Nahrain University, College of Engineering Journal (NUCEJ) Vol.15 No.1, 2012 pp.1-11

Finite Element Analysis of Composite Steel- Concrete Beams Subjected to Fire

Dr. Hani M. Fahmi,
Professor in Civil Engineering Department,
Al-Nahrain University.
e-mail: fahmihanim@yahoo.com

Tamara Tofeq
Civil Engineering Department
Al-Nahrain University.

Abstract

In the present study, a nonlinear three-dimensional finite element analysis has been used to analyze composite steel-concrete beams subjected to BS476 fire test, utilizing (ANSYS V.10) code. The analysis includes thermal and structural parts. In the thermal analysis, eight-node isoparametric brick elements (thermal solid) have been used to model the concrete slab and the steel beam. In the structural analysis, eight-node isoparametric brick elements (reinforced concrete solid) have been used to model the reinforced concrete slab. The reinforcement is assumed to be smeared throughout the concrete elements. Eight-node isoparametric brick elements (structural solid) have been used to model the steel beam, while the shear connectors are modeled by truss (spar) elements and nonlinear spring elements to resist slip and uplift separation between the steel beam and concrete slab. The nonlinear behavior of the surface between the steel and concrete is modeled using interface elements. Material nonlinearity due to cracking and crushing of concrete, yielding of steel beam and reinforcing bars and temperature dependent material properties are considered in the analysis. The nonlinear equations of equilibrium are solved by an incremental-iterative technique. The Newton-Raphson method is used as a nonlinear solution algorithm with a temperature criterion to monitor convergence in the thermal analysis and a force criterion in the structural analysis. The numerical integration has been conducted using Gauss quadrature rule. Parametric study is carried out to study the influence of several important parameters including concrete compressive strength, steel section yield stress, degree of shear connection, concrete slab thickness and the effect of applied load on the overall behavior of composite beams subjected to fire.

Keywords: composite structures, fire resistance, ANSYS modeling, structural modeling, thermal modeling.

1. Introduction

The aim of the use of composite construction is to achieve a higher level of performance than would have been when the two materials functioned separately [1]. By about 1950 the development of shear connectors to connect the slab to the beam had made the construction practicable [2]. Composite beams may be the most common form of composite elements in steel frame building construction and have been the major form for steel bridges [1]. As a basis, composite beams are defined as, “elements resisting only flexure and shear that comprise two components connected together by a series of discrete connectors”. The generic form of a composite beam comprises the combination of a solid concrete slab attached to a steel section (normally I shape), Fig.1.

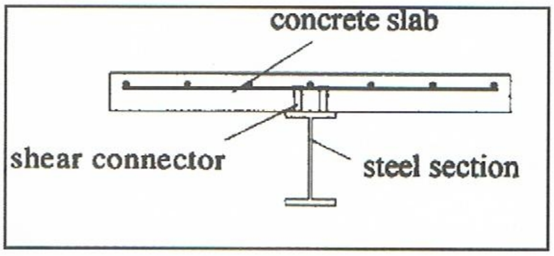
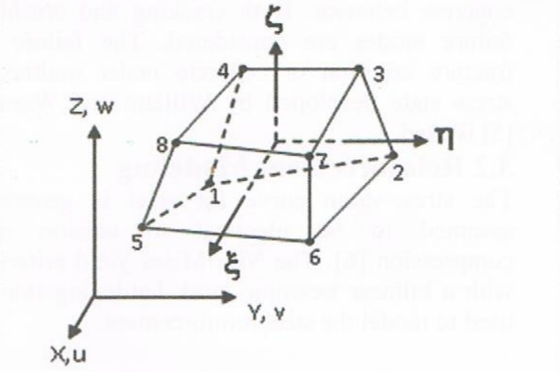


Figure (1) Composite beam components[3].

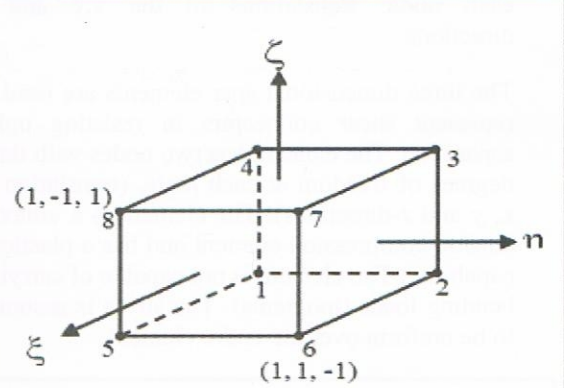
2. Finite Element Formulation

2.1 Thermal Analysis

In the present study, three dimensional isoparametric solid elements are used to model the concrete and steel sections, Fig.2. The element has 3-D thermal conduction capability and has eight corner nodes with a single degree of freedom (temperature) at each node. The element is applicable to a 3-D steady-state or transient thermal analysis. The integration rules used are 8 (2×2×2) points and 4 (2×2) points, Fig.3. Three methods are available to do a transient analysis: full, mode superposition and reduced [4]. The full method is adopted in the present study. For nonlinear thermal analysis, the ANSYS program allows a choice from three solution options: the full option, the quasi option and the linear option [4]. In this study, the full option is used which corresponds to the default full Newton-Raphson algorithm with a temperature criterion to monitor convergence in the analysis.



