The Iraqi Journal For Mechanical And Material Engineering, Special Issue (E)

THE RESPONSE OF COMPOSITE CONCRETE-I-BEAMS UNDER THE EFFECT OF BOTH FIRE AND EXTERNAL LOADINGS

Dr. Kareem N. Salloomi
AL-Khwarizmy College of Engineering
Manufacturing Processes Dept.

Dr. Laith Abed Sabri
AL-Khwarizmy College of Engineering
Mechatronics Dept.

Abstract

The study of nonlinear response of steel buildings under elevated temperatures from fire loading can be achieved by sequentially using heat-transfer analysis and stress analysis. By means of finite element analysis represented by the commercial finite element (FE) software **ANSYS** both thermal and structural response of composite concrete beams exposed to fire has been set up. The proposed three-dimensional FE model is able to simulate the overall flexural behavior of simply supported composite beams subjected to either concentrated or uniformly distributed loads in combined with temperature effect. The bond between the top flange of the steel beam and the concrete slab was assumed to be rigid in this model.

الخلاصة

في هذا البحث تم دراسة الاستجابة اللاخطية للبنايات الحاوية على الفولاذ كعنصر اساسي في الهيكل تحت تاثير درجات الحرارة العالية والناتجة بسبب احمال النيران وذلك بتحليل الانتقال الحراري اولا ومن ثم دراسة تحليل الاجهاد نتيجة للحمل الحراري . تم استخدام طريقة العناصر المحددة والممثلة ببرنامج (Ansys) لدراسة كل من الاستجابة الحرارية والهيكلية للاعمدة المركبة والمكونة من الخرسانة المسلحة بالفولاذ . ان النموذج المستخدم في هذة الدراسة قابل لنمذجة التصرف الالتوائي للاعمدة المركبة والمثبتة جزئيا والمعرضة لاحمال مرك زة في نقطة او احمال موزعة بشكل منتظم بالاضافة الى الاحمال الحرارية . في هذة الدراسة تم افتراض الربط بين عمود الفولاذ والخرسانة المسلحة هو ربط ثابت.

Keywords: Composite beams; Finite element method; Temperature; Fire; Stress analysis

1-Introduction

Basic material research of structural steel materials is becoming more important as the significance of fire engineering design of steel structures is growing and new steel materials, including high strength steels and stainless steels are going to be used more widely in steel structures in the near future. When a structural element is being subjected to an important increase of temperature, such as occurs in the case of a building fire or local fire, its behaviour is affected by the restraint offered by the surrounding parts of the complete structure that may or may not be subjected to heating. The most important effects are loss of strength, thermal expansion, and the relative stiffnesses of the surrounding elements, the development of considerable deflections, temperature gradients and buckling phenomena, among others. Traditionally, the design of structural elements under fire has been based on the behaviour of a single element that is considered to be isolated from the other elements surrounding it. Another important factor that affects the behaviour of steel frames under fire is compartmentation or the lack of it. If the building is appropriately divided into fire compartments, it can be fairly well assumed that fire will not spread beyond compartment boundaries and the design of structural members is done based on the compartment as an entity without need for the analysis of surrounding structures, or, for a more refined design, the stresses will be redistributed to the surrounding structures. In steel and concrete composite structures, this is usually achieved through the behaviour of the concrete slab as load-carrier in fire conditions. **Ranzi and et al [1]** presented novel analytical solutions to describe the behavior of composite steel–concrete beams with partial interaction at elevated temperatures. The analytical model is derived by means of the principle of virtual work and, based on its strong form, solutions are derived in closed form for the cases of a simply supported beam and of a propped cantilever subjected to a generic regime of temperature. **Panahshahi and et al [2]** used a combined computational fluid dynamics (CFD) and structural stress investigation to study the 3D response of a two-story framed steel building. **Queiroz and et al [3]** focused on the evaluation of full and partial shear connection in composite beams using the commercial finite element (FE) software ANSYS. Previous numerical studies have been conducted to investigate the behaviour of composite beams. Nevertheless, most of them are based on two-dimensional analytical models (**e.g., Gattesco [4] and Pi et al. [5]**), and are thus not able to simulate more complex aspects of behaviour, which are intrinsic for three-dimensional studies; for instance: full distribution of stresses and strains over the entire section of the structural components (steel beam and concrete slab). The present investigation focuses on the modelling of composite beams with rigid connection using the software **ANSYS version 11 [6]**. A three-dimensional model is proposed, in which all the main structural parameters and associated nonlinearities are included (concrete slab, steel beam).

