforcement often makes the serviceability limit state the determining parameter. Current applications of FRP reinforcement in reinforced concrete elements typically exhibit failure modes dominated by deflection, concrete crushing, or FRP rupture. The deflection modes are generally brittle, contrasting with the way under-reinforced steel beams in reinforced concrete behave [2].

Several researchers have investigated the use of FRP as reinforcement. One study found that beams reinforced with FRP exhibited three times more deflection than their steel-reinforced counterparts at ultimate load [3]. Another study observed that FRP reinforcement reached its tensile capacity without yielding, leading to unwanted sudden deflection in beams, indicative of brittle failure [4]. Considering the needs of the construction sector, the combination of FRP and steel reinforcement is currently being explored for potentially better results [9].

Registered: 231029 Article Info: 1801 Accepted: 240525
Reserved: 240219 Available: 240630

Glass fiber reinforcement produced using the pultrusion method stands out due to its superior mechanical strength, lightweight, strength to corrosion, minimal density, high ratio of strength to weight, minimal heat conductivity, long-term maintenance-free characteristics, ease of production with low labor, and ease of cutting and processing [5]. In light of these features, glass fiber profiles are rapidly becoming an alternative for many materials in the construction sector. However, the brittle fracture behavior and excessive deformations of glass fiber reinforcements are significant drawbacks. The numerical results obtained that the finite element model's numerical output and the experimental data agreed rather well [6].

2. MATERIALS AND METHODS OF RESEARCH

FEM is a numerical technique developed for the acceptable resolution of various engineering problems. Analytical solutions for many engineering problems are challenging due to the complexity of the problems, material properties, and boundary conditions. Therefore, numerical and experimental methods are commonly employed to solve these problems. The finite element method involves modeling a structure or structural element by dividing it into finite elements. It is widely used in various branches of engineering, including mechanics, fluid mechanics [10], heat transfer [18], electromagnetism, stress analysis for linear and nonlinear situations, deformation analysis, and applications in aerospace and automotive industries [8, 9].

2.1. Key Terms Used in the Finite Element Method (FEM)

- Element: Elements are simple geometrically shaped pieces into which a continuous medium is discretized.
- Node Point: Node points are the intersection points of the elements. These points are the physical locations where the unknowns of the field problem are numerically solved, reducing the infinite unknowns to a finite number.
- Degrees of Freedom: It is defined as the number of components of the displacement vector that an element can undergo at a node point. The sum of all freedoms of an element at its node points is defined as the degree of freedom of that element. The number of freedoms at node points varies according to the dimension of the problem. In two-dimensional elements, there are three freedoms at each node point, consisting of horizontal and vertical displacements and rotation about the plane. In three-dimensional elements, there are a total of six freedoms at each node point, including horizontal, vertical, and perpendicular displacements, as well as rotations in the horizontal, vertical, and perpendicular planes. The calculation steps in solving a problem using the finite element method can be outlined as follows:
- Meshing: In the first step of the FEM, the continuous medium is consisting of finite elements. Appropriate finite elements should be used, and their type, number, and arrangement must be determined. To obtain accurate analysis results, it is crucial to use as many elements as possible during meshing. However, increasing mesh density beyond a certain value does not significantly impact the results. The element division process should be carefully performed for result accuracy, and meshing may need to be repeated when necessary.
- Global System Setup: After the system is divided into elements, the global axes of the system, degrees of freedom (DOF) of nodes, and boundary conditions are established. Stiffness matrices for each element are calculated in their local coordinate systems.
- Transformation to Global Coordinates: The local coordinates of the elements' stiffness matrices are transformed to the global coordinate system. The global stiffness matrix of the system is formed by assembling the stiffness matrices in the global coordinate system, taking into account the elements linked to each node.
- Loading and Solution: Loads on the elements are transferred to node points, creating the global load vector {F}. Considering the boundary conditions of the continuous medium, the displacements are calculated by [15]. After calculating displacements, stress and strain can be determined. The calculated displacements are specific to the node points. Due to finite element method the acting force vector [17]:

$$\{F\} = [k]\{d\}$$

where, $\{F\}$ shows the acting force vector, $[k]$ is express matrix of stiffness and $\{d\}$ is a vector of displacement

2.1.1. Types of Elements Used in FEM

In FEM various types of elements are used according to the geometric structure of the solution region. These elements are divided into two main categories: continuous elements (solid, two-dimensional surface elements) and structural elements (beams, columns). Depending on their dimensions, they can be classified as one-dimensional, two-dimensional, three-dimensional, rotational, and isoperimetric elements (equal-parameter elements) [8].

