Nonlinear Finite Element Analysis of Perfobond Shear Connectors

Haider M. Alsaeq
M.Sc. – Structural Eng., Dept. of Civil Eng., College of Eng., University of Babylon, Babel, Iraq
Email: haidermufeed@yahoo.com

Abstract:
This study investigates the structural behavior of the perfobond shear connectors used in composite bridges. The software ANSYS 12.1 is used to handle the nonlinear finite element analysis. The difference between experimental and theoretical results for concrete is good for ± 20%. In the present research, it was noticed that the failure of this composite system initiates in the concrete slab, rather than the steel perfobond connector, in which the stresses remain relatively small. The most important factor affecting the strength of the system was found to be the perfobond connector thickness. Increasing the thickness by 80% nearly doubles the strength. Other important factor is the compressive strength of the concrete slab. When increasing the compressive strength from 30 MPa to 70 MPa, the strength of the system increases by about 80%.
Keywords: Perfobond Connectors, Composite Structures, Finite Element, Ansys.

الخلاصة:
الهدف من هذا البحث هو دراسة التصرف الانشائي لروابط القص من نوع (Perfobond) والتي تستخدم في الجسور المركبة من الخرسانة والحديد. تم استخدام البرنامج (Ansys 12.1) لغرض اجراء عملية التحليل اللاخطي باستخدام طريقة العناصر المحددة. ان التحمل الانشائي الأقصى والذي تم الحصول عليه باستخدام البرنامج (Ansys 12.1) يظهر تطابقاً جيداً مع النتائج العملية مع فرق لايتجاوز 20%. من خلال هذا البحث تمت ملاحظة أن الفشل في هذا النظام المركب يبدأ في البلاطات الخرسانة بدلاً عن روابط القص الحديدية (Perfobond) والتي تبقى فيها الاجهادات صغيرة نسبياً. تم الاستنتاج بأن أهم عامل يؤثر على التحمل الانشائي لهذه المنظومة هو سمك روابط القص. ان زيادة هذا السمك بمقدار 80% يؤدي الى مضاعفة التحمل الانشائي تقريباً. العامل المهم الآخر هو تحمل الانضغاط لخرسانة البلاطات والذي عندما تمت زيادته من (30 MPa) الى (70 MPa) أدى الى زيادة التحمل الانشائي للمنظومة بمقدار 80%.

1. Introduction:
The construction of composite highway bridges includes the implementation of several categories of steel–concrete composite deck systems. To achieve the desired composite action, longitudinal shear force needs to be transferred between the steel and the concrete. Among the several different types of shear connectors used to provide composite action, the headed shear stud is the most common. However, the stud might cause spatial obstacles during erection, and some fatigue problems of headed studs during service life have been reported [1].
Back in1987, a German consulting engineering firm introduced the "*perfobond*" rib shear connector for composite beams to overcome the fatigue problem of the studs under live load [1], [2]. The perfobond rib shear connector consists of a steel plate with a number of uniformly spaced holes. In this case, the ribs were directly welded onto the top flange of a steel beam. If the holes in the perfobond rib are filled with concrete, concrete dowels are formed, which provide longitudinal shear resistance between the steel and the concrete. The potential advantages of the perfobond rib shear connectors are: they are easy to customize and fabricate; there are smaller obstacles than the studs during erection; and a perfobond rib could replace a number of headed studs. See Fig. 1

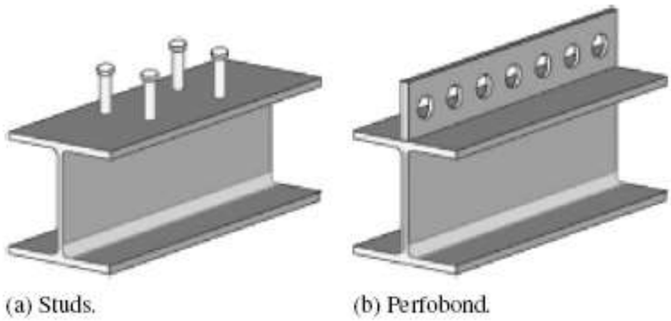


Fig. 1 (a) Traditional stud connectors, (b) Perfobond connector,[2].

In the last decades many experimental and analytical studies were performed to investigate the structural behavior of the perfobond connectors, and several attempts were made to facilitate the design process of such systems [1], [2].

2. Objectives:

In the present study; an attempt is made to understand the influence of several parameters on the structural behavior and strength of perfobond shear composite system, and to suggest the best approaches to enhance the strength of this type of systems, and this is done by investigating how altering each of the parameters under consideration will eventually increase or decrease the strength of the whole system. These parameters includes the perfobond connector dimensions, number and size of openings, as well as other parameters pertaining the reinforced concrete slab, like the compressive strength of concrete and amount of steel reinforcement.

3. The Push out Test:

In the push out test (see Fig. 2), two perfobond ribs are welded to both flanges of the steel girder, while the perfobond connectors are embedded in the concrete slab. The load then applied on the top of the steel beam, and the two slabs are supported at the bottom. The push out test is a standard evaluation of the strength of this composite system [1], [2], [3], [4].

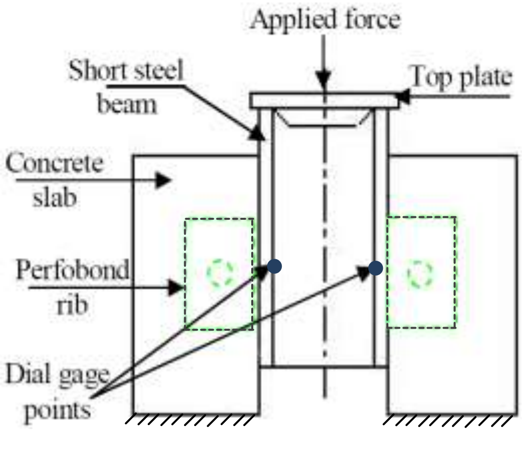


Fig. 2 Push out tests [3]

4. Material Modeling:

4.1 Concrete:

In the last decades, many stress-strain curves for concrete were suggested by several researchers [5], [6]. Shortcomings exist when manipulating the most commonly used existing stress-strain models. First, the equations cannot be easily inverted to explicitly calculate strain as a function of stress; this poses a problem when one wants to conduct rate-dependant modeling. Second, the equations cannot be easily integrated in order to determine the equivalent rectangular stress-block parameters for hand analysis and design purposes.