a) In global coordinates



b) In local coordinates

Figure (2) Eight node isoparametric solid element

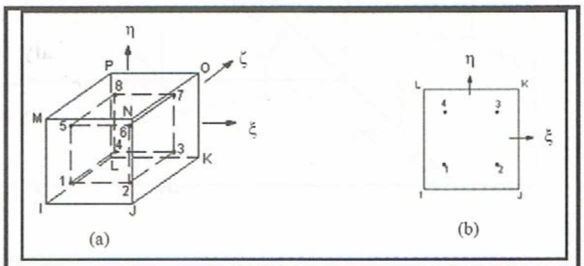


Figure (3) Distribution of integration points a) 2×2×2 b) 2×2, [4]

2.2 Structural Analysis

In the current study, three dimensional isoparametric solid elements are used to model the reinforced concrete. The element has eight nodes with three degrees of freedom at each node: translations in the x, y and z directions. The reinforcement is assumed to be smeared throughout the element.

Three dimensional isoparametric solid elements are used to model the steel section. The element has a plasticity capability and defined by eight nodes having three degrees of freedom at each node: translations in the x, y and z directions.

A three dimensional point to point contact element is used to model the nonlinear behavior of the surface between the concrete slab and the steel beam, which may maintain or break physical contact and may slide relative to each other. Also, the element is capable of supporting only compression in the direction normal to the surfaces and shear (Coulomb friction) in the tangential direction. The element is defined by two nodes and has three degrees of freedom at each node: translations in the x,y and z directions.

The three dimensional spar elements are used to represent shear connectors in resisting uplift separation. The element has two nodes with three degrees of freedom at each node, (translation in x, y and z-directions). The element is a uniaxial tension-compression element and has a plasticity capability. The element is not capable of carrying bending loads (moments). The stress is assumed to be uniform over the entire element.

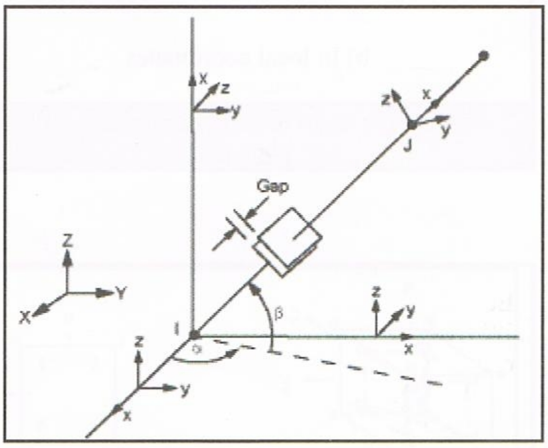


Figure (4) 3-D node to node contact element [4]

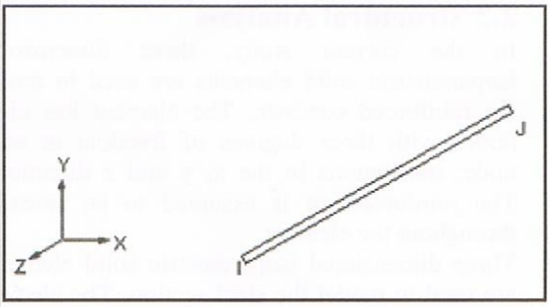


Figure (5) Three-dimensional link element [4]

Nonlinear unidirectional spring element with nonlinear generalized force-deflection capability is used to resist slip. The element has two nodes and a longitudinal capability (uniaxial tension-compression element) with up to three degrees of freedom at each node: translations in x, y and z directions. No bending or torsion is considered.

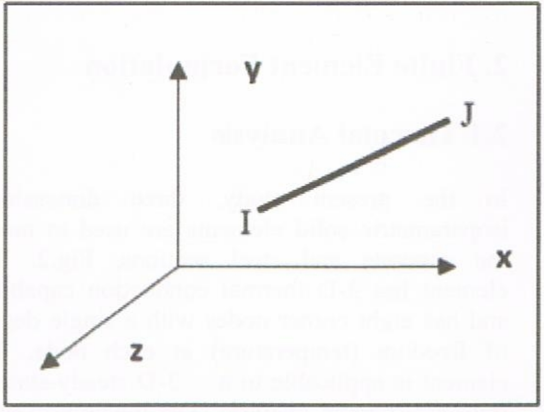


Figure (6) Nonlinear spring element [4]

The 8 (2×2×2) points Gauss quadrature integration is used in the structural analysis. The modified Newton-Raphson method is use as a nonlinear solution algorithm with a force criterion to monitor convergence in the analysis.

3. MODELING OF MATERIALS

3.1 Concrete Material Modeling

In the present work, a linear elastic brittle fracture model is used to represent the concrete behavior. Both cracking and crushing failure modes are considered. The failure or fracture criterion of concrete under multiaxial stress state developed by Willlam and Warnke [5] is used.

3.2 Reinforcement Modeling

The stress-strain curve for steel is generally assumed to be identical in tension and compression [6]. The Von Mises yield criterion with a bilinear isotropic work hardening rule is used to model the steel reinforcement.

3.3 Steel Beam Modeling

The steel beam is modeled using Von Mises yield criterion with an elastic perfectly plastic behavior.

