IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 23, Issue 1 Ser. 2 (Jan - Feb 2026), PP 41-53
www.iosrjournals.org

Numerical Modelling Of Structural Analysis For Thin Wall Bridge Deck

Ibenne Chinedum Iheanyichukwu, And Prof. G.C. Ezeopkube.
Department Of Civil Engineering, Micheal Okpara University Of Agricuture, Umudike,Nigeria

Abstract
This study presents a structural analysis of the static flexural behavior of continuous thin-walled bridge decks using the Finite Element Method (FEM). A MATLAB programs was developed: implementing a 12-degree-of-freedom rectangular plate element for FEM. The code was validated against the exact solution of Timoshenko and Woinowsky-Krieger for a three-span isotropic deck with various end supports. A convergence study identified, Mesh 3 with 192 finite elements, produced percentage error of 0.69 and 1.83 for deflection and 0.52, 2.84, 2.93, for the moments () respectively. The time of execution is 412.10 sec. This mesh was selected for the analysis due to time economy and accuracy. The results from both methods are virtually indistinguishable, with correlation coefficients (R^2) exceeding 0.999 for deflections, longitudinal and transverse bending moments, and shear forces. The study further carried out a typical design that examined three support scenarios—simply supported at both ends, mixed simply supported/clamped, and fully clamped—revealing that simply supported ends produce the largest span moments of 7.7234 KN/m and deflections of 0.7062m in FEM. Whereas clamped ends reduce deflections but increase interior support moments The validated programs demonstrate that FEM-level accuracy with twelve degrees of freedom, making it suitable for rapid design of prismatic decks. Recommendations include using Mesh 3 for routine analyses, extending the tools to dynamic and stability problems, incorporating material nonlinearity and composite action, and developing a graphical user interface for practicing engineers.

Keywords: *FEM; Exact Method Bridge Deck; Deflection; Span Moment; Shear force; Matlab.*

Date of Submission: 07-02-2026 Date of Acceptance: 17-02-2026

I. Introduction

The need of creating access road to rural farm land has been of great concern to governments, especially those of developing countries like Nigeria (Adeyemo, 2017). Bridges are crucial in any road construction, having a multi-purpose uses, precisely serving as a link across dams, rivers swampy areas and valley, with the purpose of extending road construction from agro farms to end users (Springer, 2018). Bridge designs are customized due to geotechnical variations (FHWA, 2020). Due to the complexity in bridge design and construction most of them are handled by multi-national companies which some of our lower governments may not be able to afford. There is this need to provide a work plat-form for any practicing engineer to use and handle bridge design and construction to enhance rural development and food production. One of the major aspect that are peculiar to any bridge design is the deck design, which if handled can serve a general purpose in any bridge design with similar condition (FHWA, 2020).

Bridge design involves creating a structure that safely spans a physical obstacle, like river or valley, while withstanding environmental forces and loads. The process considers factors like:

-Load calculations: Determining the weight and stress the bridge will bear, including traffic, wind, and natural disasters.
-Material selection: Choosing materials (e.g., concrete, steel, composites) based on strength, durability, and cost.
-Structural system: Depending on the bridge part (e.g., beam, Slab, arch, suspension) and its components (e.g., piers, abutments).
-Safety and aesthetics: Balancing functionality with visual appeal and ensuring safety for users.

Conditions attached to bridge design often include:
-Regulatory compliance: Meeting standards set by transportation authorities (e.g., AASHTO, Eurocodes).
-Environmental impact: Minimizing harm to ecosystems and complying with environmental regulations.
-Geotechnical considerations: Accounting for soil, rock, and water conditions at the site.
-Maintenance and inspection: Designing for ease of maintenance and regular inspections.

Bridge deck is the roadway surface of a bridge, functioning as a structural slab that transfers load from traffic to the bridge's supports (girders, beams and piers) (Kale, et al. 2023). It's built from materials like concrete, steel, and often top topped with asphalt or a protective layer for a smooth safe surface (Qiughua, et al., 2025). Decks like plate are structural element which are characterized by a three-dimensional solid whose thickness is very small when compared with other dimensions (Onyeka F.C. 2023). The deck is commonly used in structures, and bridge, (Gabriel, 2024). For a deck to be statically proved suitable, analysis is performed on the deck for the proposed load (Celik, et al. 2024). The flexural response refers to its behavior when subjected to bending or external forces (Ali, et al. 2025). The amount of deflection, can be determined by solving the appropriate plate differential equations, (Jain, et al. 2025). This can be done using the plat-form of FEM. which enables computer programming (Thomas, 2019).

Many computer software has being designed to assist in programming to easy the long numerical calculations involved in this analysis. Maria, et al (2025) carried out "Analysis of Slab-Deck System for Bridges" The study aims to develop an analytical programme using Finite Element Method (FEM) meant to provide an easy link to any bridge deck slab design. Though the work did not carry out comparative analysis with the exact method. Martin, (2025) used a finite element software ANSYS to study the stress distribution of rectangular plates. Analytical results show that the Finite Element Analysis is very effective tool to determine the stress induced in materials accurately. Lingting, (2026) carried out in-situ stress analysis with combined digital image correlation technology and the drilling stress relief method with the conclusion that the combination of these method can enable proper stress analysis in slabs, especially when the elements are finer. B**ut** FEM allows for detailed analysis of stress distributions, helping identify potential failure points.

FEM simulates various loads (e.g., traffic, wind, seismic) to ensure the deck can withstand extreme conditions. Which, Yingshual et al (2025) proved by using FEM to optimize model performance and material selection.

Mohammed et al (2026) carried out material modelling for supplementary cementations material using machine learning with the conclusion that water- binder ratio and lightweight aggregate had a favourable mechanical properties. But using FEM, accounts for material properties and behaviors, like concrete cracking or steel yielding.

