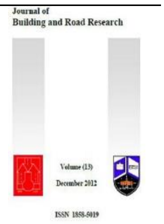
Journal of Building and Road Research Volume 21 (2019) 37- 57

Journal of Building and Road Research

http://ejournals.uofk.edu/index.php/JBRR



Experimental and Finite Element (FE) Studies on the Behaviour of Horizontally Curved Composite Bridges Deck

Badria A. M. Fageer [1,a], A. K. Ahmed [2,b] , Osama M. A. Daoud [3,c]

[1]*Department of Civil Engineering, Red Sea University,*
[2]*Department of Civil Engineering, University of Khartoum,*
[3]*Building and Road Research Institute, University of Khartoum,*

[a]B_fageery@yahoo.com, [b]akhogali@yahoo.com , [c]eng.osama72@yahoo.com

ABSTRACT

This paper aims to study the behaviour of horizontally curved composite steel-concrete bridges deck slab and beams under static load, using experimental tests and nonlinear finite element analysis. Two different specimens were examined in the structure laboratory, one of them was horizontally curved composite deck slab and steel girders with full interaction between the concrete deck and the steel girders, while the other with partial interaction. Each specimen has three curved I-girders acted parallel connected with X shape cross frames. Linear Variable Differential Transducers (LVDT) and strain gauges were used at critical points to measure deflections and strain respectively. Three dimensional finite element (FE) models with 137119 elements were built and analysed using ANSYS software version 14. Models dimensions were 1/10 in both length and radius of curvature of the Federal Highway Administration (FHWA) full scale model. Numerical results have been compared with the experimental test results. Maximum values and distribution of both stresses and strains at top flanges, bottom flanges and the web tension zone for numerical modelling results, showed close agreement with the experimental tests results in both full and partial interaction models. Also, similar failure modes were encountered from both numerical modelling and experimental tests. Maximum deflection values obtained from numerical modelling were slightly below the experimental values and this was attributed to the stiffer numerical model. Further, mesh refinement could have increase accuracy of numerical predictions.

المستخلص

تهدف هذه الورقة إلى دراسة سلوك الجسور المركبة من الفولاذ والخرسانة والمنحنية أفقياً, عن طريق التحليل اللاخطي باستخدام العناصر المحدودة والتجارب العملية والتي أجريت تحت تاثير الحمل الإستاتيكي. تم فحص عينتين مختلفتين في مختبر الانشاءات. كان أحدهما ذو ترابطا كاملا، في حين أن الآخر كان الترابط فيه جزئيا بين البلاطة الخرسانية والعارضات الفولاذية. تحتوي كل عينة على ثلاثة عارضات منحنية تعمل بشكل متوازي متصلة بإطارات عرضية على شكل X. تم استخدام المحول التفاضلي المتغير الخطي (LVDT) لقياس انحراف ومقاييس الإجهاد لقياس الضغط عند النقاط الحرجة .تم تصميم وتحليل نماذج العناصر المحدودة ثلاثية الأبعاد (FE) التي تحتوي على 137119 عنصرًا باستخدام برنامج ANSYS إصدار 14. كانت ابعاد النماذج 1/10 في كل من اطوال العارضات وانصاف اقطار الانحناء بالنسبة الى النموذج المستخدم من قبل الإدارة الفيدرالية للطرق بالولايات المتحدة. و بعد ذلك تمت مقارنة النتائج التحليلية مع نتائج الاختبارات التجريبية. نتائج التحليل العددي للنماذج، الخاصة بالقيم القصوى و توزيع كل من الإجهاد و الانفعال على الشفتين العلوية و السفلية و كذلك على العصب في منطقة الشد أظهرت توافقا و تقاربا مع نتائج التجارب العملية لكل من النماذج ذات الترابط الكامل و الجزئي. كذلك فان نمط الإنهياركان متشابها في كل من التحليل العددي و التجارب العملية. القيم القصوى للإزاحات الراسية التي تم الحصول عليها من البرنامج أقل بقليل مقارنة بالقيم التي تم الحصول عليها من الاختبارات وتم إرجاء هذا الفرق لزيادة جساءة نموذج المحاكاة مقارنة بالنموذج العملي. زيادة تقسيم النماذج العددية الى عناصر اكثر ربما تؤدي الى زيادة دقة التحليل العددي فيما يختص بالإزاحات الرأسية.

Keywords: horizontally curved, non-linear analysis, ANSYS software, experimental test

1 Introduction

Presently, many cities face problems about the space limitation for the transportation systems. One probable solution is to construct bridges with special configurations such as skewed bridges, C-bent column bridges and curved bridges. Horizontally curved composite bridges are among the most economical options for satisfying these demands.

Composite bridges have become a popular solution in many countries. The competitiveness of composite bridges depends on several circumstances such as site conditions, local costs of material and staff and the contractor’s experience. One major advantage compared to concrete bridges is that the steel girders can carry the weight of the form-work and the wet concrete during construction, which means that the need for temporary structures is reduced, beside many other known advantages. Structural analysis is a process to analyse a structural system to predict its responses and behaviours by using physical laws and mathematical equations. The main objective of structural analysis is to determine internal forces, stresses and deformations of structures under various load effects, [1].

Today, a few standards are available in order to help the designing of composite constructions such as 'AISC' Specification, 'AASHTO-LRFD' and Chinese Code. In any case, development of designing and analytical models for composite bridges must be followed by development of numerical and analytical models that are supported by experimental results in order to obtain the most accurate solutions. The application of finite element method 'FEM' adds a touch of attractiveness to modelling of all, including this type of structure as analytical apparatus, [2].

Numerous studies have been performed on the behavior of straight and curved composite steel girder bridges. To develop and improve a rational set of design guidelines, the Federal Highway Administration 'FHWA' initiated the curved steel bridge research project in 1992. As a part of this project, 'FHWA' constructed a full-scale model of curved steel girder bridge at its 'Turner- Fairbank' Structures Laboratory. This full- scale model made it possible to conduct numerous tests and collect a significant amount of data relating to the static behaviour of the curved girder bridge, [1]. Ching-Jen Chang [3], investigated construction simulation of curved steel I-girder bridges, by the development of a prototype software system for analysis of horizontally curved steel I-girder bridges using open-section thin-walled beam theory. Recommendations are provided for the use of three-dimensional grid idealizations in analysing curved I-girder bridge structural systems. The three-dimensional grid idealizations account for the general displacements and rotations common within complex curved I-girder bridge structures. D. Nevling; D. Linzell [4], investigated the accuracy of different levels of analysis used to predict horizontally curved steel I-girder bridge response, a field test was performed on a three-span continuous structure composed of five ASTM A572 Grade 50 steel plate girders. Donald White and Ozgur Cagri [5], investigated the overall design behaviour of a curved and highly skewed I-girder bridge by finite element analysis and three dimension Grid models. The 3D 'FEA' modelling approach applied in this study has been shown to predict experimentally measured responses in prior research. W. T. Jung, et al. [6], conducted loading tests and finite element analysis in order to examine the behaviour of curved Fibre-reinforced polymer (FRP)-concrete panel produced by 'pultrusion'. The test results reveal that FRP and concrete exhibit linear elastic behaviour until the maximum load. The parametric analysis with various FRP sections shows that the behaviour of the curved FRP-concrete composite panel depends on the web height of FRP, the spacing of the webs, the length of the flange and the radius of curvature. Jerad J. Hoffman [7], investigated the superstructure behaviour under design loading conditions, particularly design thermal loads for horizontally curved steel I-girder bridges with integral abutments through field evaluations and analytical finite element static analysis.

