IJAAR PUBLISHING®

Engineering Research Journal | ISSN: 2782-8212
Vol. 5, Issue 3 (2025) | www.ijaar.org/erj

Modeling and Simulation of Heavy Axle Loads on Bridge Superstructure using ANSYS Workbench

S.O. JIBRIL, A. ABUBAKAR, A. MOHAMMED, I. M. LAWAL
Civil Engineering Department, Abubakar Tafawa Balewa University Bauchi, Nigeria

ABSTRACT

This paper presents investigation on the effect of heavy axle loads on 2-span Reinforced Concrete Bridge superstructure using ANSYS workbench. The bridge is a link between the Nigeria export processing zone and Kano city. Damage to bridges is as a result of its repeated load-carrying capacity. The continuous flow of automobiles results in degradation and subsequent decline in the mechanical properties of the deck, leading to reduced reliability over an extended period. This research employed a relatively cost effective numerical approach to evaluate the possible damage accumulated and finite element model (FEM) of the bridge deck was developed based on preliminary calculations using information gathered from built drawings which help to predict the deterioration rate of the superstructure using simulation capability of ANSYS 2018. A single span of 16m by 8m width was isolated for a detailed structural analysis. The bridge 3D-geometry and material properties were modeled in ANSYS with calibration from CSiBridge v17 program and simulated in accordance with "British Code of Practice for Highway Bridges (BS 5400). At serviceability, the bridge maximum deflections at deck using ANSYS and CSI Bridge were 1.07.mm to 2.6mm and 1.1mm to 3mm respectively while at deck/girder were between 8.179mm to 11.673mm and 5mm to 11.1mm simulations respectively and values are within the acceptable range of span800-span1200mm *or (13.33 – 20mm). The deck service stresses were found to be within the endurance Stress* e *(2 - 3.5)*N/mm2 *defined by the materials data. This study provides valuable scientific and practical outcomes that can be utilized in the design and operation of bridge structures, contributing to the development of infrastructure and ensuring the safety of transportation systems. The use of ANSYS software in the analysis process enhances the accuracy, and efficiency of the research, further emphasizing its relevance in the analysis of bridge.*

KEYWORDS: Axle loads, ANSYS Workbench, Bridge Deflection, Finite Element, Modelling, Simulation.

S.O. Jibril, A. Abubakar, A. Mohammed, & I. M. Lawal. (2025). Modeling and Simulation of heavy Axle Loads on Bridge Superstructure using ANSYS Workbench. *Engineering Research Journal*. *5*(3), 29-41. DOI: https://doi.org/10.5281/zenodo.15789755

1.0 INTRODUCTION

Bridges are structures that span and provide passage over road, railway, river, or some other obstacle. They act as an important link in surface transportation network (Chen and Duan, 2014). The bridge deck is the flat surface providing direct support to the traffic loads such as vehicles, trains, pedestrians. Most Bridge decks are designed and constructed of Reinforced concrete (RC), Steel, or Wood. The dynamics of bridge structures involves an analysis of the loads and forces acting on the structure such as road traffic, wind effects, train vibrations, or other dynamic factors (Chang *et al.*, 2003; Chen and Duan, 2014). Subsequently, modelling and calculations are carried out to predict the dynamic behaviour of the structure, comprising the analysis of resonance phenomena and possible resonance frequencies (Chopra 2012 and Cunha, 2013). In addition, an assessment of the vibrational properties, such as, the amplitude of oscillations, natural frequencies, and damping are conducted (Fujino 2002; Fukuda. Kvasnikov and Stakhova, 2022; Oksen, 2016).

Researching the dynamics of bridge structures requires the application of various methods and tools. Sensors, accelerometers, deformation sensors, and other instruments can be used for data collection and vibration monitoring (Kvasnikov and Stakhova, 2022). Modern computer programs enable numerical modeling and calculations, which help predict the dynamic behaviour of the structure under different conditions and optimize its parameters (Chopra 2012).

The dynamics of bridge structures using modern technologies and software tools, such as ANSYS, also enable virtual testing and optimization of bridge designs (ANSYS, 2020). This reduces costs associated with physical testing and allows engineers and designers to more efficiently utilize resources in the construction of new bridges or the upgrading of existing ones. The results of studying the dynamics of bridge structures can have a wide range of applications. They can be used to optimize the design of new bridges and the reconstruction of existing bridge structures to enhance their durability, safety, and efficiency. Investigating the dynamics of bridge structures contributes to the development of more accurate models and predictions for determining allowable loads and forecasting the behaviour of structures under operational conditions. Thus, research on the dynamics of bridge structures using modern methods and software is of great importance for infrastructural development and ensuring the safety of transportation networks. It assists in obtaining more accurate data on the dynamic behavior of bridge structures, making informed decisions in design and maintenance, and reducing unforeseen risk. Two types of moving dynamic loads are distinguished: loads generated by the natural flow of vehicles (Kvasnikov and Stakhova, 2022; Oksen, 2016) and loads generated by specialized vehicles (Oksen, 2016)

Loads in regular traffic flow are uncontrolled, while those created by specialized vehicles (e.g., vehicles loaded with ballast) are controlled. The passage of vehicles occurs at a specified constant speed along different lanes to create maximum dynamic loads. Impact loads are typically generated by mechanical and hydraulic hammers (Cunha, 2013), loads dropped from a certain height, specialized machines for inducing impact vibrations (Cunha, 2013).

2.0 MATERIALS AND METHOD

2.1 Materials

The materials used in this study are categorized into three (3) namely:

(1) FE program software.