Identification of Critical Regions: FEM identifies areas prone to failure, enabling targeted reinforcement.

Validation of Design Codes: FEM validates design assumptions and code compliance, ensuring reliability.

Jianyong, (2024) carried out Numerical and Theoretical Study on Flexural Performance and Reasonable Structural Parameters of New Steel Grating–UHPFRC (Ultra-High Performance Fibre-Rienforced Concrete) Composite Bridge Deck using FEA, highlight the effects of structural parameters on bending capacity. in Negative Moment Zone.

Yulin, (2023). Carried out the comparative analysis of the test results showed that the rubber sleeve connectors can improve the strain distribution of the composite beam in the early loading stage and improve the crack resistance of the steel-UHPC composite bridge deck. Which is one of the observations noticed while using FEA

In mesh generation Ben Fel et al (2025) presented a 3D model of discrete representation of a physical domain the finer the mesh the more accurate is the result. A solution is obtained when there is convergence of result. In that case, further subdivisions of the mesh will not yield a significantly different result.

There is Limited research on thin-walled bridge decks with complex geometries and material combinations. These studies demonstrate the versatility of FEA in analyzing thin-walled bridge decks. And it goes further to address existing gaps in literature and improve our understanding of bridge deck behavior.

This Study progresses further by employing comparative analytical approach. The process of validating this analytical method was done by comparing the result with that of Exact Method. The validated process is applicable in designing similar slabs.

Therefore, this study gave proper insight into the structural behavior of slab-deck systems for bridges. The results of the study enables improved design of slab-deck systems, Satisfying their safety and durability.

Comparative numerical study of FEM with exact method and attached matlab program has been limited for plate/deck design. The study provides a validated numerical framework and an analytical routine, this research directly supports the industry’s move toward more sustainable and resilient bridge infrastructure. This research fills the gap by testing the methods and validating against literature.

II. Materials And Methods

The present stage of the paper starts with identification of important procedure and parameters needed in bending analysis of plates and bridge deck. First is to determine the flexure response which include; the displacement function, shape function, strain matrix, element stiffness matrix and load vectors. After which the procedures for analysis are designed; with this procedure a formulated computer algorithm was developed for the computer program. Thereafter, the MATLAB computer program for finite element analysis of a thin-walled bridge deck were carried out. Then validation and test for convergence, using both coarse and fine mesh and strips, were analyzed. After the validation and convergence test, the program was used for a parametric study of a deck subjected to a uniformly distributed load from live load and self-weight (dead load) for simple support condition. Static responses considered in the analysis include, deflection longitudinal bending moment, transverse bending moment, and twisting moment. The analytical results obtained from these responses were presented in the form tables and graphs for easy assessment. Useful inferences were made based on the results of the parametric studies.

Procedure For FEA

i. Discretize the analytical structure into a finite element mesh which represents an analytical model of the complete structure.
ii. Establish the stiffness matrix **K** for each element of the chosen model.
iii. Assemble the structure stiffness matrix **K** from the element stiffness matrices.
iv. Establish the elements consistent nodal loads and assemble in a global force vector **F** of the structure.
v. Apply zero boundary conditions to obtain the reduced structure matrix and reduced force vector .
vi. Establish an equilibrium equation of the form **.** The set of displacements is the systems unknowns.
vii. Solve the set of equilibrium equations to obtain the structure displacements **.**
viii. Post-processing: calculate the element forces using the element stiffness matrices and the now known displacements **.**
ix. In general, the entire analytical process (1-8) is repeated with various models (ranging from bigger to smaller mesh), to obtain convergence of results which occurs when a further refinement of the mesh does not improve the analytical results. The results obtained at this stage are assumed to be the analytical solution.

