International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249 – 8958, Volume-5, Issue-4, April 2016

Numerical Modeling of Innovative Connection Between RC Beam and Steel Column

Ahmed H. El-Masry, Mohamed A. Dabaon, Tarek F. El-Shafiey, Abd El-Hakim A. Khalil

Abstract—Transferring the load from beam to the column by a safe way is considered one of the critical issues which have been studied by many researchers in many fields. The failure of the connection between the column and the beam is one of the major reasons which causes structures failure and has a great effect on the acceleration of collapse. Experimental study has been provided to investigate the transfer part technique as a structural system which aims to keep the failure location at the beams side. This paper presents the finite element modeling to simulate that technique for expanding the research field. A discussion is provided for the effect of increasing the column stiffness through the experimental results. The study indicates that the use of finite element modeling by a specific materials and elements could be given an asymptotic behavior of the behavior of the experimental test.

Key words—Finite element modeling, RC beam, Steel column, Transfer Part.

I. INTRODUCTION

Using a transfer part technique to connect RC Beam to a steel or composite column is considered one of the innovative ways which opens the field for more studies.In this technique, the main beam of the frame consists of a transfer part (part of beam; Tr.P) and a common reinforcement concrete beam. The transfer part of the beam is connected to the column, whereas the rest of the beam is connected to the transfer part from each side.The study of the connection between steel column and RC beam is considered a complicated problem. It consists of many different materials; each of its components has a non-linear behavior in addition to the nonlinearity due to the interaction between the column and beam part. Furthermore, the unknown behavior of the beam components may be contributed to the connection behavior.

Researchers examined finite element modeling of different types of structural elements. Their models changed regarding to the material, structural system and the applied loads. Wei Li and Lin-Hai Han, 2011, [1] presented a finite element analysis (FEA) model for the concrete-filled steel tubular (CFST) column to steel beam joint with a reinforced concrete (RC) slab under cyclic loading. Julio Garzón-Roca et al, 2012,[2] proposed a finite element modeling of RC columns strengthened by a steel cage under axial loads and bending moments. 3D finite element model is proposed by Mohamed A. Dabaon et al, 2007,[3] for analytical investigation and comparative study with the experimental results of testing frame with rigid or semi-rigid joint. In the previous work mainly there were no changes provided at the column stiffness to modify the beam behavior. As far as we are aware, no study has been published to date dealing with a numerical simulation of an RC beam connected to a steel column by using a transfer part technique. Elremaily, Azizinamini [4], [5] presentedand developed an economical connection detail for connectingsteel beams to concrete filled tube (CFT) columns.

This paper describes an analysis by the finite element method (FEM) of the behavior of RC beam connected to a steel or composite column through using a transfer part. The work is a continuation of a previous experimental study carried out in the Faculty of Engineering at Tanta University [6] using the finite element software ANSYS multi-physics v.14 [7].

II. SUMMARY OF PREVIOUS EXPERIMENTAL WORK

Previous experimental work [5] had been carried out at the heavy structure laboratory, by Faculty of Engineering, Tanta University and consisted of a test on four full-scale specimens simulating the beam–column connection for the transfer part technique in a multi-story building.

The specimens were designed to represent an exterior beam-column connection. Each of column and beam elements represents a part of its full length and the provided length for both of them is up to contra-flexural point. Four specimens (SP1, SP2, SP3 and SP4) were built with the same dimension as shown in Fig. 1. SP1 is a control beam whereas the reinforcement bars are connected directly to the column without transfer part. SP2, SP3 and SP4 use a transfer part to transfer the load from the common reinforced concrete beam to the column. The geometry, dimensions, and reinforcement detailing of the test specimens are depicted in Fig. 1. The net height of all columns is 2220mm each. The beam length is 1200mm.Test program includes four specimen frames. The concrete cross section of all beams is (160mm) in width and(320mm) in depth. Overall length of each beam is (1200mm). The reinforcement of all beams is same; running along the span, there are four bars at the top and four bars at the bottom, each with a diameter of 12mm. For shear reinforcement, 8mm diameter stirrups are often placed 100mm apart along the entire length of each beam. All steel beams (IPE 240) connected to the column (HEB 160) in the same way; the steel beam is welded to an end plate which is connected to the column by using bolts [8]. This part of the beam is considered a transfer part; the remaining part is reinforced concrete one with top and bottom reinforcement. It is important to state that the reinforcement covers the whole span of the beam including the transfer part.

