

# Nonlinear Finite Element Analysis of Composite Cantilever Beam with External Prestressing

R. I. Liban, N. Tayşi

**Abstract**—This paper deals with a nonlinear finite element analysis to examine the behavior up to failure of cantilever composite steel-concrete beams which are prestressed externally. 'Pre-' means stressing the high strength external tendons in the steel beam section before the concrete slab is added. The composite beam contains a concrete slab which is connected together with steel I-beam by means of perfect shear connectors between the concrete slab and the steel beam which is subjected to static loading. A finite element analysis will be done to study the effects of external prestressed tendons on the composite steel-concrete beams by locating the tendons in different locations (profiles). ANSYS version 12.1 computer program is being used to analyze the represented three-dimensional model of the cantilever composite beam. This model gives all these outputs, mainly load-displacement behavior of the cantilever end and in the middle span of the simple support part.

**Keywords**—Composite steel-concrete beams, external prestressing, finite element analysis, ANSYS.

## I. INTRODUCTION

### A. General

COMPOSITE steel-concrete beams which are prestressed externally by using a high strength tendons, can be used to strengthen the existing structures and also for the new constructions, mainly for the bridges and buildings [1]. Composite steel-concrete beams externally prestressed with high strength steel tendons have many advantages when compared to the non-prestressed composite beams. These advantages of the externally prestressed tendons are: the prestressed tendons increase the elastic behavior when used at higher loads, increase the ultimate capacity of the beam, decrease the deflection under the service loads, make the crack resistance of the concrete very high, improve the fatigue and fracture behavior of the steel beams [2], and finally make the use of the material more efficient and reduce the structural steel weight [3].

### B. Finite Element Analysis

Finite element (FE) is a numerical analysis method that divides the structural member into much smaller elements and then simulates static loading conditions to evaluate the response of the element members when subjected to loadings to provide an accurate prediction. The use of FE analysis has become preferred method to study the behavior of elements like concrete and steel as it is much faster than the experimental methods and also is cost effective. In this study, nonlinear FE analysis is carried out using the computer

program ANSYS which uses Newton-Raphson method to solve the higher order differential equations. The nonlinearity of this analysis is a material nonlinearity and it arise from the different stress strain relationship of materials and hence modulus of elasticity of the materials is different. In reality, most of the problems are nonlinear in nature, hence nonlinear analysis is very good tool to obtain exact solution. Nonlinear analysis method stimulates the exact behavior of the material to estimate strength in inelastic range and to recognize the potential of high load carrying capacity of components [4].

### C. Objective

The objective of this study is to use FE method to analyse the behavior of composite cantilever steel-concrete beam when externally prestressed by using a high strength tendons with the help of the computer program ANSYS version 12.1. Five tendons profiles (locations) in the steel beam section of the composite cantilever beam have been compared to find out which profile will produce the highest failure load when the beam is subjected to static loadings.

### D. Typical Element of Composite Cantilever Beam

The typical element of the composite cantilever steel-concrete beam in this study contains five main components which are: concrete slab, steel beam, prestressed tendons, slab reinforcements, and shear connectors. All of these components are shown in Fig. 1. The whole beam length with the supports is also shown in Fig. 2.

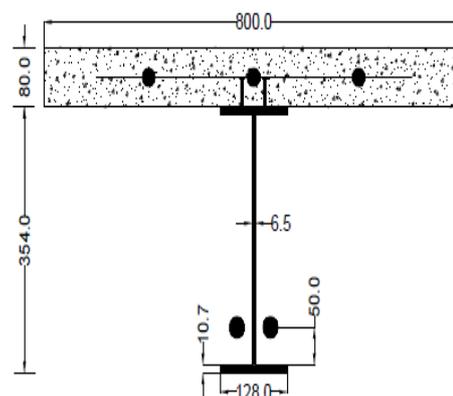


Fig. 1 The cross-section of the composite beam



Fig. 2 Length of the composite beam

Roble Ibrahim Liban is with the Department of Civil Engineering, Gaziantep University, Gaziantep, Turkey (e-mail: rooble206@gmail.com).