ANSYS, one of Finite Element Analysis packages, has two major interfaces: an old one (Classical GUI, now named "Mechanical APDL") and a new one (Workbench, also named "ANSYS Mechanical"). Mechanical APDL is not so convenient to use, but is very useful to solve some specific problems. Mechanical APDL could be driven via text commands but workbench is more user friendly, and allows the easy usage of the geometry from modern CAD packages, maintaining the associativity through the whole FEA process. However, Workbench has few limitations; it is not as flexible as APDL. Heavy trucks wheel load, tire pressure, frequency, duration and environmental factors are all important to the performance of the bridge. However, the technology and high cost associated with conducting laboratory experiments and field test to acquire data for assessing the structural integrity of a bridge especially in developing nations are factors militating against researchers willing to evaluate the extent of damage and safety reserve of bridge components. This study will employ analytical approach, which is relatively economical, reliable and timely, to evaluate the influence of heavy axle's loads on the bridge

FEA software simulates the behavior of structures and systems under various conditions. FEA ANSYS software of 2020 v20. These tools allow engineers to analyze stress, strain, deflection/deformation, and other factors modeling the behavior of Concrete and steel materials as well as their interaction are complex concept which can be implemented within the ANSYS Work Bench (WB) module either by:

i. Using the material library to assign material properties to the 3D model (Geometry)
ii. Inserting an APDL command

For this study, the choice of WB module as well as the analysis system (Transient Structural), has limited the choice of material modeling to the later. This is because the module cannot model the interaction (contact) between concrete and steel automatically as such Concrete and Steel APDL command are used to simulate the exact behavior and interaction between them.

(2) Load Model

In bridge analysis, load models represent the various forces that a bridge experiences, including static loads (dead and live), dynamic loads, and environmental loads. These models are essential for accurately simulating bridge behavior and ensuring safe and reliable designs.

This study is limited on checking the deflection/deformation, stress-stain, share, moment and based on load models recommended by BD 37/01 2001 design codes are used to simulate the bridge response and not actual load on collected on weighbridge. The axle loads for BLM is 36.9t which will be used for the simulation with additional 20% increment for worst loading condition 44.28t,51.66t,59.04t,66.42t and 73.8t.

(3) Resistance Model: This refers to the bridge superstructure, which in this study is the Airport Road Bridge Kano, Nigeria. The cross sectional details of the bridge superstructure were presented in Figure 2.

2.2 Methods

This section focuses on the methods and procedures employed to model, simulate and analyze the bridge superstructure in ANSYS Workbench

This study is basically on stress-stain, shear, moment and deflection as such; normal load models recommended by BD 37/01 2001 design codes are used to simulate the bridge response over its design. The axle loads for BLM is 145kN. Figure 1 shows the plan view of the vehicles applied on the bridge deck. The airport bridge cross section is shown in Figure 2.

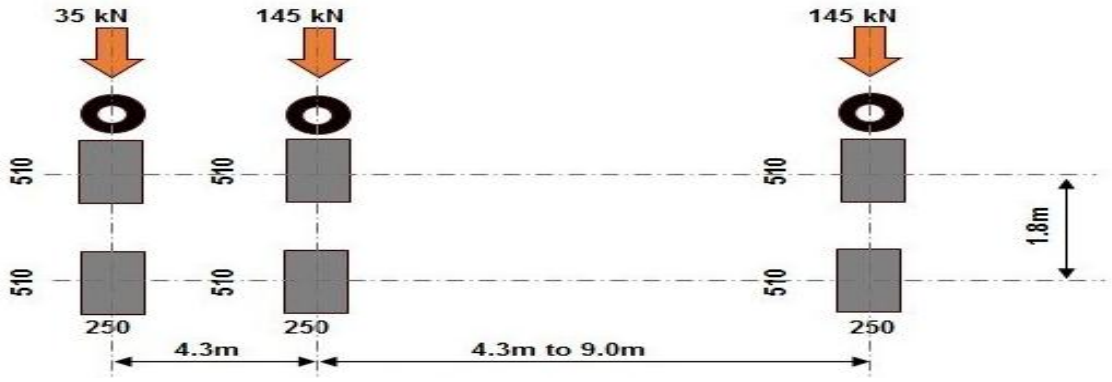


Figure 1: Plane Geometry of the BD 37/01 FLM

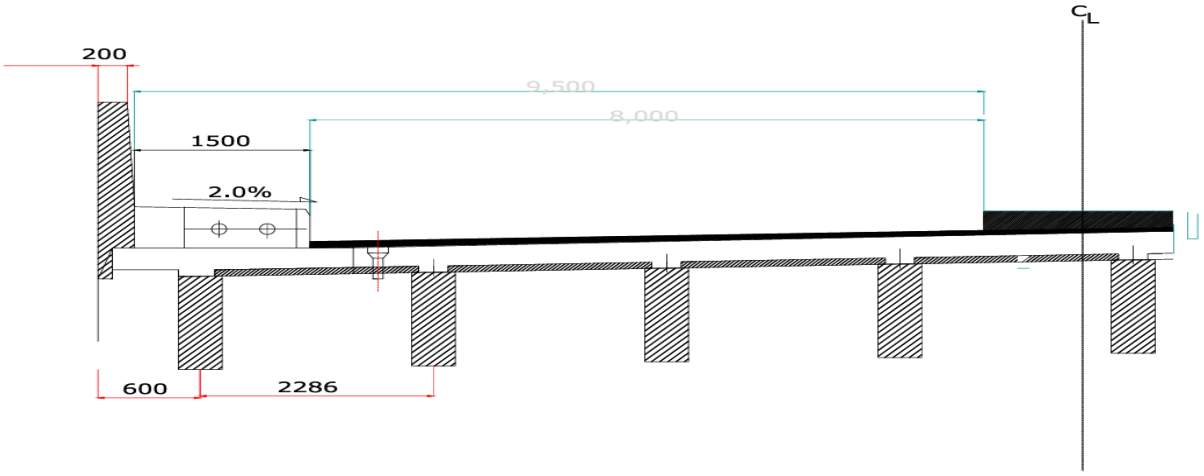


Figure 2. Airport Bridge Cross Section Details

2.2.1. 3D-Bridge modeling

For a detail stress, share, moment and deflection/deformation assessments of the bridge in ANSYS, only a segment (span) of the superstructure was considered from the entire bridge to limit the analysis time and capability of the engine to run a full model of the bridge.

