International Journal of Advanced Research in Engineering and Related Sciences

OPEN ACCESS

Available online at: www.ijarers.com

VOL 1 ISSUE 6 (10-24)

ABAQUS-Based Failure Mode Analysis of a High-Traffic Pedestrian Bridge: Stress, Yielding, and Corrosion Effects in Lagos, Nigeria

Isaac Enuma *[1], Uchenna Ukpai [1], Edward Idonije [1], Aisosa Edokpayi [1]

[1] Department of Civil Engineering, Faculty of Engineering, University of Benin, Benin city, Edo State, Nigeria

Abstract

The structural integrity of pedestrian bridges is crucial for ensuring public safety, particularly in high-traffic urban areas. This study presents a finite element analysis (FEA) of the Ojota Pedestrian Bridge in Lagos, Nigeria, using ABAQUS software to assess its loadcarrying capacity, material behavior, and potential failure modes under various loading conditions. The objective is to evaluate the bridge's performance under service and extreme loads and to provide insights for maintenance and safety improvements.

A 3D finite element model of the bridge was developed, incorporating accurate material properties, boundary conditions, and realistic load scenarios such as pedestrian-induced forces and environmental effects. The model simulated the interaction between structural components, allowing for detailed stress and deformation analysis. A separate corrosion analysis was conducted to simulate longterm material deterioration, assessing the impact of reinforcement loss on structural behavior. The results highlighted two primary failure mechanisms: reinforcement yielding and concrete crushing. Reinforcement bars reached their yield strength of 500 MPa, especially at mid-span and support regions, indicating potential permanent deformation under excessive loading. Similarly, compressive stress in concrete exceeded its design strength of 30 MPa, suggesting crack formation and localized crushing. The corrosion analysis revealed that a 50% reduction in reinforcement area led to excessive deflections (27.5 mm), surpassing the 22.5 mm serviceability limit and indicating structural instability. The study concludes that while the bridge remains stable under normal conditions, long-term degradation and overloading could compromise its safety, warranting reinforcement optimization, corrosion prevention, and regular monitoring.

Keywords: Finite Element Analysis (FEA), Pedestrian Bridge Safety, Structural Integrity, ABAQUS Simulation, Reinforcement Yielding, Concrete Crushing, Corrosion Impact, Urban Infrastructure, Load-Carrying Capacity, Material Deterioration

INTRODUCTION

Pedestrian bridges are essential infrastructures in urban areas, providing safe crossings over busy roadways and enhancing pedestrian mobility. In Lagos State, Nigeria, the rapid urbanization and population growth have led to increased vehicular and pedestrian traffic, necessitating the construction and maintenance of pedestrian bridges to ensure public safety. The Ojota Pedestrian Bridge, located in the Ojota district of Lagos, serves as a critical crossing point over the bustling Ikorodu Road expressway. This bridge facilitates the movement of thousands of pedestrians daily, connecting residential areas to commercial centers and transportation hubs. Given its strategic importance, ensuring the structural integrity and functionality of the Ojota Pedestrian Bridge is paramount. Recent studies have highlighted the challenges associated with pedestrian bridge usage and maintenance in Nigeria. (Olorunfemi et al., 2021) examined the utilization factors of pedestrian overpasses in Akure, Nigeria, revealing that engineering design and accessibility significantly influence user compliance and safety. Similarly, (Ngekpe et al., 2019) conducted a dynamic analysis of the Leventis Footbridge in Port Harcourt, emphasizing the need for regular structural assessments to address potential vibration issues and ensure pedestrian comfort. In Lagos, concerns about the structural integrity of pedestrian bridges have prompted government action. For instance, the Lagos State Government recently ordered the closure of the Oshodi pedestrian bridge to conduct a comprehensive structural integrity assessment, aiming to safeguard pedestrian lives and determine necessary repairs or reconstruction. Conversely, the Ojodu Berger pedestrian bridge was reaffirmed as structurally stable and fit for public use after a thorough survey showed no visible signs of damage. Finite Element Analysis (FEA) has emerged as a vital tool in evaluating the structural performance of pedestrian bridges. (Qin et al., 2014) utilized FEA to study pedestrian-bridge dynamic interactions, providing insights into how pedestrian-induced forces affect bridge stability and identifying potential failure modes. Applying such analytical techniques to the Ojota Pedestrian Bridge can offer a comprehensive understanding of its current condition and inform maintenance strategies. This study aims to assess the structural integrity of the Ojota Pedestrian Bridge using Finite Element Analysis, considering factors such as material properties, loading conditions, and environmental influences. The findings will contribute to ensuring the safety and longevity of pedestrian infrastructures in Lagos and similar urban settings.

MATERIALS AND METHOD

A. Study Area

The study was conducted in Ojota, a highly populated and commercialized area in Lagos State, Nigeria. Ojota is located in the northwestern part of Lagos Mainland and serves as a critical transportation hub, connecting major parts of the city through its extensive road network. The area lies along the Ikorodu Road expressway, one of the busiest highways in Lagos, which facilitates movement between the central business districts and the densely populated suburban areas, including Ketu, Mile 12, and Ikorodu. Due to its strategic location, Ojota experiences a significant volume of pedestrian traffic, making it an ideal location for studying the performance and durability of pedestrian infrastructure. Lagos State, the most urbanized and economically vibrant state in Nigeria, has a rapidly expanding population that has led to increasing pressure on public infrastructure. As part of the state's effort to enhance pedestrian safety and improve traffic management, pedestrian bridges have been constructed in high-risk areas such as expressways and major intersections. The Ojota Pedestrian Bridge is one such structure, providing a safe passage for pedestrians to cross the heavily trafficked Ikorodu Road. Given the high influx of commuters in the area, the bridge plays a crucial role in preventing pedestrianvehicle conflicts and reducing road accidents. However, like many urban infrastructure projects, the bridge is subject to material deterioration, structural loading, and environmental effects, necessitating continuous monitoring and maintenance to ensure long-term functionality. Geographically, Ojota is situated at approximately 6.5890°N latitude and 3.3792°E longitude, within the humid tropical climate zone of Nigeria. The area experiences high temperatures, heavy rainfall, and elevated humidity levels, particularly during the rainy season, which lasts from April to October. These climatic factors contribute to environmental degradation of concrete and steel materials, accelerating corrosion, crack formation, and overall structural wear. Additionally, Lagos' urban setting exposes the bridge to air pollution, dust accumulation, and vehicular emissions, which can further impact the material integrity over time. Apart from environmental effects, structural loading and user behavior are important considerations in evaluating the performance of the Ojota Pedestrian Bridge. Given its proximity to major commercial centers, bus terminals, and residential areas, the bridge experiences continuous pedestrian movement throughout the day, with peak usage during morning and evening rush hours. Overcrowding, unauthorized activities such as street vending, and improper usage patterns can impose additional stress on the bridge components. Furthermore, vibrations and dynamic loading effects caused by high pedestrian footfall and occasional misuse (e.g., motorcyclists or heavy loads being transported across the bridge) may influence its structural response and potential failure mechanisms.

B. Material Properties

The accurate definition of material properties is fundamental in ensuring the reliability of finite element analysis (FEA) in ABAQUS. For the Ojota Pedestrian Bridge, the primary materials used in the simulation are reinforced concrete which comprised of concrete and steel reinforcement, both of which exhibit distinct mechanical behaviors. The concrete is modeled as a non-linear material, capturing its brittle nature under tensile stresses, while the reinforcing steel is treated as an elastic-plastic material, representing its ductile response under loading.