Revised Version Manuscript Received on March 27, 2016.

A. H. El-Masry, PhD Candidate, Faculty of Engineering, Tanta University, Egypt.

M. A. Dabaon, professor in steel structures with Faculty of Engineering, Tanta University, Egypt.

T. F. El-Shafey, professors in concrete structures with Faculty of Engineering, Tanta University, Egypt.

A. A. Khalil, professors in concrete structures with Faculty of Engineering, Tanta University, Egypt.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

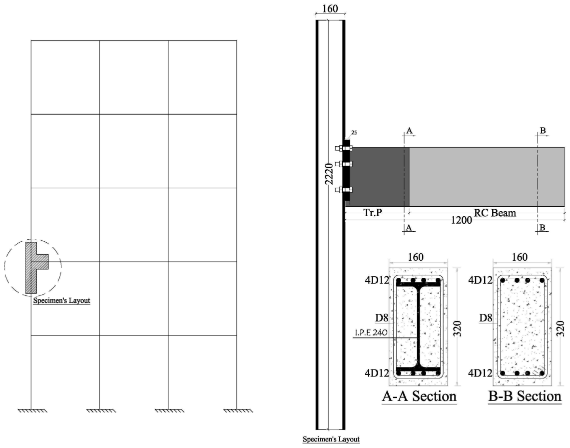

Fig.1. Experimental specimens details (dimensions in mm).

Columns of all specimens have the same support from top and bottom. Columns is allowed to rotate and prevented for vertical and horizontal movement by using hinged support as shown in Fig. 2. Static load was applied vertically at the end of the beam. Fig. 2 illustrates the test setup that was employed, indicating the ideal support and loading conditions.

Simulation of the experimental tests by using finite element modeling is one of the methods for expand the research field. Verification of the finite element model is the first essential step to ensure that the used elements, materials and modeling technique at the simulation model are approached to the experimental results.

III. FINITE ELEMENT MODEL

The laboratory tested specimens were numerically simulated by FEM on ANSYS Multi-Physics v14. All elements, materials and the modeling method which used to simulate the present problem are going to discuss at this section.

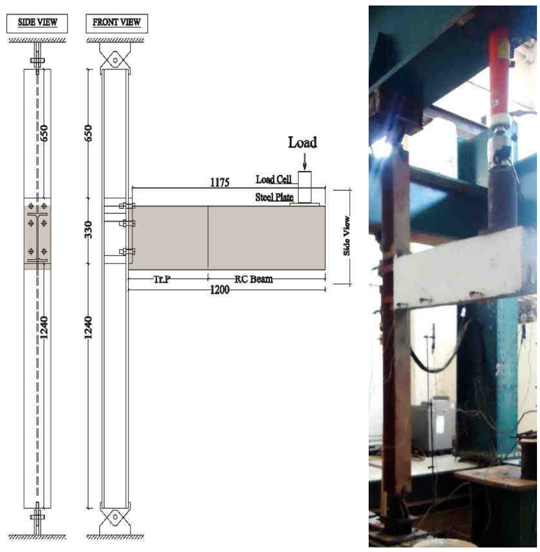

Fig.2. Experimental specimens' setup

A. Description of the finite elements used, boundary condition and loads applied

Due to the existence of symmetry in the laboratory tested specimens, only a half of each specimen was modeled; applying symmetry conditions on the corresponding plan; see Fig. 3.

