Al-Mukhtar Journal of Engineering Research 07 (1): 43-56, 2023
Doi:

Research Article **Open Access**

Modelling and simulation of flexural behavior for reinforced concrete beams using ANSYS

Mohamed Khalifa Bneni[1] and Samir Hasuna Ashour[2]

***Corresponding author:** mo.bneni@zu.edu.ly Faculty of Engineering, University of Zawia, Libya.

Second Author: S.ashour@zu.edu.ly, Faculty of Engineering, University of Zawia, Libya .

Received:
17 July 2023

Accepted:
05 December 2023

Publish online:
31 December 2023

Abstract

Over the last twenty years, many investigators are used finite element software, to validate and compare the FE results with their experimental research. This work focused on the development of a numerical model implemented by the ANSYS 2022R2 software, to simulate the flexural behavior of the RC beam. Numerical models are tested under four-point bending. To investigate the influence of reinforcement steel ratio and compressive strength of concrete on the flexural capacity of the model. The results indicated that the Finite Element model was able to predict the flexural behavior of the experimental test beam. Furthermore, the influence of different tensile reinforcement ratios has the most effect on the flexural behaviour of the FE models at maximum loads. While the change in concrete compressive strength has affected the flexural performance of the models. This influence shows slight increases in the first crack load and maximum loads of the models. Furthermore, cracking pattern behaviour at the final stage for numerical models showed a good agreement with experimental cracks behaviour.

Keywords: RC Beams, Finite Element Method, ANSYS, Steel Reinforcement Ratio, Compressive Strength.

INTRODUCTION

Experimental studies of the flexural behavior of reinforced concrete beams comprise the cost of materials, equipment for testing, workers and time. Safety and serviceability evaluation of construction structures requires the development of accurate methods and three-dimensional FE models for their analysis. For example, the numerical and experimental studies are conducted side by side. To compare the numerical and experimental results and collect detailed information using numerical models. Also, the experimental studies are becoming expensive and take a long time. The Finite Element Method (FEM) is commonly used for predicting the behavior of structures, and it is often preferred over experimental studies when investigating the behavior of concrete. Many variables have an impact on the accuracy and convergence of the results such as properties of materials, mesh and convergence criteria. Vasudevan and Kothandaraman [1] present several trial analyses of influencing factors on the flexural behavior of numerical models. As, mechanical properties of concrete, mesh density, points loaded, the influence of shear reinforcement on flexural behavior, and steel reinforcement ratios. The results of their study demonstrate the ANSYS program's ability to simulate mechanical properties in the analysis of reinforced concrete beams. Mazen Musmar [2] referred in his study to the performance of the finite element model of the reinforced concrete beams that are designed to fail in flexure. The work showed the performance of the FE model of the RC beam from where crack pattern, load-deflection curve, mode of failure and behavior material models are implemented in the FE models. This study targets to explain the effective role of FE structural modelling in simulating the performance of RC structures members.

Another study showed that modelling and simulation of reinforced concrete beams using ANSYS program to understand the effect of percentages of steel reinforcement on flexural behavior as under, balanced and over-reinforced beams implemented by Pawar and Pawar [3]. Pranata et al. [4] studied testing of the studied deep beams was performed by FE modelling using ANSYS program. To obtain useful parameters for modelling RC deep beams in FEM modelling, calibrating tests have to be done out of verification and validation processes. Moulika et al. [5] worked in their study that modelled and analyzed reinforced concrete beams when subjected to two-point loads at one-third span from each support, using the Finite Element Analysis tool, called ANSYS software. The numerical model has dimensions length of 600mm, width of 160 mm and height of 160 mm with main steel reinforcement of 3Φ12 mm and 2Φ8 mm at top reinforcement, also stirrups using Φ8 mm/100 mm. The results showed have more sensitive to mesh size, materials properties and load increments.

Tjitradi et al. [6] studied the conduct of structural members of one-layer reinforced concrete beams under tension, balanced and compressive, consequences of collapsed mechanisms with modelling and simulation using ANSYS Workbench. The outcomes displayed that the reinforced concrete members can be analysed using the ANSYS program with the modified three-dimensional model. The numerical model used simulates Multilinear Kinematic Hardening using the compression stress-strain curves of unconfined concrete. The use of the element SOLID 65 in the modelling of concrete materials can specify outcomes by the nonlinear behaviour of reinforced concrete members. Steel reinforcement is used as an axial bar element by taking the discrete engineering model Spar Link Element (LINK8). The behaviours of RC elements can be determined through the analysis of calculation and FEM that beams with the tensile collapsed condition have a lower flexural capacity and collapse behavior is more ductile. On the other hand, many studies focused on the development of a 3D FE model using ANSYS software to analysis the flexural behavior of the RC beam strengthened, and validate numerical results with the experimental results. These papers presented the applications of nonlinear finite element models in the analysis and predict the behavior of RC beams strengthened with U-jacket, CFRP sheets or rods and NSM FRP rods. The numerical results are compared to experimental results of unstrengthen beams. As well, the comparisons are carried out about of load- deflection behavior at mid-span of beams, the ultimate load at failure and cracks pattern [7-11].

The aims of this research are to:

- Select the suitable element types available in ANSYS 2022 R2 software, such as steel reinforcement, concrete, plates of loading, and steel support plates.
- modelling of a 3D model to simulate the behaviour of simply supported RC beams analysis.
- Validation of the numerical model results of the current study by comparison of the experimental results of a tested beam (reference beam CB) implemented by Sharaky et al. [12].
- Studying the effect of two variables such as tensile reinforcement steel and compressive strength of concrete on the behaviour of the numerical model.

MATERIALS AND METHODS

To model and idealize the RC beams in the ANSYS software some elements must be selected. ANSYS's element library [13] contains a lot of different element types. Each element type has a unique number and a prefix that identifies the element category. Concrete was modelled using solid element SOLID65 with 3-D 8-node solid elements as shown in Fig. 1 (a). The SOLID 65 element is capable of crushing in compression and cracking in tension. The steel reinforcement was modelled using LINK 180 as shown in Fig. 1(b). Moreover, a 3-D structural solid element 185 was used to model the plates of loading and supports as shown in Fig. 1 (c).