The 3D-model of the RC bridge superstructure was developed using the Design Modeler module of ANSYS component system. The concrete frame was modeled from solid bodies while the steel rebar was modeled from line bodies.

The basic materials as presented in Table I for the bridge superstructure are combination of concrete and steel rebars which is known as reinforced concrete deck. The basic isotropic material properties used in modeling the proposed bridge for the FE analysis are;

i. Modulus of elasticity (E)
ii. Poison's ratio (υ)

Table I.: Mechanical Properties of Materials

Bridge component	Material	Properties/Parameters		
		Ec /Es(kN/mm2)	υ	Grade (N/mm2)
RC Deck	Concrete	25x103	0.2	25
	Steel rebar	210x103	0.3	410
Longitudinal Girders	Concrete	25x103	0.2	30
	Steel rebar	210x103	0.3	410

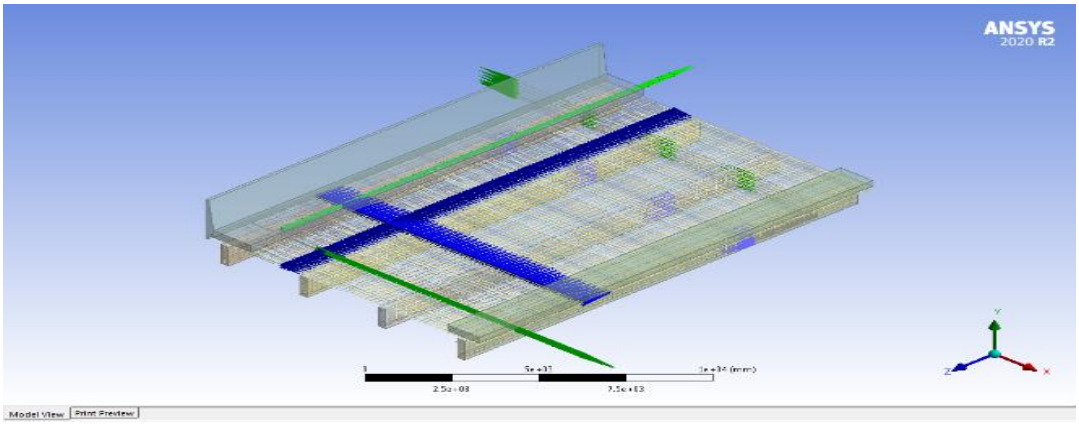


Fig. 3: 3D Model of Bridge Superstructure Steel Rebar in Design Modeler

SOLID 65 element APDL command was used to model concrete, while LINK 180 element command was used to model the steel rebar in Work Bench and an APDL code was used to simulate the contact between the two materials as shown in figure 3.

2.2.2 Meshing the Model

The entire bridge geometry was meshed (discretized) with a default mesh property as controlled by the program, only the element size was refined to 10mm with a fine relevance center as shown in figure 4.

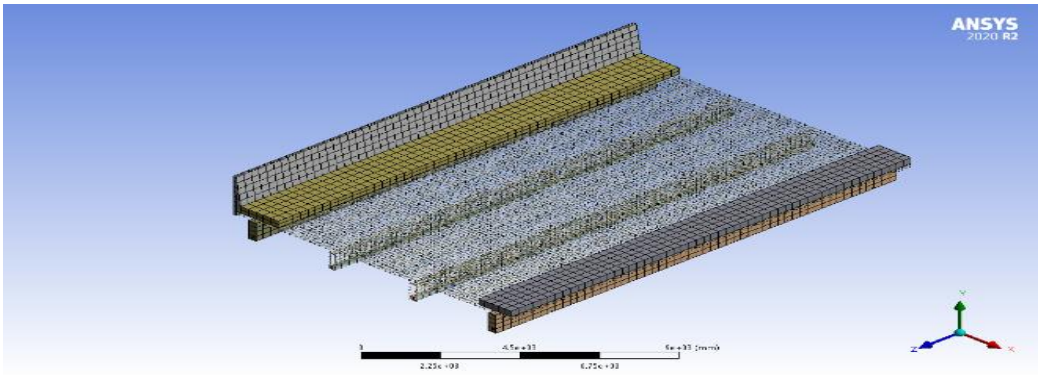


Fig. 4: Bridge Meshing Progress

2.2.3 Loading the Model (Bridge Geometry)

The normal bridge load models (LM) specified by BD 37/01 2001 were used to evaluate the bridge model (Resistance model). The loads were represented as Gross Weight-in-motion (GWIM) using the gravity and y-component options of the ANSYS program. The magnitudes of the weights were applied on the contact area of the tyres as defined in the codes and the tabular option was used to define the load for BLM is 36.9t and 44.28t,51.66t,59.04t,66.42t and 73.8t respectively, as moving load, as shown in figure 5.