- One-Dimensional Elements: These elements are used to solve one-dimensional problems.
- Two-Dimensional Elements: These are elements used to solve two-dimensional problems. The fundamental element in this category is the three-node triangular element. The number of nodes for a triangular element can be six, nine, or more depending on the degree of the chosen interpolation function. Quadrilateral elements are formed by combining two triangular elements.
- Rotational Elements: These elements are used to solve problems exhibiting axial symmetric properties. Although these elements essentially have three dimensions, they are very useful as they can be solved like two-dimensional problems by making a full rotation around the symmetry axis.
- Three-Dimensional Elements: The most basic element in this category is the triangular pyramid. Additionally, rectangular prisms and more generally hexahedral elements can be used to solve three-dimensional problems.
- Isoperimetric Finite Elements: These elements are used to solve problems defined by regions with curved boundaries. Each node point on these elements is defined

by a function. The characteristic feature of these elements is that every point's position and displacement can be defined by the same order of the same shape (interpolation) function.

2.1.2. Advantages of the FEM

The following is a list of the benefits of the FEM:

- That permits for the nonlinear analysis of reinforced concrete structures, providing a more realistic understanding of structural behavior.
- It enables the analysis of complex reinforced concrete systems by introducing flexibility in their shape and dimensions.
- In comparison to prototype experiments, finite element analyses are cost-effective and time-saving.
- Simulating design projects in a computer environment during the design phase can provide insights into potential future issues.
- It allows for the application of complex material properties, and the time-dependent properties of materials can be considered.

2.1.3. Disadvantages of the FEM

The disadvantages of the FEM include:

- ➢ Significant discrepancies from real results can occur if elements suitable for the material and geometry are not selected.
- ➢ Care should be taken regarding the accuracy of results obtained with this method. Insufficient mesh density can lead to results that differ significantly from the outcomes of the experiment.
- ➢ The program's effectiveness is dependent upon the availability of good hardware, and attention should be given to ensuring that the computer's capacity is not insufficient for the analysis.

2.2. ANSYS Finite Element Software

When creating the geometry of the structural elements to be analyzed, it is essential to consider the dimension in which the element will be modeled. In two-dimensional analyses where points, lines, and areas can be formed, the model's geometry can be created by connecting lines from points or creating areas directly from sub-menus. Three-dimensional model geometry is defined either by specifying the depth of the areas created after forming areas or by creating directly from sub-menus. In this study, a beam model was created by assembling various volumes, taking into account concrete cover. Additionally, when creating the model, the rebars were assumed to be distributed within the concrete, paying attention to their positions.

For the solution of problems using numerical methods, entering material properties in a way that closely approximates reality is crucial for the accuracy of the solution. In this study, properties of concrete, cover, longitudinal steel, and GFRP reinforcement were individually defined Figure 1 [11]. In the ANSYS program, the element library includes more than 150 types of elements, such as beams, rods, shells, plates, and contact elements.

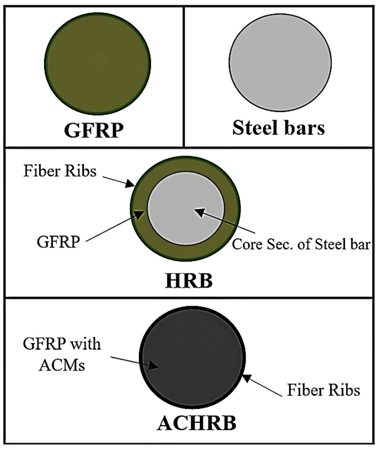

Figure 1. GFRP Reinforcement

The Solid65 element is available for modeling concrete material, and it has been used in all studies in the literature for modeling concrete. Steel and glass fiber reinforced polymer (GFRP) reinforcements are modeled using the Link180 element. The Solid65 element is an 8-nodal element with three degrees of freedom in each direction at every nodal point used for three-dimensional modeling of concrete. It can model concrete both with and without reinforcement, and its most important feature is having nonlinear material behavior.