The specimen of study is a simply supported reinforced concrete beam. The full-size specimen was 2600 mm × 160mm × 280mm and the clear length of 2400 mm. While the main steel reinforcement was 2Ø12 mm and the top reinforcement of 2Ø8 mm as shown in Fig. 2. The shear steel reinforcement was 8 mm stirrups with spacing between bars of about 100mm .Table 1. displays the concrete material properties used for current model.

Six models divided into two main groups were subjected to four-point bending. We have selected the four - points test to evaluate of behavior flexural of FE models and

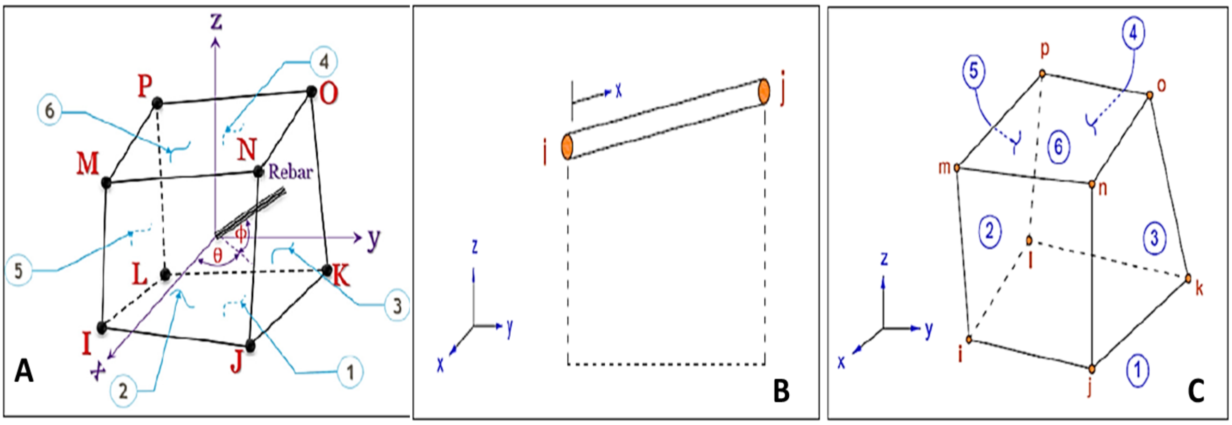


Fig. (1). FE models elements: (a) SOLID65 (b) LINK180 (c) SOLID185[13].

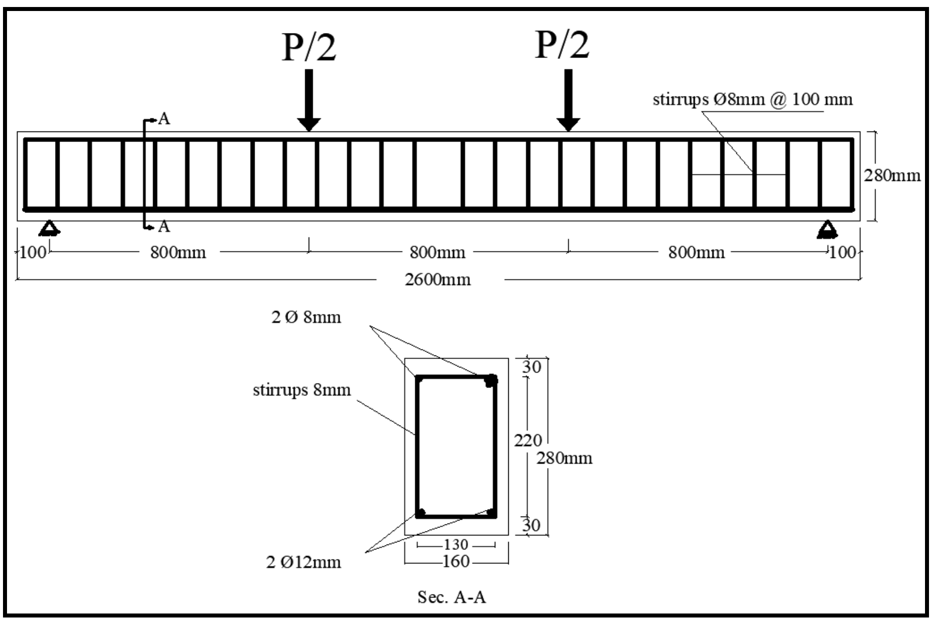


Fig. (2).Tested beam details [12].

Table (1): Concrete material properties used in ANSYS beam model

		Material properties	
		M30	M60
Concrete	Modulus of Elasticity, Ec	25000MPa	36000MPa
	Poisson Ratio, υ	0.2	0.2
	Open Shear Transfer Coef.	0.2	0.2
	Closed Shear Transfer Coef.	0.8	0.8
	Uniaxial Cracking Stress (ft)	2.8 MPa	3.50 MPa
	Uniaxial Crushing Stress (fc)	31 MPa	60 MPa
Tension Reinforcement	Modulus of Elasticity, Es	200 GPa	200 GPa
	Poisson Ratio, υ	0.3	0.3
	Yield stress	545 MPa	545 MPa
	Tangent Modulus	1200 MPa	1200 MPa
Loading & Supporting Plates	Density	7850 kg/m^3	7850 kg/m^3
	Modulus of Elasticity, Es	200 GPa	200 GPa
	Poisson Ratio, υ	0.2	0.2
	Tensile Yield Strength	550 MPa	550 MPa
	Tensile Ultimate Strength	650 MPa	650 MPa

to avoid shear failure. The models were designed as an under-reinforced section, the first group is normal strength concrete, while the second group with high strength concrete. The variable factor for each group in the numerical study using ANSYS software is steel reinforcement ratios. Details of the models for the numerical program are summarized in Table 2

Modelling and Meshing

Modelling of the RC beams is idealized in the ANSYS. The RC beam has been modelled as volumes, such as the concrete, loading plates and supports. While the steel reinforcements and stirrups are modelled as line bodies. The model of RC beam is shown in Fig. 3. Concrete was simulated using a multilinear isotropic hardening model. The stress-strain curve was used to simulate the concrete plasticity based on equations 1 and 2. The concrete material properties are given in Table 1.