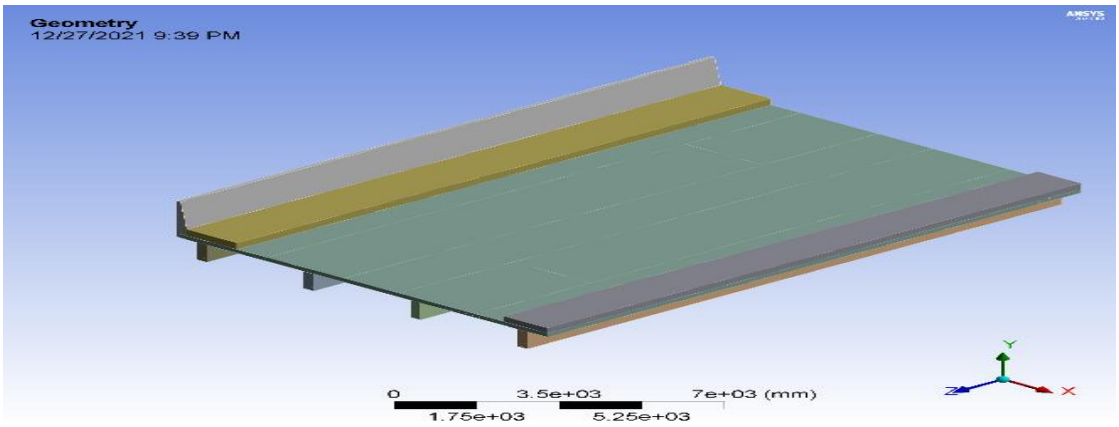


Figure 5**:** Load and Boundary Conditions

.

2.2.4 Initial and Boundary conditions

The bridge model (single span) considered was assumed to be simply supported along its width. The boundary conditions were simulated as follows:

i. Remote points were created at the start and end points of every Girder. This give the exact coordinate of the supports location in the Cartesian system, as the model is represented.

ii. The set of Remote points at the start of model were simulated as hinge/fixed supports, by arresting the three degrees of freedom in x, y, z directions, i.e. x = y = z = 0. The second sets of Remote points were simulated as Roller supports by arresting only two translational degrees of freedom in y and z directions as shown in Figure 6.

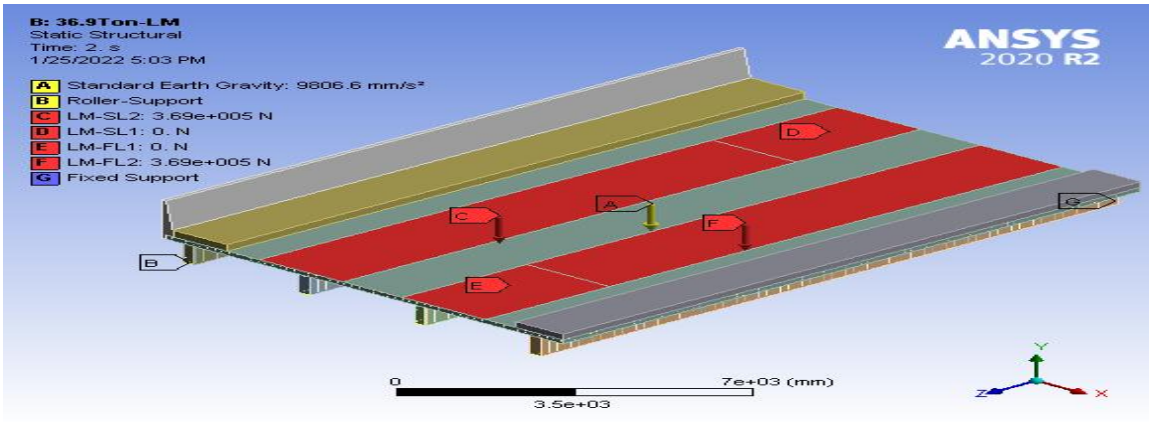


Figure 6. Boundary Conditions

2.2.5 Modelling the FLM as GWIM in ANSYS

Transient structural analysis system embedded in the Work Bench module of the ANSYS program was used, since the study involved the simulation of Moving loads (vehicles) on a static structural element(s) (bridge deck) in order to obtain the desired response (Deflection/deformation and moment) required for the evaluation.

The analysis setting to simulate the LMs as moving on the bridge deck is shown in the figure 6. The step-control was defined by sub steps, with a definite end time which defines the duration of the load on the model.

2.2.6 BS 5400 Bridge Traffic Design

The maximum 1/800 of the span length for general vehicular bridges and 1/1000 of the span length for vehicular bridges with pedestrian traffic are universally accepted criteria for the live load deflection limit, this limit will be used to evaluate the permissible deflection. Generally, deflection can be calculated by taking the double integral of the Bending Moment Equation, M(x) divided by EI (Young's Modulus X Moment of Inertia**)**.

3.0 RESULTS AND DISCUSSION

This section presents serviceability and fatigue analysis results obtained from the analytical study. Serviceability limit state results obtained from the analytical study includes: Directional deformation (deflection), total deformation, and the stresses and strain due to varies of load cycles on the bridge superstructure

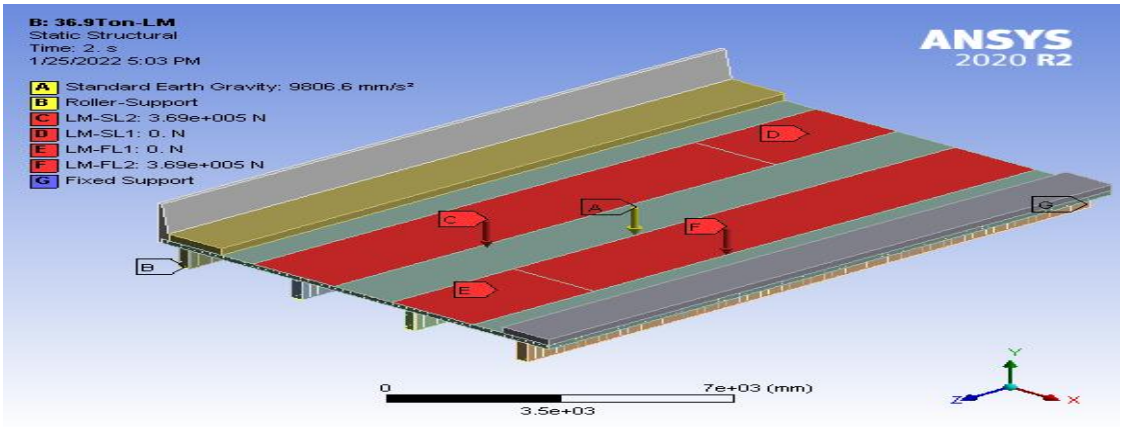


Figure 7: Simulating Moving FLM

The bridge deck system was analyzed for three responses at Serviceability limit state, i.e. the bridge directional deformation (deflection), Total deformation, and the stresses and strain due to varies of load cycles on the bridge Superstructure.