N. S. Ingawale1 et al. [8] investigated Parametric study of a horizontally curved bridge girder using a finite element software. The bridge is simply supported. IRC Class AA type of moving load is simulated on two lanes on the beam of span 66m, having a trapezoidal box type cross-section. A parametric study was done by varying the radius of curvature of the beam from 50 m to 250 m with the interval of 50 m to check the response of the beam like bending moment, shear force, torsional moment and deflection. The effect of L/R ratio i.e. ratio of span to radius of curvature was computed.

2 Experimental Tests

An experimental research program was conducted, to examine the behaviour of the composite curved bridges under static load. The tests were conducted using frame with a 5000 kN capacity with Static actuator (2000 kN) for static test, and two hydraulic power systems. Each test model was simply supported on a rigid- beams by a system of steel plates and rollers. The applied vertical load was distributed on six points on the tested models by a system of steel beams and plates as shown in Figure 1.

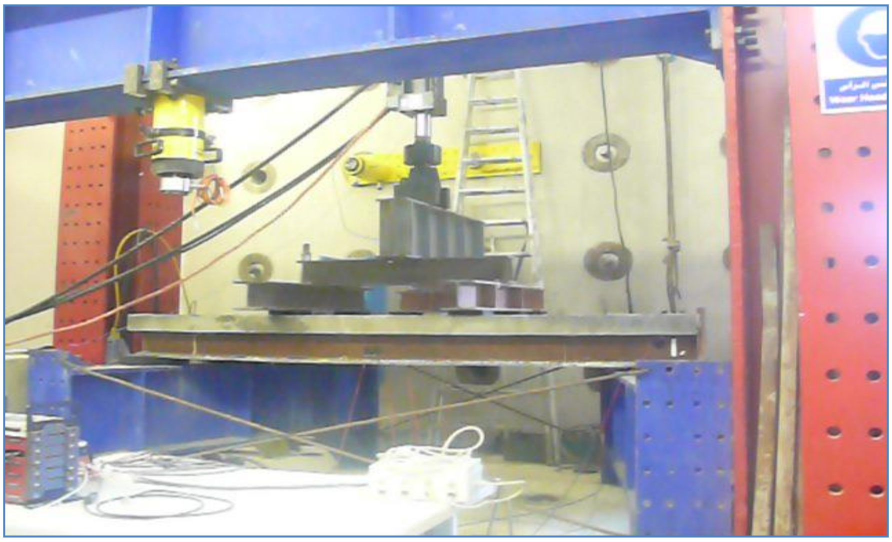


Figure 1: Test Setup

Several different types of sensors were used to acquire data relating to strain, slip, and deflections over the course of the testing. A linear variable differential transducer (LVDT) sensors were used to measure displacements. Steel and concrete Strain gauges were used to measure strain in critical positions of stresses. Additionally, data were collected from the hydraulic actuator LVDT and load cell, which provided measurements of actuator deflection and load, respectively, to be used in the analysis. All data from the strain gauges, LVDT’s, the hydraulic actuator LVDT and load cell during static testing were collected through the data acquisition system in conjunction with the computer program Strain Smart installed on a lab computer for analysis.

2.1 Material properties and details of the specimens

The specimen frame consisted of three concentric girders labelled G1, G2, and G3 with radii of curvature 4.560m, 5.673 m and 6.360 m, respectively. The lengths of centre girders of the model frame are (2.87 m, 2.6 m and 2.33 m) measured along the arcs, as shown in Figure 2 and Table 1.

Steel Grade 50 was used for all steel sections, with a mass density of 7850 kg/m^3, modulus of elasticity of 200 GPA and Poisson ratio of 0.30. The bridge slab is a conventional 100 mm thick cast-in-place concrete slab. The concrete cylinder yield strength, fc' was 28 N/mm^2, A mass density was 2400 kg/m^3, modulus of elasticity 25.379 GPa and Poisson ratio of 0.20 was used for concrete deck section.

The cross-frame members used in the design were grade 37 with 40x40x3 mm angles and the X-type configuration were chosen. Shear transfer between the girders and the concrete was provided by one and two rows - for partially interaction and full interaction respectively - of grade 37 with 13 mm diameter x 70 mm high, mild steel headed-stud type shear connectors spaced at 30 mm apart on centres. Studs were spaced longitudinally at (281-318-324 mm) on centres along the entire length of the longest beam. The deck was 100 mm thick. The final reinforcement layout utilized both ø6 and ø8 mild steel.

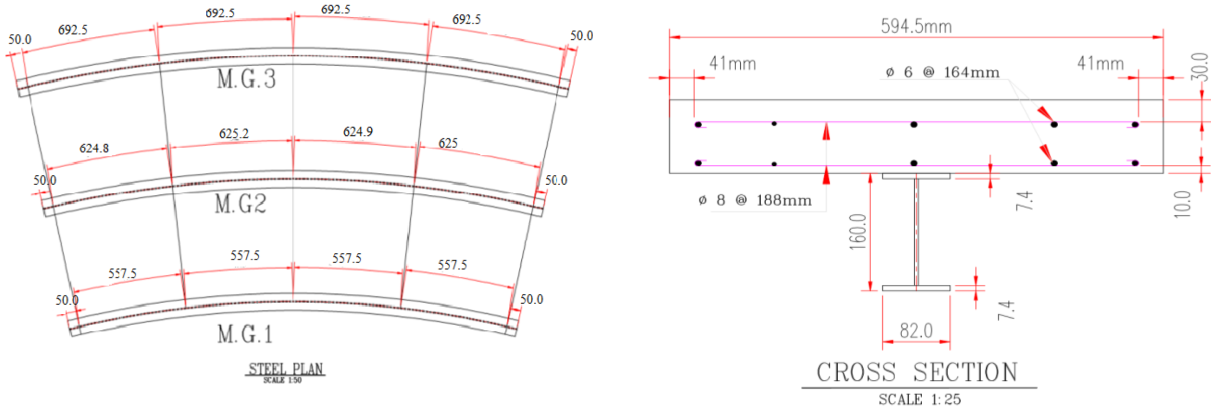


Figure 2: Specimen plan and cross-section

Table (1): Dimensions of the models

Number of girders	3 girders for each specimen
Length of spans	2.87 - 2.6 - 2.33 m
Spacing between girders	0.5945 m
Thickness of concrete deck	10 cm
Spacing between shear connecter	281-318-324 mm
Radius of curvatures	6.360 m, 5.673m, and 4.560m
Section of I-girders	IPE 160
Cover plate	150X14

2.2 Static Testing Procedure

Static test was conducted on the test specimen to determine the residual capacity (or plastic moment capacity) of each type of bridges. To generate the amount of load necessary for this test, a 200 ton capacity static hydraulic actuator was utilized. The static actuator was mounted to a steel load frame at locations directly above the test specimen steel girders, and the load applied through the distributing system setup. All instruments were zeroed following the seating of the specimen. Through the static test the load was applied in increments - tow Ton every three minutes - till collapse, ten readings were taken every one second, the full time was 97400 milliseconds for full interaction specimen.