3.4 Interface Modeling

Two combined interface models are used. The first interface is capable of supporting only compression, in the direction normal to the interface surface, and shear (Coulomb friction) in the tangential direction. The second is used to represent the normal and tangential (dowel) stiffness of the crossing connectors.

4. Mechanical Properties At Elevated Temperatures

The structural effects of a fire on the behavior of a composite structure are caused by:

a) Changes in the mechanical properties of steel and concrete. Both steel and concrete materials become weaker and more flexible at high temperatures.

b) Temperature induced strains.

In this study, the material properties used mainly follow the guidance given in Eurocode 3 Part 1.2 [7] and Eurocode 4 Part 1.2 [8].

5. Verification Of Results

Analysis of simply supported composite beams tested by Smith and Thomson [9] and [10] in accordance with BS 476: Part 8: 1972 fire test [11] is carried out.

5.1 Thermal Analysis

The thermal analysis is carried out using the finite element method utilizing ANSYS 10.0 software to compute the temperature distribution history in the beams subjected to the standard ISO temperature-time curve given by the following equation [11]:

$$T - T_\circ = 345 \log_{10}(8t + 1) \tag{1}$$

where

t = time of test in minutes

T = furnace temperature in ° C at time t

$T_\circ$ = initial furnace temperature (reference temperature) in °C

The beams are exposed to heat on their underside with reference temperatures of 11 and 15 °C for beams B1 and B2, respectively, with the temperature increasing according to Eq. (1) for 6 hours.

By taking advantage of symmetry of geometry and loading, one half of the beam has been modeled for the thermal analysis, considering insulating the surface at the line of symmetry. Half the beam is modeled using 1216 elements, as shown in Fig.7. The concrete slab is modeled using 988 eight node brick elements; whereas the steel beam is modeled using 228 eight node brick elements.

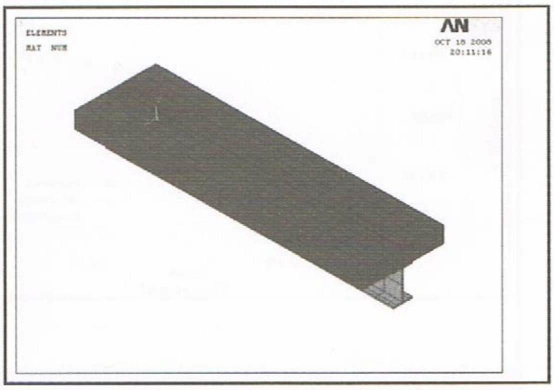


Figure (7) Finite element mesh used in thermal and structural analyses of one half of the composite beams

The thermal properties of the materials at the various temperatures used in the analysis of the composite beams B1 and B2 are as given in in Eurocode 3 Part 1.2 [7] and Eurocode 4 Part 1.2 [8]. Eurocode 1 Part 1.2 [12] recommends constant convective heat transfer coefficient of 25 W/m^2.K.

The beams B1 and B2 were heated under load for 40 and 23 minutes, respectively. The computed relations between temperature and time at different depths in the beams as well as the experimental results are shown in the following figures, Fig.8, 9, 10, 11, 12, 13, 14, 15, 16 and 17:

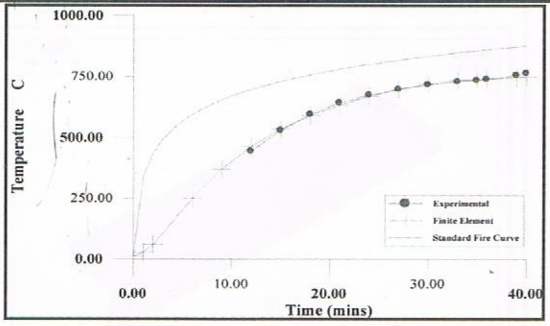


Figure (8) Finite element and experimental temperature distribution at the lower flange of composite beam B1

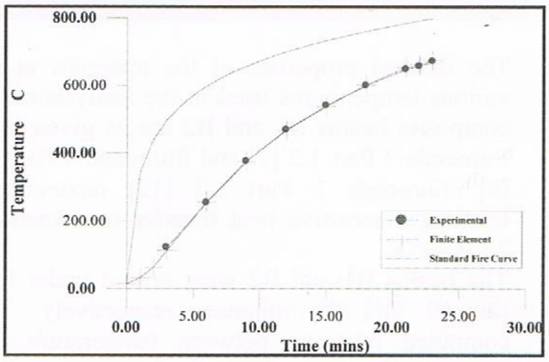


Figure (9) Finite element and experimental temperature distribution at the mid-height of the web of composite beam B1

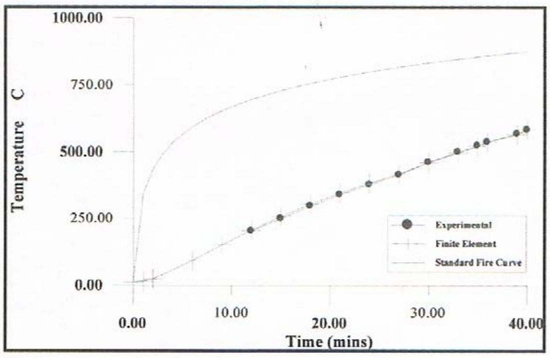


Figure (10) Finite element and experimental temperature distribution at the upper flange of composite beam B1