Directional/Total Deformation

Total deformation produced the resultant deformation due to the X, Y and Z component deformation of the Bridge Model, unlike the directional deformation (Deflection) which is a representation of deformation either in the X, Y or Z axis of the model.

From Figure 8, maximum total deformations on the bridge deck system was observed. More also, the resulting maximum total deformation occurs within the mid-span of the bridge, with the deck/girder total deformations of 11.634mm occurred at a load of 73.8Ton (738KN) and the minimum 8.148mm at LM 36.9 Ton (369KN) respectively. However, it was also observed that, the higher the load from LM, the higher is the total deformation along the bridge deck/guiders and the more sustainable to deflection. The difference in the total deformation observed from the passage of the vehicular load increments, is owing to their gross difference in Weight, which represents a 50% difference and as well accounts for 50% difference in their Total deformation. In other words, the Gross weight of the LM is proportional to the Total deformation of the Bridge components. The maximum total deformation observed from simulating the LMs from the codes considered indicates that the deformations are within the acceptable B.S limits i.e. (Span800-Span1000) or (20.000mm – 16mm).

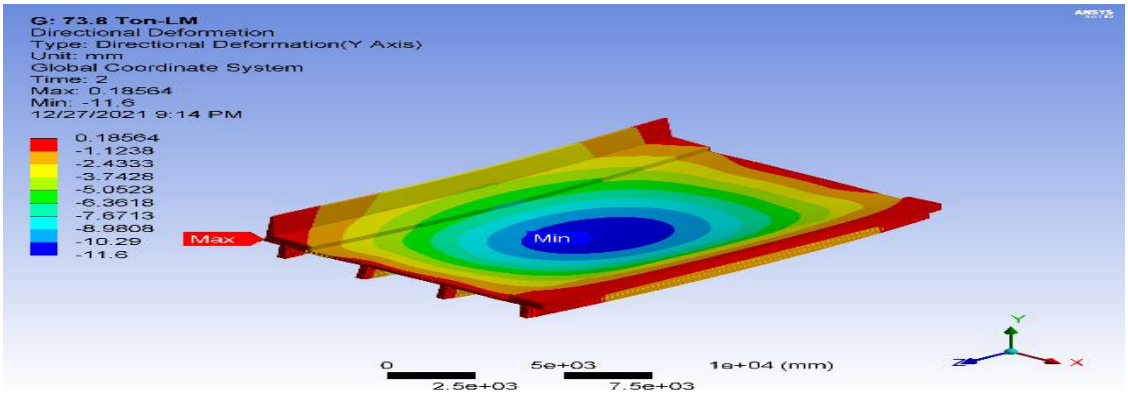


Fig. 8: Bridge Deck Directional Deformation Plot

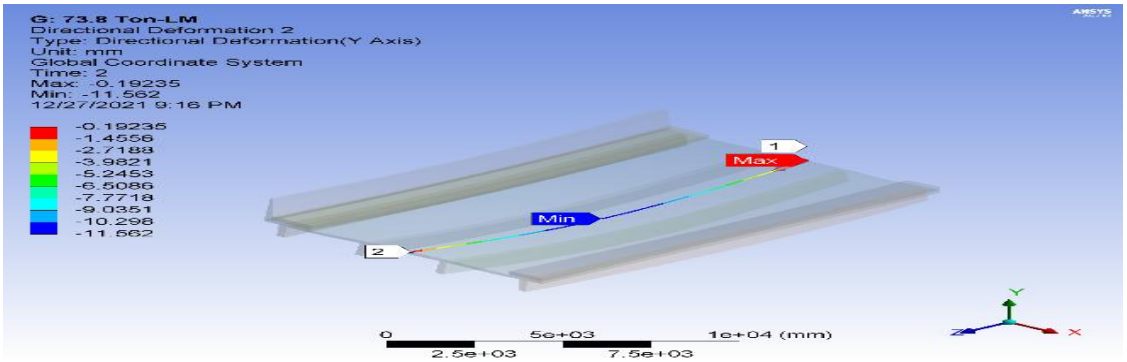


Fig. 9: Girder Critical Deformation

2. Displacement/Time-Displacement

The Load-Displacement/Time-Displacement of the bridge presented in Fig. 10 - 11 shows the relationship between the force reactions generated from the bridge due to a passage of the LM and the equivalent material response in form of deformation/Displacement within the specified simulation time frame. The combined graph for Load-Displacement, for different loading sequences from all the increments load are considered in this study.

The LM Loading sequences from all the loads produced a similar Load-Displacement curves pattern, an indication that, irrespective of the position /location of the LM, the load-displacement curve remains unchanged since the load magnitude is unchanged. Therefore, it is clear that the deformation of the Bridge deck components is largely dependent on the magnitude of the load traversing the bridge and it is not affected by the position of travel of the load. The above assertion is evident as the loading sequences in all cases of LMs, produced a greater and distinct Load-Displacement curves, showing a greater displacement of the components due to greater magnitude of loading due to LMs.