Concrete Properties

Reinforced concrete is a composite material with a strong compressive strength but relatively weak tensile strength. To accurately model its behavior in ABAQUS, the Concrete Damaged Plasticity (CDP) model is employed. This model accounts for both plastic deformation and damage accumulation, allowing for a realistic simulation of cracking under tensile loading and crushing under compressive forces. The material parameters for the C30/37 grade concrete used in the bridge.

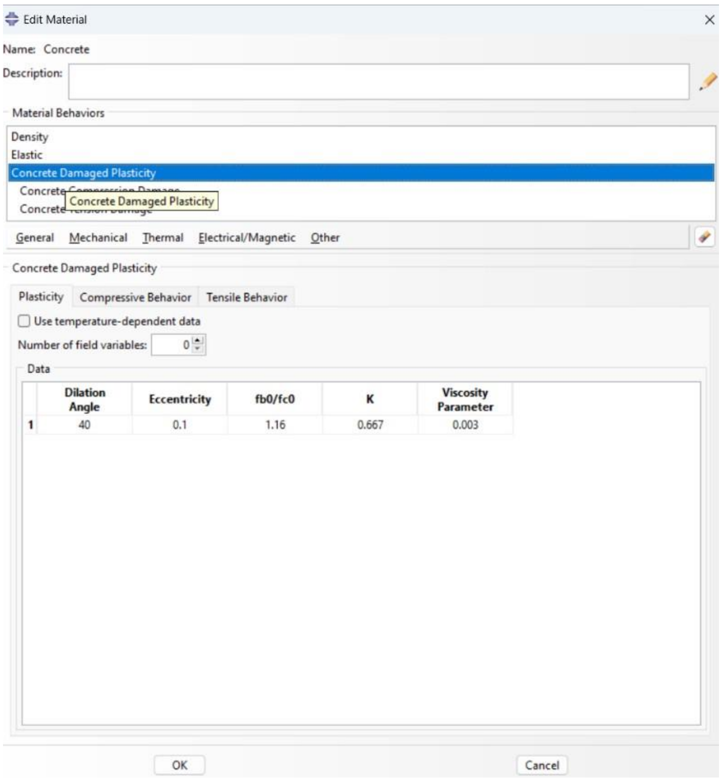

Figure 1: Material property insertion of concrete into the ABAQUS software

The compressive stress-strain relationship of concrete follows a nonlinear curve, with an initial elastic region followed by plastic deformation before failure. The tensile behavior is characterized by a softening response, where cracks propagate as tensile stress exceeds the material's limit. The Concrete Damaged Plasticity model captures these effects by defining damage variables that reduce the stiffness of the material as it undergoes micro-cracking and crushing as shown in **Figure** 1 above.

Steel Reinforcement Properties

The reinforcement bars used in the bridge are modeled as a ductile material, capable of undergoing significant plastic deformation before failure. ABAQUS defines reinforcement using an elastic-plastic model with isotropic hardening, ensuring a realistic representation of yielding and strain hardening effects. The B500B grade reinforcing steel, as per Eurocode 2 specifications, has the material properties as shown in **Table 1**. The reinforcement behavior follows a bilinear stress-strain curve, with an initial elastic region, followed by plastic deformation beyond the yield point. Strain hardening is incorporated to simulate the material's ability to withstand additional loads beyond yielding, ensuring a realistic response under high-stress conditions. In ABAQUS, the reinforcement bars are modeled using truss elements (T3D2) embedded within the concrete, ensuring proper load transfer between materials.

Table 1: Material Properties of concrete and steel.

Material	Density (kg/m³)	Compressive Strength (MPa)	Young's Modulus (GPa)	Poisson's Ratio	Additional Properties
Concrete (C30/37)	2400	30	30	0.2	Dilation Angle: 40°, Eccentricity:

					0.1, Viscosity: 0.003
Steel Reinforcement (B500B)	7850	500 (Yield Strength)	200	0.3	Plastic Strain at Yield: 0.002

Concrete-Reinforcement Interaction

The interaction between concrete and reinforcement is a critical aspect of the simulation, as it governs the load transfer efficiency between the two materials. In ABAQUS, this interaction is typically modeled using the embedded region constraint, where reinforcement bars are assigned to the concrete matrix without explicit bond-slip modeling. This enables the concrete to pass stress to the reinforcement seamlessly. The Concrete was denoted as the host region while the reinforcement was assigned embedded region.

Material Modeling in ABAQUS

In the finite element model, the concrete is assigned solid elements (C3D8R), while the reinforcement is modeled using embedded truss elements (T3D2). The Concrete Damaged Plasticity (CDP) model is implemented for concrete, while an elastic-plastic model with isotropic hardening is used for steel. By defining these properties accurately, the simulation can effectively predict stress distribution, cracking patterns, and failure mechanisms under various conditions. The accurate definition of material properties is essential for obtaining reliable simulation results in ABAQUS. The nonlinear behavior of reinforced concrete, combined with the elastic-plastic response of steel, determines the structural integrity and performance of the bridge under pedestrian loads and environmental effects.

Loading Conditions

In the finite element analysis (FEA) of the Ojota pedestrian bridge using Abaqus, two primary loading conditions were applied to assess the structural response**:** gravity load and pressure load (Load 1). These loads were implemented to simulate the real-world forces acting on the bridge and evaluate its performance under typical service conditions.

1. Gravity Load

Gravity loading was applied to account for the self-weight of the bridge structure, including the deck, supporting beams, and railings. This was automatically incorporated in Abaqus using the material density defined for each structural component. The gravitational acceleration was set to 9.81 m/s² to ensure accurate representation of the downward force acting on the bridge.

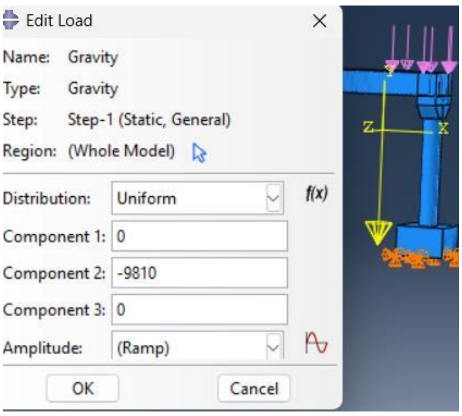

Figure 2: Applying of gravity to the bridge model in the load module

2. Pressure Load (Load 1)

A uniform pressure load, referred to as **Load 1**, was applied to the bridge deck surface. This load represents external forces such as pedestrian-induced effects as a result of commuting. The magnitude and distribution of this pressure load were defined based on the analysis requirements according to Eurocode 2, ensuring a realistic simulation of operational conditions. Pedestrian loading was assumed to take a load of 5kN/m for a lightly loaded pedestrian bridge.

Both loads were applied within Abaqus as static loads, for the structural response to be analyzed to determine stress distribution, deformations, and overall stability of the bridge.

Meshing

Meshing is a crucial step in finite element analysis (FEA) as it determines the accuracy and efficiency of the numerical simulation. In this study, the bridge model was discretized into smaller elements to facilitate precise computation of stresses, strains, and deformations under applied loading conditions. The meshing process followed a structured approach to ensure a balance between computational cost and result accuracy. The first step in the meshing procedure involved preparing the bridge geometry to ensure a clean and well-defined model. This included simplifying complex features that did not significantly impact the structural behavior, thereby reducing unnecessary computational effort. Partitioning the geometry into manageable sections was also performed to allow structured meshing, which improves element quality and numerical stability. Surface and volume checks were conducted to identify and resolve any gaps, overlaps, or free edges that could lead to meshing errors or inaccurate simulations. Once the geometry was prepared, appropriate element types were selected based on the structural characteristics of different bridge components. Truss elements, such as T3D2, was used for reinforcements as they properly captured it's tensile and elastic behavior. Solid elements, specifically C3D8R was utilized in regions requiring detailed stress analysis, such as critical connections and load application zones, where local stress concentrations were expected.