3 Finite Element Modelling:

3.1 Modelling of the curved composite bridge geometry

To build the model geometry, first Joint coordinates for three I-girders were prepared manually to draw curved I-girders by Ansys software 'APDL' graphic, then shear connectors 'studs' installed on the upper flanges to create interaction between steel girders and concrete deck, then X shape plates used to build cross frames at the ends between three I-girder to join them together at three equal spaces from the inner side and two as diaphragms. Cross frames were installed on gusset plates, on the hard points. Contact (170-174) elements were used between the web and gusset plates, concrete deck and top flanges and the shear studs with top flanges (always bonded with friction 0.5). Two layers of curved steel reinforcement installed longitudinal and transverse directions exactly in position and dimensions alike one which created in the experimental test, so 100 mm thick concrete deck built on steel I-girder. Figure 3 shows parts of steps of building geometry of the models in Ansys 'APDL' graphics software.

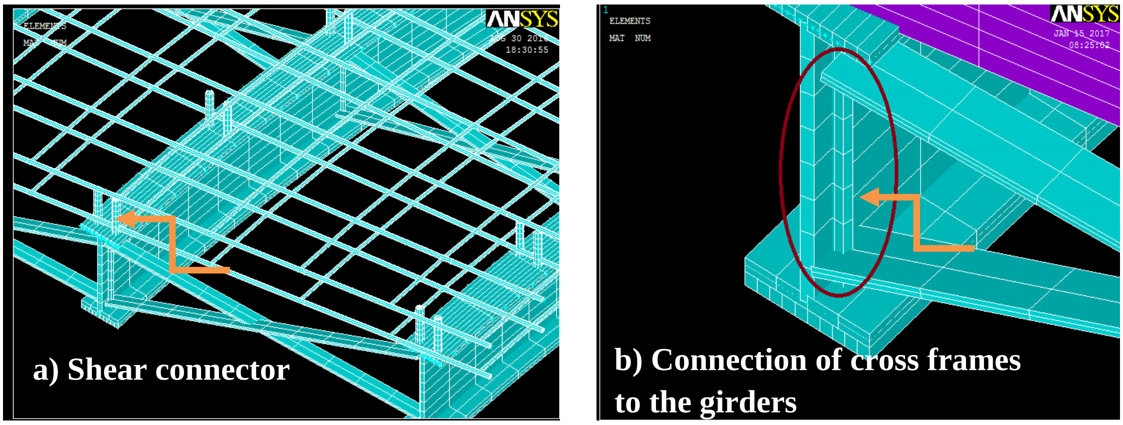


Figure 3: a) Shear connector on the top flanges, b) Cross frame fixation with the web

3.2 Elements used for modelling

Combination of elements were selected to model bridge components. Solid 186 element was used to model the concrete deck, while shell93 element was used to model the I-girders. And combin39 elements was used to model the shear connectors and Beam189 the cross frames and. The steel reinforcements were modeled using link8 elements, as shown in Table 2. The specimens' components were meshed using Ansys manual size control meshing resulting in 137119 elements, and merge node between variable elements to compose the structure configuration. The material properties and live loads used in the program were similar to values and positions of experimental test specimens.

The Solid 186 element requires linear isotropic and multi-linear isotropic material properties to properly model concrete.

The compressive uniaxial stress–strain values for the concrete model were obtained using the following equations which are used to compute the multi -linear isotropic stress–strain curve for the concrete.

$$f = \frac{E_c \varepsilon}{1 + \left(\varepsilon/\varepsilon_{\circ}\right)^2} \quad (1)$$

$$\varepsilon_o = \frac{2f_c^{\sim}}{E_c} \quad (2)$$

$$E_c = \frac{f}{\varepsilon} \quad (3)$$

Where: E_c is the modulus of elasticity of the concrete, and υ is the Poisson's ratio, f is the stress at any strain ε, ε ~ strain at stress f, ε_0 ~ strain at the ultimate compressive strength $f_c^{\sim}$.

The multi-linear isotropic stress–strain implemented curve requires that the first point of the curve to be defined by the user, this must satisfy Hooke's low.

Table (2): Element properties

Bridge component	Element used	Description
Concrete	Solid 186	Higher order 3-D 20-node solid element that exhibits quadratic displacement behavior
Reinforcement steel bars	Link 8	2 node discrete element (3 translation DOF per node)
Steel girders and cross frames Web & flanges	Shell 63	4-node shell element (3 translation DOF and 3 rotational DOF per node)
Shear friction and contact -shear connectors at top flange	Targe170 and Contact174	Nonlinear surface- to -surface interface element
Shear connectors	Beam189	Quadratic three-node beam element in 3-D. With default settings, six degrees of freedom occur at each node
Dowel Action (shear connectors inside interface)	Combin39	2-node zero length nonlinear spring element with one translational DOF per node

Steel reinforcement is modelled using element Link 8. Link8 is a uniaxial tension compression element with three degrees of freedom at each node.

The constitutive law for steel behaviour is:

$$\begin{cases} \sigma_s = E_s\varepsilon_s\ , \ \ \varepsilon_s \le \varepsilon_y, \\ \sigma_{\ s} = f_y + E_s\varepsilon_s, \ \ \ \varepsilon_s > \varepsilon_{y,} \end{cases} \tag{4}$$

Where: σ_s: is the steel stress, ε_s: the steel strain, E_s: the elastic modulus of steel, $E_s^{'}$: is the tangent modulus of steel after yielding, $E_s^{'}$ =0.01 E_s, f_y and ε_y: is the yielding stress and strain of steel, respectively.

3.3 Boundary conditions:

Simply supported boundary condition roller and hinge supports were applied at bottom flange node at each girder end. The roller support can move horizontally along tangential direction, whereas horizontal displacement of hinged support was restrained. All supports were restrained in the vertical direction, but allow rotating along support line as shown in Figure 4. Finally nonlinear analysis applied to the models.

