Sustainability, Agri, Food and Environmental Research, (ISSN:0719-3726), 12(X), 2024:
http://dx.doi.org/

Numerical analysis of GFRP rebar embedded concrete beam.

Análisis numérico de vigas de concreto con refuerzo de barras de GFRP incrustadas.

Sajith S[1],Smitha K K[2]

[1]MTech Dept. of Civil Engineering, Toc H Institute of Science & Technology, Arakkunnam, Kochi, Kerala India.

[2]Assoc. Prof Dept. of Civil Engineering,Toc H Institute of Science & Technology, Arakkunnam, Kochi, Kerala, India

Correspondingauthor'semail:sajithssanal@gmail.com(Sajith S)

ABSTRACT

The use of Glass Fiber-Reinforced Polymer (GFRP) in construction has several sustainability benefits that make it an attractive alternative to traditional reinforcement materials. One of the main sustainability problems faced by steel is its carbon footprint, 1.89 tons of CO_2 is being released into the atmosphere for every ton of steel produced. Another well-known problem faced by steel in construction is corrosion which is the leading cause of failure of a building. the use of GFRP in construction offers several sustainability benefits that make it an attractive alternative for use in a variety of applications. The combination of improved performance, reduced waste, and lower carbon footprint make GFRP a sustainable choice for the construction industry. This paper presents a numerical analysis of a concrete beam reinforced with Glass Fiber-Reinforced Polymer (GFRP) rebar. The study aims to investigate the mechanical behaviours and performance of concrete beams embedded with GFRP rebar under 2-point loading conditions. Ansys 2023 software was utilized for the modelling and analysis of the developed model. Two models were created in the Ansys workbench, one with GFRP rebar as reinforcement and other with steel rebar reinforcement. The numerical results were then analyzed to determine the load-carrying capacity and deformation behaviors of the GFRP-reinforced concrete beams compared to that of regular steel reinforced beam.

Keywords: Finite element analysis, GFRP rebar, mechanical behavior, simply supported beam

INTRODUCTION

Finite Element Analysis (FEA) is a crucial tool in experimental design and engineering, which utilizes mathematical models to simulate the behavior of physical systems under different conditions. FEA is particularly useful in cases where experimental testing is challenging or impossible, as it offers valuable insights into the behavior of complex systems. Ansys, a software package for FEA, is widely used in various industries such as aerospace, automotive, and electronics [1]. Ansys provides various tools for simulating and analyzing physical phenomena such as stress and strain analysis, thermal analysis, and fluid dynamics. FEA is essential in validating experimental design as it provides valuable insights into complex systems' behavior, helping to optimize, reduce costs, and minimize the risk of failure. By simulating the beam's behavior under various loading conditions, FEA enables engineers to optimize the design for maximum strength and durability [2]. One of FEA's key advantages in reinforced beam design is its ability to identify potential failure points in the structure. Analyzing the stresses and strains in different parts of the beam provides information that can be used to refine the beam's design, making it stronger and more resilient. In this paper, FEA is used to test and analyses a reinforced beam made of modified concrete embedded with GFRP and deformed steel bars. By simulating the behavior of different reinforcement materials and running the analysis under appropriate boundary conditions in a static structural simulation, the 3D model of the beam was prepared inside the Ansys software. Even though Ansys has an extensive library of engineering data for different materials, GFRP rebars are not included; hence new materials have to be created. The 3D model can be generated from 2D cross-sectional views, and the engineering data of the material can be assigned to the model to run the analysis. The reinforcement design of the simply supported beam of 0.15×0.15m cross section and 1m long was done with respect to steel rebar for purpose of comparison.

Glass fiber reinforced polymer (GFRP) rebar is a corrosion-resistant alternative to traditional steel reinforcement in concrete structures. When used in the construction of reinforced concrete beams, GFRP rebar can increase the durability and lifespan of the structure, particularly in harsh environments or where the use of traditional steel rebars may be limited. Additionally, GFRP rebar is lighter in weight, easier to handle. Overall, the use of GFRP rebar in concrete structures can lead to improved performance, reduced maintenance costs, and a longer service life.