Table (2): Numerical models configuration

Type of group	Model ID	Bottom reinforcement	fc	Top reinforcement	Stirrups
Group (A)	B1-Φ12-M30	2-Φ12mm	31MPa	2-Φ8mm	Φ8mm@100mm
	B2-Φ14-M30	2-Φ**14**mm		2-Φ8mm	
	B3-Φ16-M30	2-Φ16mm		2-Φ8mm	
Group (B)	B4-Φ12-M60	2-Φ12mm	60MPa	2-Φ8mm	
	B5-Φ14-M60	2-Φ14mm		2-Φ8mm	
	B6-Φ16-M60	2-Φ16mm		2-Φ8mm	

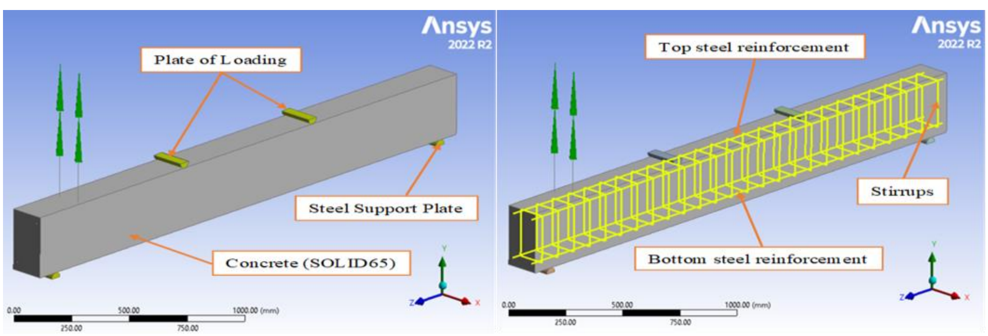


Fig. 3 Details of the numerical model in ANSYS.

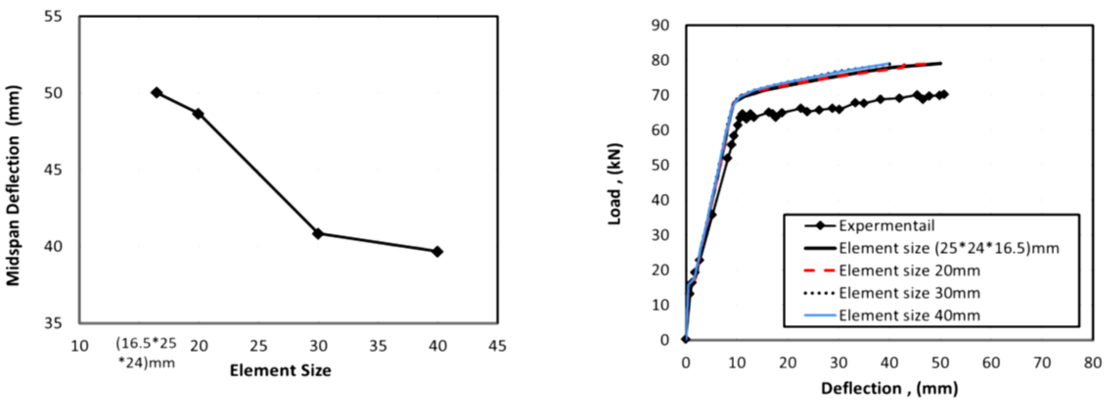
$$f = \frac{E_c \varepsilon}{1 + \left(\frac{\varepsilon}{\varepsilon_0} \right)^2} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 1$$

$$\varepsilon_0 = \frac{2 f_c^{'}}{E_c} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2$$

Where, E_c is Young's modulus for concrete, ε is the concrete strain, and ε_o is the compression failure strain.

Element mesh size sensitivity

On the other hand, sensitivity analysis was done by studying the element's sizes effect of (25*24*16.5) mm, 20mm, 30mm and 40mm, to examine the convergence of the results. Fig. 4 shows the effect of element sizes on the numerical results that were studied and compared to experimental results by Sharaky et al. [12]. The mesh size (25*24*16.5) mm was chosen as it generated good results from the solver keeping the run time at a reasonable length. Fig. 5 presents the mesh size of the numerical model in ANSYS for this study.



a) Effect of element size on the sensitivity of results

b) Comparison of P-δ curves with different mesh sizes

Fig. (4). Results verification; a) Effect of element size on the sensitivity of results, b) Comparison of P-δ curves with different mesh sizes.

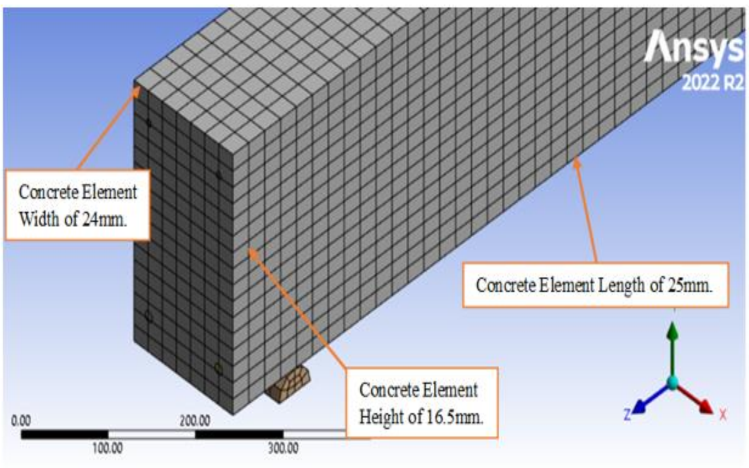


Fig. (5). The meshes of the numerical model in ANSYS

Loading and Boundary Conditions

Displacement boundary conditions are required to constrain the model to obtain a remarkable solution. To confirm that the model works similarly to the testing beam; boundary conditions must be applied to the supports. So, the support conditions in this study will be taken as a pin support with no movement in the X, Y and Z directions. In contrast, another support will be taken as a roller of which there will be only movement in the Z- direction with no movement in the other directions as seen in Fig. 6. The load plate is loaded by applying a remote force on each plate as seen in Fig. 6.

The methodology of this study was done by modelling of geometry body, selecting the type of elements, material properties of concrete and steel, meshing details, finite element analysis, and results as shown in Fig. 7.