Although the stress-strain models proposed by various researchers have varying levels of sophistication, for the best models it is difficult to check their accuracy. There is a need for a well-developed stress-strain model that can not only be used computationally, but can also be easily adapted for hand analysis to enable rapid design checks to be performed [6].

In the present work; the stress-strain equations proposed by Kent and Park in 1971 were used as follows [5], [6]:

$$f_c = f_c^{'}\left[\frac{2\varepsilon_c}{\varepsilon_{co}} - \left(\frac{\varepsilon_c}{\varepsilon_{co}}\right)^2\right] \qquad for\ 0 \le \varepsilon_c \le \varepsilon_{co}$$

$$f_c = f_c^{'} \qquad for\ \varepsilon_c > \varepsilon_{co}$$

$$\varepsilon_{co} = 2f_c^{'}/E_c$$

Where:

f_c : Stress at any strain ε_c (MPa)

ε_{co}: Strain at the ultimate compressive strength $f_c^{'}$

E_c: The modulus of elasticity of concrete (MPa)

The modulus of elasticity and modulus of rupture (f_r) for concrete, are both calculated as illustrated in the ACI 318-08 code [7]:

$$E_c = 4700\sqrt{f_c^{'}}(\text{MPa})$$

$$f_r = 0.62\sqrt{f_c^{'}}(\text{MPa})$$

Fig. 3 below illustrates the stress-strain relationship adopted in the present research, for the concrete used in section 6, which has a compressive strength of 60.3 MPa, and ultimate strain (ε_{co}) of 0.0033. These stress-strain equations are usable for other values of compressive strength as well.

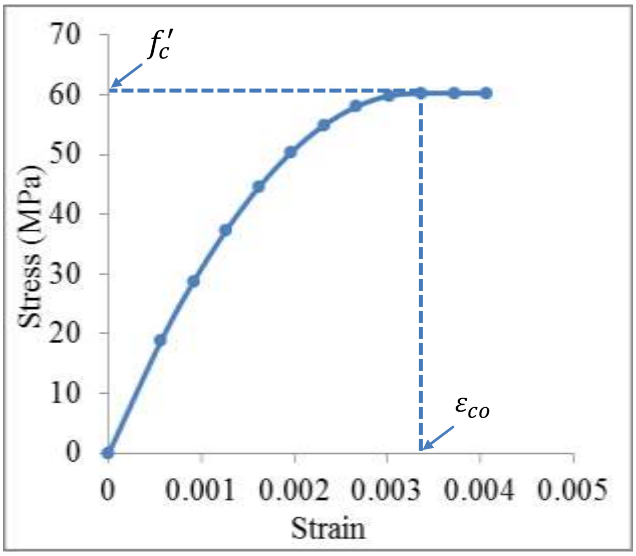


Fig. 3 Concrete stress-strain curve used in the present work

The modulus of rupture is entered separately while creating the concrete material modeling in Ansys 12.1. The Poisson ratio for concrete is usually taken as 0.2, and 0.3 for steel [8], [5].

4.2 Steel Bars and Steel Girder:

The model proposed by Park and Paulay in 1975 [8], is used in the present study for modeling both the reinforcement steel and the steel girder. In this model, the stress-strain curve is divided into 3 regions: AB, BC, and CD, as shown in Fig. 4

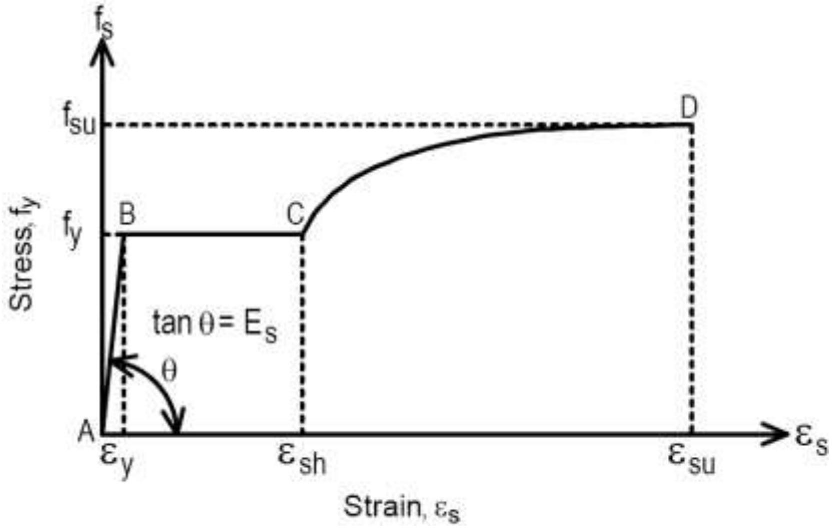


Fig. 4 Stress-strain relationship for steel proposed by Park and Paulay[8]

The equations used to constitute this model are as follows;

For region AB $0 \leq \varepsilon_s \leq \varepsilon_y$:

$f_s = \varepsilon_s E_s$, $\varepsilon_y = f_y / E_s$

For region BC $\varepsilon_y \leq \varepsilon_s \leq \varepsilon_{sh}$:

$f_s = f_y$, $\varepsilon_{sh} = 16\varepsilon_y$

For region CD $\varepsilon_{sh} \leq \varepsilon_s \leq \varepsilon_{su}$:

$$f_s = f_y\left[\frac{m(\varepsilon_s-\varepsilon_{sh})+2}{60(\varepsilon_s-\varepsilon_{sh})+2} + \frac{(\varepsilon_s-\varepsilon_{sh})(60-m)}{2(30r+1)^2}\right] \quad , \quad m = \frac{\left(\frac{f_{su}}{f_y}\right)(30r+1)^2+60r-1}{15r^2}, \quad r = \varepsilon_{su} - \varepsilon_{sh}$$

Where:

f_s: Stress at any strain ε_s (MPa)

E_s: The modulus of elasticity of steel.(MPa)

f_y: yield stress. (MPa)

ε_y: Strain at yield

ε_{sh}: Strain at the end of yielding region BC.