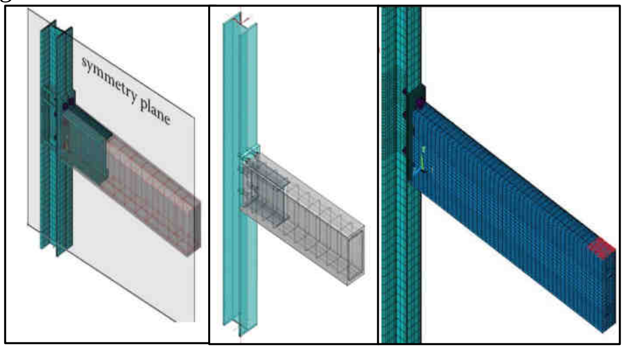

Fig. 3. Modeling 1/2 of the specimen, boundary condition and applied loads for all models

The steel elements of beam, column, end plate, bolt head and shank were modeled using solid eight-node brick element which has plasticity and large deflection capabilities (SOLID45)[9]. Eight-node brick element (SOLID65) with three translational degrees of freedom at each node was used to model the concrete core, while steel rebars and links were modeled using link elements (LINK8) with three translational degrees of freedom at each node[7]. Full bond was assumed between the rebars and concrete. The weld was idealized by providing common nodes for the girder and the steel tube in the weld locations.

Test program includes four specimen frames; all of them have a composite beam connected to a hinged-to-hinged column. Loading of all specimens was by vertical load uniform acted in the last 100mm within the full simulated beam width; see Fig. 3.

B. Concrete constitutive model

The cracking or crushing of the concrete of composite section at column or beam is considered in the current model by using a constant stress cutoff criterion. This means that once the maximum principal tensile stress reaches the tensile strength, f_t, independent of the principal stresses, a crack is initiated perpendicular to the maximum principal stress. The orientation of the crack is then stored and the material stiffness perpendicular to the crack vanishes. The post cracking behavior (softening) of the concrete is not available in ANSYS. The shear stiffness of opened and closed crack introduced as ratios of the initial shear stiffness. The shear transfer coefficients in ANSYS for closed and open cracks in the concrete element are assumed to be 1.0 and 0.5, respectively. In compression the stress –strain relation suggest is multi-isotropic. The elastic stress state is limited by von Mises. The plasticity is done before cracking and crushing check. The concrete material model check the crushing failure mode every load step.

Many numerical equations have been used to express the stress-strain relationships for concrete under uniaxial compression. The stress-strain relationship in current model is furnished by the following function [10, 11]:

$\delta = E.\varepsilon/[(1+\left(\frac{\varepsilon}{\varepsilon_o}\right)^2]$ Where, $E = 2.\delta \max/\varepsilon_o$, ε is the strain at the point of calculate stress, ε_o is the strain at the maximum stress. Concrete compressive failure occurs at the

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

strain ε = 0.0033. This curve is reflected for tension stress but with cracking control with value of f_t that the cut off criterion will be presented.

C. Steel constitutive model

When RC beam contains reinforcement and also this reinforcement is continued inside the composite part and surrounded by concrete each side, the reinforcement can be modeled with either embedded reinforcement option or as a truss element. The von Mises yield criterion with associated flow and isotropic hardening rule is used to describe the constitutive behavior of the reinforcements. In the current model, the reinforcement modeled as a truss element. The stress-strain relationship for steel reinforcement is modeled as elastic perfectly plasticity. Elastic modulus and yield stress, values of 200,000MPa and 530MPa are taken in the case of the reinforcement while multi-linear isotropic hardening plasticity curve is used to model the stress-strain relationship of steel beam, steel plate, steel column and bolts. Elastic modulus and yield stress for steel beam, steel plate and steel column is 210,000MPa and 336MPa while 209,090MPa and 575MPa is the elastic modulus and yield stress for bolts. Poisson ratio for all steel models was assumed to be 0.3.Time-dependent nonlinear such as creep, shrinkage and temperature change is not considered in current model. The coefficient of friction between steel and concrete was assumed to be 0.3.

D. Interaction between the different materials

All materials, used in the numerical analysis model, were idealized according to material modeling available in the ANSYS program [7].

E. Contact between end plate and column flange

The interaction between column flange and end plate had to define by 3D interface elements because of the two surface not be joint together at one point that at this case the program consider the end plate welded to the column which is not true so at the modeling kept 1mm gap between end plate and column flange; connection between them is made by contact element called CONTAC 52, These elements prevent the penetration of end plate into column flange but allow separation of the end plate and column flange and also, allowing the end plat to slide up and down. The stiffness value of the contact element "Contac 52" is calculated as per equation $k = 5\times AE/(n\times t)$ [10] where A is the end plate surface area, E is the plate modulus of elasticity, n is the number of interface elements connecting the two surfaces and t is the minimum of end plates and column flange thicknesses.