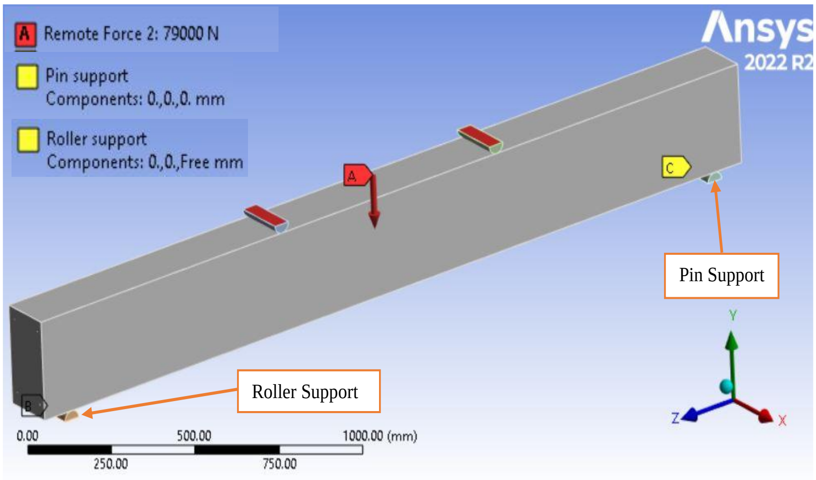


Fig. (6). Loading and boundary conditions of model.

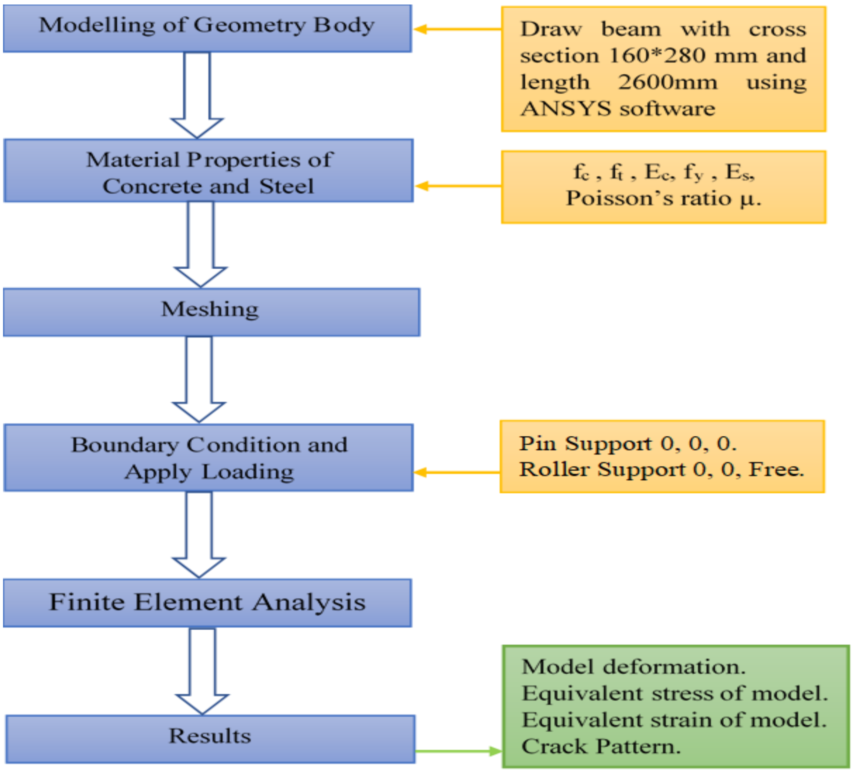


Fig. (7). A flowchart displays the process of the methodology.

RESULTS AND DISCUSSION

Validation of the model

The validation of the FE model must be done to determine the accuracy of the results. The numerical model was simulated and compared to an experimentally tested beam of the study published in the international journal, by Sharaky et al. [12]. Comparing the numerical results to the experimental results, the flexural behavior of the finite element model is acceptable through the load-displacement curve as shown in Fig. 8. The data in the figure clearly shows that the first crack load registered about 15.72kN for the current numerical model compared to 14.7kN for experimental beam [12]. While the yield load, P_y, and mid-span deflection, Δ_y, registered about 67.79kN and 9.43mm for the current numerical model compared to 64.5kN and 11.20mm for the experimental beam, respectively.

On the other hand, Fig. 9. shows the numerical deflection distribution along the model and compares it with that experimentally measured by [12]. It can be noticed that the deflection at the mid-span for the experimental recorded about 1.95mm, 4.12mm, 6.90mm and 9.75mm compared to 2.18mm, 3.62mm, 5.87mm and 8.11mm for deflection of the numerical model with percent difference by (-11.79%), (12.13%), (14.92%) and (16.82%) at loading steps 20kN, 30kN, 45kN, 60kN, respectively. The distribution shows convergent results in the elastic zone as a linear relationship exists and an increase in the percentage difference at the end elastic region and after yield stress of steel, due to the plastic region as seen in Fig.8. Also, there are microcracks in the concrete for the tested beam due to the production of concrete by shrinkage that it is not included in the numerical model and it affects numerical results. Overall, there was an agreement and convergence in the load-deflection behaviour.