ε_{su}: Ultimate strain

f_{su}: Ultimate stress (MPa)

The value of E_s is assumed as 200000 MPa, and the value of ε_{su} as 0.1

5. Finite Element Modeling:

5.1 Steel Girder and Perfobond Connectors:

For the purpose of finite element modeling of the steel beam andperfobondconnectors, the Solid45 element is used. This element is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. See Fig. 5.

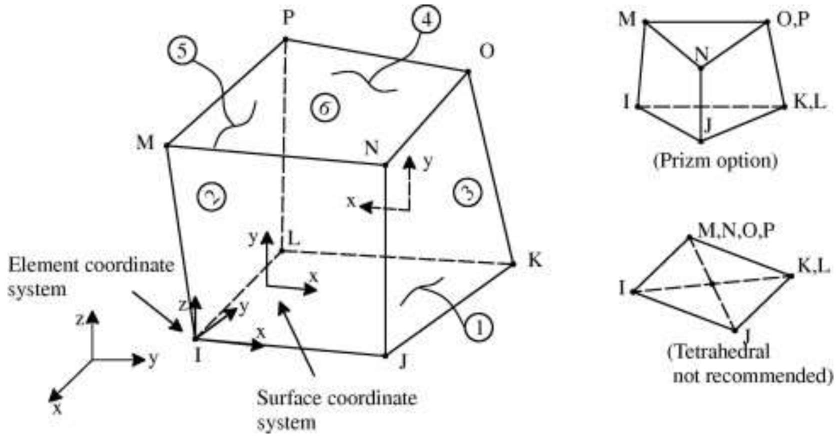


Fig. 5 Solid45 element used to model steel girder [9]

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities [9].

5.2 Steel Reinforcement:
The element Link8, as shown in Fig. 6, is used in the present study for modeling the reinforcement bars in the concrete slab. This element is a spar which may be used in a variety of engineering applications. Depending upon the application, the element may be thought of as a truss element, a cable element, a link element, a spring element, etc. The three-dimensional spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, swelling, stress stiffening, and large deflection capabilities are included [9].

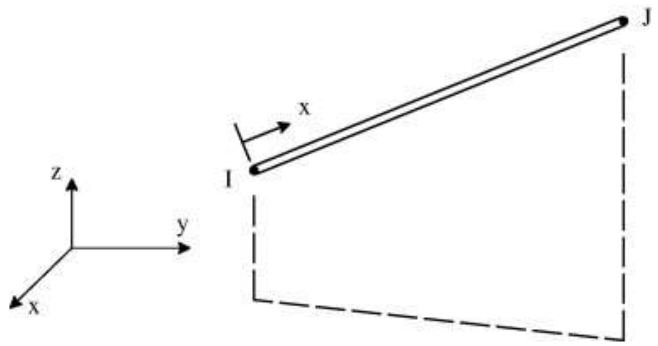


Fig. 6 Link8 element used to model steel bars [9]

5.3 Concrete:
The three dimensional solid concrete element “Solid65” is used in the present work for the modeling of the concrete. This element is used for the 3-D modeling of solids with or without reinforcing bars (rebar). The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. Other cases for which the element is also applicable would be reinforced composites (such as fiberglass), and geological materials (such as rock). The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions, as shown in Fig. 7. Up to three different rebar specifications may be defined [9]. In the present study; the discrete approach is used to model the reinforcement, in which reinforcement bars are modeled separately using the Link8 element, and then attached to concrete nodes at the conjunction points.

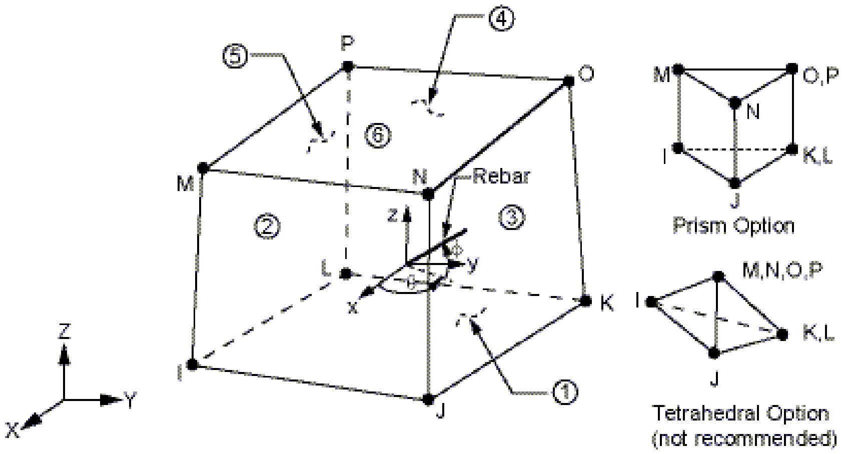


Fig. 7 Solid65 element used to model concrete [9]

The concrete element is similar to the Solid45 element (which is implemented in the present work for the modeling of the steel girder) with the addition of special cracking and crushing capabilities. The most important aspect of this element is the treatment of nonlinear material properties. The concrete is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep [9].

6. Experimental Data:

The standard push out test for specimen CP1.1 tested by Valente M., I, B.[10] is considered in the present study to confirm the validity of the Ansys 12.1 finite element model. The Perfobond connectors used in this specimen has the dimensions and configuration shown in Fig. 8. The entiredimensions are in (mm).