## II. ANALYSIS USING ANSYS SOFTWARE

The FE analysis calibration study for this study, includes modeling a composite cantilever beam with the dimensions and properties [5]. To perform an analysis by an FE model in ANSYS, three steps must be carried out (1) build the model (2) apply loads and obtain the solution and (3) review the results [6]. To create the FE model, there are multiple tasks that requires to be completed for the model to run accurately. Models can be created using two methods, command prompt line input and graphical user interface. In this model, the graphical user interface method was used to create the model. This section describes the different tasks and entries to be used to create the FE calibration model [5].

### A. Element Type

TABLE I  
 ELEMENT TYPES

Material Type Element	ANSYS Representation
Concrete Slab	Solid65
Steel Beam	Shell181
Slab Reinforcement & Tendon Steel	Link8
Shear Connectors	Combin39 and Link8
End Plates	Shell181
Surface between Concrete slab and Steel beam	Conta173 and Targe170

A Solid65 element was used to model the concrete [7]. This element has eight nodes with three degrees of freedom at each nodes translations in the x, y, and z directions. The element is capable of plastic deformation, cracking, and crushing. The most important aspect of this element is the treatment of nonlinear material properties [8]. A schematic Solid65 is shown in Fig. 2.

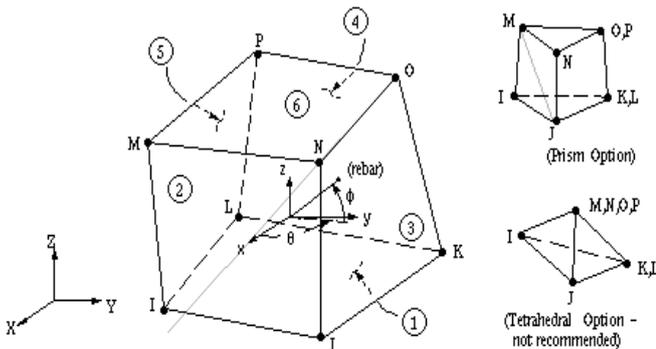


Fig. 3 Solid65 element in ANSYS

A Shell181 element was used to model the steel beam. This element has four nodes with six degrees of freedom at each node: translations in the x, y, and z directions and rotations about the x, y, and z axes. The element is used for linear and nonlinear applications. The element has also stress stiffening, large deflections, and plasticity. Schematic diagram of Shell181 is shown in Fig. 3.

A Link8 element was used to model slab reinforcement, external tendons, and shear connectors. This Link8 is a uniaxial tension-compression three-dimensional spar element and it has two nodes with three degrees of freedom:

translations in x, y, and z direction. The element has plasticity, creep, swelling, stress stiffening, and large deflection capabilities. A schematic diagram of Link8 is shown in Fig. 4.