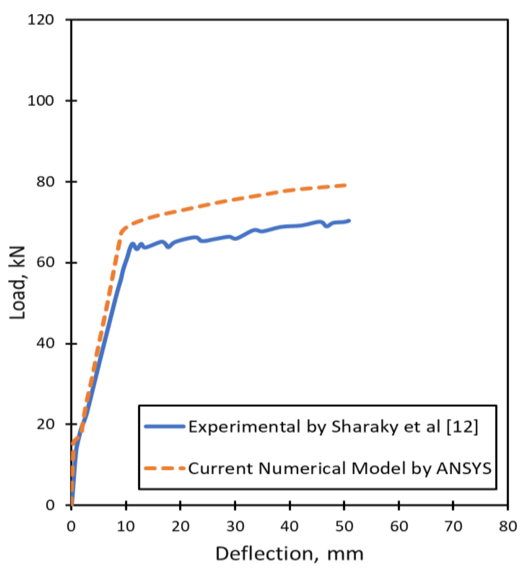


Fig.8 Load-deflection curve of experimental results by [12] and numerical model

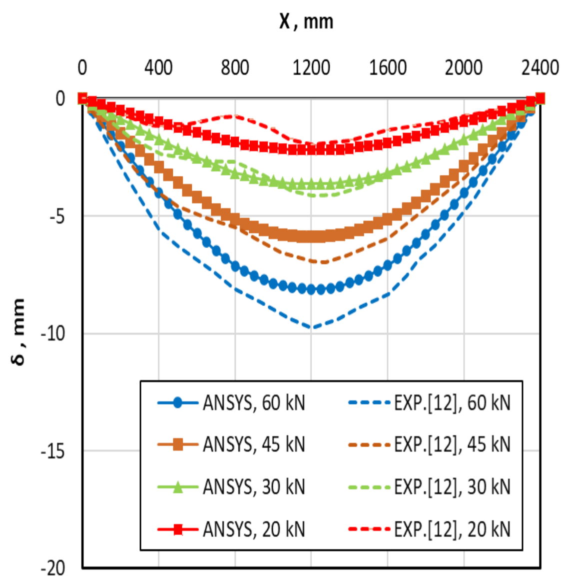


Fig. 9 Details deflection of numerical model compared to experimental results

Parametric study

Table 3. presents the numerical results of finite element models in this research for all specimens using ratios of steel reinforcement and various compressive strengths of concrete. It includes the first crack load, P_{cr}, yield load, P_y, yield deflection, Δ_y, the maximum load, P_u, the maximum deflec-

tion, Δ_u, ductility index, μ, and the failure modes. The numerical models failed due to concrete crushing after yielding the steel reinforcement.

Table (3): Numerical results of the FE Models

Model ID	P_{cr} (kN)	P_y (kN)	Δ_y (mm)	P_u (kN)	Δ_u (mm)	μ	Failure mode
B1-Φ12-M30	15.72	67.79	9.43	79	50.02	5.30	concrete crushing
B2-Φ14-M30	16.13	90.73	10.16	103	41.09	4.04	concrete crushing
B3-Φ16-M30	17.80	119.8	12.18	136	39.13	3.21	concrete crushing
B4-Φ12-M60	16.52	71.22	9.39	83	29.34	3.12	concrete crushing
B5-Φ14-M60	18.87	94.25	9.81	107	32.75	3.33	concrete crushing
B6-Φ16-M60	21.50	123.3	10.86	140	34.54	3.18	concrete crushing

Effect of tensile reinforcement ratios

The effect of the tensile reinforcement ratios on the flexural behavior of the numerical models was investigated in the difference between numerical results and experimental results due to the toughening mechanisms. So, the model was able to predict the experimental outcomes in an acceptable manner. This section. Fig.10 illustrate the effect of tensile reinforcement steel ratios on load-deflection behavior of the numerical models having the normal strength concrete M30 and tensile reinforcement ratios of 0.6%, 0.8% and 1%, which equals the area steels about by 2Φ12mm, 2Φ14mm and 2Φ16mm, respectively. We can notice that the numerical model of B3-Φ16-M30 recorded an ultimate load of 136 kN, while values of 103 kN and 79 kN were recorded for models B2-Φ14-M30 and B1-Φ12-M30 respectively. The increasing percentages are 72.1% and 30.3% for tensile steel ratios using 1% and 0.8% compared to 0.6% respectively.

Fig.11 illustrate the effect of tensile steel reinforcement ratios on load-deflection behavior of the numerical models having the high strength concrete M60 and tensile reinforcement ratios of 0.6%, 0.8% and 1%, which equals the area steels about by 2Φ12mm, 2Φ14mm and 2Φ16mm, respectively. We can notice that the numerical model of B6-Φ16-M60 recorded an ultimate load of 140 kN while values of 107 kN and 83 kN were recorded for models B5-Φ14-M60 and B4-Φ12-M30 respectively. The increasing percentages are 68.6% and 28.9% for tensile steel ratios

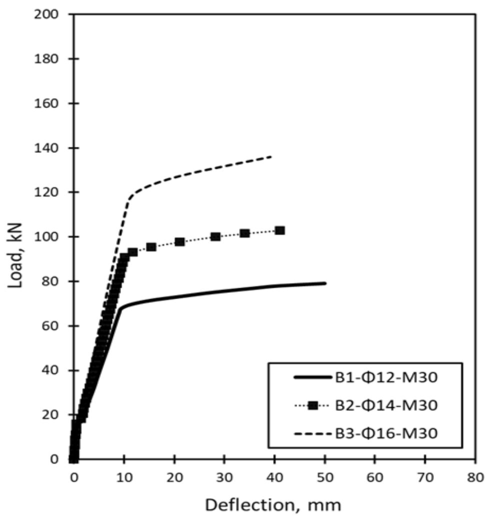


Fig .(10). Load - deflection curves of numerical models with different areas steel for M30 using 1% and 0.8% compared to 0.6% respectively. These results that it confirms the effect of change longitudinal steel reinforcement ratios on the load-deflection curve for the numerical models by ANSYS software. On the other hand, the mid-span deflection of B3-Φ16-M30 model was decreased at the same applied load of other models which means a raise stiffness due to increasing of tensile steel reinforcement ratios. FE model is able to simulate the flexural behaviors of RC beams using a change of tensile steel reinforcement ratios.

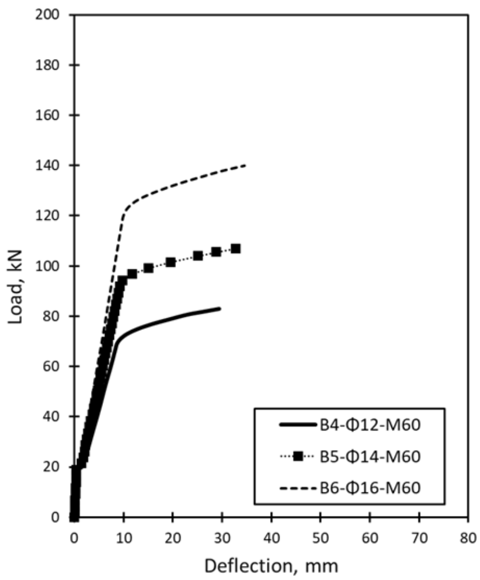


Fig.11 Load - deflection curves of numerical models with different areas steel for M60

Effect of compressive strength of concrete

This section presents, a comparison between the effect of change compressive strength proposed in this work on the flexural behavior of the numerical models. The load-deflection behavior is considered to be an indication of the effect of compressive strength on behavior for the FE models. This analysis aims to determine the efficiency of the model. The effect of the compressive strength on the flexural strength of FE models, with concrete compressive strength of M30, M60 and changing of tensile steel areas are given in Fig. 12.