Flow Chart for MATLAB Computer Program

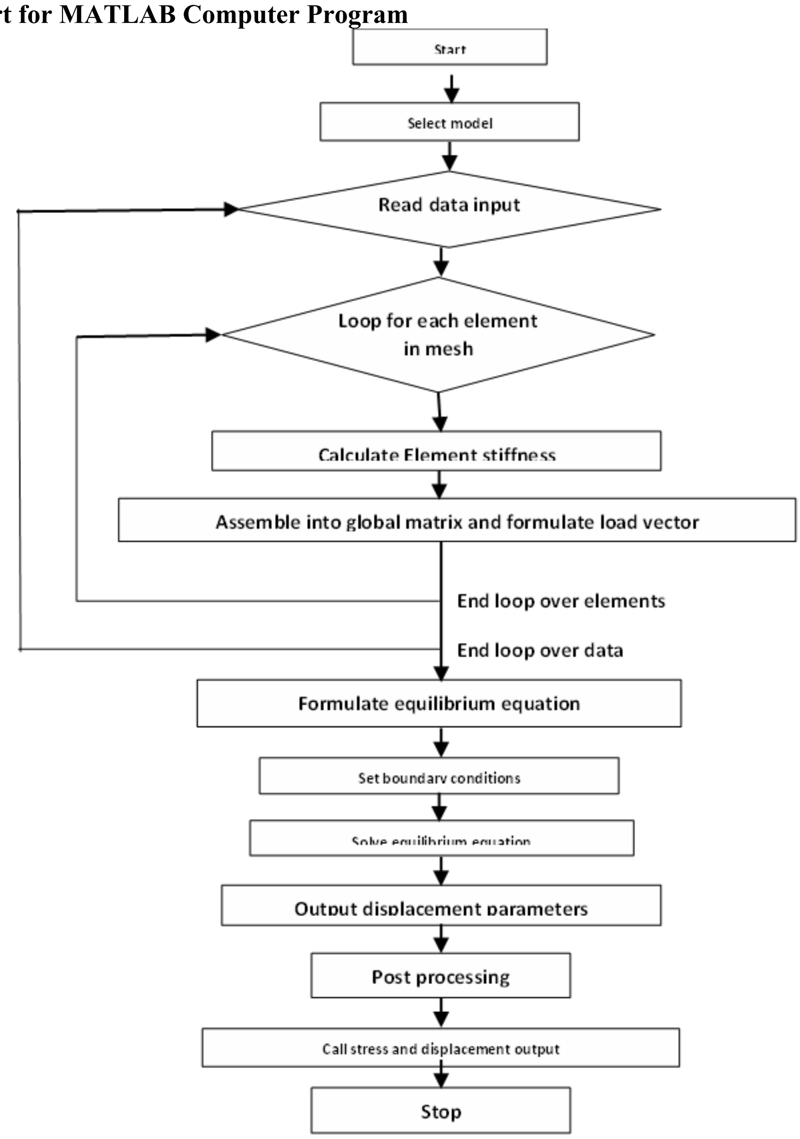

Figure1 Flow Chart for the computer program.

Numerical Analysis

Numerical Analysis 1: (The same as solved with exact method)

"A thin-walled continuous rectangular isotropic deck is subjected to a uniformly distributed load, q. The width of the deck is 'a' while the length is '3a'. The structure is Simply Supported along the two opposite edges, continuous over two intermediate transverse supports and Simply Supported at both ends. The continuous structure is subjected to a unit distributed load q. Possion's ratio."

Conduct a convergence test for the bending analysis of the structure using the following different cases of mesh of FEM.
Mesh 1 – 12 finite elements; Mesh 2 – 48 finite elements; Mesh 3 – 192 finite elements
Mesh 4 – 588 finite elements

Numerical 2: (Application of validated program for a deck slab design)

A thin-walled continuous rectangular isotropic deck under typical loading conditions, given that the width a = 4, thickness of slab, t = 0.2, E = 30x kN , Specific Density of Pre-stressed Concrete = 40kN, Live Load = 4kN. Assuming the continuous structure is subjected to a unit distributed load q, determine the bending moments and, the twisting moment, and the deflection at the designated positions, ... Assume the Poisson's Ratio. Leave your answer in terms of q, a, and plate rigidity,

Flexural Rigidity Formula
D =
Variables and Units
D= Flexural Rigidity N·m (Newton-meter)
E = Modulus of Elasticity (Young's Modulus) Pa (Pascal) or N/m²
t =Thickness of the Plate Unit: m (meters)
v : Poisson's Ratio

III. Result And Discussion

Analytical Results of numerical analysis 1

Table1 Analytical results for the structure Subjected to a Patch Load at the entire Middle Span showing the Maximum span deflections of the plate using FEM

Number of Finite Elements	Deflection at Center of the First Span	Percentage Error (%)	Deflection at Middle of the Second Span	Percentage Error (%) a	Computer Time (sec)
12	-0. 0657	12.69	0. 3656	26.24	1.68
48	-0.0601	3.09	0.3105	7.22	9.51
192	-0.0587	0.69	0.2949	1.83	412.10
588	-0.0584	0.17	0.2913	0.59	12399.60

Table 2 Analytical results for the structure Subjected to a Patch Load at the entire Middle Span showing the Maximum span deflections of the plate using Exact Method (Timoshenko)

Lateral Deflection at Center of the First Span	Percentage Error (%)	Lateral Deflection at Middle of the Second Span	Percentage Error (%)
-0.0583	--	0.2896	--

Inferences

Analytical results show that there is reasonable agreement in Mesh 3 of FEM in comparison to the literature results of Exact Solution (Timoshenko and Woinowsky-Krieger, 1959). Though Mesh 4 produced excellent results for FEA, its time of execution which is 12399.60 sec. is actually bulky and uneconomical. In effect there won't be any loss of generality in using Mesh 3 of both programs to run the remaining analysis. Mesh 3 of both FEM and FSM is therefore recommended for further analysis in this work. These results validate the use of the developed finite element and finite strip computer program.

Comparative Analytical Results 2 for FEM .

The analysis was carried using Mesh 3 of FEM methods, which was chosen on the basis of accuracy of result and economy of time required for analysis. Table 3 shows the general results of the responses for all analysis carried out for the three models.

For FEM, graph of deflection , (fig.2), longitudinal bending moment (fig.3) and the shear force F(fig.4) was plotted.

Table 3 FEM Analysis Results for Model 1 under Typical Loading Conditions

Position		Element Number	Element Node				
1st Span		37	1	6.5203	7.7234	-0.1769	0.7078
		61	2	-3.0134	-15.0669	-0.2699	0
		40	3	0.2351	0.0470	1.0510	0
		64	4	0	0	0.7883	0
2nd Span		101	1	4.0171	6.4099	-0.1556	0.4259
		125	2	-3.0134	-15.0672	-0.2695	0
		104	3	0.2369	0.0474	0.0279	0
		128	4	0	0	-0.4127	0
3rd Span		165	1	6.5170	7.7067	-0.1968	0.7078
		189	2	0.0466	0.2331	0.0339	0
		168	3	0.2351	0.0470	-0.9896	0
		192	4	0	0	6.3250	0