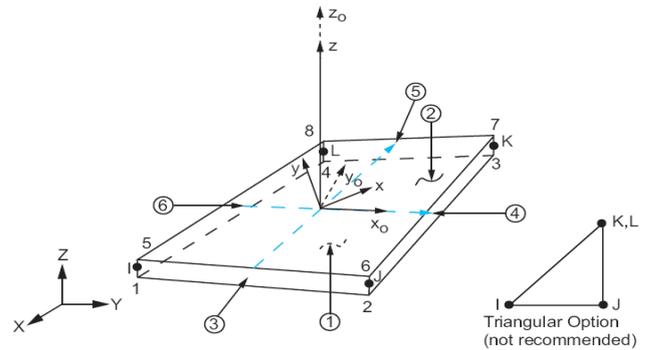


Fig. 4 Shell181 element in ANSYS

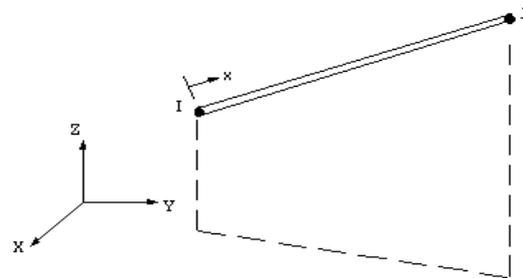


Fig. 5 Link8 element in ANSYS

Combin39 and Link8 were used to model the shear connectors. Combin39 is used to prevent the normal force between the concrete slab and steel beam, while the Link8 works as stirrups in resisting the vertical shear at the concrete layer. Combin39 is a unidirectional element which generalized nonlinear force-deflection capability that can be used in any analysis. The element has large displacement ability, and there are two or three degrees of freedom at each node [8]. A schematic diagram of Combin39 is shown in Fig. 5.

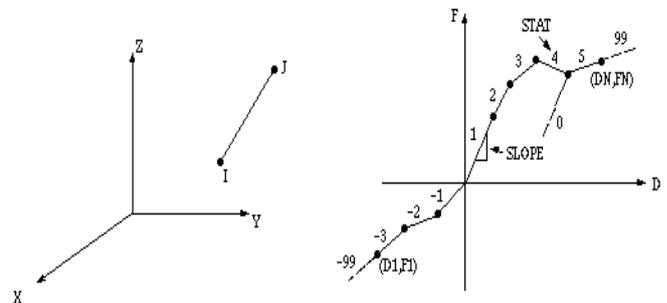


Fig. 6 Combin39 element in ANSYS

A three-dimensional nonlinear node to area "contact-pair" element (Conta173 and Targe170) is used to model interface surface between concrete slab and steel beam. The contact-pair consists of the contact between two boundaries. Conta173 in ANSYS represents contact, slid, and deformable surface taken as contact surface and Targe170 in ANSYS represents

the rigid surface taken as a target surface. A schematic diagram of Conta173 and Targe170 is shown in Fig. 6.

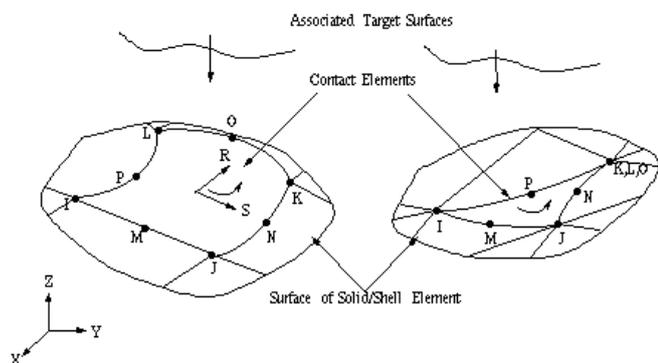


Fig. 7 Conta173 and Targe170 elements in ANSYS

### B. Real Constant

Element real constants are properties that depend on the element type, like cross-sectional properties of an element. For example, area (AREA), moment of inertia (IZZ), height (HEIGHT), initial strain (ISTRN), and added mass per unit length (ADDMAS). Not all element types need real constants, and different elements of the same type may have different real constant values [6]. Real Constant Set 1 was used for Solid65 element and it needs real constant rebar assuming a smeared model. Values can be inserted for material number,

volume ratio, and orientation angles. In the present study, the concrete slab was modeled using discrete reinforcement. Therefore, zero value was entered for real constants, which changed the smeared reinforcement capability of the Solid65 element. Real Constant Set 2 was used for Shell element. Real Constant Set 3 refers to the slab reinforcement with 12 mm diameters bars, and the initial value strain of zero is entered because no initial stress is the reinforcement. Real Constant Set 4 refers to prestressed tendons with cross-sectional area of 175 mm<sup>2</sup>, and initial strain of 0.0056 is entered. Here, the tendons are prestressed, that is why we entered the initial strain. The Real Constant Set 5 is Link8 used for the shear connectors with number 18 mm diameter bars and zero initial strain is entered. Lastly Real Constant Set 6 refers to the Combin39 which has force-deflection curve.