F. Contact the bolt to end plate and column flange

Contact pair 170, 174 elements is used to model the complex interaction between the bolt shank -to- end plate and column flange [7] and also, the contact surface between the bolt head and nut -to- end plate and column flange as shown in Fig.4. These elements prevent the penetration of bolt head or bolt nut into end plate or column flange.

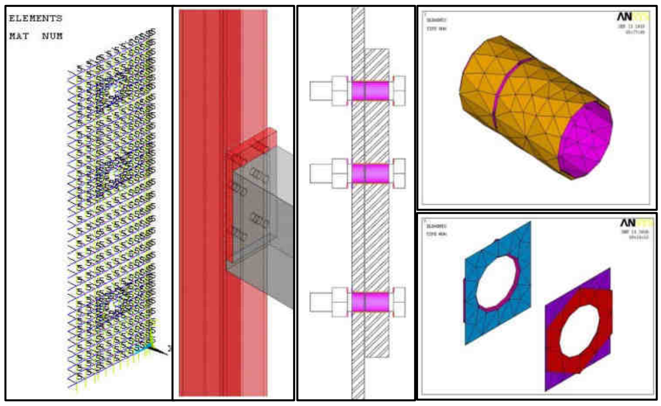

Fig. 4.Configuration of 3D interface element, Contact elements between bolt shank, head and nut-to-end plate and column flange

In problems involving contact between two boundaries, one of the boundaries is conventionally established as the target surface and the other as the contact surface. These two surfaces together comprise the contact pair. TARGE170 element is used to represent the target surface and CONTA174 element is used to represent the contact surface These elements are locates on the surfaces of 3D solid elements and have the same geometric characteristics as the solid elements face as shown in Fig. 4 that there is gap between bolts shank and both of column flange and end plate, that gap is 1mm only around the bolt shank.

IV. COMPARISON BETWEEN NUMERICAL AND EXPERIMENTAL RESULTS

The procedure used in the numerical modeling was validated by comparing the results obtained from the FE models with the Experimental results [6], See Fig. 5.for specimens' details.

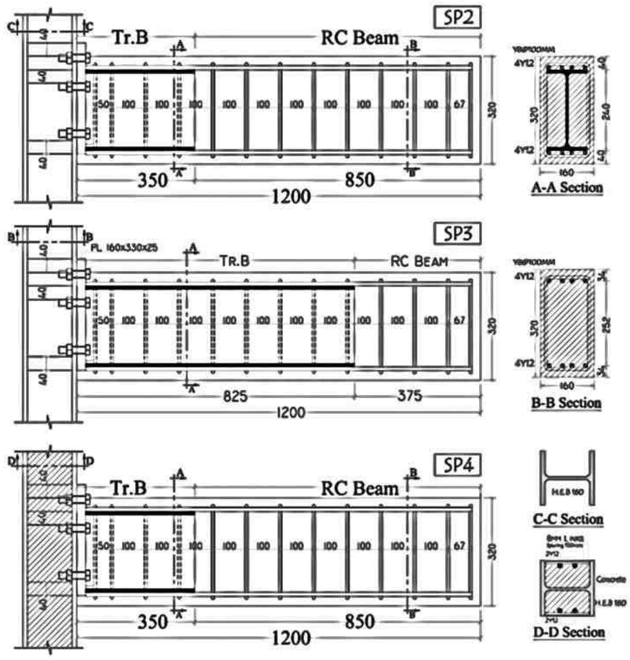

Fig. 5. Experimental specimens Details

Studying the beam behavior in case of using a transfer part technique is one of the main targets of the experimental study, so load-displacement relationship is considered one of the important curves which give an impression about the

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

beam behavior. Load-displacement relationship is used at this section to compare between the experimental and numerical results. As shown in Fig. 6, beams load-displacement relationship of the simulated model is close to the experimental one and the deformed shape of the numerical analysis is almost same as the deformation of the experimental test. So, the simulation model using the proposed elements and materials could be considered as one of the options that could be used to expand the research scope.Table 1 shows the maximum load and the corresponding deflection of SP2, SP3 and SP4 from the experimental test and the numerical analysis.