For tensile reinforcement steel of 2Φ12mm as shown in Fig. 12 (a), models of B1-Φ12-M30 and B4-Φ12-M60 recorded the first cracking load increases from 15.72 kN to 16.52 kN by a percentage enhancement equals 5.0%, respectively. While the value of the ultimate load increases from 79 kN for model B1-Φ12-M30 to 83 kN for model B4-Φ12-M60, the percentage increase is 5.0 %. Moreover, the mid-span deflection at ultimate loads of models is 50.02 mm and 29.34 mm, with percentage decreases of 41.3%, respectively. Fig. 12 (b) presents the behavior of FE models B2-Φ14-M30 and B5-Φ14-M60 with reinforcement steel area of 2Φ14mm. It is observed that the first cracking load increases from 16.13 kN of specimen B2-Φ14-M30 to 18.87 kN of specimen B5-Φ14-M60, by a percentage enhancement equals 17.0%. While the value of the ultimate load increases from 103 kN for model B2-Φ14-M30 to 107 kN for model B5-Φ14-M60, the percentage increase is 5.0 %. Moreover, the mid-span deflection at ultimate loads of models is 41.09 mm and 32.75 mm, with percentage decreases of 20.3%, respectively. Fig. 12 (c) presents the behavior of FE models B3-Φ16-M30 and B6-Φ16-M60 with reinforcement steel area of 2Φ16mm. It is observed that the first cracking load increases from 17.80 kN of specimen B3-Φ16-M30 to 21.5 kN of specimen B6-Φ16-M60, by a percentage enhancement equals 20.7%. While the value of the ultimate load increases from 136 kN for model B2-Φ14-M30 to 140 kN for model B5-Φ14-M60, the percentage increase is 3 %. Moreover, the mid-span deflection at ultimate loads of models is 39.13 mm and 34.54 mm, with percentage decreases of 11.7%, respectively.

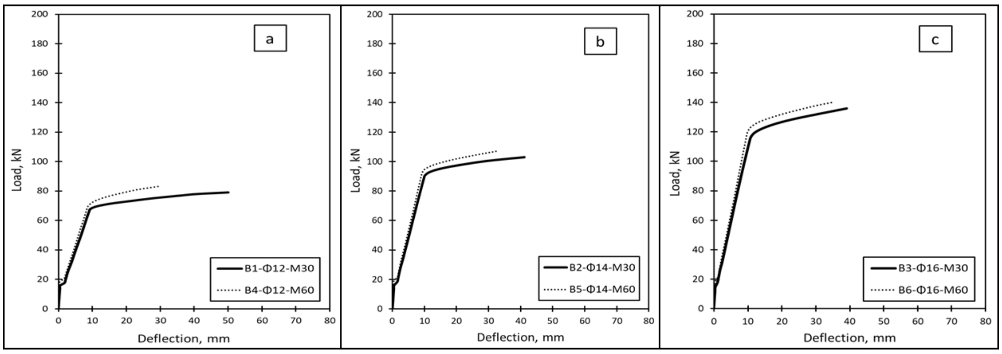


Fig. (12). Load - deflection curve of FE models with change compressive strength

Stress contours and crack pattern of the FE models

The compressive, tensile stress contours and cracks pattern are shown in Figs. 13-18, for numerical models at the ultimate stage loading. As seen in the figures the stress contours of the ANSYS software can effectively display stress prediction and development in the concrete for each mode which depends on various parameters such as change of reinforcing steel ratio and concrete compressive strength. The crack patterns noticed from the finite element analysis at ultimate loads that the FE models using a variety of ratios reinforcement steel failed in flexural by yielding main steel followed by concrete crushing and a high spread of cracks in the mid-span of the models.

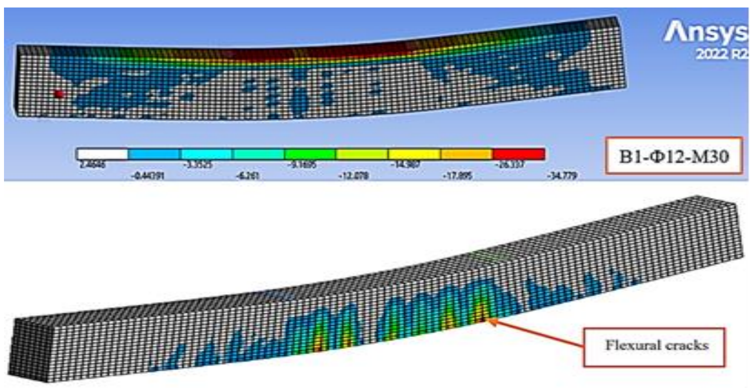


Fig. 13 Numerical stress contours and crack patterns of model B1-Φ12-M30.

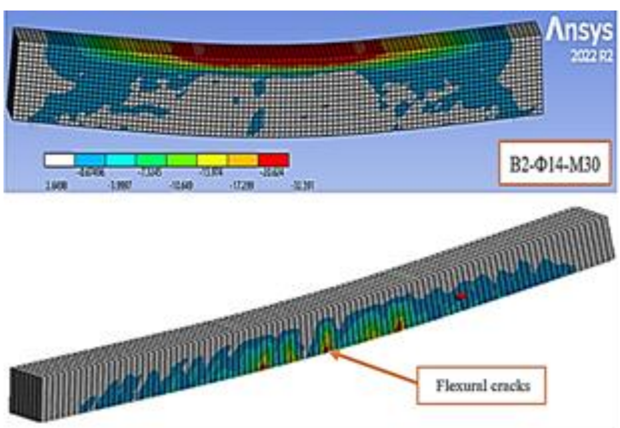


Fig. 14 Numerical stress contours and crack patterns of model B2-Φ14-M30.

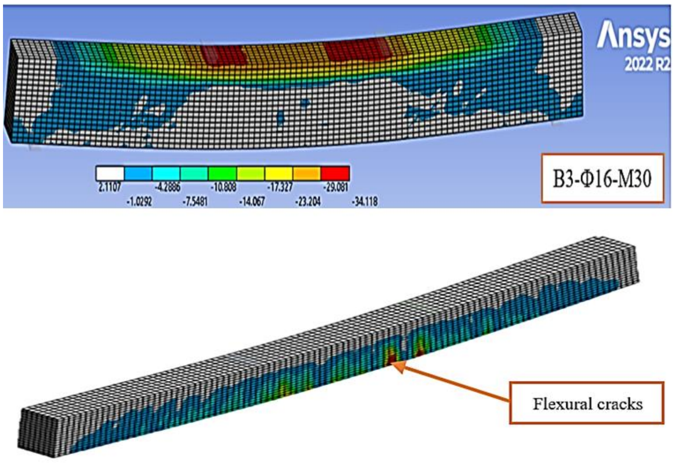


Fig. 15 Numerical stress contours and crack patterns of model B3-Φ16-M30.