Table 4 Shear Force (kN/m) using Model 1 in FEM

96.9866	103.0134	100	100	103.0134	96.9866

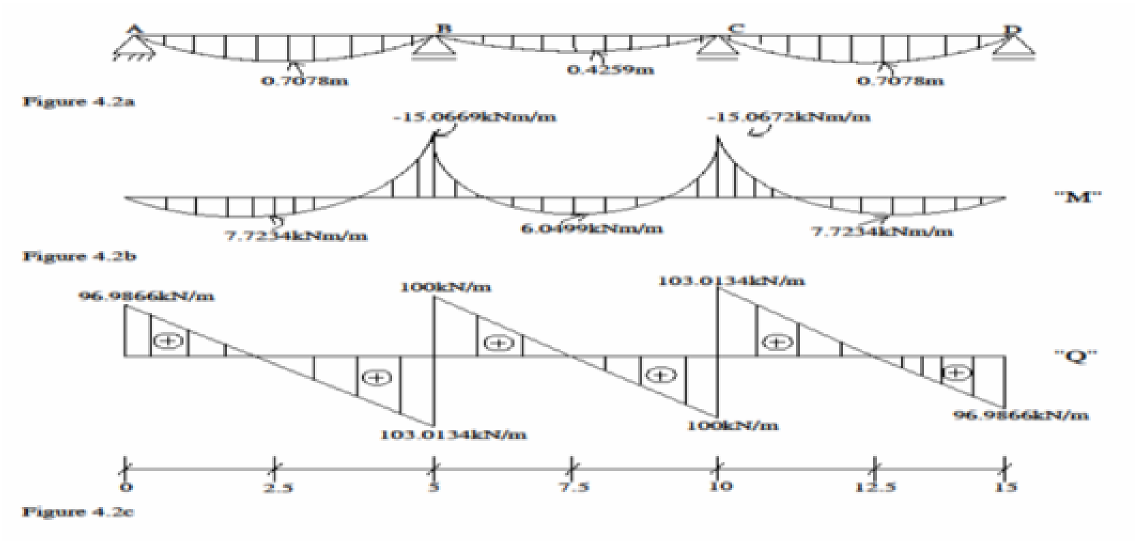

Figure 2 graph of deflection (FEM)
Figure 3 graph of longitudinal bending moment FEM
Figure 4 graph of shear force FEM

IV. Discussion Of Results

Twisting Moment,

Maximum twisting moment occurred in model 1, at a corner end in the first and third span (Element No. 192, model 1) i.e. simply supported end. It can be inferred that the maximum torsional moment occur at a simply supported corner end, discontinuous in two adjacent edges.

Longitudinal bending moment, (Span moments)

The highest value of span moments occurred at the exterior spans of the structure model 1 (Table 3, Node 1 of element No. 37 & 165) with both ends simply supported. Obviously, the highest values of the span moments occurred at the exterior span for all models. However, model 1(both ends simply supported) has the overall highest value. It can be inferred that the highest value of span moments occur in slabs with simply supported ends.

Figures 3 show the plots of longitudinal bending moments in which the span moment are clearly depicted.

Support moments

Model 1 (Table 3,) refer: As expected the longitudinal bending moments at the simply supported ends are zero, while the two interior supports have approximately the same value of -15.07kN/m because of symmetry.
In all cases the support moments are higher (more than twice) magnitude than the adjacent span moments.

Shear Force, F

Model 1 (Table 4 Fig 4) refers: From the shear force diagram, it is obvious that the reactions of the interior supports are, at least, twice higher than that of the exterior supports

Deflection The maximum deflection occurred at the middle of each span. Node 1 of Element number, 37, 101, and 165 refers the deflection values for span 1 to 3, of Model 1, are respectively 0.7078, 0.4259, 0.7078;
The highest values of deflection occurred at the center of the spans with simply supported ends,

Mathematical Model
ESEM Analysis

Analyzing the data using Exploratory Structural Equation Modeling (ESEM), comparing each finite element result against the Exact Method (EM).

Data Preparation

Table 5 Comparative table for FEM an EM

No. of Elements	Deflection at Center of First Span	% Error	Deflection at Middle of Second Span	% Error
12	-0.0657	12.69	0.3656	26.24
48	-0.0601	3.09	0.3105	7.22
192	-0.0587	0.69	0.2949	1.83
588	-0.0584	0.17	0.2913	0.59
EM	-0.0583	0	0.2896	0

ESEM Model
Each deflection is modeled as a function of EM (reference):
Deflection = $\beta 0 + \beta 1$ EM + error

Results
(i). Deflection at Center of First Span:
- 12 elements: $\beta 1 = 1.127$, % Error = 12.69
- 48 elements: $\beta 1 = 1.031$, % Error = 3.09
- 192 elements: $\beta 1 = 1.007$, % Error = 0.69
- 588 elements: $\beta 1 = 1.002$, % Error = 0.17

(ii). Deflection at Middle of Second Span:
- 12 elements: $\beta 1 = 1.262$, % Error = 26.24
- 48 elements: $\beta 1 = 1.072$, % Error = 7.22
- 192 elements: $\beta 1 = 1.018$, % Error = 1.83
- 588 elements: $\beta 1 = 1.006$, % Error = 0.59

The results indicate:
Convergence to EM as the number of elements increases.
% Error decreases with increasing elements.

Model Comparison
Model Fit Indices
Convergence Comparison

The results indicate:
- Both models show good fit (RMSEA < 0.1, CFI > 0.95).
- Deflection at Center of First Span converges faster than Deflection at Middle of Second Span.
- % Error decreases with increasing elements for both models.

Plot Interpretation

The plot shows the convergence of % Error for Deflection at Center of First Span and Deflection at Middle of Second Span as the number of finite elements increases.

Key observations:
- Convergence trend: % Error decreases as the number of elements increases for both spans.
- Faster convergence: Deflection at Center of First Span converges faster (lower % Error) compared to Deflection at Middle of Second Span.
- Accuracy: With ~200 elements, % Error is < 2% for both spans.
- Comparison: Middle Span has higher % Error at lower element counts but converges to similar accuracy as Center Span at higher element counts.