This enables modeling of cracking, crushing, plastic deformation, and yielding behaviors of concrete in three orthogonal directions, as well as the behaviors of reinforcement under compression and tension, plastic deformation, and yielding behavior [15, 16]. Meshing, or creating a grid, refers to the process of dividing a physical domain into smaller subdomains. The goal is to simplify the solution of differential equations. In the finite element method, structural elements are divided into smaller elements according to the geometry of the element. Maintaining high mesh density is crucial for the accuracy of the results during meshing. Attention should be paid to not having a significant difference in the sizes of elements during the meshing process.

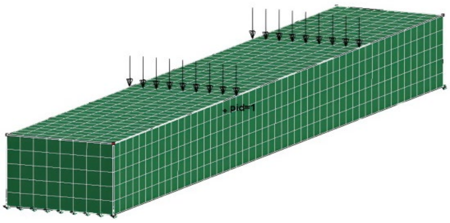

Figure 2. Modeling the loading on the beam

In the numerical study, the beams were prepared with one end fixed and the other end sliding (simple) support. These boundary conditions were applied as they are during the modeling of the beam. In modeling, loads were applied to the beam not from a single point but in the form of singular loads from multiple points. This prevented the occurrence of cracks and local displacements due to increased local stresses in the regions where the load was applied Figure 2.

2.2.1. Introduction of Beams

This work encompassed the modeling of six beams in total. The test specimens consist of 6 reinforced concrete beams with measurements of 150mm×300mm and 3200 mm of length. In all elements, $2\phi10$ steel reinforcement was used as compression reinforcement. As stirrup reinforcement in all elements, plain steel reinforcements with $\phi5/75\,\text{mm}$ spacing were used. For tensile reinforcement, 5 bars were used in each element, with a diameter of 10mm for steel reinforcement and 13 mm for glass fiber reinforcement (Table 1). The parameters used for variable steel and glass fiber reinforcements in the tensile zone of the test specimens are the number and arrangement. The same concrete class was used for all test specimens. The concrete class is C30 (30 MPa).

Table 1. Tensile strength values of reinforcements used in test specimens

Material	Yield strength (f_{yk})	Ultimate tensile strength (f_u)
5 mm diameter plain steel reinforcement	220 MPa	340 MPa
10 mm diameter deformed steel reinforcement	420 MPa	500 MPa
13 mm diameter glass fiber reinforcement	----------	350 MPa

Table 2. Properties of test specimens

Element Name,	Number of Steel Reinforcements Used in Tensile Zone	Number of Glass Fiber Reinforcements Used in Tensile Zone	Percentage of Glass Fiber Reinforcement Used in Tensile Zone
C1	5	0	% 0
C2	4	1	% 20
C3	3	2	% 40
C4	2	3	% 60
C5	1	4	% 80
C6	0	5	% 100

The test specimens are denoted by symbols in the form of "letter-number-letter-number" in a consecutive manner. The first capital letter represents the steel reinforcements used in the tensile zone of the element, and the following number indicates the number of steel reinforcements. Next, the second capital letter represents the GFRP reinforcements used in the tensile zone of the element, and the following number indicates the number of GFRP reinforcements. The names of the test specimens are explained below and properties of test specimens presents in Table 2.