METHODS AND ANALYSIS

Ansys software employs a five-step process for numerical analysis which includes adding engineering data, designing the geometry, assigning material data and meshing, assigning boundary conditions and generating results. The successful completion of all these steps are necessary to obtain accurate results. In this particular analysis, static structural simulation is chosen as the analysis system. Two static structural simulations are created for analyzing two different beams named "Beam Steel" and "Beam GFRP".

A. Adding Experimental Data

One of the crucial steps in conducting an analysis is to add experiment data and define material properties, which the software uses to ensure accurate simulations. Ansys provides a comprehensive library of experimental data, which can be utilized directly or modified as needed. In this analysis, three materials were used: GFRP, bearing, and concrete. GFRP material properties were obtained from the manufacturers and entered into the software to create a new material. Bearing was an imaginary material which is used for creating the simply supported condition in beam, it was created with high young’s modulus and Poisson’s ratio values, to prevent deformation and minimize errors. Concrete was already present in the software, but it was modified to reflect the compressive and tensile experimental data of metakaolin incorporated concrete. Structural steel was used without any changes from the software’s library. Table 1 outlines the various engineering data that was entered into the software.

Table 1 Engineering data

Material	Young's Modulus (MPa)	Poisson's Ratio	Density (Kg/m3)
Modified Concrete	32250	0.18	2550
GFRP Rebar	71457	0.33	1850
Bearing	1.2×10^6	0.499	3530
Steel Rebar	2×10^5	0.3	7850

B. Design of Beam Geometry

The Ansys software provides users with three different geometry engines, each of which can be utilized to create 3D models. For this analysis, the reinforced beam's 3D model was created using one of these geometry engines, called "DesignModeler." This engine is specifically designed to generate 3D models from 2D cross-sectional drawings, which makes it ideal for reinforced concrete beam design. DesignModeler makes use of a Tree-outline to generate the model, where each element is categorized under its own branch. Figure 1 illustrates the 3D model of the Reinforcements, including the tree outline created using the DesignModeler.

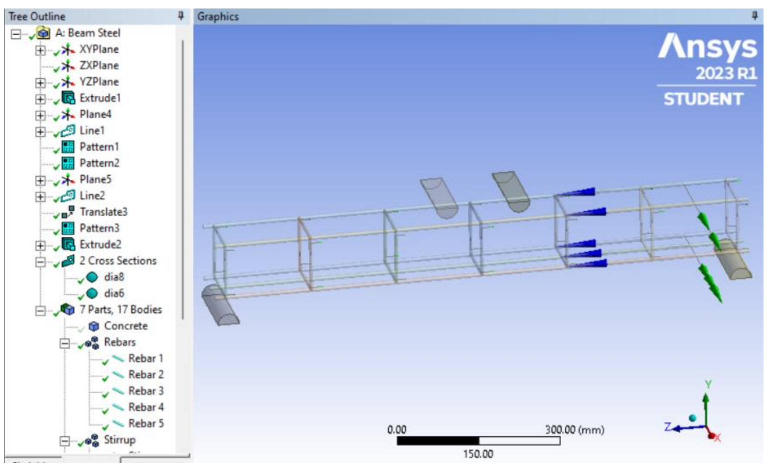


Fig. 1 3D model of Reinforcements

C. Assigning Material Data and Meshing

The subsequent step following geometry creation is to assign material properties to the model, which requires the use of the Ansys Mechanical window. It is necessary to assign each section of the geometry with the appropriate material and model type. In this case, the concrete material is assigned as a "Beam" type geometry, while the rebar is assigned as a "Reinforcement" type geometry. This aids the analysis software in comprehending that the rebars are embedded in the concrete and factoring in the bonding strength between them during calculations. The cross-sectional diameter of the bars is also assigned to the rebars.