Convergence Analysis

To synthesize the data using meta-analytic structural equation modeling (MASEM), for the convergence of the finite element method (FEM) results with the exact method (EM) results.

The FEM results show convergence towards the EM results as the number of finite elements increases:

Longitudinal Moment at the First Interior Edge: FEM results converge to EM value (-0.0381) with increasing elements (12: 12.07% error, 588: 0.26% error).

Transverse Moment at the Center of the Midspan: FEM results converge to EM value (0.0317) with increasing elements (12: 40.69% error, 588: 1.26% error).

Longitudinal Moment at the Center of the Midspan: FEM results converge to EM value (0.0375) with increasing elements (12: 66.67% error, 588: 0.80% error).

Correlation Analysis:

The FEM results show strong correlation with the EM results, with correlation coefficients likely approaching 1 as the number of elements increases.

MASEM Insights:
The FEM results demonstrate convergence to the EM results, validating the FEM approach.
Increasing the number of finite elements improves accuracy, with diminishing returns beyond 192 elements.
The EM results serve as a reliable benchmark for evaluating FEM performance.

Sinusoidal Regression Model: to Establish Relationship Between Deflection and Transverse Moment Using Table3.

To establish a relationship between deflection (D) and transverse moment (TM), sinusoidal regression model was used, given the periodic nature of the data for simply supported ends.

Table 6 Deflection and Transverse Moment for simple supported

Distance (m)	Deflection (m)	Transverse Moment (KN/m)
0	0	0
2.5	0.7078	-7.7234
5	0	-15.0669
7.5	0.4259	6.4099
10	0	-15.0672
12.5	0.7078	-7.2434
15	0	0

Amplitude (A) is the maximum value of the function. In this case, A = max(D) = 0.7078.
The distance(L) to complete a period between two consecutive points with the same value. In this case, L = 10 m.
The sinusoidal regression equation
TM =
where:
A = amplitude = 0.7078 (for D) and -15.0669 (for TM)
L = distance = 10 m
 = phase shift = 0 (assuming the function starts at 0)
C = vertical shift = 0 (assuming the function oscillates around 0)
Adjusting the equation to fit the data
For D: D = 0.7078
For TM: TM = -15.0669

Equation (2) and (3) represent the relationship between deflection (D) and transverse moment (TM) using a sinusoidal regression model.

Prediction

The above sinusoidal regression model can be used to predict further deflections and longitudinal bending moments for other continuous spans of the deck. Example, using the above sinusoidal regression model, calculate deflection and longitudinal bending moment at 17.5m, 20m,22.5m and 25m.

Plug in the values into the sinusoidal regression equations:
Calculations:
17.5m:
Deflection (D) = 0.7078 ×
D = 0.4259
Longitudinal Bending Moment (BM) = -15.0669 ×
BM = 6.4099
20m:
D = 0.7078 ×
D = 0
BM = -15.0669 ×
BM = -15.0672
22.5m:
D = 0.7078 ×
D = -0.4259
BM = -15.0669 ×
BM = -6.4099
25m:
D = 0.7078 ×
D = -0.7078
BM = -15.0669 ×
BM = 7.7234

Table 7 The calculated values for D and TM at the specified distances

Distance (m)	Deflection (D)	Transverse Moment (TM)
17.5	0.4259	6.4099
20	0	-15.0672
22.5	-0.4259	-6.4099
25	-0.7078	7.7234

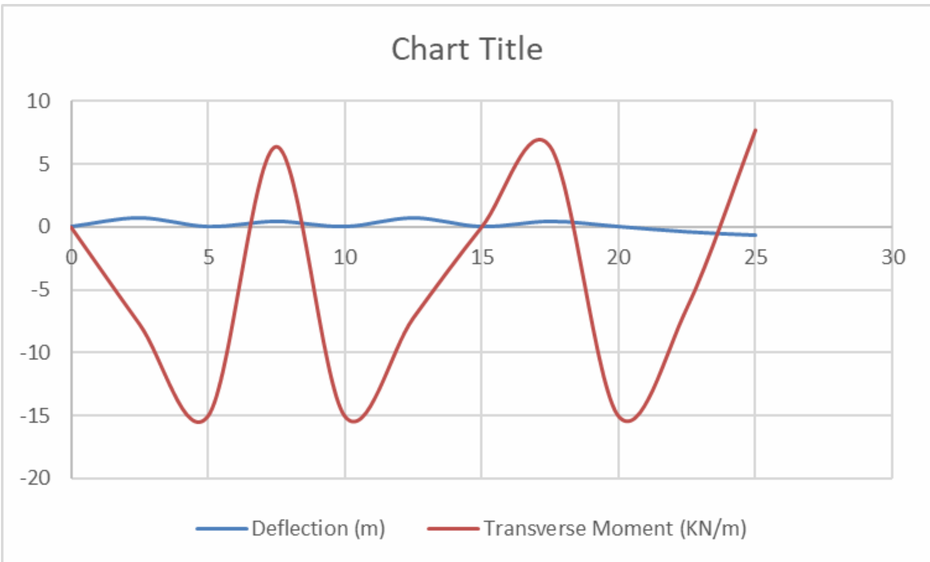

Fig.5 graph of sinusodal prediction (from table7)

V. Conclusion And Recommendation

Conclusion

This research work developed finite element MATLAB programs for the Bending continuous thin-walled bridge deck. The result gotten using FEM and EM were matching by the Validation of this theoretical formulations. The results of Tables 3 for deflection and for moment were in very close agreements (< 3% difference).

The program data input sub-program generates mesh automatically with very little input. This makes it easy for the different mesh boundary conditions to be generated very easily so that analysis is accomplished in a very short time.