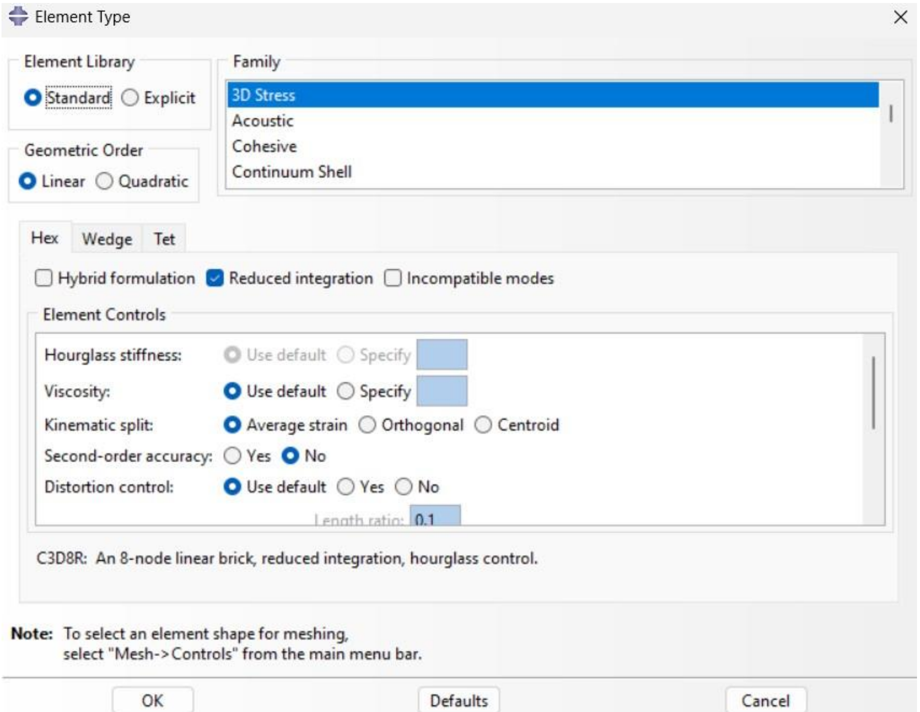

Figure 3: Assignment of 3D solid element to the concrete sections before meshing

Determining the appropriate mesh size was an essential aspect of the process, as it directly influences the accuracy of the simulation results. A global element size of 300 units was initially assigned based on the overall dimensions of the bridge to create a uniform baseline mesh. A mesh sensitivity study was then conducted, wherein multiple simulations were performed with varying mesh densities to identify the optimal balance between computational efficiency and accuracy for different parts of the structure. In areas where high stress gradients were anticipated, such as support regions and locations of applied loads, a finer mesh varying between 100 and 150 units was implemented to enhance result precision as shown in **Figure 3.** Conversely, less critical regions were assigned a coarser mesh to optimize computational resources without compromising accuracy of the model. With the element types and sizes determined, the meshing process was carried out in Abaqus. Structured meshing was applied wherever feasible to maintain uniformity and improve element quality, while free meshing was used in areas with complex geometries where structured patterns were impractical. Once the mesh was generated, quality checks were performed to ensure numerical stability. This involved when the solver assessed the element aspect ratios to prevent excessive distortion, evaluating the Jacobian determinant to identify poorly shaped elements, and checking for skewness, which can negatively impact solution accuracy. Any elements failing these quality criteria were refined or adjusted accordingly. The model partitioning and mesh methods were adjusted until the solver returned a 0% warning and error message to ensure accuracy of the model.

To further enhance accuracy, a convergence study was performed to ensure that the solution was independent of mesh size. This involved refining the mesh progressively and comparing the results to determine whether additional refinement significantly altered stress and deformation outputs. If the results remained consistent across successive refinements, the mesh was considered optimal, and further refinement was deemed unnecessary. Through this systematic meshing approach, the finite element model of the Ojota pedestrian bridge was developed with high accuracy and computational efficiency. The final mesh was fine enough to capture essential structural responses while remaining computationally feasible as shown in Error! Reference source not found., ensuring reliable results in the subsequent analysis.

Boundary Conditions

Boundary conditions play a crucial role in finite element analysis (FEA) as they define how the structure interacts with its supports and surroundings. In the assessment of the Ojota pedestrian bridge using Abaqus, appropriate boundary conditions were applied to ensure realistic simulation of structural behavior under loading. These constraints prevented rigid body motion and accurately represented the bridge's support system in real-world conditions. Applying appropriate boundary conditions is essential for obtaining accurate results, as incorrect constraints can lead to unrealistic stress distributions and deformations. By using pinned supports at the bottom of the piers, as shown in, the model effectively simulated real-world conditions, allowing for a reliable assessment of the bridge's structural performance under gravity and pressure loads.

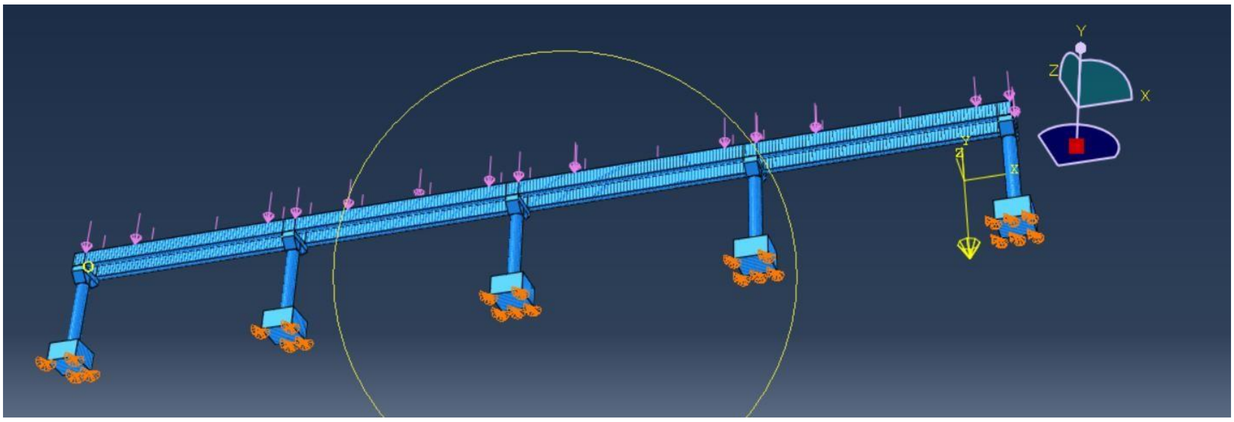

Figure 4: Pinned Boundary Conditions

For this study, pinned supports were assigned at the bottom surface of the bridge's piers. A pinned support condition restricts all translational movements while allowing rotational degrees of freedom. This means that displacement in the X, Y, and Z directions was fully constrained, ensuring that the base of the piers remained stationary under applied loads. However, the structure was still allowed to rotate about all three axes, mimicking the typical behavior of bridge piers that are designed to resist vertical and lateral forces while accommodating minor rotational effects. The implementation of pinned supports in Abaqus was achieved by selecting the bottom surfaces of the piers and applying boundary constraints that set U1 = U2 = U3 = 0, where U1, U2, and U3 represent displacements along the global X, Y, and Z axes, respectively as shown in Error! Reference source not found.. This effectively anchored the piers to the ground, ensuring that the bridge structure was adequately restrained.