### C. Element Properties

Modeling an element for the behavior of concrete is challenging work. Concrete is a quasi-brittle material and it shows different behavior in compression and tension. A modulus of elasticity of 30000 MPa and Poisson's ratio of 0.2 is used for concrete. The shear transfer coefficient for open crack is 0.3 and for closed crack it is 1. Uniaxial tensile cracking stress is 3.9 MPa and uniaxial crushing stress is used for 40 MPa. Properties of the materials are shown in Table III.

TABLE II  
 REAL CONSTANT FOR THE ELEMENT

Real Constant Set	Element Type		Constants		
			Real Constant for Rebar 1	Real Constant for Rebar 2	Real Constant for Rebar 3
1	Solid65	Material Number	0	0	0
		Volume Ratio	0	0	0
		Orientation Angle	0	0	0
		Element x-axis rotation		0	
2	Shell181	Add mass/unit area		0	
		Transverse shear stiffness		0	
3	Link8	Area (mm <sup>2</sup> )		113	
		Initial Strain		0	
4	Link8	Area (mm <sup>2</sup> )		175	
		Initial Strain		0.0056	
5	Link8	Area (mm <sup>2</sup> )		201	
		Initial Strain		0	
6	Combin39	Data set 1	-0.10		-3200
		Data set 2	0		0
		Data set 3	0.10		3200

The ANSYS needs the uniaxial stress-strain relationship for concrete in compression. The Solid65 element needs linear isotropic and multilinear isotropic material properties to properly model the concrete. The multilinear isotropic material uses the Von-Mises failure criterion to define the failure of the concrete. Fig. 8 shows the stress-relation of concrete.

Material model number 2 is used for the slab reinforcement, the steel beam, and the shear connectors. This material is steel of grade Fe415. In this study, the elastic modulus of this

material is 200,000 MPa and the Poisson's ratio of 0.3 has been used. This material was modeled as bilinear isotropic with yield strength of 415 MPa and Tangent modulus 6000 MPa.

The material model number 3 is just used for the modeling of the prestressed tendons. This material is modeled as multilinear isotropic hardening material since prestressed tendons are very high strength steel bars. The elastic modulus and Poisson's ratio used for this model are 206,290 MPa and 0.3 respectively.

TABLE III  
MATERIAL MODEL NUMBER 1

Material Model Number	Element Type	Material Properties		
1	Solid65	<b>Linear Isotropic</b>		
		EX	30000 MPa	
		PRXY	0.2	
		<b>Multilinear Isotropic</b>		
			Strain	Stress (MPa)
		Point 1	0.00025	7.5
		Point 2	0.00050	13.3
		Point 3	0.00075	18.8
		Point 4	0.00100	22.9
		Point 5	0.00125	26.9
		Point 6	0.00150	29.6
		Point 7	0.00175	31.3
		Point 8	0.00200	32.7
		Point 9	0.00225	33.7
		Point 10	0.00240	34.3
	<b>Von-Mises Failure Criterion for Concrete</b>			
	Open Shear Transfer Coefficient	0.3		
	Closed Shear Transfer Coefficient	1		
	Uniaxial Cracking Stress	3.9 MPa		
	Uniaxial Crushing Stress	40 MPa		
	Tensile Crack Factor	0.6		

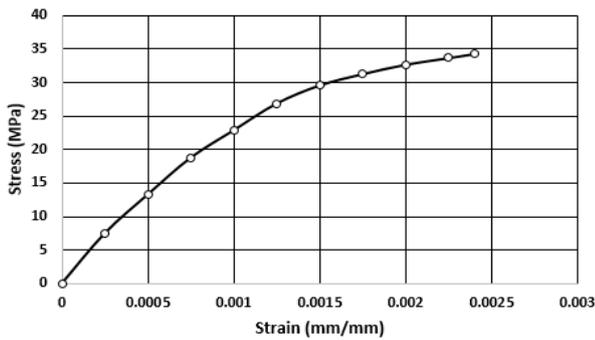


Fig. 8 Stress-strain relationship for concrete

TABLE IV  
MATERIAL MODEL NUMBER 2

Material Model Number	Element Type	Material Properties	
2	Shell181 Link8 Combin39	<b>Linear Isotropic</b>	
		EX	200 000 MPa
		PRXY	0.3
		<b>Bilinear Isotropic</b>	
		Yield Strength	415 MPa
		Tangent Modulus	6000