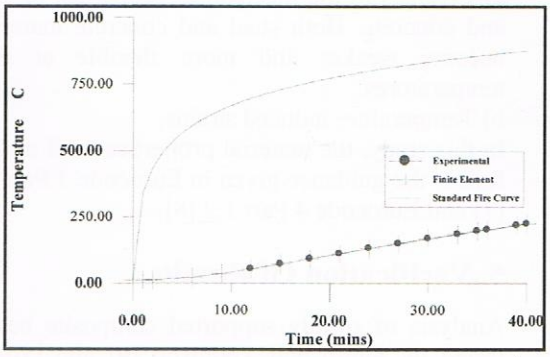


Figure (11) Finite element and experimental temperature distribution at 100mm depth of concrete slab of composite beam B1

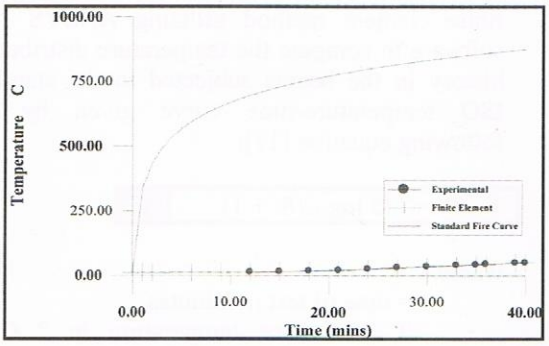


Figure (12) Finite element and experimental temperature distribution at 30mm depth of concrete slab of composite beam B1

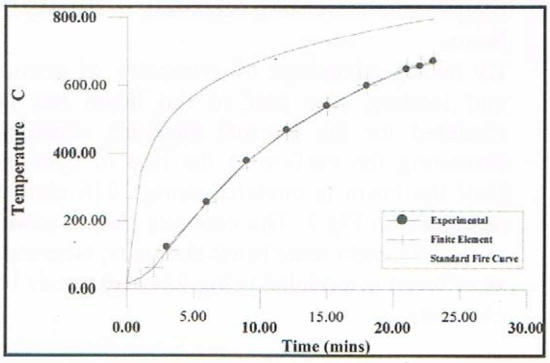


Figure (13) Finite element and experimental temperature distribution at the lower flange of composite beam B2

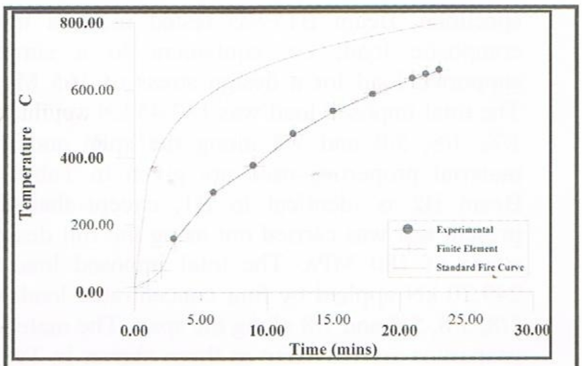


Figure (14) Finite element and experimental temperature distribution at the mid-height of web of composite beam B2

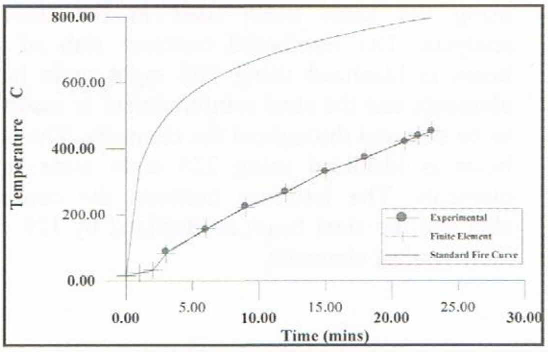


Figure (15) Finite element and experimental temperature distribution at the upper flange of composite beam B2

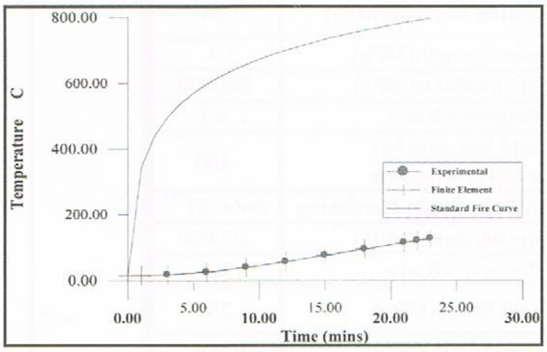


Figure (16) Finite element and experimental temperature distribution at 100 mm depth of concrete slab of composite beam B2

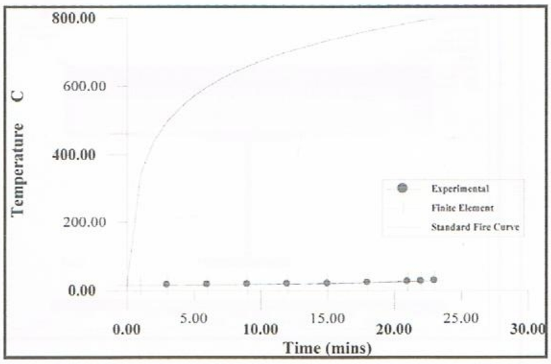


Figure (17) Finite element and experimental temperature distribution at 30 mm depth of concrete slab of composite beam B2

From these figures, it is clear that the temperature decreases towards the top surface of the concrete slab; also it is seen that the temperature in the steel beam is higher than that in the concrete slab.
These figures show that the predicted temperatures are in very good agreement with the experimental values which confirm the validity and accuracy of the thermal analysis model and the program used.