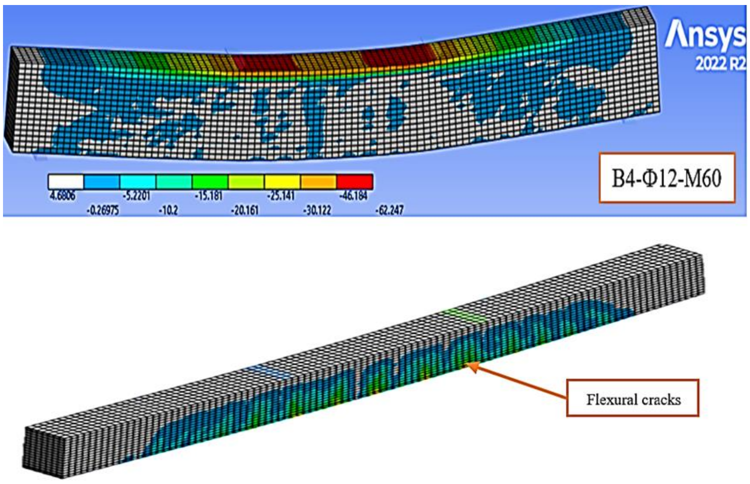


Fig. 16 Numerical stress contours and crack patterns of model B4-Φ12-M60

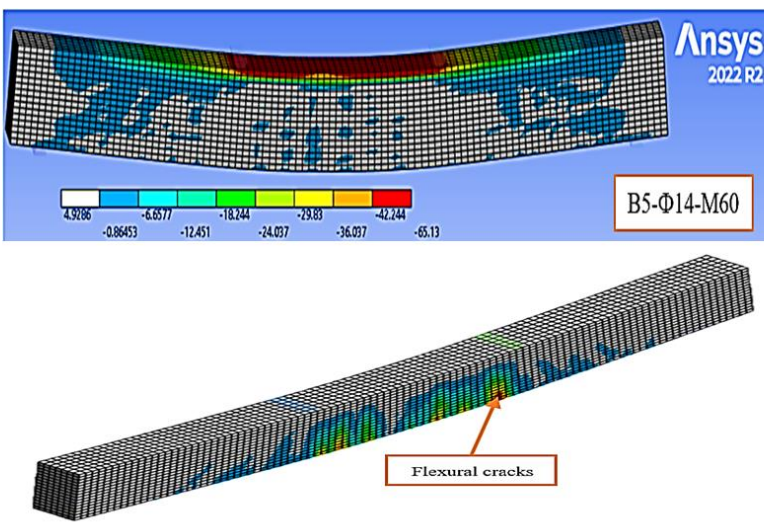


Fig. 17 Numerical stress contours and crack patterns of model B5-Φ14-M60

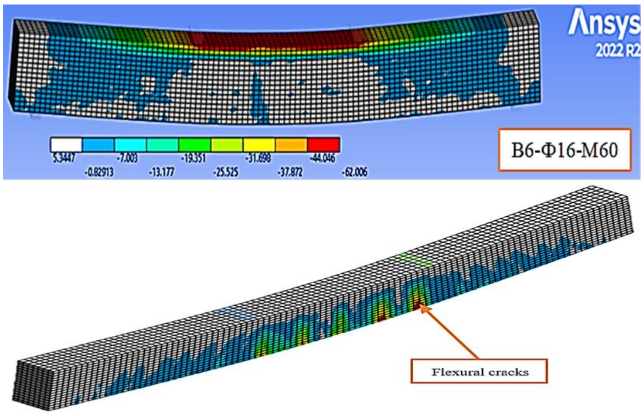


Fig. 18 Numerical stress contours and crack patterns of model B6-Φ16-M60

Stresses in main steel bars

The axial tensile stresses in steel rebars were mapped from FE models as shown in Fig. 19. It can be seen that the upper contour lines which represent the top steel reinforcement showed negative stress values indicating