TABLE V  
MATERIAL MODEL NUMBER 3

Material Model Number	Element Type	Material Properties		
3	Link8	<b>Linear Isotropic</b>		
		EX	206290 MPa	
		PRXY	0.3	
		<b>Multilinear Isotropic</b>		
		Point 1	0.007	1444 MPa
		Point 2	0.010	1740 MPa
		Point 3	0.026	1970 MPa
		Point 4	0.040	2050 MPa

#### D. Modeling

As it was stated in the introduction, the composite cantilever steel-concrete is analysed using FE analysis tool ANSYS version 12.1. The total span of the beam is 6400 mm. The depth of the steel beam is 354 mm with top and bottom flanges of same thickness 10.7 mm where the web is 6.5 mm. The concrete slab was modeled as volume block, and the dimensions for the concrete volume were shown in Table VI.

TABLE VI  
CONCRETE SLAB DIMENSION 3

ANSYS Coordinate	Concrete Dimension (mm)
X <sub>1</sub> , X <sub>2</sub>	0 6400
Y <sub>1</sub> , Y <sub>2</sub>	354 434
Z <sub>1</sub> , Z <sub>2</sub>	-400 400

1 AREAS  
TYPE NUM

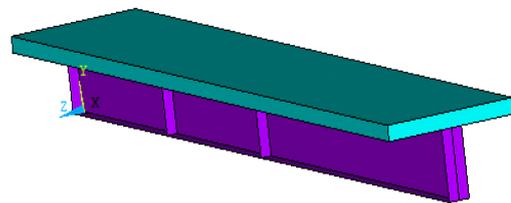


Fig. 9 FE model

#### E. Meshing

The concrete slab and the steel beam are meshed differently. To get a good result from the Solid65 element, the use of a rectangular mesh was suggested [5]. Therefore, the mesh was set up so that rectangular or square elements were created. The meshing of the steel beam is meshed as an area to create the thickness of the flanges and web. The meshing of

the reinforcement, prestressed tendons, contact surface, shear connectors was a special case compared to the volumes. No mesh was needed for these elements because individual

elements were created in the modeling through the nodes created by the mesh of the concrete volume and the steel beam.