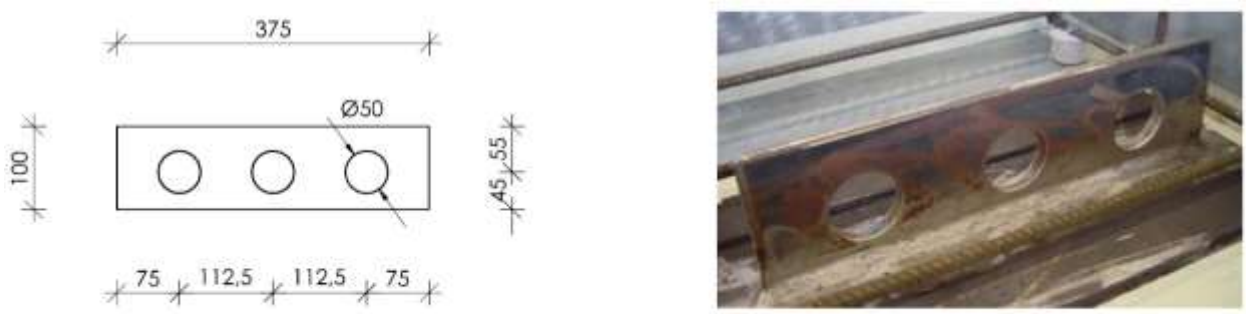
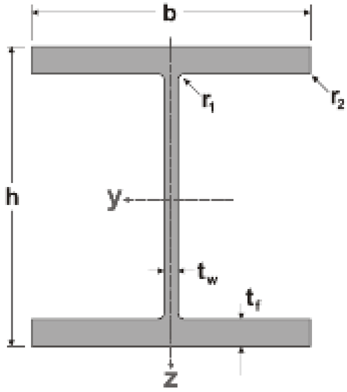


Fig. 8Perfobond rib configuration and dimensions [10]

Two ribs are welded to both flanged of HEB260 girder. The geometry of this section is illustrated in Fig. 9. All the symbols represent the dimensions of this section in (mm), except G which is the weight of unit length of the girder (kg/m).



h= 260 mm
b= 260 mm
t_w= 10 mm
t_f= 17.5 mm
r_1= 24 mm
r_2=1 mm
G= 93 kg/m

Fig. 9 Geometry of the HEB260 section

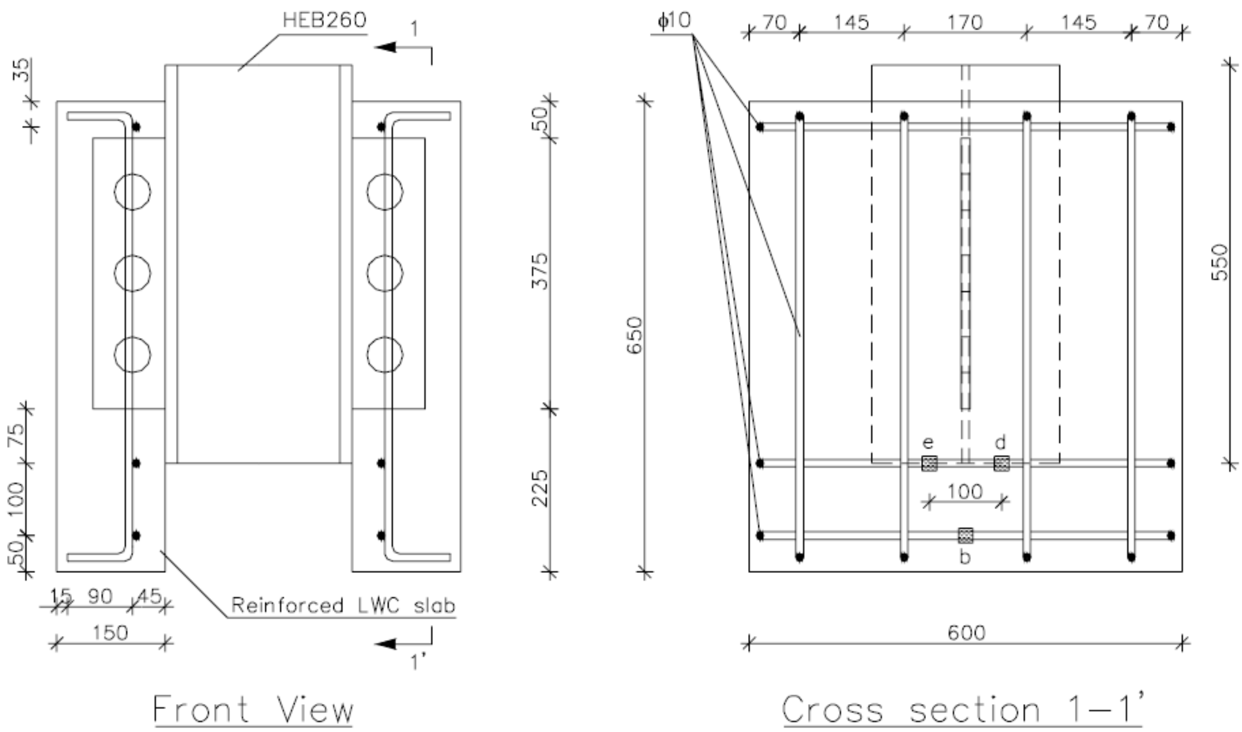


Fig. 10 The geometry of the CP1.1 specimen [10]

Fig. 11 Top view of this specimen CP1.1 before pouring the concrete [10]Supports

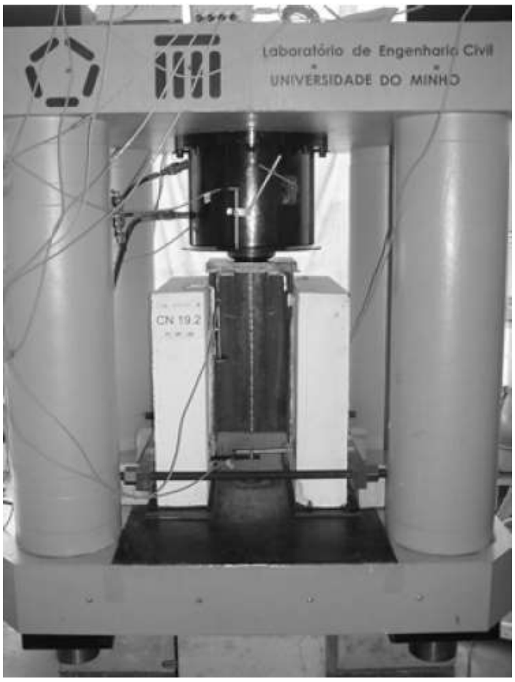


Fig. 12Testingmechine[10]

Fig. 10 shows the geometry of the CP1.1 specimen. Letters b, d and e represent the dial gauges that measure the relative movement (slip) between the concrete slab and steel beam. During the preparation of the specimen; special consideration was taken to prevent attaching the concrete surface to the steel beam flanges, to transfer the entire applied load to the perfobond shear connector. Fig. 11 shows a top view of this specimen before pouring the concrete, while Fig. 12 shows specimen during testing.