- C1 specimen: An element with 5 steel tensile reinforcements and no GFRP tensile reinforcement.
- C2 specimen: An element with 4 steel tensile reinforcements and 1 GFRP tensile reinforcement.
- C3 specimen: An element with 3 steel tensile reinforcements and 2 GFRP tensile reinforcements.
- C4 specimen: An element with 2 steel tensile reinforcements and 3 GFRP tensile reinforcements.
- C5 specimen: An element with 1 steel tensile reinforcement and 4 GFRP tensile reinforcements.
- C6 specimen: An element with no steel tensile reinforcement and 5 GFRP tensile reinforcements.

2.3. Material Properties and Behaviors

The test specimens' characteristic compressive strength on average is 30MPa. Reinforcement consists of 5 mm smooth steel bars, 10mm ribbed steel bars, and approximately 13mm ribbed glass fiber bars.

➢ Mechanical Properties of Concrete: Concrete is a composite building material obtained by mixing cement, water, and aggregates in specific proportions. It is a non-linear material whose properties can change with time and load. Concrete exhibits linear behavior up to a certain load level, beyond which it shows non-linear behavior. Micro-cracks develop and propagate into macro-cracks, resulting in a decrease in strength and load-bearing capacity. Therefore, assuming linearity up to a certain load level and non-linearity beyond that level is crucial for the accuracy of results when modeling concrete. Concrete's important mechanical properties include compressive and tensile strength, Young's modulus and Poisson's ratio.

➢ Compressive Resistance of Concrete; Compressive resistance is a crucial mechanical property of concrete, and the concrete class is determined based on it. The compressive strength of concrete is determined through standard cylinder or cube tests. After curing for 28 days to gain strength, the samples are subjected to axial compression using presses.

➢ Tensile Strength of Concrete; The tensile strength of concrete generally varies depending on the quality of the concrete and is approximately between 7% and 17% of the compressive strength. Tensile strength is determined through direct tension, splitting tension, and flexural tension tests.

2.3.1. Properties of Steel Reinforcement

Steel reinforcements are preferred in reinforced concrete structures due to their high tensile strength. In the initial part of the stress-strain curve, steel exhibits linear-elastic behavior in Figure 3 [12].

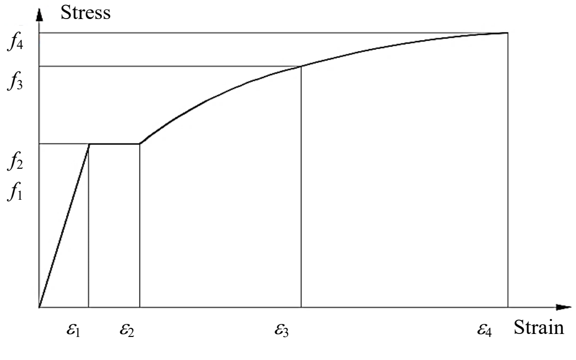

Figure 3. Stress-strain curve for steel reinforcement

After reaching the yield point, stress in steel remains constant while strain increases. The steel undergoes strain hardening, and after reaching the ultimate strain, rupture occurs [14]

2.3.2. Properties of GFRP Reinforcement:

GFRP reinforcement is a high-tensile, corrosion-resistant, lightweight, and insulating anisotropic material.

It exhibits linear stress-strain behavior until failure. In the study, GFRP reinforcement is considered a linear elastic material and modeled using the Link180 element.

2.3.3. Assumptions Made During Modeling

Briefly, the assumptions made during finite element modeling of beams are as follows:

- Solid65 elements are used for concrete, while Link180 elements are used for reinforcements.
- Concrete is modeled considering elastic, plastic, and collapse behaviors. Additionally, no crushing in concrete is assumed.
- Steel reinforcements are modeled considering yield stress, modulus of elasticity, and Poisson's ratio.
- Glass fiber reinforcements are modeled as linear-elastic.
- Full bonding is assumed between concrete-reinforcement and concrete-glass fiber reinforcement.

3. RESULTS AND DISCUSSION

Numerical load-deflection graphs for each element from Figure 4 is provided.

❖ Element C1: In the numerical study, the first crack load occurs at approximately 18.5 kN.

❖ Element C2: In the numerical study, the first crack load is approximately 16 kN.