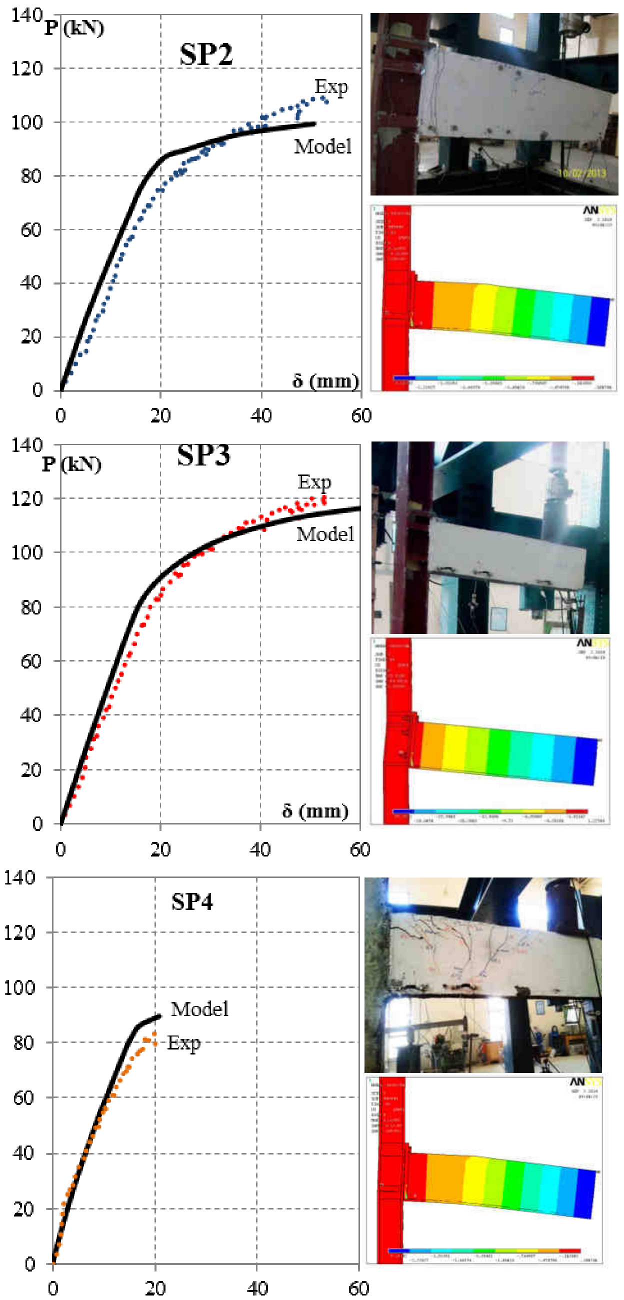

Fig. 6.Load-deflection relationship and deformation shape of the experimental and the numerical results of SP2, SP3 and SP4.

It has to be stated that the used numerical models is shown a high performance at the simulation of the experimental specimens.

Table 1: Comparing between the maximum load and the corresponding deflection from the experimental results and the numerical analysis.

ID	Experimental Results		Numerical Results	
	Maximum Load (kN)	Maximum Deflection (mm)	Maximum Load (kN)	Maximum Deflection (mm)
SP2	109.50	52.95	99.20	50.40
SP3	121.28	52.81	118.20	68.70
SP4	83.40	19.91	92.80	25.15

V. HIGHLIGHT THE IMPACT OF THE USE OF A COMPOSITE COLUMN IN THE PROPOSED TECHNIQUE

The above mention results in Table 1 show that the maximum load and deflection for both of SP2 and SP4 is different; both of them has the same details for the beam but the column of SP4 is a composite column while the column of SP2 is a bare steel column. The use of composite column instead of bare steel column affects at increasing the stiffness of the system along the test up to failure, see Fig10 for the load-rotation relationship for both of SP2 and SP4. Maximum load of SP4 is less than SP2 by 24%, toughness and beam ductility index of SP4 are less than SP2 by 54% and 73% respectively.