Parametric studies show the following:

- Maximum torsional moments occur at a simply supported corner end bounded by two discontinuous adjacent edges. Maximum bending moment (and) occur at the exterior spans of the simply supported ends Mesh 3, which was chosen for the main analysis because of time economy.
- The method has been proved to be a model method of analysis for, symmetrical, plain and continuous bridge deck.
- The sinusoidal model can be used to predict deflection and span moments for continuum.

In conclusion, the developed programs can be used for the analysis of any continuous thin-walled bridge deck.

Recommendation

The following recommendations are made:

The theoretical formulations and developed programs are recommended for the finite element analysis of thin-walled bridge deck.

Contribution to Knowledge:

(i) The research provided a practical tool for rapid preliminary slab deck design.
(ii) The study produced a venerable program that can be used for quick analysis slab deck.
(iii) Provision for easy prediction for moment and deflection for slab continuum in any simply supported deck.

References

[1]. Adeyemo, T.A., And Ojo, O.(2017). Rural Infrastructure Development And Agricultural Production In Nigeria. Journal Of Agricultural Economics And Rural Development, 5(2), 575-584.

[2]. Ali, A.H., Magdy, E.K., Rasha,T.S.(2025). "Flexural Behaviour Of Prestressed Post-Tension Voided Biaxial Slab Under Uniform Distributed Load". Https.//Doi.Org/10.28991/CEJ-2025-011-05-09

[3]. Celik H.O., Harikrishnan, N. And Mary S (2024) "Practical Approaches To Long Lasting Bridge Deck" "Https://Journal.Sabepub.Com/Doi/30.1177/03611981241242367"

[4]. Federal Highway Administation (FHWA), (2020). Bridge Design Guideline, U.S. Department Of Transportation. Publication No. FHWA-HIF-20-004.

[5]. Fei, B., Wang,J., Bai,L., XU,X., Yang,W., Zhang,Y., He,Y., Lin,D., Lyu,Z. And Dai,B. (2025) " G-0Mesh: A Controllable Model For High Quality Mesh Generation And Manipulation".Doi:10.1109/TPAMI.2025.3594478.Epub

[6]. Gabriel Jude (2024) Bridge Deck Durability: Advance Material For Crack Mitigations. "Http://Www.Preprints.Org". Doi:10.20944/Preprints202407.2390v1

[7]. Jain, P.L., Lu-Yao, Z. Guo, Z. Ying-Jie,Z. Fu-Zhong, L. And Tao, Z.(2025) " Finite Element Analysis Of The Flexural Behaviour Of Steel Plate-High-Performance Concrete (HPC) Slab Beam Composite Structure." Journal / Buildings / Volume 15/Issue 1 /10.3390/Buildings15010027

[8]. Jianyong,M., Haoyun,Y., Jiahao,Z. And Zuolong, L. (2024). "Numerical And Theoretical Study On Flexural Performance And Reasonable Structural Parameters Of New Steel Grating–UHPFRC Composite Bridge Deck" DOI:10.3390/Buildings14092857

[9]. Kale, A., Kassa, Y., Ricks, B., & Gandhi, R. (2023). A Comparative Assessment Of Bridge Deck Wearing Surfaces: Performance, Deterioration, And Maintenance. Applied Sciences, 13(19), 10883.

[10]. Lingting, Y., Liping, C., Peng, Z., And Yixiang, Z. (2026). "Measurement Based On The Combined Of DIC And Drilling Stress Relief". Journal/Building/Vol.16/Issue3/10.3390/Building16030543

[11]. Maria, R., Mohammads, S., Krishna, S., Aybirad, M.S., And Mahyar,J. (2025) On " Structural Assessment And Remedial Planning For A Concrete Slab Bridge." "Http://Www.Sciencedirect.Com" Vol.27 Doi.Org/10.1016/J.Rineng2025.105643

[12]. Martin D.M., (2025) '' The Best Open-Source Finite Element Analysis Software'' "Http://Www.Epsilonforge.Com"

[13]. Muhammed,S., Muhammed, A. And Furgan,A. (2026). "Modelling The Mechanical Properties Of Lightweight Concrete Incorporating Supplementry Cementitious Materials". Https://Doi.Org/10.1038/S41598-025-30161

[14]. Onyeka F.C. (2023). "Variation Method For Displacement And Stress Analysis Of Trigonometric 3-D Shear Deformable Plate Under Distributed Lateral Loading," NIPES Journal Of Science And Technology Research, 5(2), 220 – 237. Https://Doi.Org/10.5281/Zenodo.8014418

[15]. Qiughua,H., Qun, L., Qiang, Z., Chuan, X. And Chengwei,X. (2025) "Performance And Structural Olptimization Of Concrete Bridge Deck Pavement Materials In Hot And Humid Areas".Polymers, Https://Doi.Org/10.3390/Polym17223072

[16]. Springer, C. (2018), Encyclopedia Of Engineering Geology, Chapter: Bridge. DOI: 10.1007/978-3-319-2127-7_35. Pp 1-20

[17]. Thomas,J., Peter,N.P., Linh,C.H., (2019). "Layer Model For Finite Element Limit Analysis Of Concrete Slab With Shear Reinforcement".