❖ Element C3: In the numerical study, the first crack load is approximately 17 kN.

❖ Element C4: In the numerical study, the first crack load occurs at approximately 15.7 kN.

❖ Element C5: In the numerical study, the first crack load occurs at approximately 17 kN.

❖ Element C6: In the numerical study, the first crack load occurs at approximately 15.6 kN.

Table 3 presents the final load values derived from numerical research and analysis.

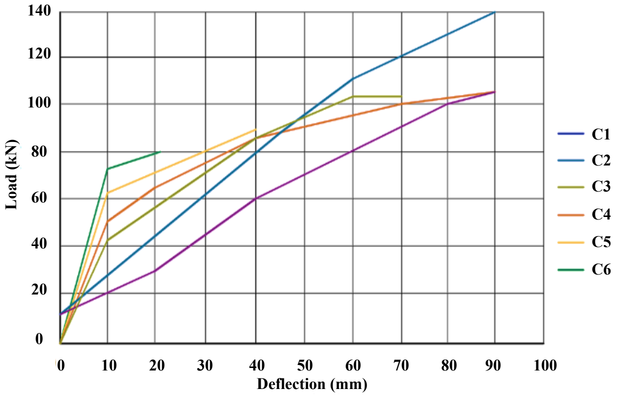

Figure 4. Numerical load-deflection graphs for all elements

Table 3. Ultimate load marks of numerical model

Element	P_{ud} (kN)
C1	83.67
C2	87.53
C3	98.25
C4	103.1
C5	123.28
C6	111.1

Loadings were applied in 40 steps in numerical studies based on the ultimate load values obtained from experimental results. However, during the analysis, it was determined that the beams reached their ultimate load capacities and yielded between the 35th and 45th load steps. When all analysis results are compared with experimental results, discrepancies in ultimate load values ranging from 1.8% to12.5% are obtained. These deviation percentages in ultimate loads are considered acceptable for the finite element method. The ultimate load values obtained from experimental studies and analyses are given in Table 3. When analyses are compared with experimental results in terms of displacements, it is observed that the difference exceeds acceptable limits in some beams, similar to the ultimate loads. This difference is thought to be due to the effect of cracks in the concrete, which is not fully reflected in ANSYS's crack model for concrete.

4. CONCLUSIONS

In reinforced concrete elements exposed to environmental conditions, corrosion over time can lead to section loss in steel reinforcement. This section loss diminishes the load-carrying capacity of the structural element. The aim of this study is to investigate the effects of Glass Fiber Reinforced Polymer (GFRP) reinforcement used as tensile reinforcement in beams, in conjunction with steel reinforcement, to mitigate the adverse effects of corrosion on reinforced concrete elements. In the scope of this work, a previous study was modeled using the ANSYS computer program, and the ratios and arrangements of steel and GFRP reinforcement used as tensile reinforcement were examined as parameters. The load-deformation curves obtained from the analyses were compared with experimental results.

During numerical modeling, complete bond between concrete and reinforcement was assumed, although this bond was not perfect in the experimental study. Additionally, microcracks that occurred during the experiments were disregarded during the analyses. Due to these two reasons, it was observed that, when looking at the load-deformation graphs, the flexural rigidities obtained from the analyses were larger than those obtained from the experimental results. A decrease in slopes was observed in the load-deformation graphs for all elements after the development of the first crack. Generally, an increase in the GFRP reinforcement ratio led to a tendency of decreasing flexural rigidity, while an increase in the steel reinforcement ratio led to a tendency of increasing flexural rigidity in beams. In terms of ultimate load values, the analysis results showed a maximum deviation of 12.5% from the experimental results, which is within acceptable limits.

Generally, an increase in the GFRP reinforcement ratio led to an increasing trend in ultimate load values, while an increase in the steel reinforcement ratio led to a decreasing trend in ultimate load values. When the results obtained from the analyses were compared with the experimental results in terms of deformation values, it was observed that the difference exceeded acceptable limits for some beams. This discrepancy is thought to originate from the inability of ANSYS to fully reflect the behavior of cracks in concrete in the crack model it assumes. Moreover, generally in beams, an increase in the GFRP reinforcement ratio resulted in an increase in deformation values, while an increase in the steel reinforcement ratio resulted in a decrease in deformation values. This is believed to be a result of the significantly lower elasticity modulus of GFRP reinforcement compared to steel reinforcement.