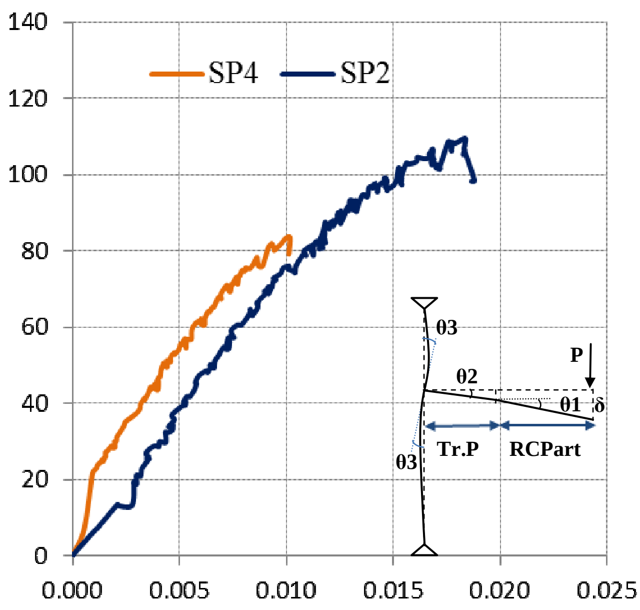

Fig.7. Load-rotation relationships of SP2 and SP4

Load-rotation relationship is shown that the linear stage of SP4 is occurred before SP2 yield stage. The ability of SP2 column to rotate allow it to pass the crack stage without change at its stiffness while SP4 stiffness is decreased at 21.50kN which caused a moment value at the transfer part by 16.55kN.m "M_{Tr}=Load x distance from load to the intersection point between the transfer part and the RC part" which is more than the crack moment of reinforced concrete section as per the next calculations. After this drop at SP4 stiffness, its stiffness value is continued higher than SP2 up to end of loading.

$E_s = 2 * 10^5 N/mm^2$

As per ECP[13]→ $Ec = 4400\sqrt{f_{cu}}$

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

$$Ec = 4400\sqrt{50} = 31112.7N/mm^2$$

$$\eta = \frac{E_s}{E_c} = 6.43$$

$$I_{cr} = \frac{160 * 320^3}{12} + 2[4 * 113 * 6.43(160 - 34)^2]$$
$$= 529189409.4mm^4$$

$$f_{ctr} = 0.6\sqrt{f_{cu}} = 0.6 * \sqrt{50} = 4.243$$

$$M_{cr} = \frac{f_{ctr}.I_{cr}}{h/2}$$

$\rightarrow$ *while h is the height of the section*"320mm"

$$\mathrm{M_{cr}} = \frac{4.243 * 529189409.4}{160} = 14033441.65\mathrm{N.mm}$$
$$\cong 14\mathrm{kN.m}$$

The applied moment at the transfer point "M_{Tr}" is exceeded the crack moment value by 15%. Both of specimens; SP2 and SP4 have the same transfer part (Tr.P) specification. Steel Tr.P of SP4 is recorded a high increase in strain value than steel Tr.P of SP2 after crack load. The use of composite column instead of bare steel column increased the stress at the beam which causes its strain increase. It has to be notified that the ability of SP2 column to rotate allow the stresses to be shifted to the column which keeps the steel beam strain in its gradually linear increase up to end of loading as shown in Fig.8

Table. 2 Beam deflection, toughness, ductility index and rotation angles at the maximum load level

ID	Rotation angles at SP1 failure mode (39.67kN)			Rotation angles at SP4 failure load (83.40kN)		
	θ1	θ2	θ3	θ1	θ2	θ3
SP2	0.0437	0.0514	0.0061	0.0199	0.0260	0.0144
SP4	0.0051	0.0064	0.0032	0.0169	0.0182	0.0080