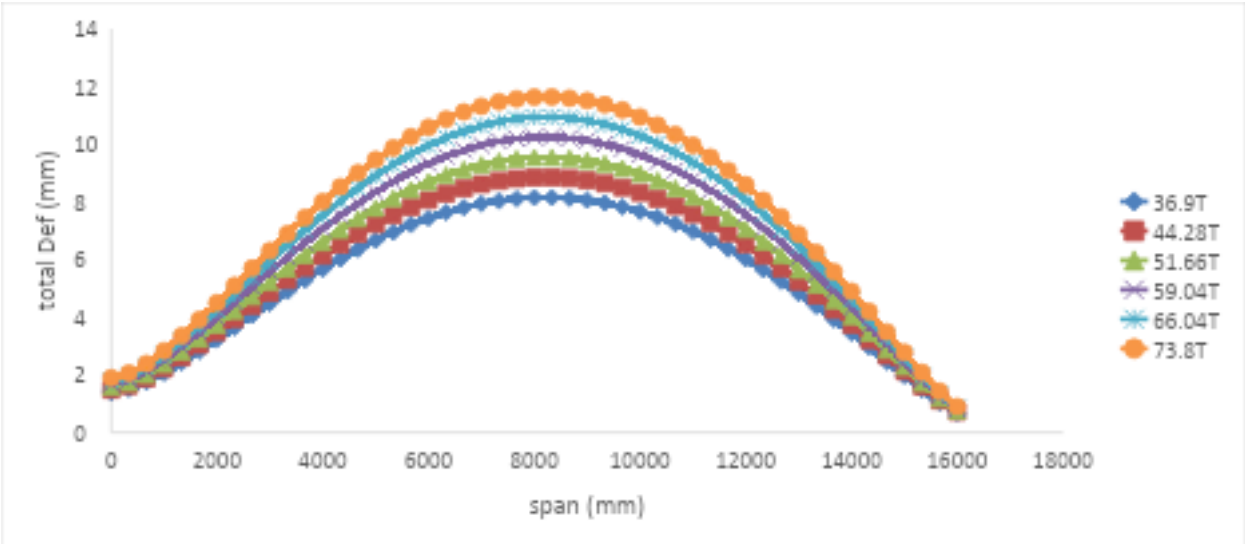


Fig. 10: Total Deformation of Bridge Deck/Girder

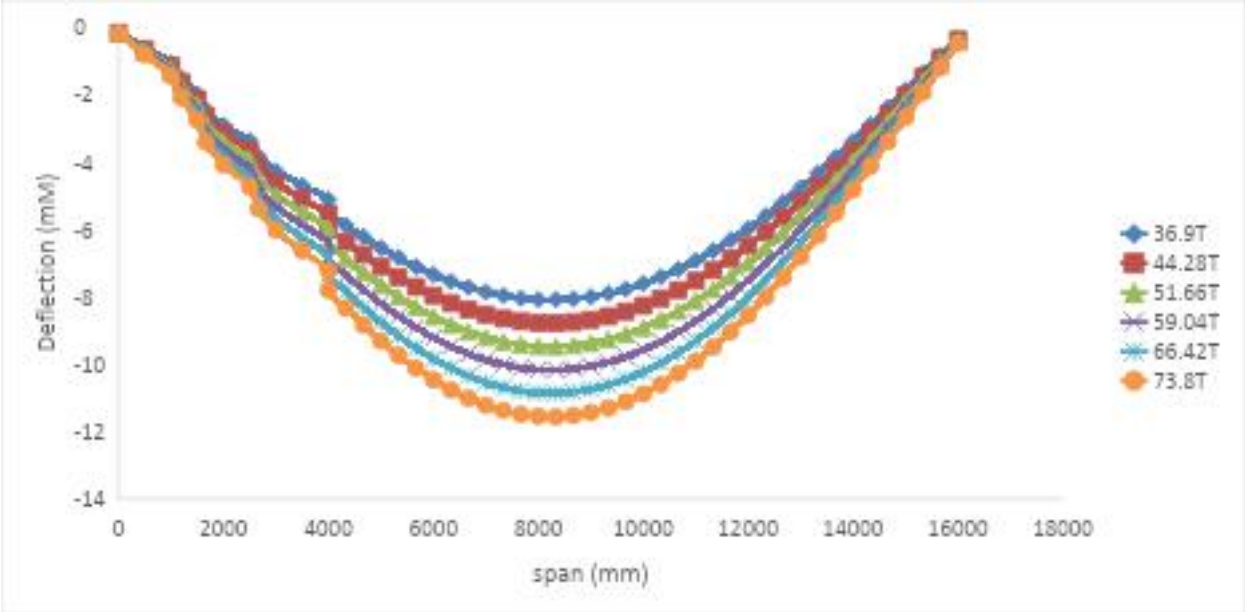


Fig.11: Deflection of Bridge Deck/Girder

From Fig 12, it is evident that the load and displacement are proportional (i.e. Hooke's law), in other words the Bridge is said to be operating within its Elastic limit.

The slight curve points observed in the charts explain the Non-linearity considered in the simulation. The slope from each curve determines the stiffness of the components.

There is no visible sign from the curves, that the materials are approaching their yield or Ultimate strengths. Therefore, the bridge deck system is deemed durable in service.