The material properties used in this test are as follows [10]:

Concrete: compressive strength f_c' = 60.3 MPa,

Steel bars: yield stress f_y = 576 MPa, ultimate stress f_{su} = 675 MPa.

Perfobond rib and steel girder: f_y = 415 MPa, ultimate stress f_{su} = 550 MPa.

It is possible to produce lightweight concrete that is comparable to normal weight concrete in terms of compressive strength.To produce high strength lightweight concrete, it is necessary to use lightweight aggregates with higher compressive strength, and high strength cement [10].

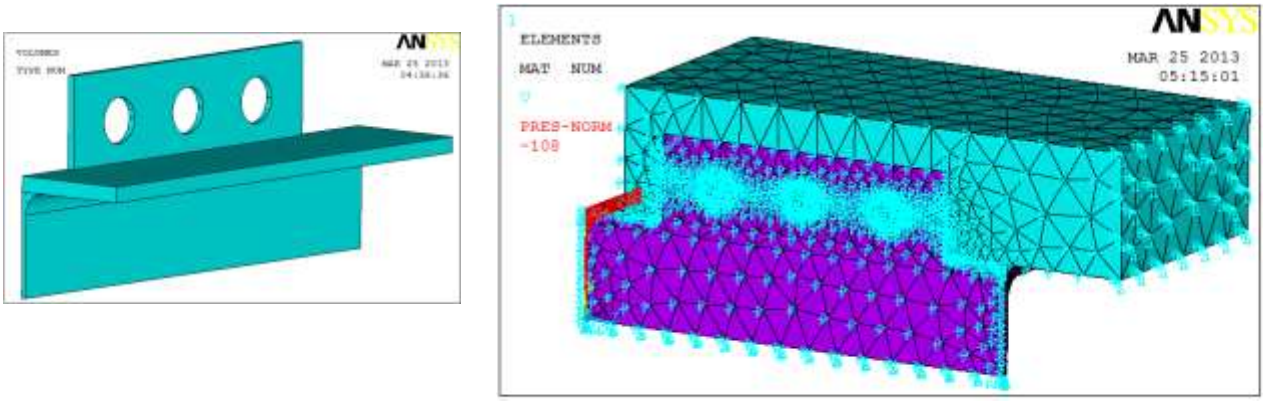


Fig. 13 One quarter of the composite system model in Ansys12.1

Only one quarter of the composite system is considered in the ANSYS 12.1 model due to symmetry, as illustrated in Fig. 13. It is clear that the 6 node option (prism option) is used in the finite element mesh for both steel and concrete. It also clear that the mesh is fine near points of stress concentration.The cracked concrete slab and the deformed perfobond connector after completing the test are shown in Fig. 14, while Fig. 15 shows the experimental vs. finite element results obtained by the present ANSYS 12.1 model. The curves show good agreement, with a difference percentage of 20%. In this graph; the applied load in the perfobond rib is plotted against the relative slip between surfaces of concrete slab and steel girder.Fig. 16 shows the strain contour in concrete at failure,the darker the color, the higher the absolute value of strain. The total number of elements used to obtain these results is 4810, and it was found out that increasing the number of elements above that number will have insignificant effect on the results, as shown in Fig. 17. The failure criterion in ANSYS can describe when brittle failure (fracture) or ductile failure (yielding) occurs. Concrete element SOLID65 can include cracking/crushing behavior. The Mises yield criterion is the default for most of the plasticity models in ANSYS (and is used in the present study). Use of the Misespotential allows specification for anisotropic yield criteria [9].

Fig. 14 Cracked slab and deformed perfobond rib at failure [10]