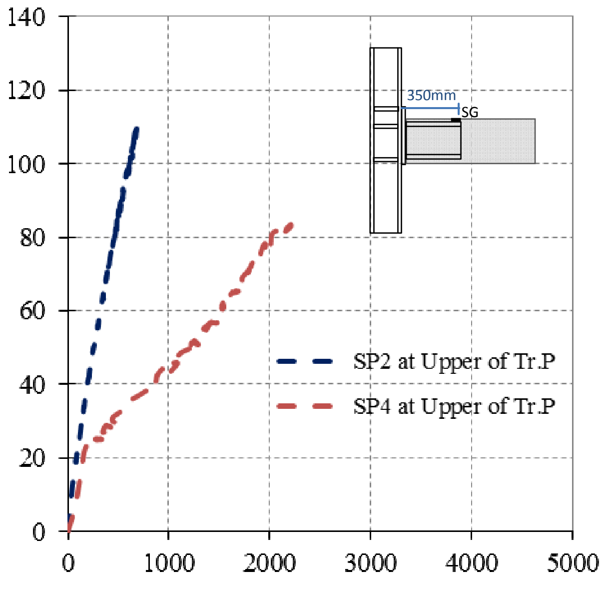

Fig.8. Load-strain relationships forthe Tr.P of SP2 and SP4

The different rotation angles of SP4 are less than the rotation angles of SP2 at all stages of loading. There is no rotation angle values could be considered for SP4 column. Rotation angles along the beams of SP2 and SP4 (θ1 and θ2) did not has a big difference which indicate that the two parts of the beam (Tr.P and RC part) is behaves as a one part until achieved the maximum capacity of the RC part. After passing the maximum capacity of the RC part, plastic hinge is created near to the transfer part edge of SP4 specimen while the rotation ability of SP2 column provides other location for the plastic hinge creation but at the column not at the beam as has occurred for SP4.

VI. Conclusions and Recommendations

It can conclude from the comparison between the numerical and the experimental results that the numerical simulation can be used to expand the research field and give an idea of the general behavior of the frame and understood how loads are transferred through the system. The finite element simulation as a primary study by using the appropriate elements and the right material definitions could be a considerable guide to the performance of the chosen experimental program. The use of composite column increases the beam stiffness and decreases its maximum deflection value. Increasing the stiffness of the column either as a composite or bare steel is recommended for avoiding column buckling and decreasing the cumulative deflection at the beam side. Use of the composite column at the proposed technique is recommended for keeping the failure at the beam side.

References

1. W. Li, L Hai Han, ''Seismic performance of CFST column to steel beam joints with RC slab: Analysis'' Journal of Constructional Steel Research, 2011.
2. J. Garzón-Roca, J.M. Adam, P.A. Calderón and I.B. Valente, ''Finite element modelling of steel-caged RC columns subjected to axial force and bending moment'' Journal of Engineering Structures, February 2012.
3. Dabaon M. A, El-Boghdadi M. H and Kharoob O. F., ''Semi-Rigid Joints in Bare Steel and Composite Frames Subjected to Loading in Space, Part II: finite element modeling'' Twelfth International colloquium on Structural and Geotechnical Engineering, December 2007.
4. A.-Elremaily, A. Azizinamini"Experimental Behavior of Steel beam to CFT Column Connections," Journal of Constructional Steel Research, June 2001.
5. A.Elremaily, A. Azizinamini"Design Provisions for Connections between Steel Beams and Concrete Filled Tube Columns," Journal of Constructional Steel Research, April 2001.
6. A.H. El-Masry, M.A. Dabaon, T.F. El-Shafiey and A.A. Khalil, ''Investigate on an innovative way to connect RC Beam and Steel Column '', International Journal of Civil, Structural, Construction and Architectural Engineering Vol:9, No:4, 2015.
7. ANSYS multiphysis v14 user's manual.
8. Eurocode 3.,''Design of Steel Structures''part 1.8 Design of joints CEN.2005.
9. T.M. Khalifa, " Static and Dynamic nonlinear analysis of frames with semi rigid joint" Ph. D thesis (2008), Tanta University, Egypt.
10. P. Desayi and S.krishnan, "Equation for the stress0strain curve of concrete," ACI jornal, Proc.61 (3), PP. 345-350 (1964).
11. J.D. Carreira and. K.H. CHU, ''Stress-Strain relationship for plain concrete in compression'' ACI Journal, 82, No. 6, PP. 797-804 (1985).
12. A.N. Sherbourne and M.Bahaari, ''Finite Element Prediction of End Plate Bolted Connection Behavior; Parametric Study'' Journal of Structural Engineer February (1997) 157-164.
13. Egyptian Standard Specifications No 262-1999, "Tensile testing of materials".

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.