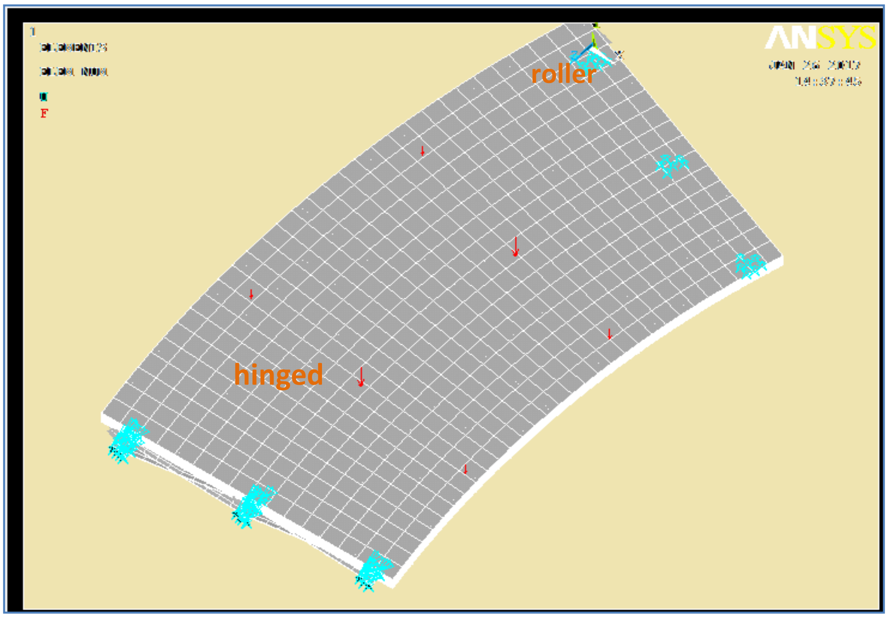


Figure 4: Boundary condition and load distributions.

4 Results and discussions:

In this section the results of numerical analysis were evaluated. Two horizontally curved bridges; one with full interaction between the concrete and the steel girder, while the other with partial interaction between the concrete, the steel girders were simulated in Ansys software 'APDL' graphic. The two models were analyzed during the static loading acted incrementally. Displacements, and strains were discussed and compared with those obtained from laboratory experiments.

4.1 Numerical modeling stresses results:

Figure 5 shows the distribution of stresses in Y direction and Von Mises stresses of full and partial interaction curved composite models. The maximum stress was detected at third length of the curved exterior girder. Some of stresses results were positive in spite-of the concentrated load was in y-direction and negative. The lateral moments and rotations in the all structures induce changing in the distribution of stresses.

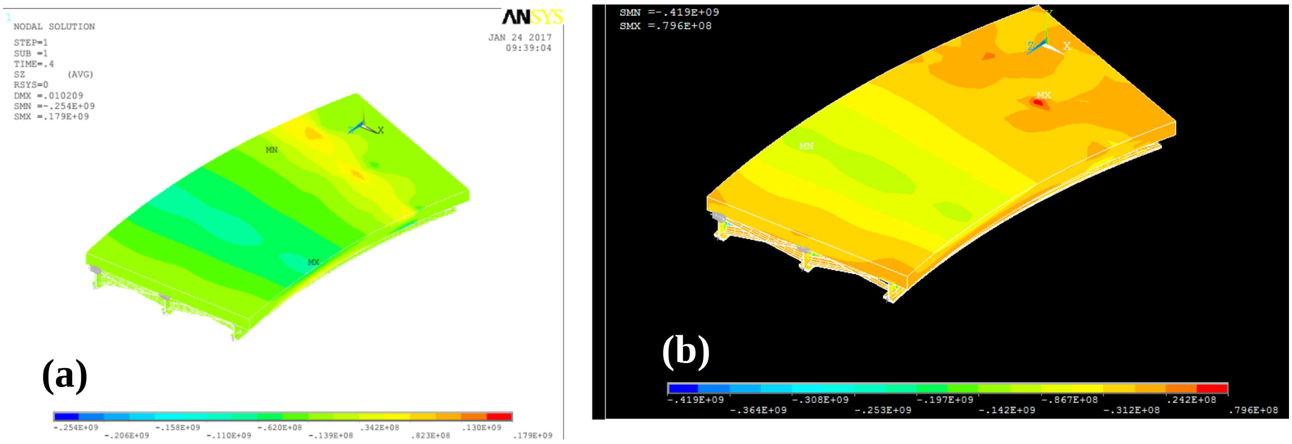


Figure 5: The distribution of stresses of curved composite model by ANSYS software: a) Full interaction and b) Partial interaction

The stress distributions across the concrete deck slab for the curved model along the radius of curvature are shown in Figure 6b and Figure 7b. It is clear from both figures the maximum value is always appears toward the exterior girder.

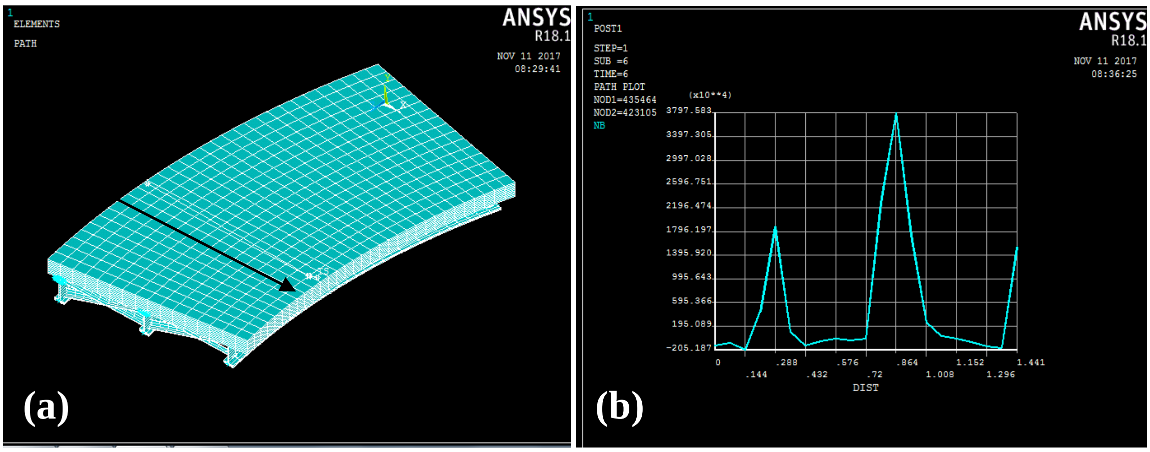


Figure 6: a) The path of stresses for full interaction model, b) The stresses vs. distance for the path