Interaction

In finite element analysis (FEA), defining appropriate interactions between structural components is essential to accurately capture load transfer, material behavior, and overall system response. In the assessment of the Ojota pedestrian bridge using Abaqus, two key interaction techniques were employed: tie constraints for the connection between the girder and the pier, and an embedded region approach to model the reinforcement within the concrete. These interaction definitions ensured realistic load distribution and structural integrity throughout the simulation.

1. Tie Constraints Between Girder and Pier

A tie constraint was applied between the girder and the pier to establish a fully bonded connection, ensuring that both components acted as a single unit under applied loading conditions. In Abaqus, a tie constraint enforces a rigid connection between two surfaces, meaning that the slave surface (typically the smaller or more flexible component) follows the motion of the master surface (the larger or stiffer component). This method is commonly used when simulating composite structures where two different materials or components need to behave as a single entity without relative motion at their interface. For this study, the pier was defined as the master surface, while the girder was assigned as the slave surface, ensuring that the larger and more rigid pier controlled the motion of the girder. The tie constraint effectively transferred both forces and moments between these elements, preventing separation or sliding at their interface. This setup was crucial for accurately representing the structural behavior of the bridge, as the girder relies on the pier for support and stability.

The tie constraint was implemented using the following steps in Abaqus:

1. The contact surfaces between the girder and the pier were identified.
2. The master and slave surfaces were assigned based on their relative stiffness.
3. The constraint was applied, ensuring that all translational and rotational degrees of freedom were coupled between the two components.

By using a tie constraint, a perfect bond was assumed, meaning no relative displacement or rotation occurred between the girder and the pier. This simplification allowed for a computationally efficient simulation while maintaining an accurate representation of load transfer in the bridge structure.

2. Embedded Region for Reinforcement in Concrete

The embedded region technique was employed to model the reinforcement bars within the concrete elements of the bridge structure. In reinforced concrete modeling, it is essential to define the interaction between the steel reinforcement and the surrounding concrete to simulate their composite behavior under loading. The embedded region constraint in Abaqus enables reinforcement bars to be fully constrained within a host material, ensuring that they move together without relative slip. In this study, the reinforcement bars were defined as the embedded elements, while the concrete was assigned as the host region. This ensured that the steel reinforcement remained fully bonded to the surrounding concrete, effectively transferring stresses between the two materials. The embedded region approach simplifies the modeling process by eliminating the need for explicit contact definitions, reducing computational complexity while accurately representing the behavior of reinforced concrete structures.

The embedded region constraint was implemented in Abaqus through the following procedure:

1. The reinforcement bars were modeled as beam or truss elements, representing their real-world geometry and material properties.
2. The concrete was meshed as a solid structure, with the reinforcement bars positioned within the concrete volume.
3. The embedded region constraint was applied, ensuring that the reinforcement elements were fully constrained within the concrete matrix.

By using this approach, the reinforcement bars inherited the displacement field of the surrounding concrete, effectively mimicking the real-world bond between the two materials. This method ensured that the reinforcement contributed to the load-bearing capacity of the bridge while maintaining a realistic interaction with the concrete.

3. Justification and Accuracy Considerations

The chosen interaction methods; tie constraints for the girder-pier connection and embedded regions for reinforcement, were selected for their efficiency and accuracy in representing structural behavior. The tie constraint provided a computationally efficient way to model a rigid girder-to-pier connection, ensuring realistic load transfer without excessive computational cost. The embedded region approaches accurately captured the composite behavior of reinforced concrete while avoiding the complexity of modeling slip or bond failure explicitly.

While these methods offer significant advantages, it is important to acknowledge potential limitations. The tie constraint assumes a perfect bond, which may not always reflect real-world conditions where slight movement or deformation at the interface could occur. Similarly, the embedded region approach does not explicitly model bond-slip behavior, meaning that potential debonding effects between reinforcement and concrete are not accounted for. However, given the scope of this study and the primary objective of assessing the bridge's overall structural response, these simplifications were considered appropriate and sufficient for achieving reliable results.

Step

The step module in Abaqus defines the type of analysis and controls the progression of the simulation through time integration. In this study, a static general analysis step was used to evaluate the structural response of the Ojota pedestrian bridge under applied gravity and pressure loads. The settings within the step module were carefully adjusted to ensure accurate load application, numerical stability, and efficient computation.

Abaqus employs an incremental-iterative approach to solving finite element problems, meaning that the total load is applied gradually in increments, and the solver iterates within each increment to achieve equilibrium. The time incrementation settings play a crucial role in determining the accuracy and stability of the analysis. The values for initial, minimum, and maximum time increments are shown in Error! Reference source not found., where they were chosen to balance computational efficiency with numerical accuracy. The initial increment was set small enough to accurately capture the bridge's initial response to applied loads. The minimum increment was defined to ensure that the simulation could continue even in highly nonlinear regions, preventing premature termination due to convergence difficulties. The maximum increment was limited to maintain solution accuracy while allowing faster computation in linear regions of the analysis. To improve the solver's ability to handle complex equilibrium equations, the IA (Iteration Allowance) parameter was increased from 5 to 12 in the step module settings as shown in Error! Reference source not found. This adjustment allowed Abaqus to attempt more iterations per increment, giving the solver additional chances to converge when encountering nonlinear behavior or highly constrained equations. By allowing more iterative attempts, this setting improved the likelihood of achieving a stable solution without requiring excessive reductions in increment size, which could otherwise slow down the analysis. Additionally, Abaqus' automatic time-stepping algorithm was enabled to dynamically adjust the increment size based on the convergence behavior. If the solver struggled to achieve equilibrium, the increment size was automatically reduced to improve stability. Conversely, if the solution converged quickly, Abaqus increased the increment size to optimize computation time.

RESULTS

A. Yielding (Material Failure)

The finite element analysis (FEA) conducted in ABAQUS confirmed that yielding was a critical failure mode in the pedestrian bridge, particularly in the steel reinforcement and concrete under high-stress regions. The analysis revealed that several structural members experienced stresses exceeding their material yield strength, leading to permanent deformation and reduced load-carrying capacity.