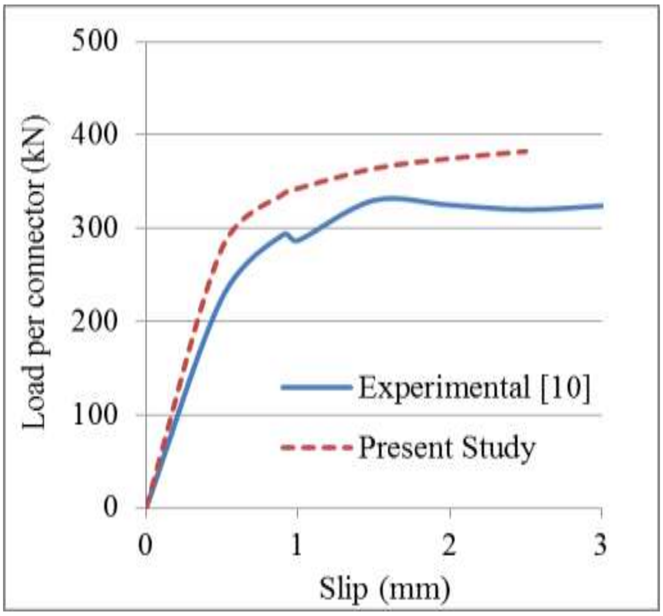


Fig. 15 experimental vs. current model

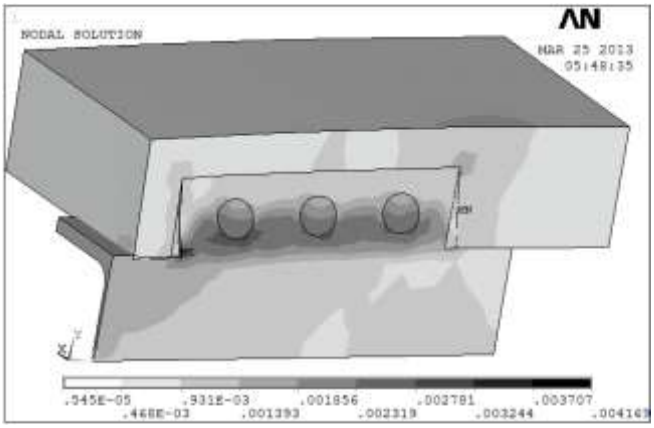


Fig. 16Strain contour at failure for quarter of the composite system

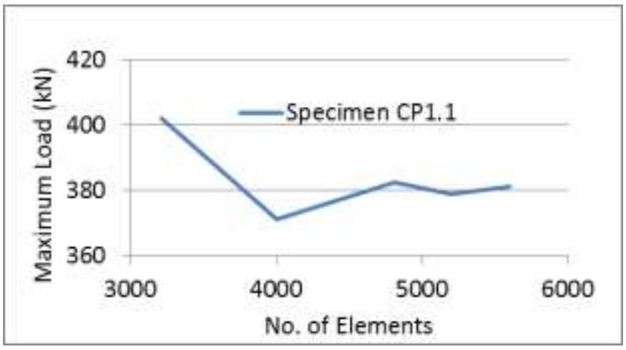


Fig. 17Maximum load per connector vs. No. of elements for specimen Cp1.1

7. Parametric Study:

The effects of several parameters on the strength of the perfobond connector-concrete slab composite system are studied using the Ansys 12.1 software. Three cases are considered each time, which Perfobond rib with three openings, two openings and single opening. For example; Fig. 18 shows the finite element mesh and strain contour for the same specimen (CP1.1) but when reducing the number of openings in the perfobond rib to 2 openings and keeping all the other parameters fixed, and Fig. 19 shows similar visualization for the single opening case.

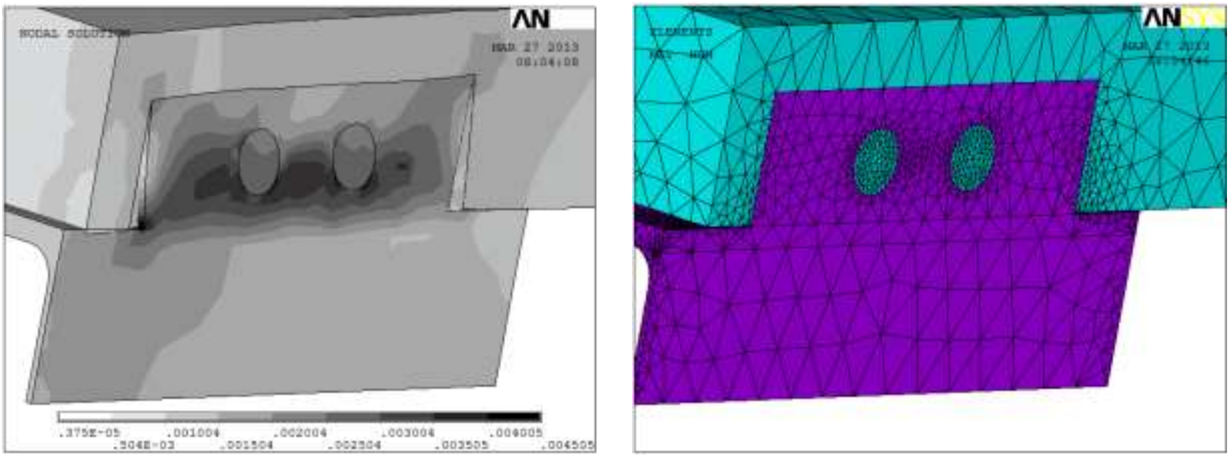


Fig. 18 Finite element mesh and strain contour for the two opening case

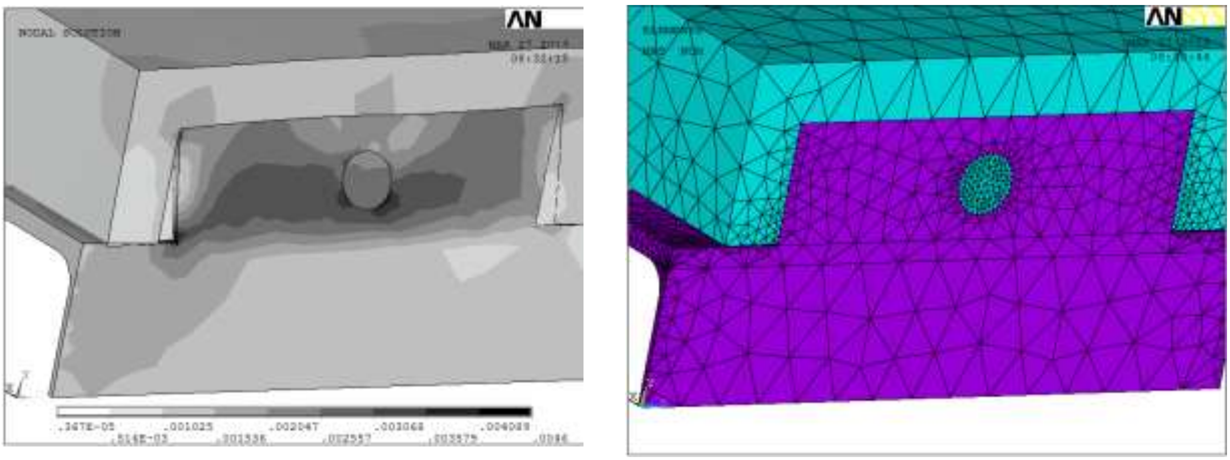


Fig. 19 Finite element mesh and strain contour for the single opening case

By investigating the strain contours in figures 16, 18 and 19; there are two apparent points: the first is that the strains (and therefore stresses) are intense at the conjunction between the concrete and the lower edges of the openings. The second is that the strains in the three opening case are more uniformly distributed, than strains in the two or single opening case.

Fig. 20 illustrates the effect of changing the perfobond rib thickness (t) on the structural strength of the composite system. In this case, all the dimensions and material properties of the specimen CP1.1 considered in the present research are kept fixed, except the connector thickness. Fig 21 shows the effect of changing the perfobond connector height (h_r) on the strength of specimen Cp1.1.

It is clear from Fig. 19 that increasing the connector thickness has much more influence on the overall strength of the system than increasing the number of openings. Using three openings instead of one increased the strength by about 25%, while increasing the connector thickness by 80% (from 10 mm to 18 mm) resulted in increasing the strength by about 100%.