The next step in the analysis process is mesh generation, which is also referred to as meshing or discretization. The objective of meshing is to create a mesh that precisely represents the geometry of the object under scrutiny while also enabling effective and precise numerical simulation. An ideal mesh should have a sufficient number of elements to accurately capture the solution without becoming computationally expensive [3]. In this analysis, the mesh size was set to 15 mm to comply with the software limitations, as the Ansys student version has a maximum node limit of 512k. After the mesh is created, it serves as the foundation for numerically solving the governing equations of the problem being analyzed. The solution is computed at each node and/or element of the mesh, after which the results are interpolated to provide a continuous solution throughout the entire domain. The accuracy of the solution is influenced by several factors, including the quality of the mesh. Fig 2 depicts the meshed model of the beam.

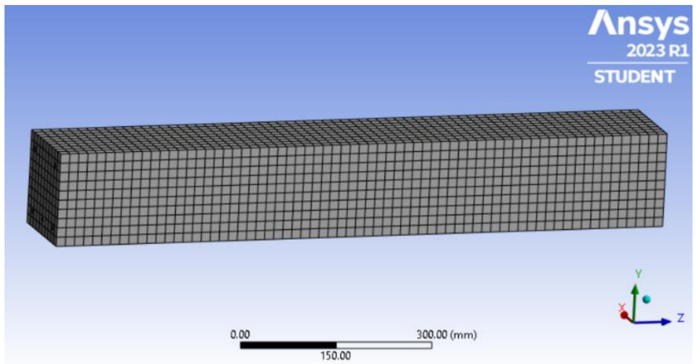


Fig 2 Meshed model

D. Assigning Boundary Conditions

Assigning boundary conditions is a crucial step in finite element analysis using ANSYS or any other FEA software. It allows for the realistic modelling of physical behaviors, accurate simulation results, optimization of system design, and the identification of potential failure points for safety and reliability. Without boundary conditions, FEA models are incomplete and do not account for the interactions between the system and its environment. Engineers use boundary conditions to define loads, constraints, and other variables that can be analyzed to simulate the response of the system under different operating conditions. Therefore, it is critical to assign appropriate boundary conditions to achieve accurate simulation and analysis of real-world systems and structures. In this analysis, there are four boundary conditions, including two load conditions and two support conditions, where one acts as a fixed-but support and the other is a roller support.

RESULTS AND DISCUSSION

Analysis was successfully done with necessary geometry, boundary conditions and material properties for both steel and GFRP rebar beam

A. Directional Deformation

The figure displays the directional deformation, which is taking place in the negative Y direction. In both the steel rebar beam and the GFRP rebar beam, the maximum deformation is indicated as "Min," with a value of 0.4258 mm and 0.4491 mm, respectively which is at the mid of the beam.

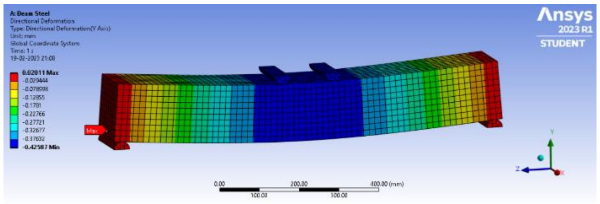


Fig 3 Deformation of steel rebar beam

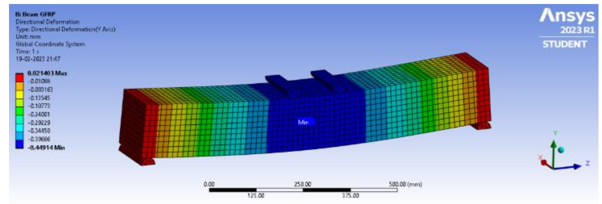


Fig 4 Deformation of GFRP rebar beam

B. Equivalent Strain in Beam

The figures illustrate the equivalent strain distribution in the beam. While the maximum equivalent strain appears to be around the supports, it may be attributed to the crushing of concrete mesh in the contact region. The actual equivalent strain values for the steel-reinforced beam and GFRP-reinforced beam were determined as 3.472×10^{-4} and 3.694×10^{-4}, respectively. Figure 5 and figure 6 shows the equivalent strain in both beams