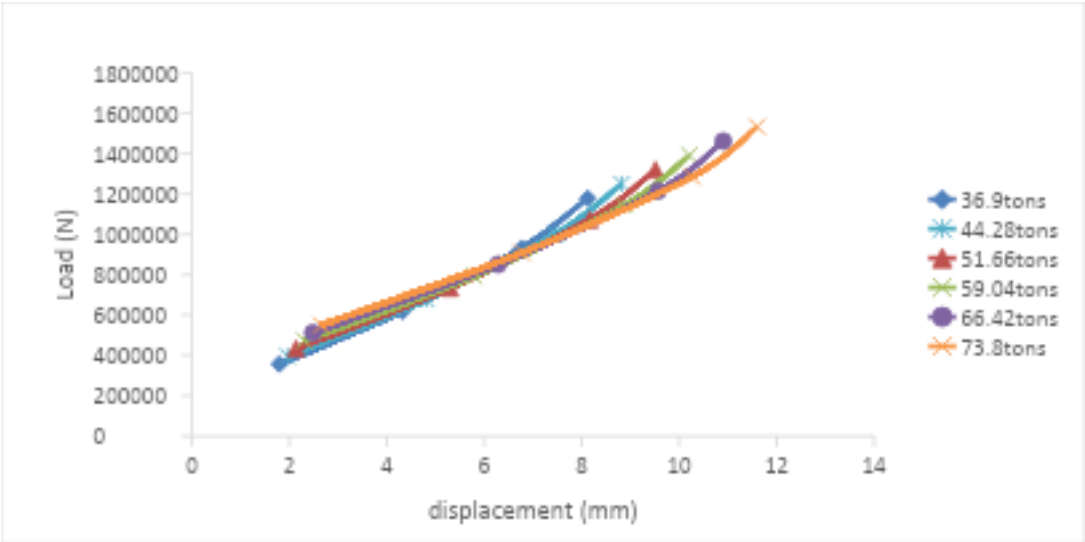


Fig. 12: Bridge Load-Displacement Performance yy (Combined)

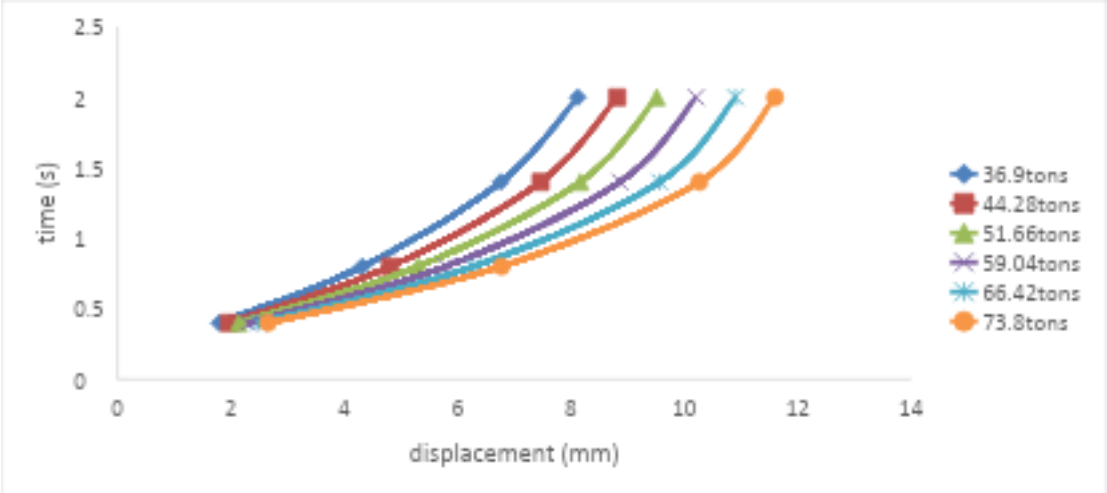


Fig. 13: Bridge Time-Displacement Performance yy**.**

3.3 Stress and Strains

Fatigue safety factor contour plots results presented for the Bridge performance in response to the applied loads of the bridge simulation analysis based on the provisions of the codes considered, the Stress Strain for all the appalled loads show that the airport road bridge is safe as the relationship is still within the elastic zone. Fig 14, show the relationship for combined stress-stain.

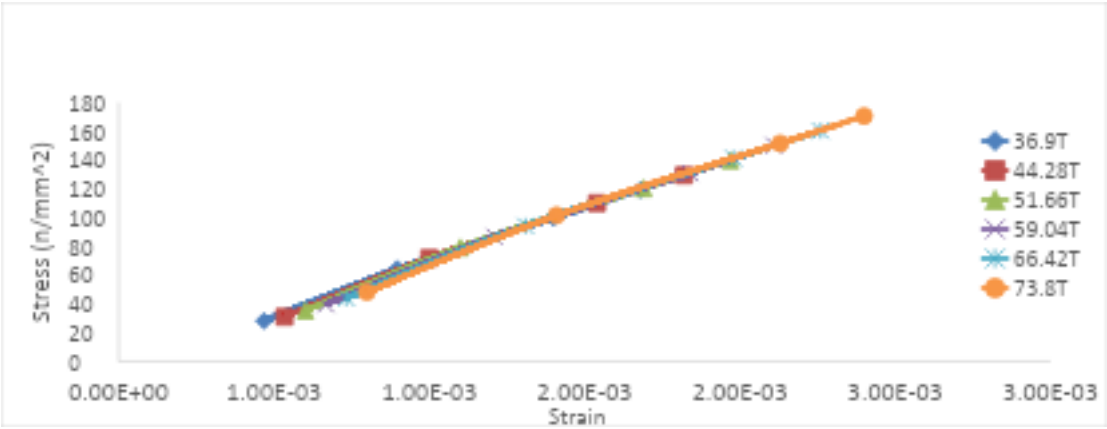


Figure 14: Bridge Stress-Strain Performance

The strain curve on the other hand presents the contour plots as the ratio of the linear distortion in the model due to passage of the LM, to the undistorted form of the bridge model as shown in figure 15