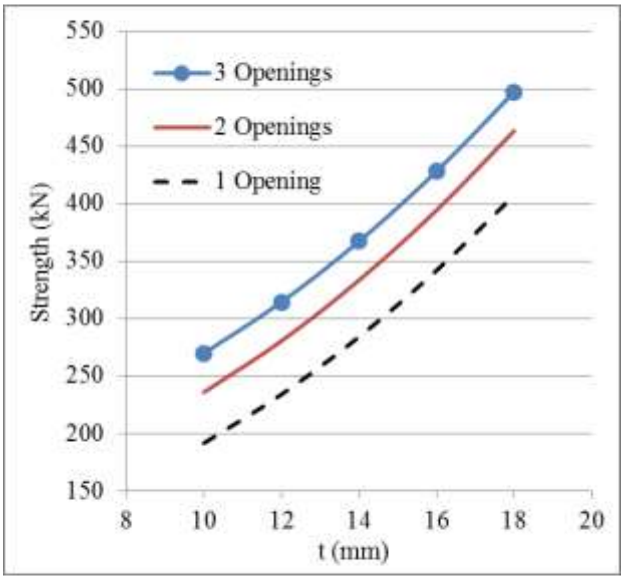


Fig. 20 Strength vs. rib thickness (t)

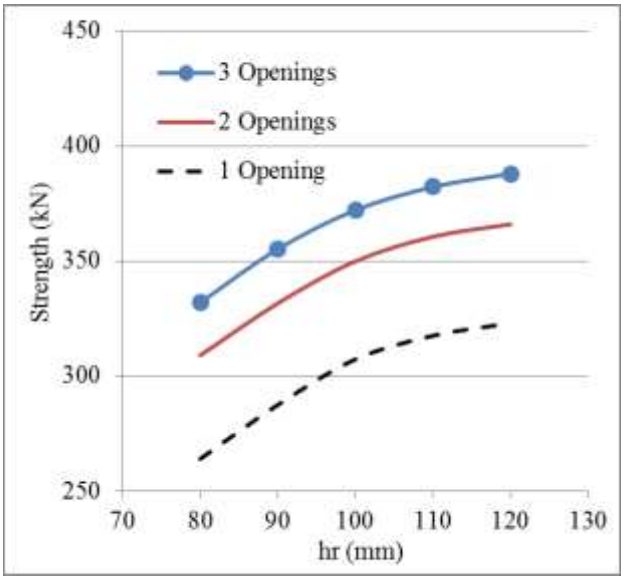


Fig. 21 Strength vs. rib height (h_r)

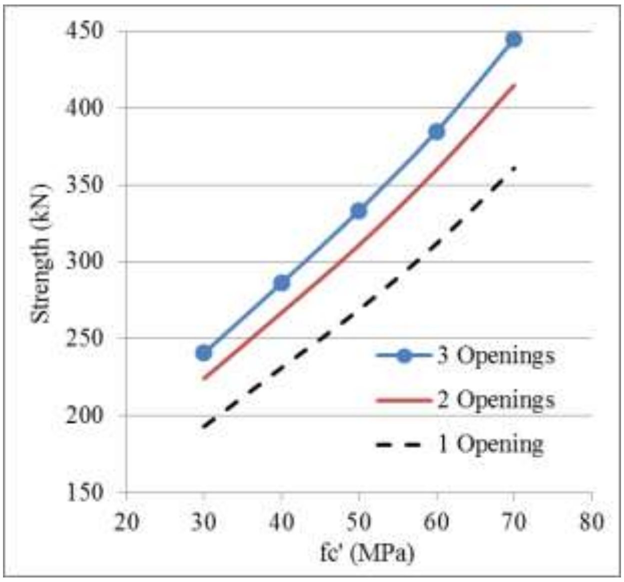


Fig. 22 Strength vs. concrete slab compressive strength f_c'

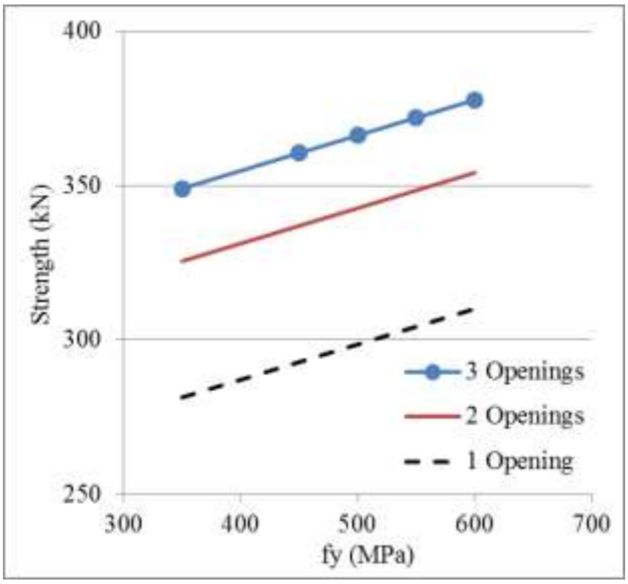


Fig. 23 Strength vs. transverse steel bars yield stress f_y

In Fig. 22 the effect of the compressive strength of the concrete slab is considered, while Fig. 23 illustrates the sensitivity of the composite system to changes in the yield stress of the reinforcement steel bars embedded in the concrete slabs of the specimen CP1.1, while keeping the remaining parameters unchanged.Fig. 24 shows the effect of changing the area of transverse steel bars on the structural strength of the composite system. In this case, all the dimensions and material properties of the specimen CP1.1 considered in the present research are kept fixed, except the number of Ø 10 bars placed in the transverse direction (originally three bars with area A_{tr} = 237 mm^2). When the number of these bars is increased, and while creating the model in Ansys 12.1; the additional bars are placed to pass through the openings. Fig 25 shows the effect of changing the perfobond connector opening diameter (d) on the strength of specimen CP1.1.