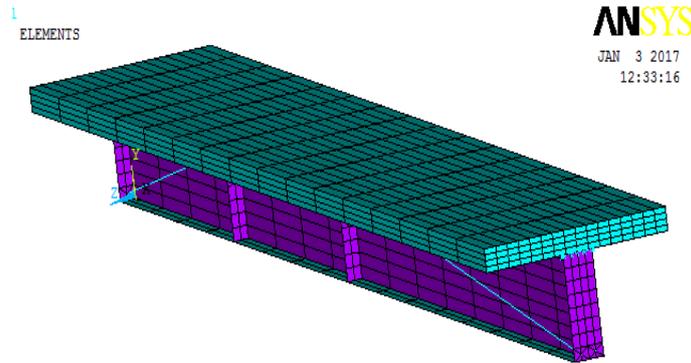


Fig. 10 FE mesh

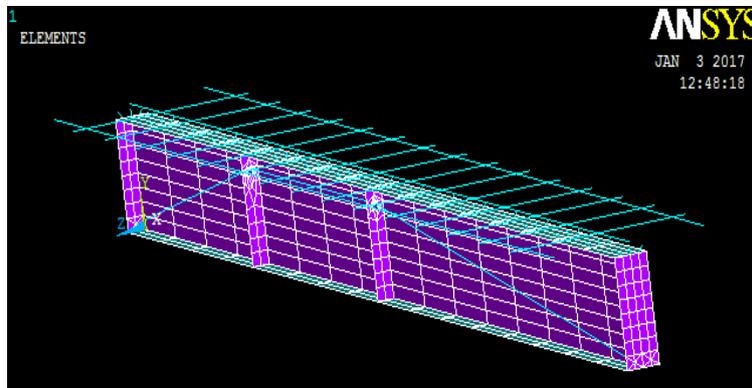


Fig. 11 FE mesh of slab reinforcement and tendons

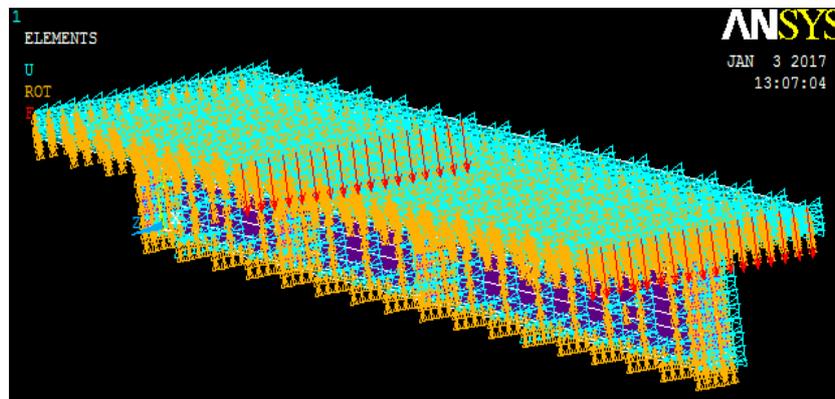


Fig. 12 Loading and boundary conditions

#### F. Loads and Boundary Conditions

Displacement boundary conditions are needed to constrain the model to get a unique solution for each model. The supports were modeled as a roller support and pinned support. The force, P, applied at the concrete slab is applied across the centerline of the plate. The force applied at each node on the plate is 1/34 of the actual force applied. Fig. 10 shows the loading and boundary condition of the composite cantilever beam.

#### III. RESULTS AND DISCUSSIONS

After the composite cantilever steel-concrete was modeled using ANSYS program, the locations of the prestressed tendons were changed to analyze the behavior of this beam, and the effects will be seen by looking at the failure loads and the displacements of the middle points and the cantilever end of the overhanging part of the cantilever beam. The five prestressed tendon profiles and their dimensions are shown in Fig. 13.

The analysis carried out the prestressed force in the external tendons, and all the other parameters like the applied loads and concrete and steel properties are kept constant, and the profile of the tendons is just changed. The loads are applied at two points, the mid-point of the simple support between the two supports and the cantilever end of the overhanging part. Fig. 14 shows the loading condition of the composite cantilever beam used this model.

In this study, there are mainly two shapes of the prestressed tendons which are straight tendons and draped tendons. First, the effects of the straight tendons on two different profiles have been compared: the top and the bottom of the steel beam sections. The failure loads and the displacement of these two profiles will be discussed here to see which profile is the best and increases the failure load. Two positions will be analyzed throughout the length of the beam, the mid-point between the two supports and the cantilever end.