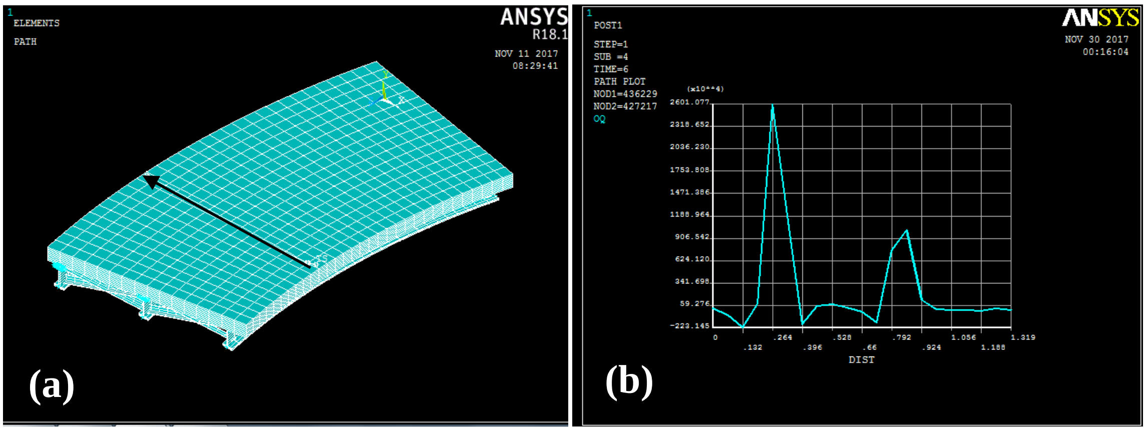


Figure 7: a) The path of stresses for partial interaction model, b) The stresses vs. distance for the path

4.2 Experimental test's results - girder's deflections:

In this section the part of results of center deflection are presented for curved specimens. The displacements were meager by LVDT acted vertically at the maximum moment points. The result and its discussions are in the figures and tables in flowing sections.

The maximum vertical displacements are presented in the Table 3 and blotted in Figure 8 and Figure 9 respectively. Regarding to values shown in table 3 the variation between the applied load on full interaction and partial interaction specimens was 16% and the deflection at three I-girders was varied clearly. Figure 8 and Figure 9 shows charts of comparisons of all deflections of all girders for two specimens.

Table (3): Maximum deflections for full interaction and partial interaction curved specimens

Specimen	Interior girder G_1 (mm)	Middle girder G_2 (mm)	Exterior girder G_3 (mm)	Load (kN)
Full interaction curved Specimen	11	28	83	1042
Partial interaction curved specimen	14	46	111	876
Variation percentage	21%	39%	25%	16%

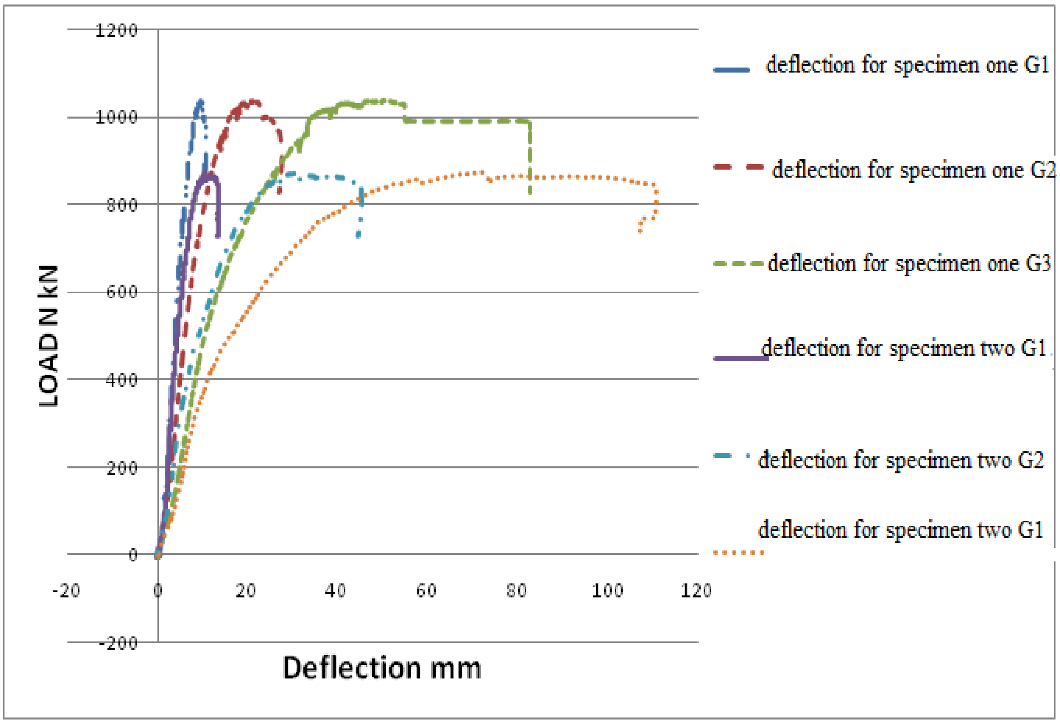


Figure 8: Deflections at the centers of girders for the two specimens

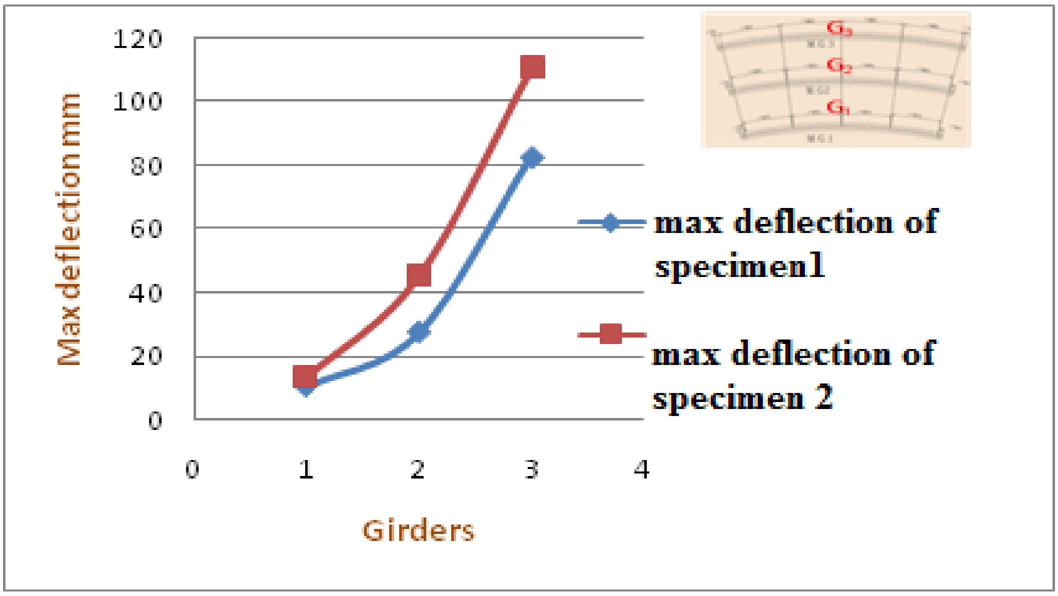


Figure 9: Comparisons of the maximum deflections for two specimens

5 Comparison of numerical and experimental tests results:

In this section comparison between Ansys software results (deflections and strains) and the experimental work will be discussed and evaluated, as shown in Figure 10 - 15 and Table 4 - 7.