With the increase of applied load, microcracks in the concrete transform into macrocracks. This leads to an increase in nonlinear behavior in concrete and a decrease in load-carrying capacity. When the analysis results were compared with the experimental results, it was observed that ANSYS predicted the load-deformation behavior, collapse load, and displacement values for beams C2, C3, C4, and C5 in accordance with the experimental results. However, ANSYS failed to capture the agreement in displacement values while accurately predicting collapse loads for beams C1 and C6. Upon examining the load-deformation graphs, it was found that an increase in the GFRP reinforcement ratio led to a softening in beam behavior and an approach of the load-deformation curve to linearity. Conversely, a decrease in this ratio resulted in stiffening of beam behavior.

REFERENCES

[1] F.A. Aydın, "Investigation of the Flexural Behavior of GFRP-Concrete-Steel Hybrid Beams", International Construction Congress, Isparta, Turkey, 11-13 October 2012.

[2] E.G. Nawy, G.E. Neuwerth, "Behavior of Fiber Glass Reinforced Concrete Beams", Journal of the Structural Division, ASCE, pp. 2203-2215, September 1971.

[3] M.S. Ozel, "Behavior of Concrete Beams Reinforced with 3-D Fiber Reinforced Plastic Grids", Ph.D. Thesis, University of Wisconsin, Madison, USA, 2002.

[4] M.A. Yaz, "The Effect of Glass Fiber Reinforcement on the Flexural Behavior of Reinforced Concrete Beams", Master's Thesis, Kirikkale University, Ankara, Turkey, 2014.

[5] R.A. Hawileh, "Finite Element Modeling of Reinforced Concrete Beams with a Hybrid Combination of Steel and Aramid Reinforcement", Materials and Design, Vol. 65, pp. 831-839, January 2015.

[6] M.S. Aiello, L.A. Ombres, "Structural Performances of Concrete Beams with Hybrid (Fiber-Reinforced Polymer-Steel) Reinforcements", Journal of Composites for Construction, ASCE, Vol. 6, No. 2, pp. 133-140, 2005.

[7] T.R. Chandrupatla, A.D. Belegundu, "Introduction to Finite Elements in Engineering", Prentice-Hall International, pp. 1-2, USA, 1997.

[8] X.Z. Lu, J.J. Jiang, J. G. Teng, L.P. Ye, "Finite Element Simulation of Debonding in FRP-to-Concrete Bonded Joints", Construction and Building Materials, Vol. 20, Issue 6, pp. 412-424, 2006.

[9] D.U. Choi, S.C. Chun, S.S. Ha, "Bond Strength of Glass Fiber-Reinforced Polymer Bars in Unconfined Concrete", Engineering Structures, Vol. 34, pp. 303-313, January 2012.

[10] R.S. Akbarli, "Features of the Behavior of Gas-Liquid Mixture Tubes", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 43, Vol. 12, No. 2, pp. 71-77, June 2020.

[11] C.K. Mias, L.A. Torres, A.L. Turon, C.H. Barris, "Experimental Study of Immediate and Time-Dependent Deflections of GFRP Reinforced Concrete Beams", Composite Structures, Vol. 96, pp. 279-285, February 2013.

[12] S.H. Alsayed, "Flexural Behavior of Concrete Beams Reinforced with GFRP Bars", Cement and Concrete Composites, Vol. 20, Issue 1, pp. 1-11, 1998.

[13] E.G. Nawy, G.E. Neuwerth, C.J. Phillips, "Behavior of Fiber Glass Reinforced Concrete Beams", Journal of the Structural Engineering Division, ASCE, 97 ST9, pp. 2203-2215, 1971.