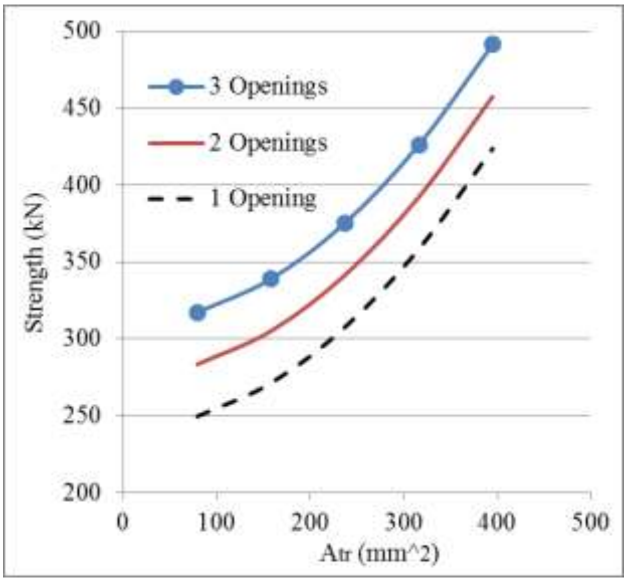


Fig. 24 Strength vs. area of transverse steel bars (A_{tr})

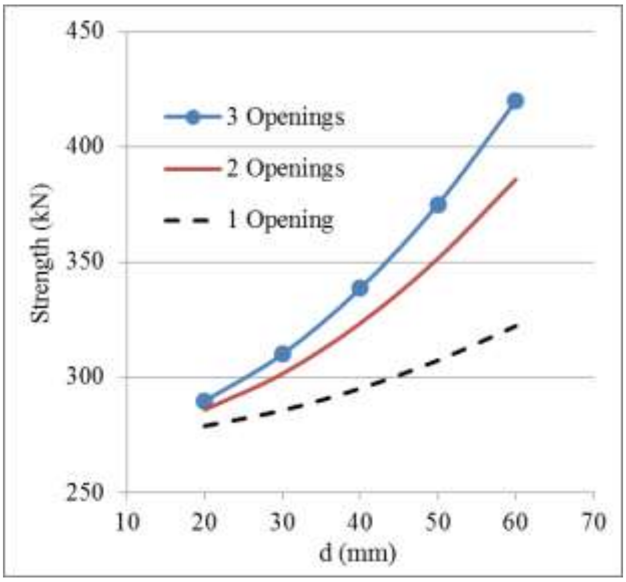


Fig. 25 Strength vs. diameter of the opening (d)

By studying figures 20 through 25, it is noticeable that the most influential parameter that affects the strength of this composite system is the connector thickness. The other two important parameters are the compressive strength of the concrete slab and the amount of the transverse reinforcement.

8. Conclusion:

It was noticed in the present study that the failure of the perfobond connector-concrete slab composite system, occurs in the concrete slab, while the stress and strain in the steel perfobond rib remain far below the ultimate values. Hence, the material properties of the steel used in manufacturing the perfobond rib will have minor effects on the strength of the system. By increasing the rib thickness from 10 mm to 18 mm the strength almost doubled, and increasing the compressive strength of the reinforced concrete slab from 30 MPa to 70 MPa increased the strength of the system by about 80%. The number of transvers steel bars is also an important factor in determining the structural behavior of the system. By increasing the number of Ø 10 bars from one bar to five bars; the strength increased by about 53%. The yield stress of these bars has less effect on the strength of the system. It is also noticed that using a perfobond rib with a single large opening will produce nearly the same strength of a rib with three small openings, if the total area of these three opening is made equivalent to the single one. Finally, the Ansys 12.1 model implemented in the present research was proved to be successful in handling the nonlinear analysis of this composite system, with a difference of about 18% from the experimental data.

9. References:

[1] Kim H. Y., and Jeong Y. J., 2005, "Experimental Investigation on Behavior of Steel–Concrete Composite Bridge Decks with Perfobond Ribs",Iraqi Virtual Science Library (www.ivsl.org), Journal of Constructional Steel Research, Vol. 62, pp. 463–471.

[2] Vianna J. da. C., Andrade S.A.L., Vellasco P.C.G. da S., and Costa-Neves L.F., 2013, "Experimental Study Of Perfobond Shear Connectors In Composite Construction", Iraqi Virtual Science Library (www.ivsl.org), Journal of Constructional Steel Research, Vol. 81, pp. 62–75.

[3] Al-Darzi S. Y. K., Chen A. R. and Liu Y. Q., 2007, "Finite Element Simulation and Parametric Studies of Perfobond Rib Connector",Iraqi Virtual Science Library (www.ivsl.org), American Journal of Applied Sciences, Vol. 4, pp. 122-127.

[4] Shariati M., Sulong N. H. R., Arabnejad K. H. and Mahoutian M., 2011, "Shear Resistance of Channel Shear Connectors in Plain Reinforced and Lightweight Concrete", Scientific Research and Essays, Vol. 6(4), pp. 977-983.

[5] Kachlakev D., Miller T., Yim S., Chansawat K., and Potisuk T. ,2001 , "Finite Element Modeling of Reinforced Concrete Structures Strengthened with FRP Laminates", Final Report SPR 316, Oregon Department of Transportation & Federal Highway Administration, USA.

[6] Reddiar M. K. M., (2009) "Stress-Strain Model of Unconfined and Confined Concrete and Stress-Block Parameters", MSc thesis, Department of Civil Engineering, Texas A&M University, USA, available at www.xa.yimg.com

[7] ACI 318-08, American Concrete Institute, "Building Code Requirements for Reinforced Concrete," American Concrete Institute, Farmington Hills, Michigan, 2008.

[8] Tavio, T. and Tata, A., 2009, "Predicting Nonlinear Behavior and Stress-Strain Relationship of Rectangular Confined Reinforced Concrete Columns with ANSYS", Civil Engineering Dimension Journal, Vol. 11, No. 1, p.p. 23-31.

[9] ANSYS, ANSYS User's Manual Revision 12.1, ANSYS, Inc., Canonsburg, Pennsylvania, 2012.

[10] Valente M., I, B., (2007) "Experimental Studies on Shear Connection Systems in Steel and Lightweight Concrete Composite Bridges", PhD. thesis, Civil Engineering Department - School of Engineering, University of Minho, Portugal., available at www.repositorium.sdum.uminho.pt