5.1 Comparison of numerical and experimental deflection results:

Table 4 and Figure 10 and Figure 11, display maximum deflection obtained from Ansys software and maximum deflection from practical experiments. Comparison between results shows that the difference between the numerical analysis results and experimental results is 16.78%, which indicates that the Ansys software presents an acceptable degree of accuracy. It should be noted that accuracy of this result could have been improved by introducing more mesh refinement. Due to time limitation, further mesh refinement could not be performed in this study.

Table (4): Comparison of numerical and experimental deflection results

Specimen	Experimental results	Ansys program software results	Variation percentages	Load (kN)
Full interaction curved specimen	83 mm	69 mm	16.78%	1040

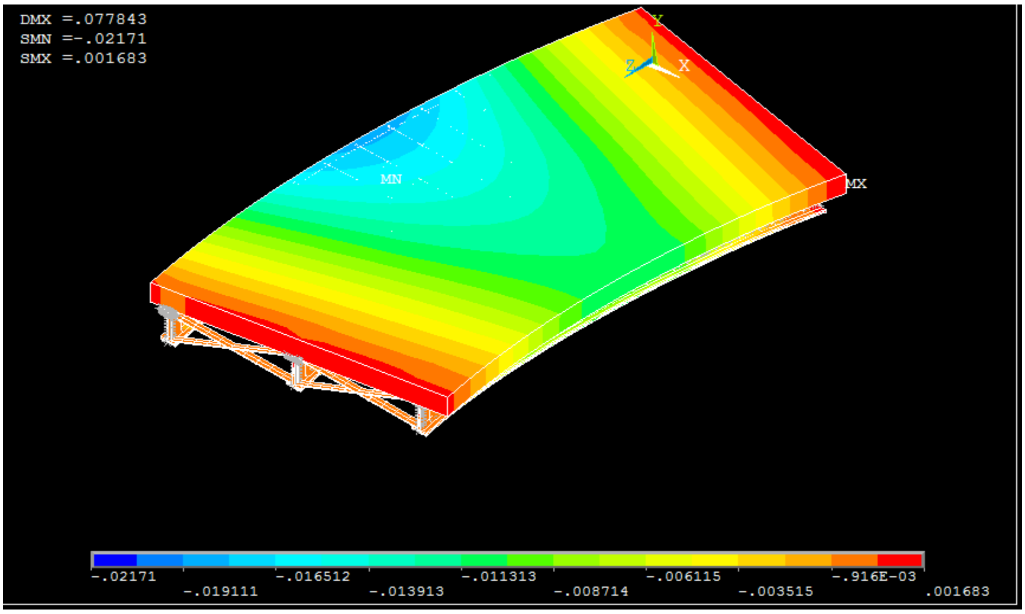


Figure 10: Deflections for full interaction curved numerical model

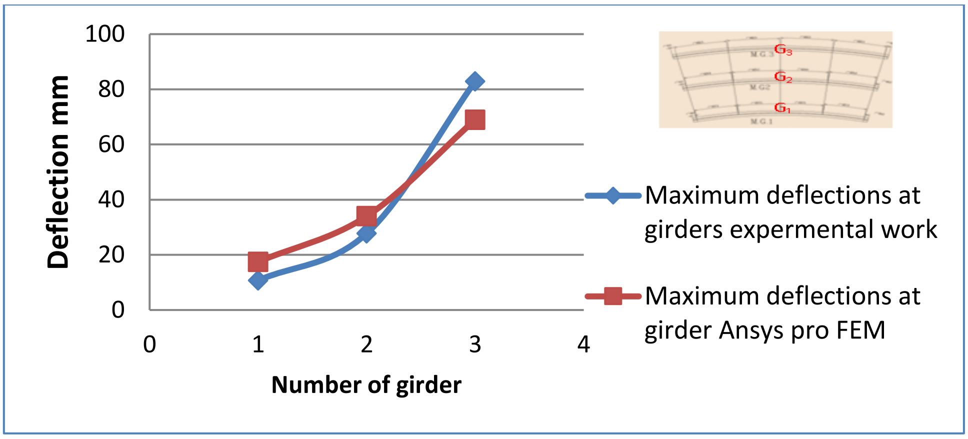


Figure 11: Comparison of maximum deflection in the girders for experimental model

5.2 Comparison of numerical and experimental strains results for full interaction curved model

Table 5 show a comparison between the maximum strains obtained from the numerical analysis and maximum strains from practical experiments. Comparison between results shows that maximum strain at web tension zone was detected at the exterior girder; they were (-0.00283 and -0.00255) mm/mm in experimental test and numerical analysis respectively, Figure 12.

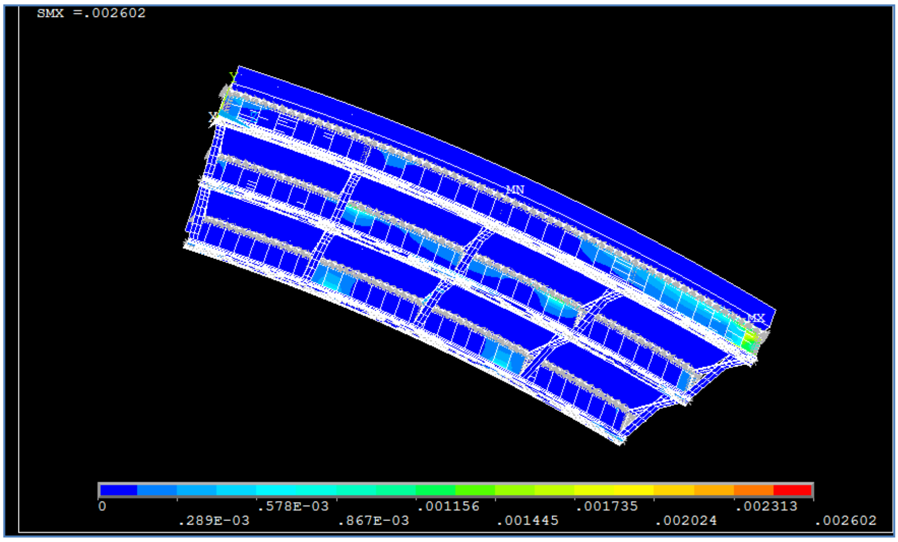


Figure 12: Von Mises Strain results of composite curved model using Ansys APDL

Also, maximum strain at top flange was detected at exterior girder; they were (0.0028 and 0.00143) mm/mm in experimental test and numerical analysis respectively. The strains at top flanges of other girders elucidate linear behavior. Finally, maximum strain at bottom flange was detected at the middle girder; they were (0.0017 and 0.0023) mm/mm in experimental test and numerical analysis respectively. It is clear that strain results, comparisons reflect a very high accuracy for Ansys software.