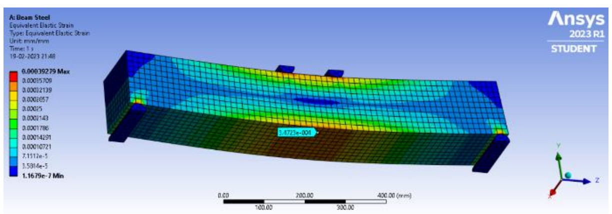


Fig 5 Equivalent strain in steel rebar beam

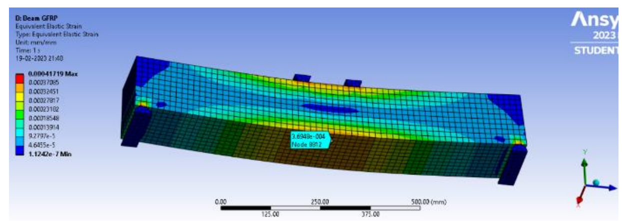


Fig 6 Equivalent strain in GFRP rebar beam

C. Equivalent Stress in Beam

The equivalent stress analysis results were obtained for both beams, revealing that the maximum equivalent stress was observed around the surface where the load was applied and in between the points where the loads were applied. The steel reinforced beam exhibited a maximum equivalent stress of 12.331 MPa, whereas the GFRP reinforced beam exhibited a maximum equivalent stress of 12.908 MPa. Figure 7 and figure 8 shows the equivalent stress in both beams.

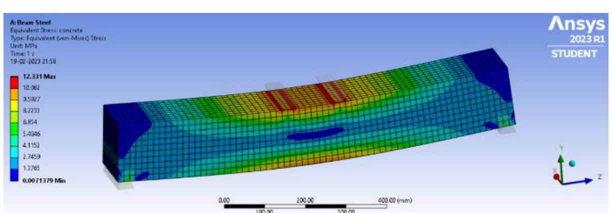


Fig 7 Equivalent stress in steel rebar beam

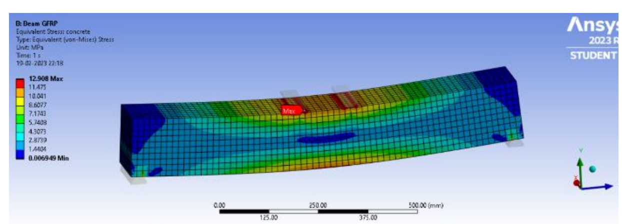
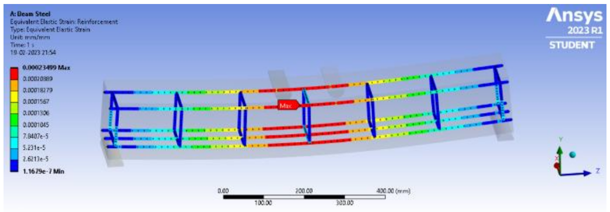


Fig 8 Equivalent stress in GFRP rebar beam

D. Strain In Reinforcement

The strain in the reinforcement was determined under the loading condition, as depicted in the figure, for both beams. The middle section of the reinforcement is uniformly subjected to high strain, although the maximum strain of 2.34×10^{-4} for the steel-reinforced beam and $2.48 \times 10^{-4}$for the GFRP-reinforced beam is observed at the midpoint between the load applied cross-section and the connection point of the main bars to the stirrup. Figure 9 and figure 10 shows the strain in the reinforcements of both beams.