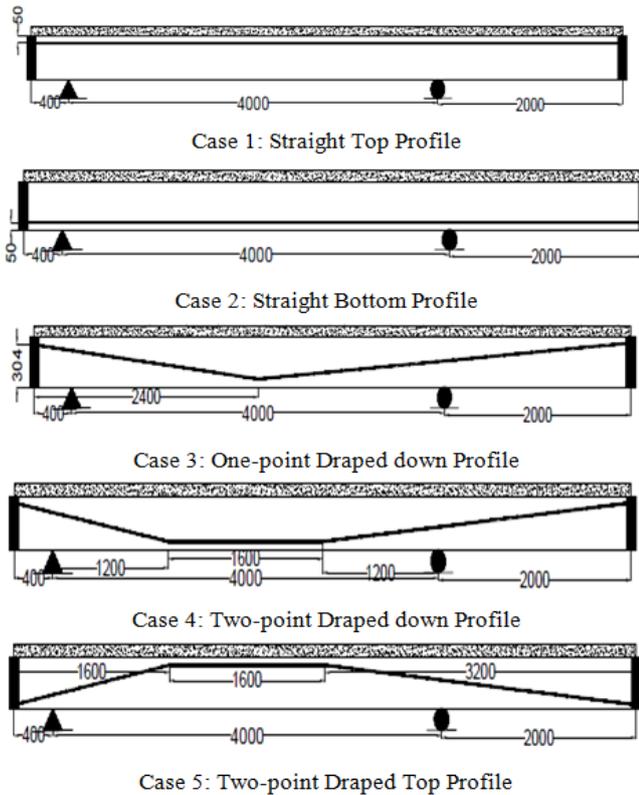


Fig. 13 Tendon profile

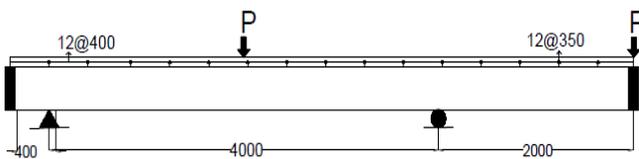


Fig. 14 Loading conditions

From these curves, it can be seen that when the external prestressed tendons are placed near the top flanges of the steel beam, the failure is higher, and displacement is lower than when the tendons are in the bottom of the steel beam.

The draped tendon shapes are located at three different places to investigate the best profile, and their load against displacement curves is also shown in Fig. 16.

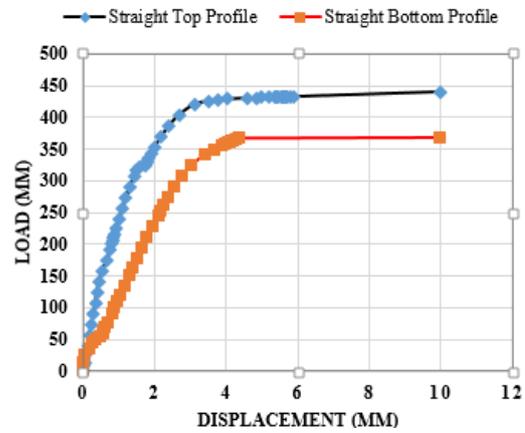


Fig. 15 Load vs displacement curve at the mid-span

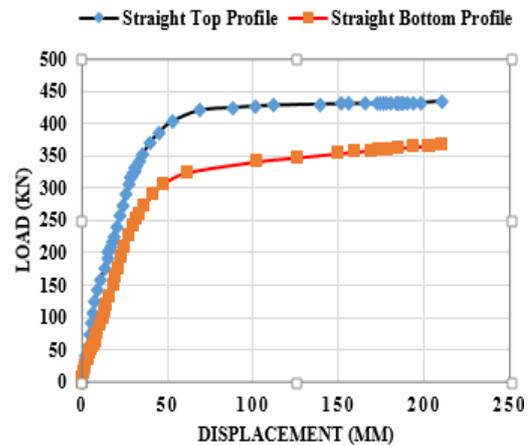


Fig. 16 Load vs displacement curve at the cantilever

#### IV. CONCLUSION

- The FE model used in the present work is able to simulate the behavior of prestressed composite cantilever steel-concrete beams. The analytical tests were carried out for the different cases (straight and draped) indicating the load-deflection behavior and the ultimate loads, keeping the prestressing force constant and changing the tendon profile. The analysis is repeated and checked the best profile for producing the highest failure load.
- When the prestressed tendons are placed in the top of the steel beam section, they produce a higher failure loads and smaller displacement than the other location like putting them in the bottom.
- Adding prestressed tendons to composite beams significantly increases the yield load and the ultimate load.
- The farther the tendons are located from the neutral axis, the greater is the increase in strength. This is why, tendons are placed near the flanges of the steel beam.