Contour plots of the temperature distribution through B1 and B2 after 6 hours of heating are shown in the following figures:

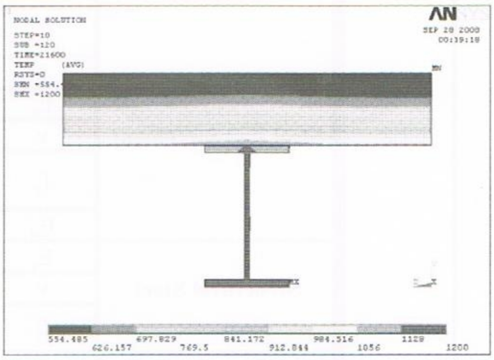


Figure (18) Contour plot of temperature distribution in beam B1 after 6 hours of heating

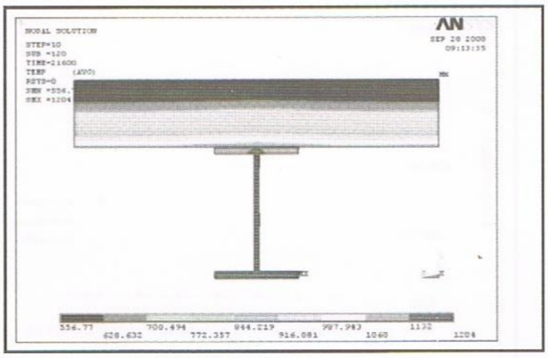


Figure (19) Contour plot of temperature distribution in beam B2 after 6 hours of heating

5.2 Structural Analysis

Beams B1 and B2 were analyzed by the finite element to compute mid span deflections of the loaded beams during standard fire exposure. The beams were subjected to imposed load applied by four concentrated loads and then to a standard time-temperature curve, Eq.1, with a reference temperature of 11 and 15 °C for beams B1 and B2, respectively. BS476: 1972[11] permits the test load to produce stresses lower than the maximum permissible value in the beam specimen. Beam B1 was tested using a non-composite load, i.e. equivalent to a simply supported load for a design stress of 165 MPa. The total imposed load was 132.45 kN applied at 1/8, 3/8, 5/8 and 7/8 along the span and the material properties used are given in Table 1. Beam B2 is identical to B1, except that the present test was carried out using the full design stress of 180 MPa. The total imposed load is 249.20 kN applied by four concentrated loads at 1/8, 3/8, 5/8 and 7/8 along the span. The material properties are the same as those shown in Table 1, with the difference is that the yield stress of the steel section is 299 MPa for the web and 273 MPa for the flanges. Fig.20 shows the boundary conditions used for the beam analysis. Due to symmetry of geometry and loading, half of the beam is analyzed. The analysis is carried out using the same mesh used in the thermal analysis. The reinforced concrete slab of the beam is idealized using 988 eight node brick elements and the steel reinforcement is assumed to be smeared throughout the elements. The steel beam is idealized using 228 eight node brick elements. The interface between the concrete slab and the steel beam is idealized by 120 two node contact elements.

Table 1 Material Properties used in analysis of beam B1 and B2 [9], [10]

	Symbol	Definition	Value
Concrete	E_c	Pa) Elastic Modulus(	23025*
	ν	Poisson's Ratio	0.2
	f'_c	Compressive Strength(MPa)	24
	f_t	Tensile Strength(MPa)	2.4**
Steel Reinforcement	E_s	Elastic Modulus(MPa)	200000
	ν	Poisson's Ratio	0.3
	f_y	Yield Stress(MPa) for top and bottom	460
	E_{st}	Modulus(MPa) Strain Hardening	6666.67***
Structural Steel	E_s	Elastic Modulus(MPa)	200000
	ν	Poisson's Ratio	0.3
	f_y	Yield Stress(MPa)	281
Stud Shear Connector	E_s	Elastic Modulus(MPa)	200000
	ν	Poisson's Ratio	0.3
	f_y	Yield Stress(MPa)	460
	E_{st}	us(MPa) Strain Hardening Modu	6666.67***

*$E_c = 4700\,(f'_c)^{0.5}$
**$f_t = 0.1\, f'_c$
*** $E_{st} = E_s/30$

Shear connectors are idealized by 16 link elements to resist uplift separation, they are attached to the nodes at the top surface of the steel beam and the bottom surface of the concrete slab. The effect of dowels passing through the interface between the top flange of the steel beam and the concrete slab is modeled by 16 nonlinear spring elements to resist slip, Thus the total number of elements used for half of the beam is 1368 elements. The fire resistance is defined as the time at which the maximum deflection is in excess of L/30 [11].