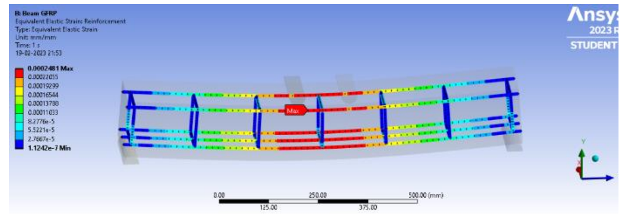
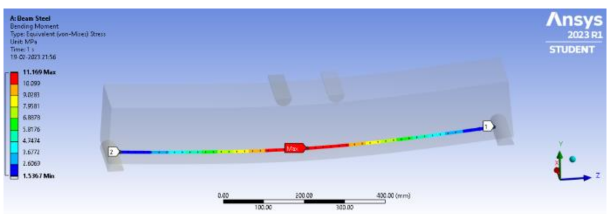


Fig 9 Strain in steel reinforcement   Fig 10 Strain in GFRP reinforcement

E. Bending Stress of Beam

The analysis also involved calculating the bending stress in the extreme fiber of the beam. However, the fiber that was in direct contact with the support underwent crushing under the load and showed very high stress values. To obtain accurate results, the reading of the extreme fiber right above the support was omitted. The maximum bending stress was obtained as 11.16 MPa for the steel reinforced beam and 11.9 MPa for the GFRP reinforced beam. Figure 11 and figure 12 shows the Bending stress in both beams.



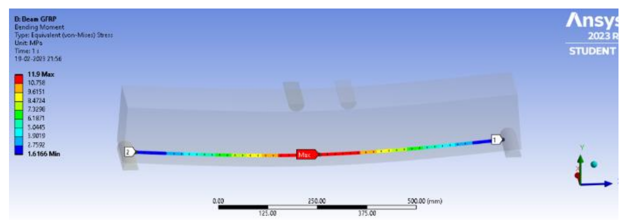


Fig 11 Bending stress in steel rebar beam   Fig 12 Bending stress in GFRP rebar beam

CONCLUSION

Based on the analysis results obtained, it is evident that GFRP reinforcement bars are not as strong as steel reinforcement bars. However, despite this difference in strength, GFRP reinforcement bars are capable of achieving comparable results when used as reinforcement.

- GFRP rebar showed about 5% more deflection compared to steel reinforcement. Lower stiffness or Young's modulus of GFRP rebar is the reason for increased deflection.
- In the case of equivalent strain and equivalent stress, GFRP reinforced concrete beams showed 6% more strain and 4% more stress compared to steel reinforced beams.
- GFRP rebars also showed 6% more strain in concrete compared to steel rebars.
- Bending stress in GFRP reinforced beams showed 6% more bending stress than steel reinforced beam
- Despite these differences, it can be concluded that GFRP reinforcement bars can perform satisfactorily when used as reinforcement bars in concrete structural elements while having no corrosion.

REFERENCES

Alhayek A., Syamsir A., Anggraini V., Muda Z.C. 2019. Numerical Modelling of Glass Fiber Reinforced Polymer

(GFRP) Cross Arm, International Journal of Recent Technology and Engineering, vol 8.

Jayajothi P., Kumutha R., Vijai, K. 2013. Finite element analysis of FRP strengthened RC beams using Ansys, Asian journal of civil engineering, vol 14, pp 631-642.

Moulika D.N., Vasireddy R., Raju P.P. 2017. Modelling and analysis of reinforced concrete beam under flexure using ansys, International Journal of Civil Engineering and Technology, vol 8, pp 1103-1111.

Mukherjee K. 2022. Numerical Simulation of Reinforced Concrete beam Under Flexure Using Finite Element Method, International Journal for Research in Applied Science & Engineering Technology,vol 10.

Nubailah A.H., Muhammad H.I., Azmi L. 2018. Finite element analysis of smart reinforced concrete beam with super elastic shape memory alloy subjected to static loading for seismic mitigation, AIP Conference Proceedings.

Sunan M., Unnikrishnan S., Menon A. 2019. Load deflection behaviour of RC beams: nonlinear analysis, International Journal of Civil Engineering and Technology, vol 10, pp 191-197.

Vasudevan G., Kothandaraman S., Azhagarsamy S. 2013. Study on non-linear flexural behavior of reinforced concrete beams using ANSYS by discrete reinforcement modeling, Strength of Materials, vol 45.

Received: 16th March 2023; Accepted: 03th August 2023; First distribution: 01th November 2023.