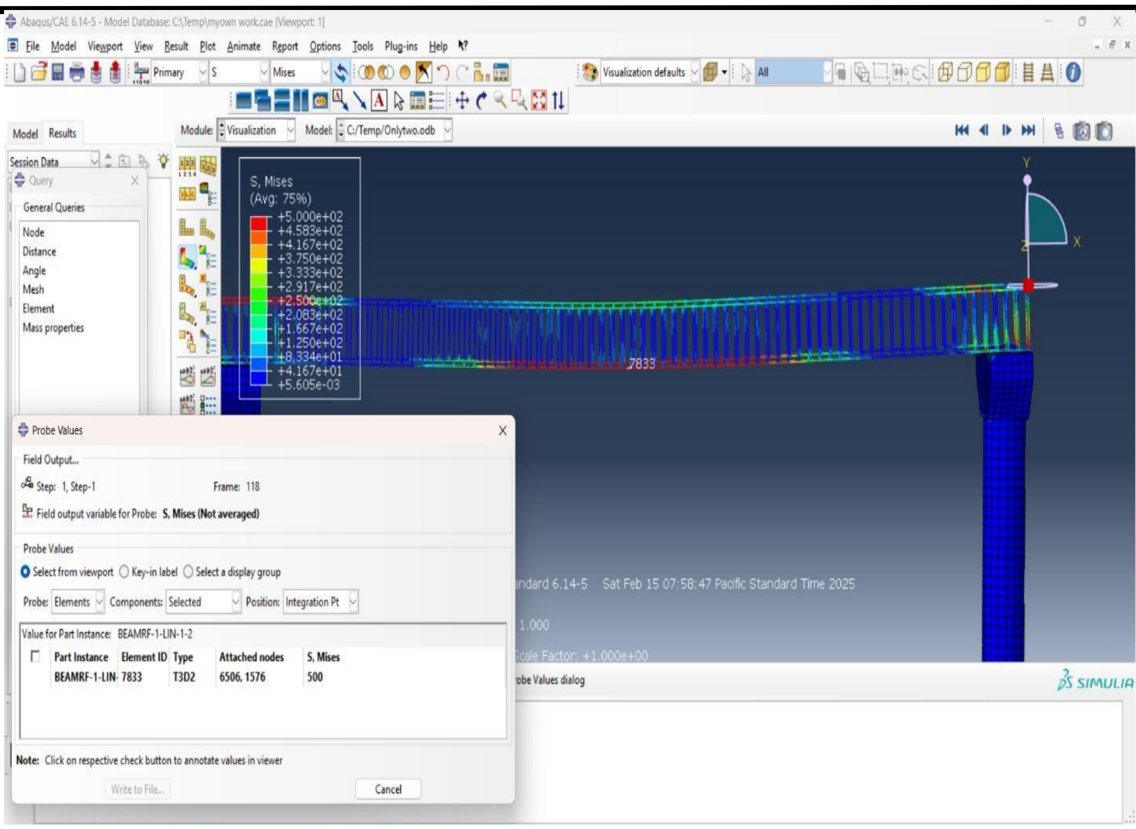

Figure 5: Probe value of a T3D2 Element at midspan of the girder.

The maximum von Mises stress recorded in the steel reinforcement was observed to exceed the material's yield strength of 500 MPa at the very last frame of the Job (118) as shown in Fig 4.1. This result confirmed that the steel had entered the plastic deformation range, particularly at the mid-span of the bridge and near support regions where bending moments were highest. The yielding was most pronounced in the tensile reinforcement layers, indicating that the steel was unable to sustain additional load without undergoing further deformation. This failure compromised the bridge's structural integrity, as the yielded reinforcement could no longer effectively resist applied loads. A graphical representation of the failure of A T3D2 element at the mid span of the reinforcement in the tensile region is shown in Fig. 4.2. As shown in this figure, it is evident that the stress increased with time gradually from 0 Time units up until A few units after 0.8time units where the stress was constant for the remaining period of the job. From this behavior it is obvious that the reinforcement reached its plastic state at 0.8-time units and the straight line after the gradual increase in the graph is a depiction of the fact that plasticity has been reached.

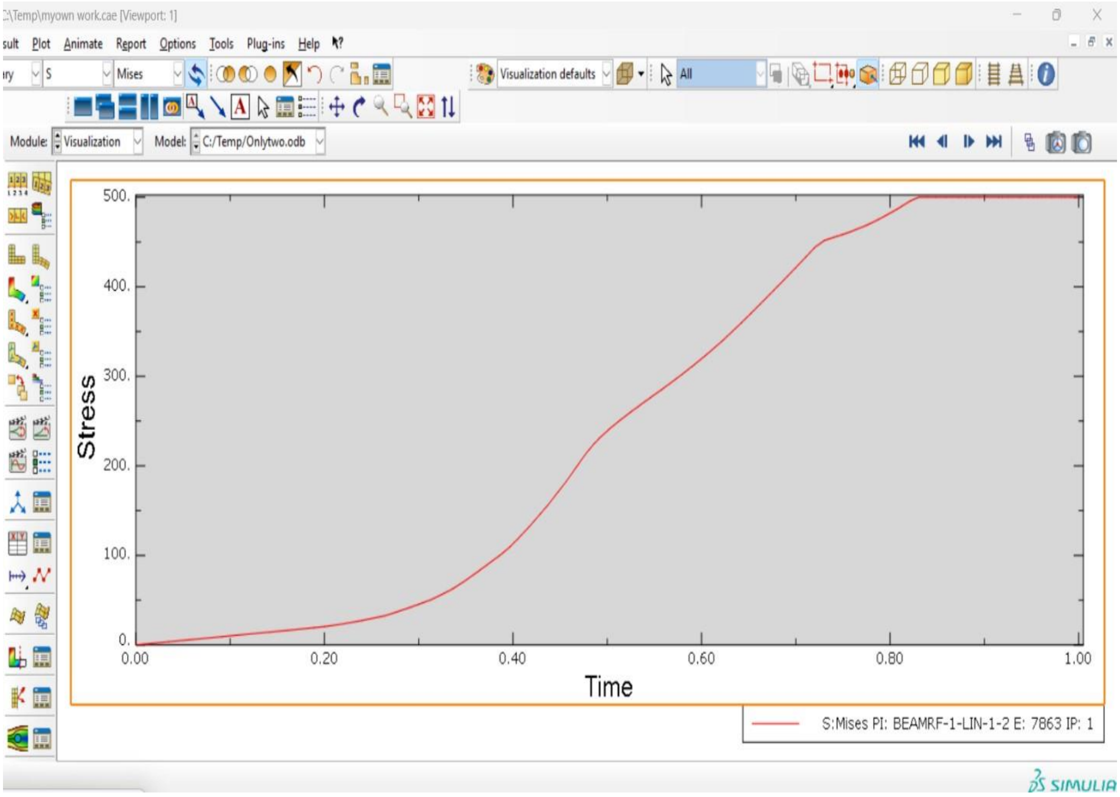

Figure 6: Graph showing yielding of reinforcement at 500Mpa

In addition to reinforcement yielding, the analysis showed that compressive stresses in the concrete exceeded its compressive strength (fck) in certain areas. At the most highly stressed regions, the recorded stress reached **31.85 MPa**, exceeding the **30 MPa** strength of the concrete.

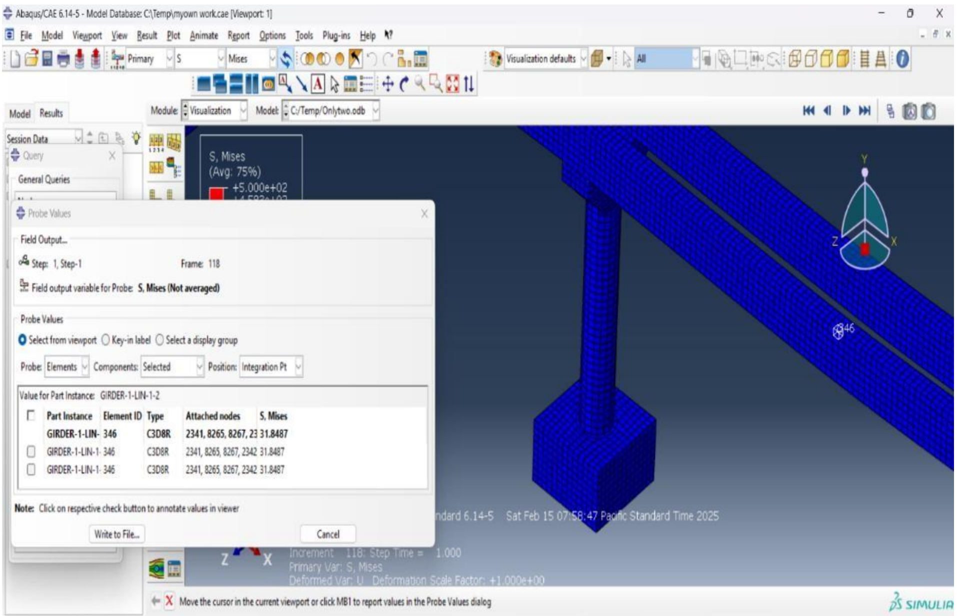

Figure 7: Probe value showing the maximum stress in the concrete under excessive loading

This results in crushing failure, particularly in areas near the top midspan where compressive forces were concentrated.