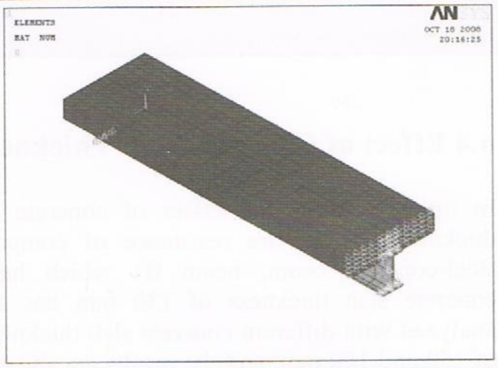


Figure (20) Boundary conditions used in the structural analysis of one half of composite beams.

The computed results of the mid deflection with time for the composite beams B1 and B2 loaded by different loads along with the experimental results are presented in Figs.21 and 22, respectively.

Failure of beams B1 and B2 is defined by the L/30 deflection criterion. The results show that the fire resistance decreases with the increase in applied load of beams B1 and B2 when subjected to standard fire, due to the decrease in the strengths of the steel and concrete.

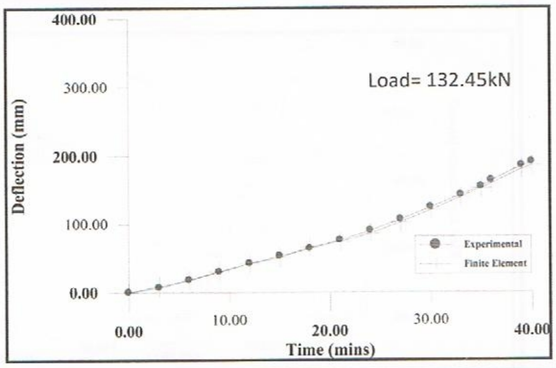


Figure (21) Finite element and experimental results for mid span deflection with time for beam B1

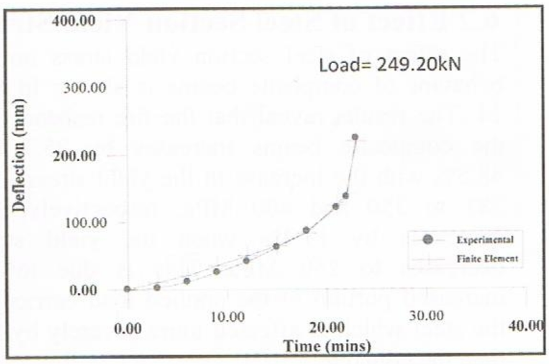


Figure (22) Finite element and experimental results for mid span deflection with time for beam B2

It is seen from the results that, there is a good agreement between the computed and experimental results, which confirms the validity and accuracy of the structural analysis model and the program used, and the variation between the computed and experimental values in Fig.22 during the first 18 minutes is expected, because the experimental deflection values of B2 are much lower than those for beam B1 during the same period, even thought (as mentioned previously), the latter is applied by a lower load, thus this variation may be due to variations in the loading conditions applied in the laboratory.

6. Parametric Study

A parametric study was performed to investigate the influence of several important parameters on the behavior of composite steel-concrete beams.

In this section, the simply supported steel-concrete composite beam B1 has been selected to carry out the parametric study. The beam was first analyzed to verify the validity of the analytical results with respect to the experimental data. The parameters considered are concrete compressive strength, steel section yield stress, partial connection, concrete slab thickness and applied load. In each case, one parameter has been changed, while the other parameters are kept constant.

6.1 Effect of Concrete Compressive Strength

The effect of the concrete compressive strength is studied and the results are shown in Fig. 23. When the concrete compressive strength is increased from 24 MPa to 30,40 and 50 MPa, the fire resistance is found to increase by 11.4, 17.1 and 22.8%, respectively.

6.2 Effect of Steel Section Yield Stress

The effect of steel section yield stress on the behavior of composite beams is shown in Fig. 24. The results reveal that the fire resistance of the composite beams increases by 25.7 and 48.5% with the increase in the yield stress from 281 to 350 and 400 MPa, respectively, and decreases by 11.4% when the yield stress decreases to 250 MPa. This is due to the increased portion of the applied load carried by the steel which is affected more severely by fire than the concrete.

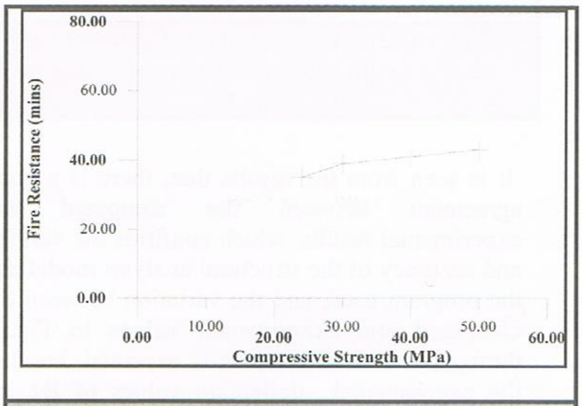


Figure (23) Effect of concrete compressive strength on the fire resistance of composite beam B1