Table (5): Comparison of numerical and experimental strain results at three I-girders for full interaction specimen

Deformations In full interaction specimen	Top flange Exp	Top flange Ansys	Bottom flange exp	Bottom flange Ansys	Web Tension zone exp	Web Tension zone Ansys
Interior girder G_1 mm/mm	-0.000342	-0.00029	-0.0006	-0.00029	0.000195	0.00029
Middle girder G_2 mm/mm	-0.000213	- 0.0019	0.0017	0.0023	- 0.00201	- 0.0014
Exterior girder G_3 mm/mm	0.0028	0.00143	0.0014	0.00183	- 0.00283	- 0.00255
Total load	1042 kN	For experimental test and Ansys software				

5.3 Comparison of numerical and experimental deflections and strains results for the partial interaction specimen:

In this section comparison between numerical analysis results (deflections and strains) and the experimental work for partial interaction curved model will be evaluated and discussed, as shown in Figure 13 - 15 and Table 6 and Table 7.

5.3.1 Comparison of deflection results between numerical analysis and experimental test

Table 6 and Figure 13 and 14 show the maximum deflection results for analysis which was demonstrated using numerical analysis. The comparison shows that the difference between the two methods is 26% (at exterior girder), which indicates that the program represents an acceptable degree of accuracy. Accuracy could be improved by mesh refinement and using more accurate contact element between concrete deck and steel I-girders.

Table (6): Comparisons of deflection results in experimental and numerical analysis

Source of results	Experimental results	Ansys program software results	Variation percentages	Load kN
Partial interaction composite Curved specimen	111 mm	82 mm	26%	875.54

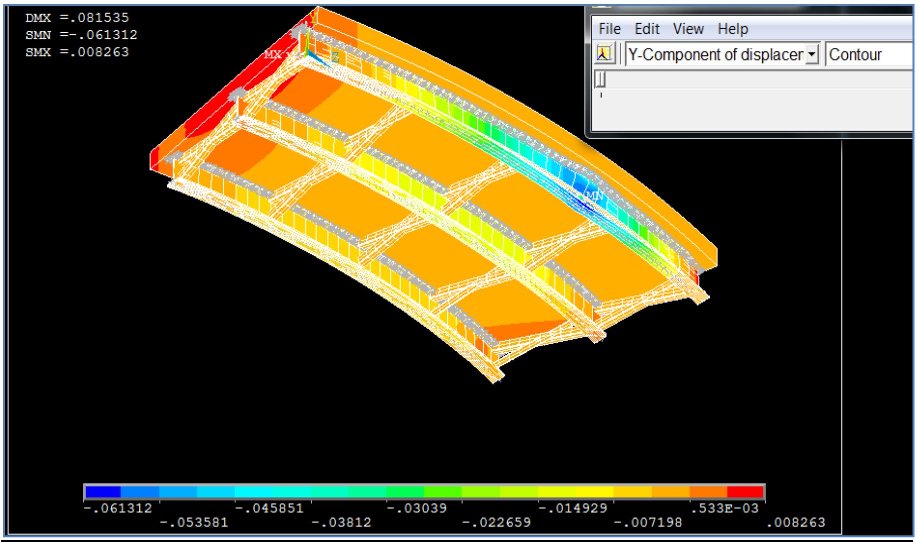


Figure 13: Deflections of partial interaction curved composite model using Ansys software

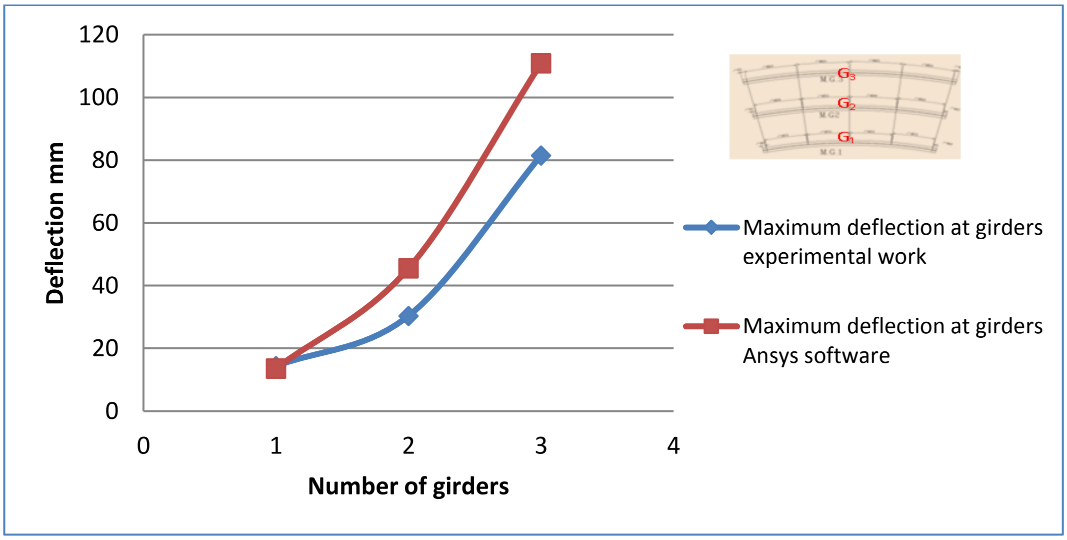


Figure 14: Comparisons of maximum deflection of three girders of experimental test and Ansys software

5.3.2 Comparison of numerical and experimental strain results:

In this section numerical analysis results of maximum strains at the flanges and webs will be evaluated, discussed and compared with experimental work strain results. Table. 7 display the maximum strains obtained from the numerical analysis and maximum strains from practical experiments. Comparison between results shows that maximum strains at top flange were detected at the exterior girder, the maximum strains were (0.00215 and 0.00219) mm/mm from experimental test and numerical analysis respectively. Also, strains at other two girders were very small at top flanges they elucidate linear behavior. Also, Maximum strains at bottom flanges were (0.002087 and 0.002365) mm/mm at middle and exterior girders respectively as a result from the experimental work and (0.00218 and 0.00253) at middle and exterior girders using numerical analysis, also, the accuracy of the model using ANSYS program were very high. The maximum strains at web tension zone detected at the exterior girder were 0.01 mm/mm from experimental test and 0.0109 mm/mm using numerical analysis, Figure 15, at middle girder was 0.00211mm/mm from experimental work and 0.00217 mm/mm using numerical analysis. It is clear that strain results, comparisons reflect a very high accuracy of Ansys software, which the strain in numerical analysis was less than a strain of experimental work by about (8% to 13%).