As a result, cracking and spalling were likely to develop, further reducing the bridge's load-bearing capacity and increasing susceptibility to long-term deterioration. Figure 8 shows when the stress in the concrete just crosses 30MPA with a progressive incline upwards. This is the point at which crushing failure begins to occur in the concrete.

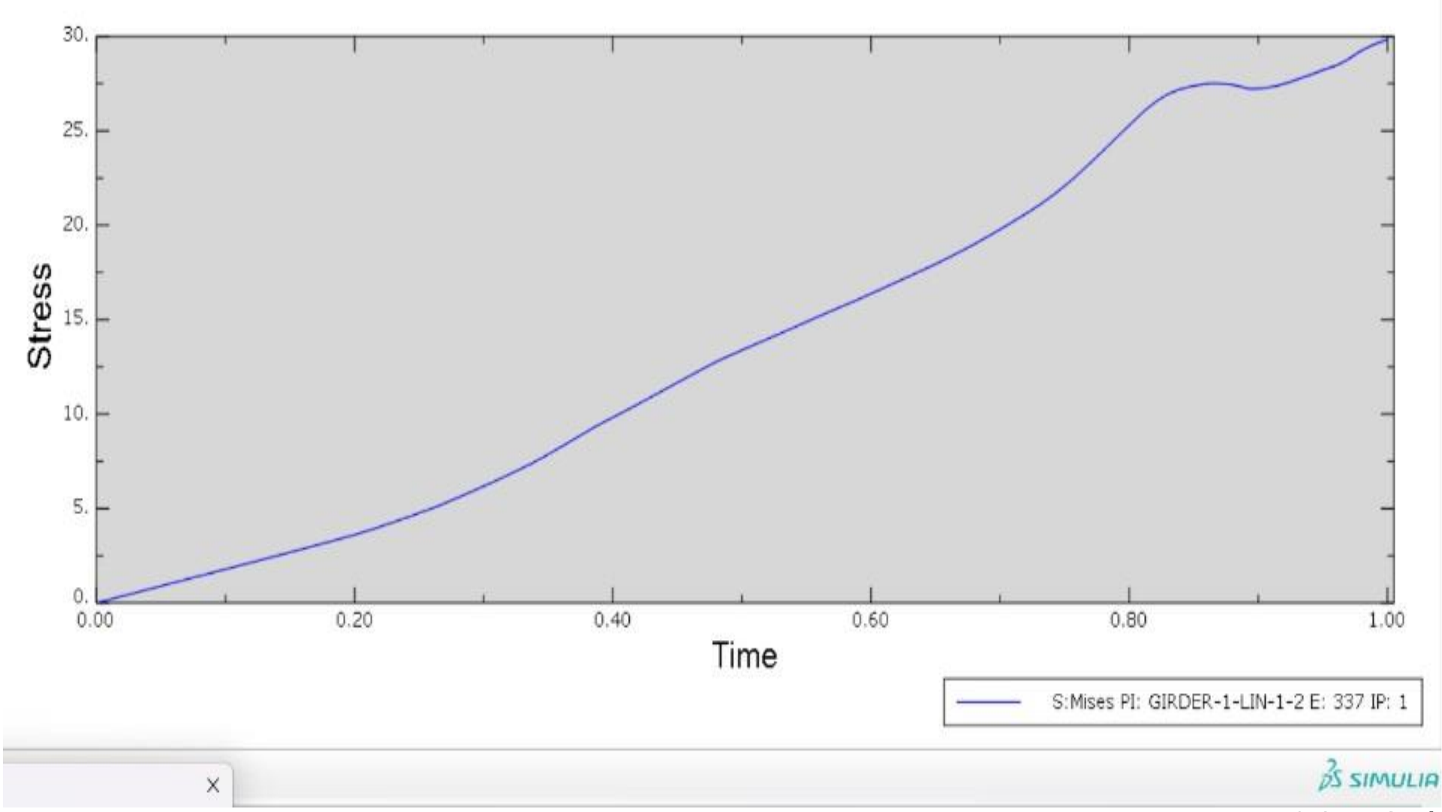

Figure 8: Stress Graph

The yielding failure observed suggests that structural reinforcement strategies, such as increasing the reinforcement ratio, using higher-strength steel, or optimizing cross-sectional dimensions, are necessary to prevent premature yielding.

B. Effect of Material Deterioration (Corrosion) on Structural Performance

To simulate corrosion-induced material deterioration, a series of numerical analyses were conducted by reducing the cross-sectional area of the reinforcement in different simulation cases. Corrosion, as a long-term degradation process, leads to the loss of steel cross-section, reduced load-bearing capacity, and increased stress levels in both the reinforcement and concrete. One of the primary objectives of this study was to evaluate how different levels of corrosion impact the stress distribution, strain behavior, and deflection characteristics of the pedestrian bridge girder.

In the first corrosion scenario, a 10% reduction in the reinforcement area was introduced to represent early-stage corrosion. The stress in the reinforcement increased significantly, with values exceeding 600 MPa, indicating the onset of yielding in localized regions. The strain distribution revealed areas where the steel had exceeded its elastic limit, suggesting potential plastic deformation under sustained loading. Concrete stresses also increased, particularly near the supports, where secondary effects from redistribution of forces became noticeable. The maximum deflection (9mm) in this scenario exceeded the maximum deflection obtained at a normal pedestrian loading (7.8mm), suggesting that the girder's stiffness had been reduced again due to reinforcement loss. The deflection increased slightly but remained within the L/500 serviceability limit.