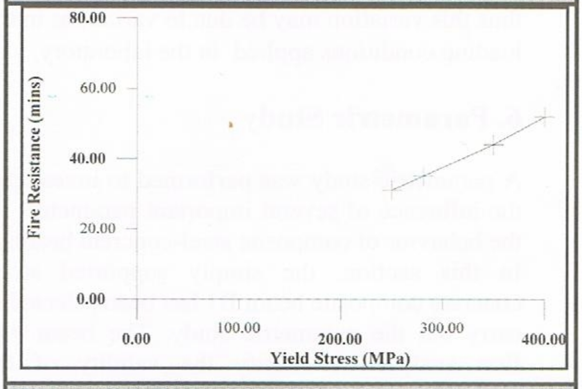


Figure (24) Effect of steel section yield stress on the fire resistance of composite beam B1

6.3 **Effect of Partial Connection**

The effect of partial connection is investigated by reducing the number of connectors. As expected, in comparison with the degree of connection used in the analysis, the effect of 62.5% and 50% connection tend to reduce the fire resistance of the beam. The reduction is 11.4% and 20%, respectively Fig. 25

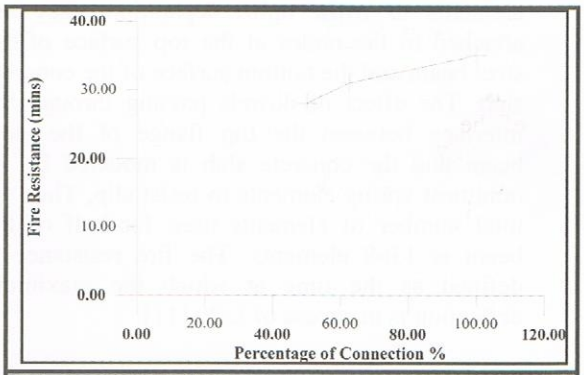


Figure (25) Effect of degree of connection on the fire resistance of beam B1

6.4 Effect of Concrete Slab Thickness

In order to show the effect of concrete slab thickness on the fire resistance of composite steel-concrete beam, beam B1 which had a concrete slab thickness of 130 mm has been analyzed with different concrete slab thicknesses of 150 and 100 mm, and the results are shown in Fig. 26.

The fire resistance is increased by 12.5% when using a concrete slab thickness of 150mm, in comparison with 130mm slab thickness, and is decreased by 11.4% when a slab thickness of 100mm is used.

6.5 Effect of Applied Load

When the composite beam is loaded using a non-composite load, i.e. equivalent to a simply supported design stress of 165 MPa, the fire resistance is 35 minutes, as shown previously in the verification part. While when the composite beam is tested at full design stress of 180 MPa, its fire resistance is reduced to 22 minutes, i.e. a decrease in the fire resistance of 37% occurs.

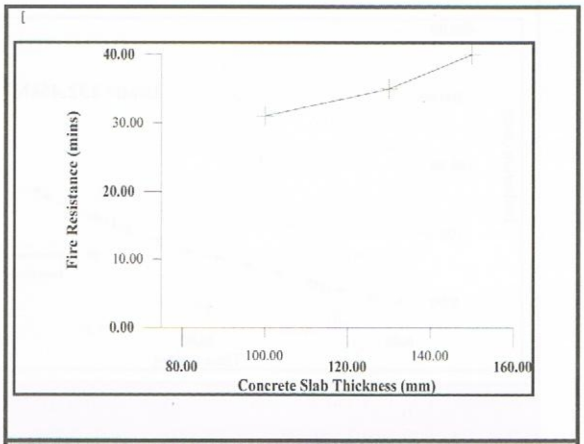


Figure (26) Effect of concrete slab thickness on the fire resistance of composite beam B1

7. Conclusions

Based on the results of the finite element analysis of the composite steel-concrete beams subjected to BS476 fire test [11], the following conclusions are drawn:

1. Comparison of the computed results with the available experimental data confirms the accuracy and validity of the finite element method, materials models, constitutive relations and the computer program to predict the behavior of composite steel-concrete beams under fire.
2. When heating is applied to the bottom surface of the composite beam, the temperature decreases towards the top surface of the concrete slab, and the temperatures in the steel beam are higher than that in the concrete slab due to the low thermal conductivity and high specific heat of the concrete.
3. An increase in the concrete compressive strength causes an appreciable increase in the fire resistance of the composite beams. An increase in the compressive strength from 24 to 30, 40 and 50 MPa causes an increase in the fire resistance of 11.4, 17.1 and 22.8%, respectively.
4. The yield stress of the steel beam section has a large influence on the behavior of composite beams under fire, and the results show that increasing the yield stress from 281 to 350 and 400 MPa increases the fire resistance by 25.7 and 48.5 %, respectively.
5. Under fire, the fire resistance of the composite beam deceases by 11.4% and 20% when the number of shear connectors reduced by 37.5% and 50% respectively.
6. The fire resistance of composite steel-concrete beams exposed to standard fire is influenced by the concrete slab thickness, it increases by 12.5% when the slab thickness is increased to 150mm from 130mm, and decreases by 11.4% when decreased to 100mm thickness
7. An increase in the applied load causes an increase in the mid-span deflection and then a decrease in the fire resistance of the beam.