Table (7): Comparison of experimental and numerical strain results at three I-girders for partial interaction specimen

Deformations In partial interaction specimen	Top flange Exp	Top flange Ansys	Bottom flange exp	Bottom flange Ansys	Web Tension zone exp	Web Tension zone Ansys
Interior girder G_1 mm/mm	0.000767	0.000725	0.00127	0.00144	0.000773	0.000725
Middle girder G_2 mm/mm	0.000848	0.00073	0.002087	0.00218	0.00211	0.00217
Exterior girder G_3 mm/mm	0.00215	0.00219	0.002365	0.00253	0.01	0.0109
Total load	875.54kN	For experimental test and Ansys software				

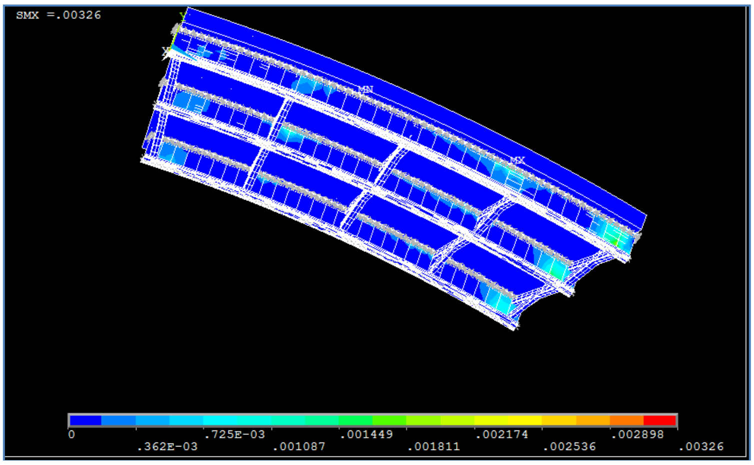


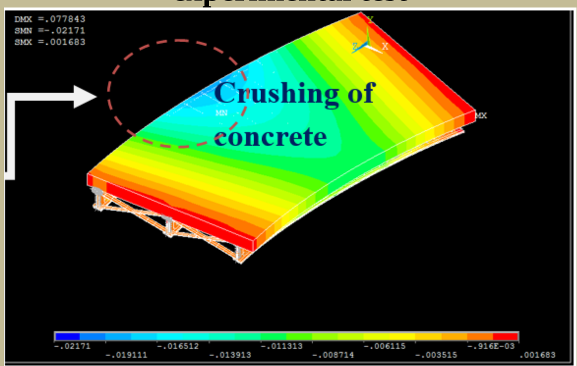
Figure 15: Von Mises Strain results for partial interaction curved numerical model

5.4 Comparisons of failure modes

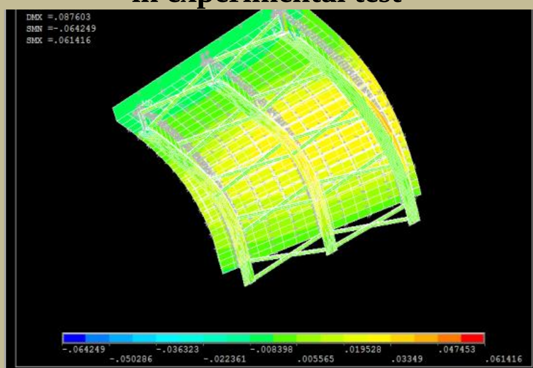
Figure 16 shows the close similarity of these failure modes in the experimental and numerical results. Three different failure modes were encountered in the curved composite bridge steel beams and deck slabs, namely are crushing concrete deck slab, torsional deflection of external steel I-girders as well as buckling in the middle cross- frame.

a) Mode of failure of concrete in experimental test

b) Torsion and buckling of steel I- girders in experimental test



c) Concrete crushing in ANSYS software



d) Torsion and deflection at steel I-girders in ANSYS software

Figure 16: Failure patterns in the curved composite bridges super structure

6. Conclusions:

- The results of the study showed that the majority of the load distributed towards the exterior girders, because the spatial configuration of the curved structures.
- Flexural deformations occurred firstly in the exterior girder in curved composite specimen for both full and partial interaction.
- The load carrying capacity of the partial interaction specimen in the plastic zone was less than that of the full interaction by 16%, but the failure behavior of the two specimens was almost similar.
- The maximum deflection values obtained from numerical model compared to the values of the experiments show differences of (16.78% and 26%) for curved full and partial interaction specimens, respectively.
- Also, the maximum strain values obtained from numerical model compared to the values of the experiments shows differences of (8% and 13%) for curved full and partial interaction specimens, respectively.
- These results indicated that Ansys non-linear finite element software can be used for analysis of this complex structure.

References

[1] Matthew R. Tilley, Furman W. Barton and José Pantaleon Gomez,.Dynamic. "Analysis and Testing of a Curved Girder Bridge", Virginia Transportation Research Council. Virginia Department of Transportation and U.S. Federal Highway Administration 71624, 2006.

[2] MirsadTarić and EnisSadović, "Application of Discrete Model in Analysis of Built and Tested Composite Bridge" Construction, Architecture and Civil Engineering Vol. 9, No1, 2011, pp. 119 - 132 DOI: 10.2298/fuace1101119T, 2011.

[3] Ching-Jen Chang, Construction "Simulation of Curved Steel I-Girder Bridges" Doctor of Philosophy in Civil Engineering Georgia Institute of Technology.

[4] Nevling, D and Linzell, Daniel, "Examination of Level of Analysis Accuracy for Curved I-Girder Bridges through Comparisons to Field Data". Journal of Bridge Engineering - J bridge eng 10.1061/(ASCE)1084-0702(2006)11:2(160), 2006.

[5] Cagri Ozgur, "Behavior and Analysis of a Horizontally Curved and Skewed I-girder bridge", Georgia Institute of Technology. Georgia Tech Theses and Dissertations [20366], 2007.

[6] W. T. Jung, J. S. Jong, S. H. Kim, “A Study on the Behavior Characteristics of Curved FRP-Concrete Composite Panel", Advanced Materials Research, Vols. 557-559, pp. 375-380, 2012.

[7] Hoffman, Jerad James, "Analytical and field investigation of horizontally curved girder bridges" *Graduate Theses and Dissertations*.13104. https://lib.dr.iastate.edu/etd/13104, 2013.

[8] N. S. Ingawale and D. B. Kulkarni, "Parametric Study of Horizontally Curved Bridge Girder". International Journal of Engineering Research - International Journal of Engineering Research-Online. (IJOER) (ISSN: 2321-7758), 2015.