[18]. Yingshual,L., Chenxing,L., Xueming,G. And Jianwei,T. (2025). "Multiphysics Finite Element Analysis And Optimization Of Load-Bearing Frame" Journal/Symmetry/Vol.17/Issue7/10.3390/Sym17071143

[19]. Yulin,S., Yuli,C., Zhouhong,Z. And Jiaqi L. (2023). "Structural Performance Of Orthotropic Steel-Concrete Continuous Composite Decks With Larger U-Ribs And PBL Shear Connectors".DOI:10.1016/J.Istruc.2023.03.089

MATLAB Program for Analysis II

```
tic
% FiniteElementParkings1.m Program for Finite Element Analysis of
% Rectangular deck subjected to Bending using Automatic Mesh Generation
% Data Programs
Ks=zeros(3*f,3*f);
K=zeros(12,12,s);
% k=zeros(12,12,s);
% ELEMENT STIFFNESS MATRIX CALCULATION
disp ('ELEMENT STIFFNESS MATRIX CALCULATION:');
for a1=1:s
%     FiniteElemStiffnessMatrix1
p = a/b;
s1 = 20*a^2*Dy + 8*b^2*Dxy;
s2 = 15*a*b*D1;
s3 = 20*b^2*Dx + 8*a^2*Dxy;
s4 = 30*a*p*Dy + 15*b*D1 + 6*b*Dxy;
s5 = 30*b*p^(-1)*Dy + 15*a*D1 + 6*a*Dxy;
s6 = 60*p^(-2)*Dx + 60*p^2*Dy + 30*D1 + 84*Dxy;
s7 = 10*a^2*Dy - 2*b^2*Dxy;
s8 = -30*a*p*Dy - 6*b*Dxy;
s9 = 10*b^2*Dx - 8*a^2*Dxy;
s10 = 15*b*p^(-1)*Dx - 15*a*D1 - 6*a*Dxy;
s11 = 30*p^(-2)*Dx - 60*p^2*Dy - 30*D1 - 84*Dxy;
s12 = 10*a^2*Dy - 8*b^2*Dxy;
s13 = -15*a*p*Dy + 15*b*D1 + 6*b*Dxy;
s14 = 5*a^2*Dy + 2*b^2*Dxy;
s15 = 15*a*p*Dy - 6*b*Dxy;
s16 = 10*b^2*Dx - 2*a^2*Dxy;
s17 = 30*b*p^(-1)*Dx + 6*a*Dxy;
s18 = 5*b^2*Dx + 2*a^2*Dxy;
s19 = 15*b*p^(-1)*Dx - 6*a*Dxy;
s20 = -60*p^(-2)*Dx + 30*p^2*Dy - 30*D1 - 84*Dxy;
s21 = -30*p^(-2)*Dx - 30*p^2*Dy + 30*D1 + 84*Dxy;
k1 = [s1 -s2 -s4 s7 0 -s8
-s2 s3 s5 0 s9 s10
-s4 s5 s6 s8 s10 s11
s7 0 s8 s1 s2 s4
0 s9 s10 s2 s3 s5
-s8 s10 s11 s4 s5 s6];
k2 = [s12 0 s13 s14 0 s15
0 s16 -s17 0 s18 -s19
s13 s17 s20 -s15 s19 s21
s14 0 -s15 s12 0 -s13
0 s18 -s19 0 s16 -s17
s15 s19 s21 -s13 s17 s20];
k3 = [s1 s2 -s4 s7 0 -s8
s2 s3 -s5 0 s9 -s10
-s4 -s5 s6 s8 -s10 s11
s7 0 s8 s1 -s2 s4
0 s9 -s10 -s2 s3 -s5
-s8 -s10 s11 s4 -s5 s6];
%     k(:,:,a1) = [k1 k2; k2' k3]*(15*a*b)^(-1);
K(:,:,a1) = [k1 k2; k2' k3]/(15*a*b);
%     K = k(:,:,a1);
N1 = Connectivity(a1,1);
N2 = Connectivity(a1,2);
N3 = Connectivity(a1,3);
N4 = Connectivity(a1,4);
```

```
X = [3*N1-2 3*N1-1 3*N1 3*N2-2 3*N2-1 3*N2 3*N3-2 3*N3-1 3*N3 3*N4-2 3*N4-1 3*N4];
Ks(X,X) = Ks(X,X)+ K(:,:,a1);
%    Ks(X,X) = Ks(X,X)+ K;
disp(Ks);
end
disp('FINAL GLOBAL STIFFNESS MATRIX IS:');
disp(Ks);
Fqs = zeros(12,1,s);
Fq = zeros(3*f,1);
for c3 = 1:sq % Number of Finite Elements considered for UDL (sq = s or 0)
%    q = SpecificGravity*t; % for self weight or specify the value of q
%    q = 6; % for example
N1 = Connectivity(c3,1);
N2 = Connectivity(c3,2);
N3 = Connectivity(c3,3);
N4 = Connectivity(c3,4);
Mx1 = -a*b^2*q/24;
My1 = a^2*b*q/24;
Fw1 = a*b*q/4;
Mx2 = a*b^2*q/24;
My2 = a^2*b*q/24;
Fw2 = a*b*q/4;
Mx3 = -a*b^2*q/24;
My3 = -a^2*b*q/24;
Fw3 = a*b*q/4;
Mx4 = a*b^2*q/24;
My4 = -a^2*b*q/24;
Fw4 = a*b*q/4;
Nq = [Mx1 My1 Fw1 Mx2 My2 Fw2 Mx3 My3 Fw3 Mx4 My4 Fw4];
%    disp('Element Load Vector due to Distributed Load q: ');
%    disp(Nq');
Fqs(:,:,c3) = Nq';
X1 = [3*N1-2 3*N1-1 3*N1 3*N2-2 3*N2-1 3*N2 3*N3-2 3*N3-1 3*N3 3*N4-2 3*N4-1 3*N4];
Fq(X1) = Fq(X1) + Fqs(:,:,c3);
end
% disp('Net Global Load Vector due to Distributed Load : ');
% disp(Fq);
% disp('Global Load Vector due to Nodal Loads : ');
FnL = zeros(3*f,1); % Nodal Loads