References

[1] Nethercot, D. A. (2003) "Composite Construction", London: Spon Press, 232pp.

[2] Johnson, R. P. (1994) "Composite Structures of Steel and Concrete, Beams, Slabs, Columns, and Frames for Buildings", volume 1, 2nd edn, Oxford: Blackwell Scientific publications, 210pp.

[3] Kruppa, J. and Zhao, B. (1995) "Fire Resistance of Composite Beams to Eurocode 4 Part 1.2", *Journal of Constructional Steel Research* **33**: 51-69.

[4] ANSYS. ANSYS release 10.0 manual. SAS IP, Inc.;2005

[5] Willam, K. J. and Warnke, E. D. (1975) "Constitutive Model for the Triaxial Behavior of Concrete", *Proceedings, International Association for Bridge and Structural Engineering, ISMES*, Bergamo, Italy. (cited by ANSYS, 2005).

[6] Chen, W. F. (1982) "Plasticity in Reinforced Concrete", USA: McGraw-Hill, Inc., 474pp.

[7] (CEN) European Committee for Standardisation (2005) "*prEN1993-1-2, Eurocode 3: Design of Steel Structures, Part 1.2: General Rules, Structural Fire Design*", British Standards Institution, London.

[8] (CEN) European Committee for Standardisation (2003) "*prEN1994-1-2, Eurocode 4: Design of Composite Steel and Concrete Structures, Part 1.2: Structural Fire Design*", British Standards Institution, London.

[9] Smith, C. I. and Thomson, G. (1981a) "Report 4: Simply Supported Composite Steel/Concrete Beam Designed to CP117.Tested at Non-Composite Load", *British Steel Corporation, Teesside Laboratories Report T/RS/1380/12/81/D*, 27pp.

[10] Smith, C. I. and Thomson, G. (1981b) "Report 6: Simply Supported Composite Steel/Concrete Beam Designed to CP117.Tested at Full Design Load", *British Steel Corporation, Teesside Laboratories Report T/RS/1380/14/81/D*, 25pp.

[11] British Standards Institution (BSI) (1972) "*British Standard BS 476, Fire Tests on Building Materials and Structures, Part 8: Test Methods and Criteria for the Fire Resistance of Elements of Building Construction*", British Standards Institution, London, 15pp.

[12] (CEN) European Committee for Standardisation (2000) "*prEN1991-1-2, Eurocode 1: Basis of Design and Actions on Structures, Part 1.2: Actions on Structures-Actions on Structures Exposed to Fire*", British Standards Institution, Lond

سلوك العتبات المركبة المعرضة للحريق

د. هاني محمد فهمي
استاذ
قسم الهندسة المدنية / جامعة النهرين

تمارا تةفيق يونان عبو يونان
ماجستير
قسم الهندسة المدنية / جامعة النهرين

الخلاصة:

استخدمت في هذا البحث طريقة لا خطية ثلاثية الأبعاد تعتمد على طريقة العناصر المحددة لتحليل عتبات مركبة من الحديد والخرسانة معرضة إلى فحص الحريق (BS476:Part8). يتضمن التحليل التوزيع الحراري و السلوك الإنشائي باستخدام برنامج (ANSYS V.10).

في التحليل الحراري, استخدمت عناصر طابوقية (حرارية) ذات ثمان عقد لتمثيل كل من البلاطة الخرسانية و العتبة الحديدية.

في التحليل الإنشائي, استخدمت عناصر طابوقية (خرسانية مسلحة) ذات ثمان عقد لتمثيل البلاطة الخرسانية المسلحة, كذلك تم فرض حديد التسليح منتشر داخل العنصر الخرساني. استخدمت عناصر طابوقية (إنشائية) ذات ثمان عقد لتمثيل العتبة الحديدية. تم تمثيل روابط القص باستخدام عناصر محورية وعناصر نابضة لا خطية لمقاومة الانفصال والانزلاق جزئي الخرسانة والحديد وتم تمثيل السلوك اللاخطي للسطح البيني بين الخرسانة والحديد باستخدام عناصر بينية.

أخذ التصرف اللاخطي للمواد، بسبب تشقق وتهشم الخرسانة، خضوع حديد العتبة وقضبان التسليح واعتماد خواص المواد على درجات الحرارة في التحليل.

تم حل معادلات التوازن اللاخطية بطريقة تزايدية- تكرارية باستخدام طريقة (Newton-Raphson) مع استخدام معيار الحرارة والقوة لإيجاد مقدار التقارب في الحل للتحليل الحراري والإنشائي, على التوالي. وأجريت التكاملات العددية باستخدام طريقة (Gauss quadrature).

تم دراسة بعض المتغيرات على سلوك العتبة المركبة كمقاومة انضغاط الخرسانة، إجهاد الخضوع للعتبة الحديدية، درجة الترابط بين العتبة والبلاطة، سمك البلاطة الخرسانية ومستوى الأحمال المسلطة لإيجاد مدى تأثيرها على تصرف عتبة مركبة معرضة إلى حريق قياسي.

أظهرت النتائج التحليلية باستخدام طريقة العناصر المحددة توافقاً جيداً مع النتائج المختبرية المتوفرة مما يؤكد ملائمة وصحة الطريقة المتبعة والنماذج المستخدمة.