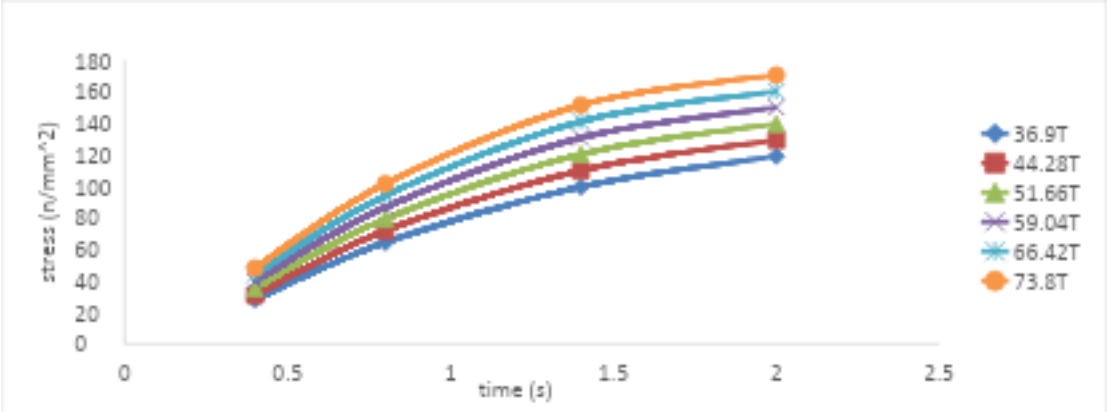


Fig. 15: Bridge Stress-Time Performance

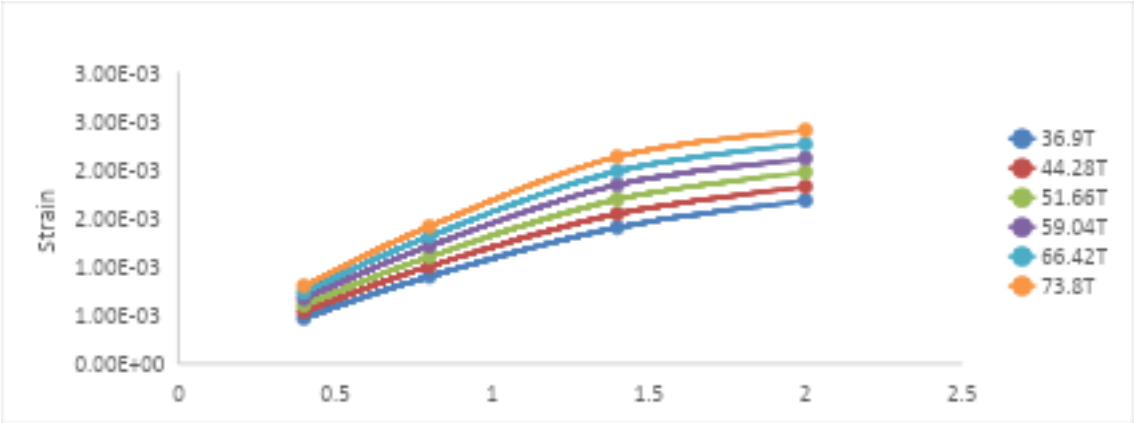


Fig. 16: Bridge Strain-Time Performance

The stress curve as presented in figure 16 shows linearity in material behavior, an indication that the bridge operates within its elastic limit.

4.0 CONCLUSIONS AND RECOMMENDATION

This study investigates the effect of heavy axle loads on bridge superstructure using ANSYS workbench. From the results of the heavy axle analysis and evaluation carried out on the case study Bridge deck super structural components based on the BS 5400 Bridge codes considered using FE approach, the following conclusions were reached:

1. The study revealed that, the deck and the girder deflections at worst loading condition show that the resulting maximum total deformation occurs within the mid-span of the bridge, with the Deck/Girder total deformations of 11.634mm occurred at a load of 73.8Ton (738KN) and the minimum 8.148mm at LM 36.9 Ton (369KN) respectively.

2. ANSYS design modeler was found to be capable of developing a 3D-model of the Airport Road Bridge deck and components as it also allows for the model to be calibrated with outputs from other external programs and adequately model and simulate concrete and steel rebars material properties taking non-linearity into cognizance

3. It was found that ANSYS transient structural analysis in the workbench platform was able to simulate the effect of moving LMs on the bridge deck and produced serviceability response such as deflection, Stress and strains that indicated that the bridge is safe in service.

4. The stress-strain analysis results of the bridge deck simulation showed a linear relationship, an indication that the bridge components operate within its service stresses.

5. The influence of axles load results obtained showed that the bridge deck components are safe, the analysis of the different axles results under the traffic loading of the LMs still possesses the maximum available strength up to 73.8T (738KN) at a varied time without failure.

6. ANSYS FE program is a good virtual tool for axle analysis of RC bridge components and should be employed as a validatory tool to validate field experiments.

7. ANSYS Workbench should be adopted for bridges of similar construction material even if the geometry differs.

REFERENCES

Analysis System, Inc. ANSYS, (2020). ANSYS Academic Research, ANSYS Swanson Product, Release 20.2, 2020.

Chang P. C., Flatau A., Liu S.-C.: Health monitoring of civil infrastructure. Structural health monitoring 2(3), 2003, 257–267.

Chen W.-F., Duan L. (2014): Bridge engineering handbook: construction and maintenance. CRC Press, 2014.

Chopra A. K.: Dynamics of structures: Theory and Applications to Earthquake Engineering. Prentice Hall, 2012.

Cunha A. et al. (2013): Recent perspectives in dynamic testing and monitoring of bridges. Structural Control and Health Monitoring 20(6), 2013, 853–877.

Fujino Y.: (2002). Vibration, control and monitoring of long-span bridges—recent research, developments and practice in Japan. Journal of Constructional Steel Research 58(1), 2002, 71–97.

Kvasnikov V., Stakhova A. (2022): Vibration Measurement Technologies and Systems. Safety in Aviation and Space Technologies: Select Proceedings of the 9th World Congress" Aviation in the XXI Century". Springer International Publishing, 2022.

Oksen E.: (2016) Defining the parameters of loading of concrete bridges superstructures basing on the level of vibroacoustic emission signals. Transportation Research Procedia 14, 2016, 3935–3942.