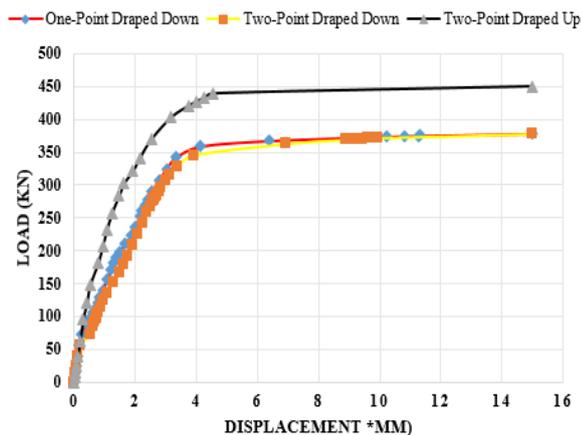


Fig. 17 Load vs displacement curve at the midspan

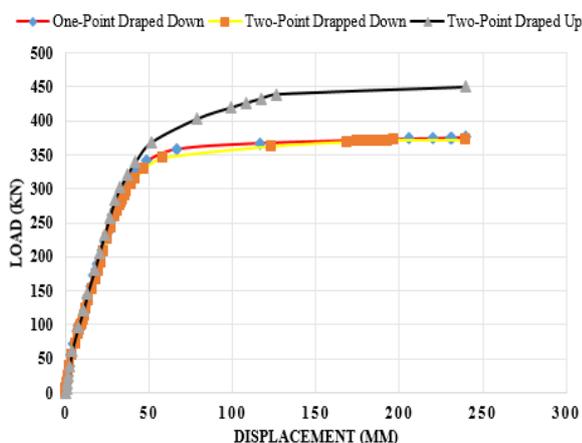


Fig. 18 Load vs displacement curve at the cantilever

#### REFERENCES

- [1] Saadatmanesh H., Albrecht P., and Ayyub B., (1989), "Experimental study of prestressed composite beams". *ASCE, Journal of structural engineering*, 115(9), pp 2348-2363.
- [2] Shiming Chen, a. P. (2005). Load Carrying Capacity of Composite Beams Prestressed with External Tendons under Positive Moment. *Journal of Constructional Steel Research*, pp 515-530.
- [3] Ayyub, B. M., Sohn, & Saadatmanesh, H. (1990). Prestressed Composite Girders under Positive Moment. *ASCE, Journal of Structural Engineering*, pp 2931-2951.
- [4] Badiger, M., Neha, S., & M, K. (2014). Parametric Study on Reinforced Concrete Bam Using ANSYS. *Journal of Civil and Environmental Research*, pp 2225-0514.
- [5] Prof. S. S. Pital, A. N. (2012). Nonlinear Finite Element Method of Analysis of Reinforced Concrete Deep Beam. *International Journal of Modern Engineering Research*, pp 4622-4628.
- [6] ANSYS Inc., S. (2013). *ANSYS Mechanical APDL Basic Analysis Guide*. Canonsburg, PA 15317, USA: 275 Technology Drive.
- [7] Amer M, I., Mohaisen, S. K., & Ahmed, Q. (2012). Finite Element Modeling of Composite Steel-Concrete Beams with External Prestressing. *International Journal of Civil and Structural Engineering*, pp 0976-4399.
- [8] ANSYS Inc., S. (2013). *ANSYS Mechanical APDL Element Reference*. Canonsburg, PA 15317, USA: 275 Technology Drive.

**Roble Ibrahim Liban** is a student at Gaziantep University in Civil engineering department as a master's degree student; He studied civil engineering at Eelo University in Borama, Somalia.

**Asst. Prof. Dr. Nildem TAYŞI** is an assistant professor at Gaziantep University. She has graduated with PhD from Gaziantep University; since then she has published many works in the field of civil engineering, mechanics division.