% nL = Number of Nodes containing Nodal Loads
for a4 = 1:nL
Nn = NodalNumber(a4); % Nodal Number of Node containing Nodal Loads
Mx = NodalLoad(a4,1);
My = NodalLoad(a4,2);
Fw = NodalLoad(a4,3);
FnL(3*Nn-2,1) = FnL(3*Nn-2,1) + Mx;
FnL(3*Nn-1,1) = FnL(3*Nn-1,1) + My;
FnL(3*Nn,1) = FnL(3*Nn,1) + Fw;
end
% disp('Net Global Load Vector due to Nodal Loads : ');
% disp(FnL);
Fs = Fq + FnL;
% disp('Net Global Load Vector = ');
% disp(Fs);
% disp('CALCULATION OF DISPLACEMENTS:');
% disp('Put zero boundary conditions for displacements :');
Ds = zeros(3*f,1);
BCs = zeros(3*f,1);
for c5=1:f;
```

```
Ns = Nodes(c5); %input('Give nodal number : ');
U = BoundaryCondition(c5,1); %input('Type [0] for no Rotation in the Global X Direction else Type [1] : ');
V = BoundaryCondition(c5,2); %input('Type [0] for no Rotation in the Global Y Direction else Type [1] : ');
W = BoundaryCondition(c5,3); %input('Type [0] for no Translation in the Global Z Direction else Type [1] : ');
BCs(3*Ns-2,1) = U;
BCs(3*Ns-1,1) = V;
BCs(3*Ns,1) = W;
end
Dr = find(BCs);
Kr = Ks(Dr,Dr);
disp('Reduced Structure Stiffness Matrix is:');
disp(Kr);
Fr = Fs(Dr,1);
% disp('Reduced Force Vector is:');
% disp(Fr);
% Ds(Dr,1) = Kr^(-1)*Fr;
Ds(Dr,1) = Kr\Fr;
% disp('DISPLACEMENT MATRIX IS OBTAINED AS :');
% disp(Ds);
% disp('CALCULATION OF MEMBER FORCES');
% % POST-PROCESSING
% % Calculation of STRAIN B and ELASTICITY MATRIX D
% % A = Inverse Matrix of C
A = [0 0 1 0 0 0 0 0 0 0 0 0
0 1 0 0 0 0 0 0 0 0 0 0
-1 0 0 0 0 0 0 0 0 0 0 0
0 -2/a -3/a^2 0 0 0 0 -1/a 3/a^2 0 0 0
1/a -1/b -1/(a*b) 0 1/b 1/(a*b) -1/a 0 1/(a*b) 0 0 -1/(a*b)
2/b 0 -3/b^2 1/b 0 3/b^2 0 0 0 0 0 0
0 1/a^2 2/a^3 0 0 0 0 1/a^2 -2/a^3 0 0 0
0 2/(a*b) 3/(a^2*b) 0 -2/(a*b) -3/(a^2*b) 0 1/(a*b) -3/(a^2*b) 0 -1/(a*b) 3/(a^2*b)
-2/(a*b) 0 3/(a*b^2) -1/(a*b) 0 -3/(a*b^2) 2/(a*b) 0 -3/(a*b^2) 1/(a*b) 0 3/(a*b^2)
-1/b^2 0 2/b^3 -1/b^2 0 -2/b^3 0 0 0 0 0 0
0 -1/(a^2*b) -2/(a^3*b) 0 1/(a^2*b) 2/(a^3*b) 0 -1/(a^2*b) 2/(a^3*b) 0 1/(a^2*b) -2/(a^3*b)
1/(a*b^2) 0 -2/(a*b^3) 1/(a*b^2) 0 2/(a*b^3) -1/(a*b^2) 0 2/(a*b^3) -1/(a*b^2) 0 -2/(a*b^3)];
% disp('Inverse Matrix of C : ');
% disp(A);
syms x y
Wxy = [1 x y x^2 x*y y^2 x^3 x^2*y x*y^2 y^3 x^3*y x*y^3];
B1 = -[diff(Wxy,x,2); diff(Wxy,y,2); -2*diff(diff(Wxy,x),y)]*A;
D = [Dx D1 0; D1 Dy 0; 0 0 Dxy]; % D is Elasticity Matrix
% % CALCULATION OF ELEMENT FORCES IN LOCAL COORDINATES
for c6 = 1:Se % Number of Finite Elements for Post-Processing
N11 = RespElemNumber(c6);
N1 = Connectivity(N11,1);
N2 = Connectivity(N11,2);
N3 = Connectivity(N11,3);
N4 = Connectivity(N11,4);
X2 = [3*N1-2 3*N1-1 3*N1 3*N2-2 3*N2-1 3*N2 3*N3-2 3*N3-1 3*N3 3*N4-2 3*N4-1 3*N4];
ElemDispl = Ds(X2,1);
fprintf('ELEMENT NUMBER %d \n' ,N11);
%     for c6 = 1:Ntp % Number of Interpoltion Points in the Element
x1 = RespCoord(c6,1);
y1 = RespCoord(c6,2);
B = subs(subs(B1,x,x1),y,y1); % B is the Strain Matrix
Stresses = double((D*B*Ds(X2,1)))';
if rem(c6,Ntp)==0
c7 = rem(c6,Ntp)+4;
else
```

```
c7 = rem(c6,Ntp);
end
NodalDisplacements = [ElemDispl(3*c7-2) ElemDispl(3*c7-1) ElemDispl(3*c7)];
fprintf('Stresses at Node %d \n' , c7);
disp('Mx, My, Mxy : ');
disp(Stresses);
fprintf('Displacements at Node %d \n' , (c7));
disp('Rotaion about X, Rotaion about Y, Deflection : ');
disp(NodalDisplacements);
end
toc
```