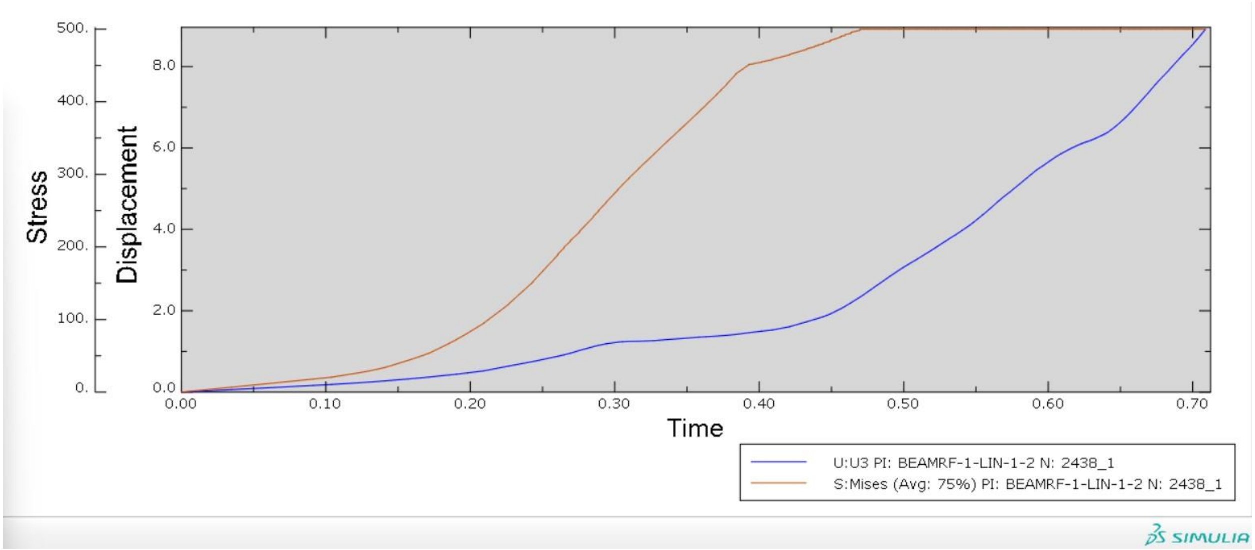

Figure 9: A stress displacement- time graph at 10% reduction in area

In the second corrosion scenario, a 30% reduction in the steel cross-section was applied, representing a more advanced stage of deterioration. The stress in the reinforcement increased significantly, with values exceeding 600 MPa, indicating the onset of yielding in localized regions. The strain distribution revealed areas where the steel had exceeded its elastic limit, suggesting potential plastic deformation under sustained loading. Concrete stresses also increased, particularly near the supports, where secondary effects from redistribution of forces became noticeable. The maximum deflection (19mm) in this scenario exceeded the maximum deflection obtained at a normal pedestrian loading (7.8mm) and was very close to the maximum acceptable deflection for the beam as calculated in chapter 3 (22.5mm), suggesting that the girder's stiffness had been reduced drastically due to reinforcement loss.

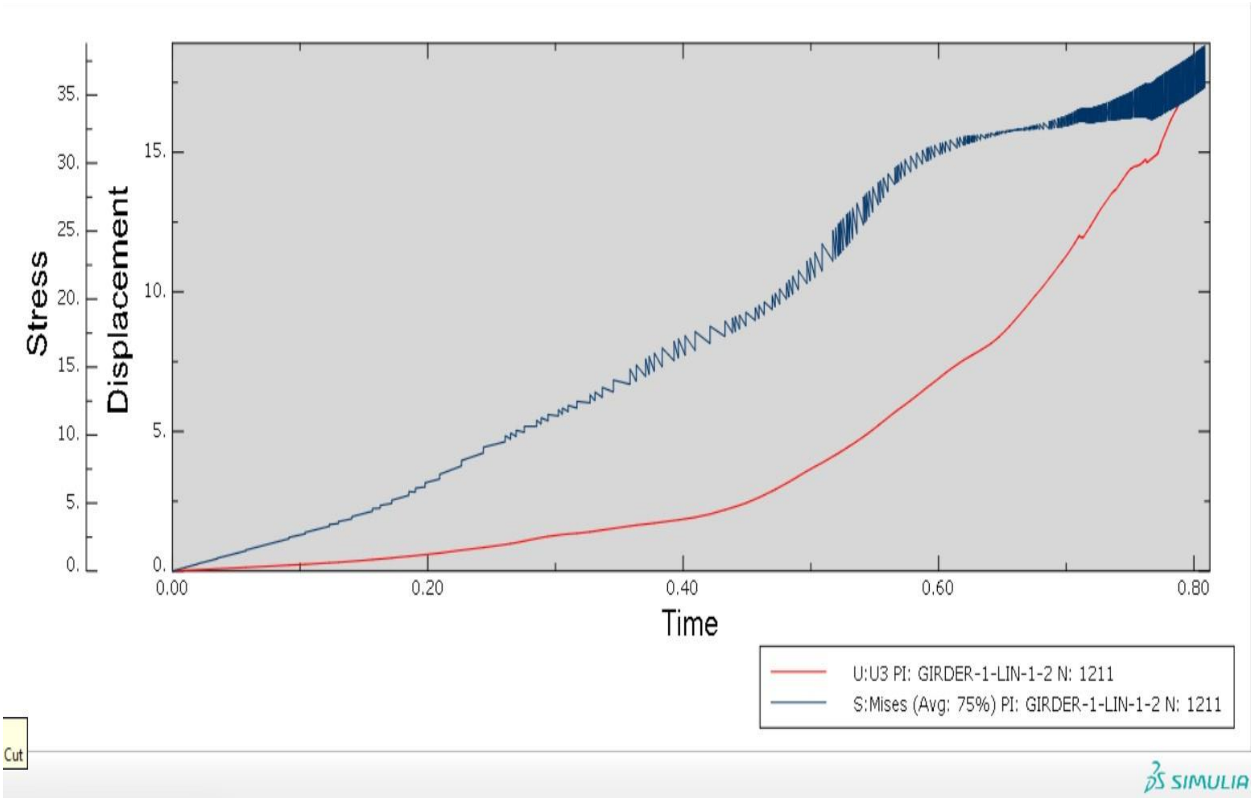

Figure 10: A stress displacement- time graph at 30% reduction in area

In the third and most severe corrosion scenario, a 50% reduction in the steel cross-section was applied, simulating extreme corrosion conditions. In this case, the reinforcement stress values reached 1000 MPa, well beyond the steel yield strength, confirming failure in tension. The concrete exhibited significantly higher compressive stresses, surpassing 45 MPa in certain areas, which is beyond the crushing strength of C30/37 concrete. The deflection increased drastically to 27.5mm, exceeding safe serviceability limits at 22.5mm, and the model showed indications of failure due to loss of loadcarrying capacity. These results indicate that severe corrosion poses a critical threat to structural integrity, leading to failure under standard loading conditions.