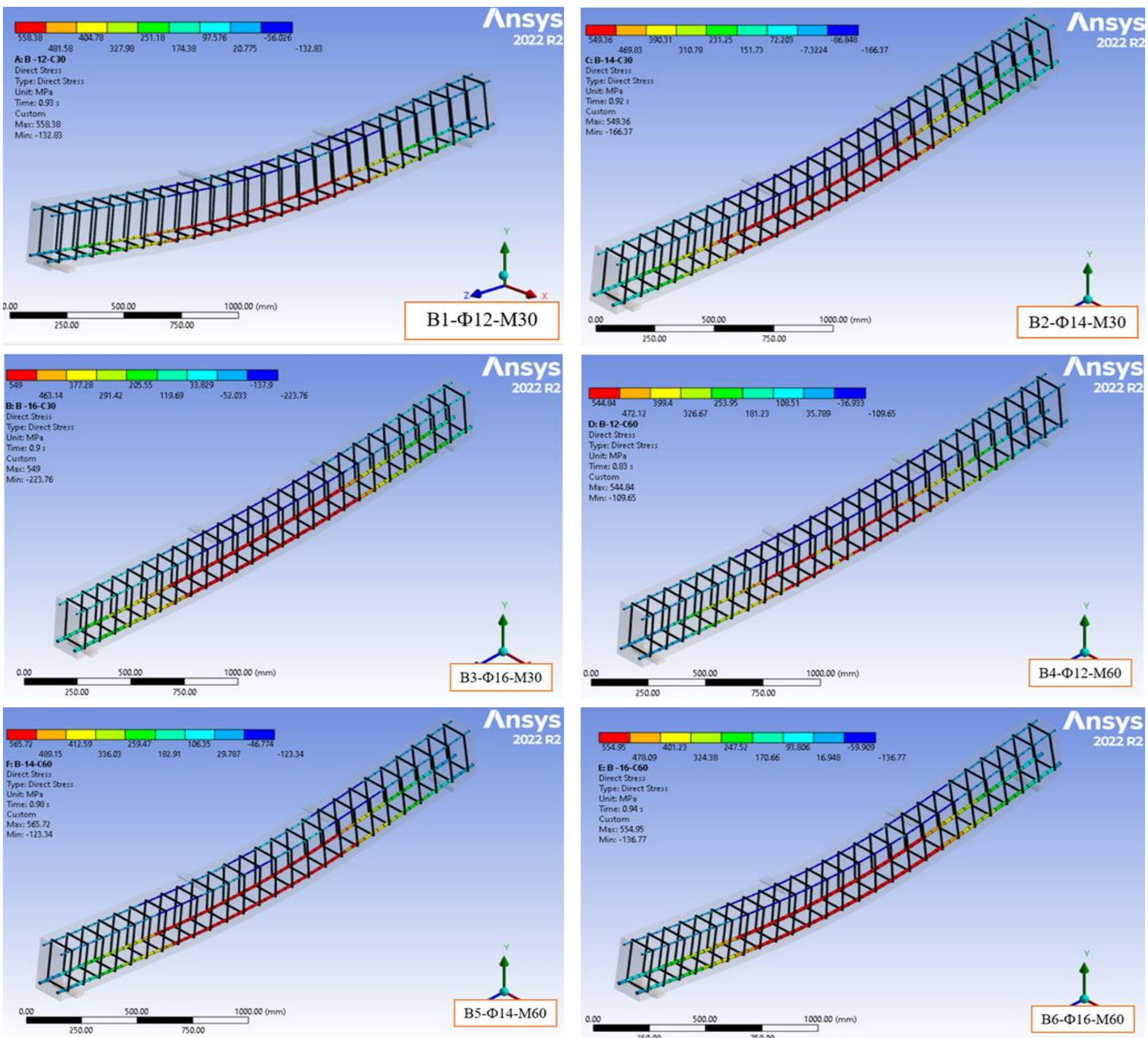


Fig. 19 Stress in the reinforcement steel of FE models at yield point.

compressive stresses while the lowest contour lines which represent the main steel reinforcement showed positive stress values indicating tensile stresses. Moreover, the stresses in the steel reinforcement of all models reached the yield strength.

CONCLUSIONS

The The numerical study allows us to conclude the following:

- The current numerical model was developed using ANSYS 2022 R2 software that was able to predict and simulate of the flexural behavior of reinforced concrete beam tested.
- According to the results, increasing the tensile steel reinforcement ratios has a considerable effect on the load-deflection capacity for the FE models.
- It was concluded that increasing the concrete compressive strength was found to remarkably affect the load-deflection behaviour of the numerical models. Furthermore, it has slightly affected the structural stiffness of the numerical models.
- The cracking pattern of the FE models in ANSYS, compatible with experimental manners.

REFERENCES

[1] Vasudevan, G. and Kothandaraman, S., "Parametric study on Nonlinear Finite Element Analysis on flexural behaviour of RC beams using ANSYS", International Journal of Civil and Structural Engineering, Vol. 2, No. 1, pp. 98 – 111, 2011.

[2] Musmar, M., "Nonlinear Finite Element Flexural Analysis of RC Beams", International Journal of Applied Engineering Research, Vol. 13, No. 4, pp. 2014 – 2020, 2018.

[3] Pawar, V. S. and Pawar, P. M., "Modelling of Flexural Failure in Reinforced Concrete Beams as Under, Balanced and Over-reinforced", International Journal of Engineering Trends and Technology, Vol. 36, No. 8, pp. 394 – 400, 2016.

[4] Pranata, A. Y., Tjitradi, D. and Prasetia, Horizontal I., "Web Reinforcement Configuration Analysis of Deep Beam Capacity and Behavior using Finite Element Model in, Engineering", Technology & Applied Science Research, Vol. 10, No. 1, pp. 5242 – 5246, 2020.

[5] Diyyala Naga Moulika, Reshma Vasireddy and P. Polu Raju "Modelling and Analysis of Reinforced Concrete Beam under Flexure using ANSYS", International Journal of Civil Engineering and Technology, Vol. 8, No. 3, pp. 1103 – 1111, 2017.

[6] Tjitradi, D., Eliatun, E. and Taufik, S., "3D ANSYS Numerical Modeling of Reinforced Concrete Beam Behavior under Different Collapsed Mechanisms", International Journal of Mechanics and Applications, Vol. 7, No. 1, pp. 14 – 23, 2017.

[7] Majeed, S. A., "Finite Element Analysis of Strengthened Reinforced Concrete Beams", Al-Rafidain Engineering Journal, Vol. 21, No. 1, pp. 134 – 145, 2011.

[8] Al-Jurmaa M. A., "Non-Liner Three Dimensional Finite Elements Analyses of Reinforced Concrete Beams Strengthened by CFRP", The Iraqi Journal for Mechanical and Material Engineering, Special Issue (D), pp. 62 – 71.

[9] Martin, A. M. and Kuriakose, M., "Finite Element Modeling and Analysis of Reinforced Concrete Beam Retrofitted with Fibre Reinforced Polymer Composite", International Journal of Engineering Trends and Technology, Vol. 38, No. 4, pp. 190 – 197, 2016.

[10] Ortes, F., Sayin, B., Bozkurt, T. S. and Akcay, C., "Numerical Simulation of Structural Behavior of NSM CFRP Strengthened RC Beams Using Finite Element Analysis", The 12th International Symposium on Fiber Reinforced Polymers for Reinforced Concrete Structures (FRPRCS-12) & The 5th Asia-Pacific Conference on Fiber Reinforced Polymers in Structures (APFIS-2015) Joint Conference, Nanjing, China, 2015.

[11] Sharaky, I.A., Reda, R.M., Ghanem, M., Seleem, M.H., and Sallam, H.E.M., "Experimental and numerical study of RC beams strengthened with bottom and side NSM GFRP bars having different end conditions", Construction and Building Materials, Vol. 149, pp. 882-903,2017.

[12] Sharaky, I.A., Torres, L., Comas, J. and Barris, C., "Flexural response of reinforced concrete (RC) beams strengthened with near surface mounted (NSM) fibre reinforced polymer (FRP) bars", Composite Structures, Vol. 109, pp. 8 – 22, 2014.

[13] ANSYS, "Release 2022/R2 Documentation", ANSYS Inc.,2022, http://www.ansys.com.