[14] K.V. Nakano, Y.S. Matsuzaki, H.K. Fukuyama, M.C. Teshigawara, "Flexural Performance of Concrete Beams Reinforced with Continuous Fiber Bars", In ACI Int. Symposium 1993, Fiber Reinforcement for Concrete Structures, pp. 743-751, Detroit, Michigan, USA, 2003.

[15] R.A. Iskanderov, J.M. Tabatabaei, "Vibrations of Heterogeneous, Fluid-Filled Cylindrical Shells Stiffened by Longitudinal Ribs", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 43, Vol. 12, No. 2, pp. 11-15, June 2020.

[16] A.P. Dzyuba, R.A. Iskanderov, Y.M. Selivanov, "Models and Technologies of Experimental Studies of Properties of Inhomogeneous Power Structural Elements with Optimal Parameters", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 55, Vol. 15, No. 2, pp. 263-273, June 2023.

[17] T.R. Chandrupatla, A.D. Belegundu, "Introduction to Finite Elements in Engineering", Prentice-Hall International, No. 1-2, USA, 1997.

[18] Y.M. Sevdimaliyev, G.M. Salmanova, R.S. Akbarli, N.B. Nagieva, "Propagation of Waves in a Hydro Elastic Shell-Viscous Liquid System, in the Presence of Gas Bubbles", Mathematical Methods in Applied Sciences, Vol. 46, Issue 12, pp. 12176-12189, August 2023.

BIOGRAPHIES

Name: **Amirkhan**
Middle Name: **Bahsat**
Surname: **Sazairov**
Birthday: 26.07.1956
Birthplace: Tashkent, Uzbekistan
Master: Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1978
Doctorate: Ph.D., Department of Mechanics, Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1983
The Last Scientific Position: Assoc. Prof., Department of Mechanics, Azerbaijan University of Architecture and Construction, Baku, Azerbaijan, Since 1996
Research Interests: Applied Mathematics, Mechanics of Deformable Solids
Scientific Publications: 40 Papers, 5 Books

Name: **Aydin**
Middle Name: **Imran**
Surname: **Mehdiyev**
Birthday: 13.07.1959
Birthplace: Baku, Azerbaijan
Master: Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1979
Doctorate: Ph.D., Azerbaijan Civil Engineering University, Department of Mechanics, Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1988
The Last Scientific Position: Prof., Department of Mechanics, Azerbaijan University of Architecture and Construction, Baku, Azerbaijan, Since 1999
Research Interests: Applied Mathematics, Mechanics of Deformable Solids
Scientific Publications: 23 Papers, 1 Book

Name: **Zohrab**
Middle Name: **Musurza**
Surname: **Khalilov**
Birthday: 22.03.1960
Birthplace: Kafan, Armenia
Master: Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1982
Doctorate: Ph.D., Department of Mechanics, Azerbaijan Civil Engineering University, Baku, Azerbaijan, 1984
The Last Scientific Position: Assoc. Prof., Department of Mechanics, Azerbaijan University of Architecture and Construction, Baku, Azerbaijan, Since 2005
Research Interests: Applied Mathematics, Mechanics of Deformable Solids
Scientific Publications: 25 Papers, 3 Books

Name: **Reyhan**
Middle Name: **Sayyad**
Surname: **Akbarli**
Birthday: 18.07.1989
Birthplace: Neftchala, Azerbaijan
Bachelor: Mechanics, Faculty of Mathematics-Mechanics, Baku State University, Baku, Azerbaijan, 2010
Master: Fluid Mechanics, Faculty of Mathematics-Mechanics, Baku State University, Baku, Azerbaijan, 2012
Doctorate: Ph.D., Faculty of Mathematics-Mechanics, Baku State University, Baku, Azerbaijan, 2022
The Last Scientific Position: Senior Lecturer, Department Mechanics, Azerbaijan University of Architecture and Construction, Baku, Azerbaijan, Since 2012
Research Interests: Theoretical Mechanics, Hydromechanics, Solid State Mechanics and Mechanics of Composite Materials
Scientific Publications: 18 Papers, 1 Book, 25 Theses