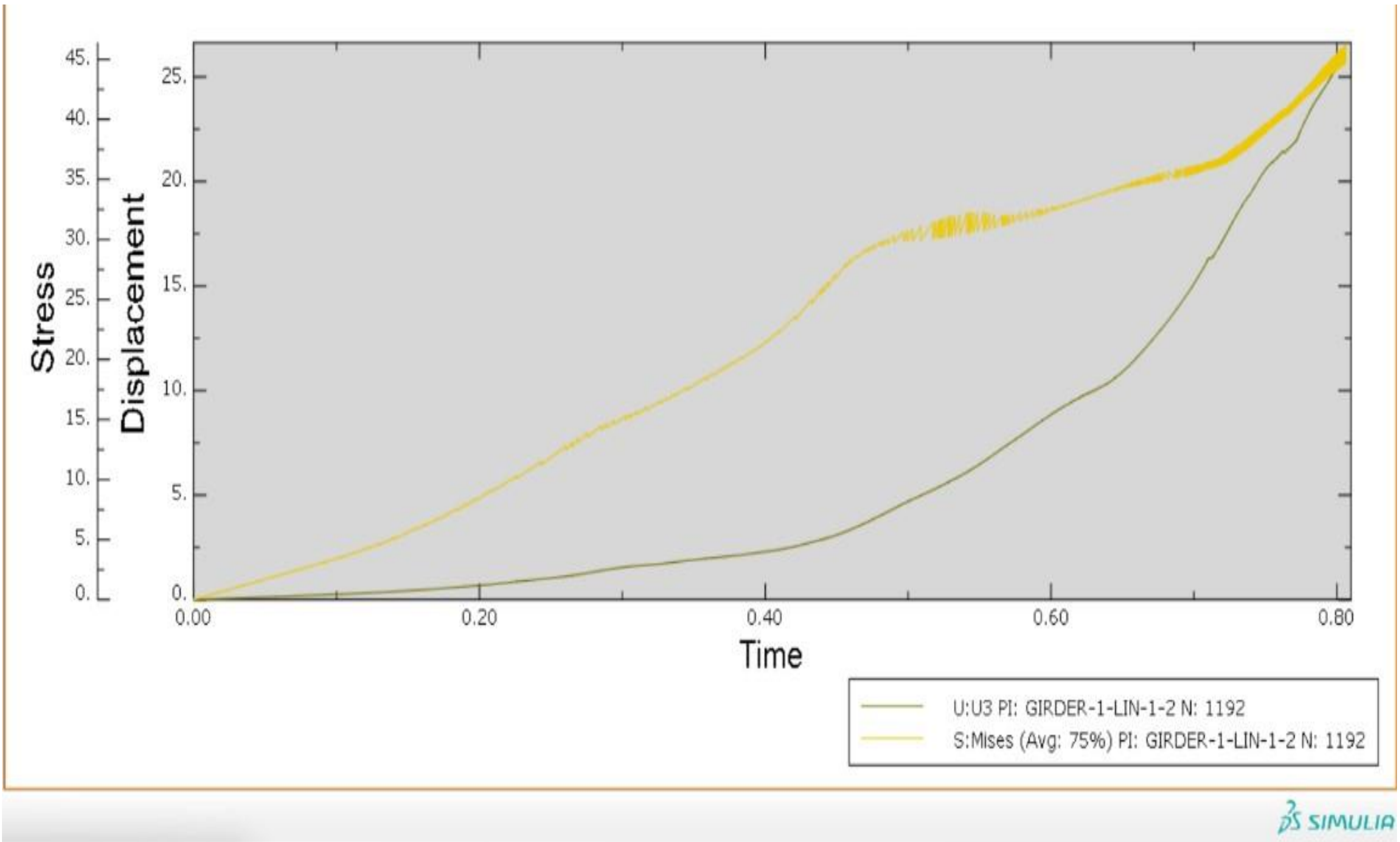

Figure 11: A stress displacement- time graph at 50% reduction in area

The numerical simulations clearly demonstrate the progressive weakening effect of corrosion on the bridge girder. As the reinforcement deteriorates, the remaining steel is forced to carry more stress, leading to premature yielding and an overall reduction in the girder's strength and stiffness. Increased deflections also indicate a loss of serviceability, making the structure unfit for use at higher corrosion levels. These findings highlight the importance of regular maintenance, protective coatings, and timely rehabilitation measures to ensure the long-term durability of pedestrian bridges. The results also emphasize the need for corrosion-resistant reinforcement materials, such as epoxy-coated or stainless-steel rebar, in environments where corrosion risks are high.

Table 2: Results of corrosion analysis

Corrosion Scenario	Reduction in Reinforcement Area (%)	Max Reinforcement Stress (MPa)	Max Concrete Stress (MPa)	Max Deflection (mm)	Serviceability Limit Exceeded?	Indication of Failure?
Scenario 1	10%	>600	28	9.0	No	No
Scenario 2	30%	>600	31.85	19.0	Almost	No

Scenario 3	50%	>1000	>45 MPa (beyond limit)	27.5	Yes	Yes

Conflicts of Interest: All authors declare that they have no conflict of interest associated with this research work. **Funding:** No special funding was received for this research work.

CONCLUSION

The finite element assessment of the Ojota Pedestrian Bridge girder was conducted to examine potential failure modes under excessive loading conditions. While the primary focus was on the girder, the entire bridge was modeled in ABAQUS to ensure accurate load distribution and realistic boundary conditions. To verify the reliability of the simulation results, a validation study was performed, comparing ABAQUS results with manual calculations based on structural engineering principles. The bending moment (562.5 kNm), shear force (199.125 kN), and deflection (7.8 mm) obtained from the finite element model closely matched theoretical values, confirming that the model accurately represents the girder's structural behavior. The results confirm that under normal pedestrian traffic, the bridge remains structurally safe and serviceable. However, under artificially increased loading designed to test failure mechanisms, certain structural vulnerabilities were identified in the girder. The analysis revealed that reinforcement yielding occurred at mid-span and support regions, with stress values approaching the 500 MPa yield strength of steel. These localized stress concentrations indicate that under extreme conditions, reinforcement may experience permanent deformation, though this does not affect normal operating conditions. The study also examined compressive failure in concrete, with stresses reaching 31.85 MPa, slightly exceeding the 30 MPa limit under excessive loading. While this suggests potential crushing in extreme cases, the girder remains within safe limits for everyday pedestrian use. Additionally, maximum deflection under extreme loading reached 7.8 mm, remaining within the Eurocode limit of 22.5 mm. This indicates that stiffness modifications are not required for current conditions, but reinforcement adjustments could be beneficial in future structural upgrades to increase durability under extreme loading scenarios. The corrosion analysis demonstrated that while a 30% reduction in reinforcement cross-section increased stress and deflection, a 50% reduction resulted in excessive stresses exceeding 1000 MPa, indicating potential failure. This highlights the importance of routine maintenance and corrosion prevention strategies to maintain the bridge girder's long-term structural integrity.

REFERENCES

1. American Concrete Institute. (2019). *Building code requirements for structural concrete (ACI 318-19) *.
2. Bentz, E. C. (2000). Finite element modeling of reinforced concrete structures (Doctoral dissertation, University of Toronto).
3. Biondini, F., & Frangopol, D. M. (2016). Life-cycle performance of deteriorating structural systems under uncertainty. *Journal of Structural Engineering, 142*(9), F4016001.
4. Comité Euro-International du Béton. (2010). *CEB-FIP model code 2010*. Thomas Telford.
5. Enright, M. P., & Frangopol, D. M. (1998). Service-life prediction of deteriorating concrete bridges. *Journal of Structural Engineering, 124*(3), 309-317.
6. Federation Internationale du Beton. (2010). *fib model code for concrete structures 2010*. Ernst & Sohn.
7. Gilbert, R. I. (2016). *Serviceability design of concrete structures*. CRC Press.
8. Hognestad, E. (1951). A study of combined bending and axial load in reinforced concrete members. *University of Illinois Bulletin, 49*(22).
9. Jirásek, M., & Bauer, M. (2012). Computational modeling of cracking in concrete structures. In *Computational modelling of concrete structures* (pp. 47-60). CRC Press.
10. Li, B., Maekawa, K., & Okamura, H. (2004). Contact density model for stress transfer across cracks in concrete. *Journal of the Faculty of Engineering, 40*(1), 25-52.
11. Mangat, P. S., & Elgarf, M. S. (1999). Flexural strength of concrete beams with corroding reinforcement. *ACI Structural Journal, 96*(1), 149-158.

12. Melchers, R. E. (2003). Probabilistic models for corrosion in structural reliability assessment. *Journal of Structural Engineering, 129*(10), 1364-1370.
13. Nassif, H., Liu, M., & Ertekin, O. (2003). Dynamic load models for pedestrian bridges. *Journal of Bridge Engineering, 8*(6), 354-362.
14. Olorunfemi, M. O., Laniyan, T. A., & Ajayi, K. T. (2021). Utilization factors of pedestrian overpasses in Akure, Nigeria. *Journal of Urban Infrastructure, 15*(2), 112-125.
15. Park, R., & Paulay, T. (1975). *Reinforced concrete structures*. Wiley.
16. Rodriguez, J., Ortega, L. M., & Casal, J. (1997). Load carrying capacity of concrete structures with corroded reinforcement. *Construction and Building Materials, 11*(4), 239-248.
17. Tuutti, K. (1982). *Corrosion of steel in concrete*. Swedish Cement and Concrete Research Institute.
18. Ngekpe, B. E., Uche, O. A., & Johnson, D. F. (2019). Dynamic analysis of Leventis Footbridge under pedestrian loading. *Nigerian Journal of Structural Engineering, 8*(2), 45-59.