2. Finite element model

2.1. Software, element types and mesh construction

Advances in computational features and software have brought the finite element method within reach of both academic research and engineers in practice by means of general-purpose nonlinear finite element analysis packages, with one of the most used nowadays being ANSYS. The program offers a wide range of options regarding element types, material behaviour and numerical solution controls, as well as graphic user interfaces (known as GUIs), auto-meshers, and sophisticated postprocessors and graphics to speed the analyses. In this paper, both thermal and structural systems modelling is based on the use of this commercial software. Because of the available element types of the program, the concrete and structural and main steel reinforcing is represented by a 3D isoparametric solid element with eight nodal points having three degrees of freedom at each node (translations in the nodal X-, Y-, and Z-directions). The element used for thermal analysis is (solid 70) this element type has a three-dimensional thermal conduction capability and eight nodes with a single degree of freedom (temperature) at each node. The element is applicable to a three-dimensional, steady-state or transient thermal analysis. For structural analysis element (solid 65) was used for the concrete, this element is used for three dimensional modelling of solids with or without reinforcing bars (rebar capability). The element has eight nodes and three degrees of freedom (translations) at each node. The concrete is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep [6]. The rebars are capable of sustaining tension and compression forces, but not shear, being also capable of plastic deformation and creep. Both longitudinal and transverse reinforcing bars are modelled as smeared throughout the solid finite elements. For the steel beam element we used solid45 element type, the element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Symmetry of the composite beams is taken into account by modelling only one half of the beam span. A typical FE mesh for a composite beam is shown in Fig. 1. A full description of the model used in this study is shown in Fig. 2. The width of both flanges and hub were taken as 10mm.

2.2. Material modeling

The von Mises yield criterion with isotropic hardening rule (bilinear-hardening material) is used to represent the steel beam (flanges and web) behaviour. This option is often preferred for large strain analyses. The material behavior is described by a bilinear stress-strain curve starting at the origin with positive stress and strain values. The initial slope of the curve is taken as the elastic modulus of the material. At the specified yield stress, the curve continues along the second slope defined by the tangent modulus (having the same units as the elastic modulus). The tangent modulus cannot be less than zero nor greater than the elastic modulus. The concrete slab behaviour is modelled by a multilinear isotropic hardening relationship, using the von Mises yield criterion coupled with an isotropic work hardening assumption. The uniaxial behaviour is described by a piece-wise linear total stress–total strain curve, starting at the origin, with positive stress and strain values. The modulus was based on the equation [7],

$$E_C = 57000\sqrt{f'_C} \qquad \text{. ....1}$$

with a value of f_c' equal to 4,800 psi (33.1 MPa). The compressive uniaxial stress-strain relationship for the concrete model was obtained using the following equations to compute the multilinear isotropic stress-strain curve for the concrete [8]

$$f = \frac{E_C \varepsilon}{1+\left(\dfrac{\varepsilon}{\varepsilon_\circ}\right)^2} \qquad \text{.........2}$$

$$\varepsilon_\circ = \frac{2f'_c}{E_C} \qquad \text{..........3}$$

$$E_C = \frac{f}{\varepsilon} \qquad \text{..........4}$$

where:

f = stress at any strain ε

ε = strain at stress f

ε_0 = strain at the ultimate compressive strength f_c'

The multilinear isotropic stress-strain implemented requires the first point of the curve to be defined by the user. It must satisfy Hooke’s Law

$$E = \frac{\sigma}{\varepsilon} \qquad \text{.........5}$$

The multilinear curve is used to help with convergence of the nonlinear solution algorithm.

Figure 3. shows the stress-strain relationship used for this study and is based on work done by [9]. In the present study the beam is modeled using discrete reinforcement. Therefore, a value of zero was entered for all real constants which turned the smeared reinforcement capability of the Solid65 element off.

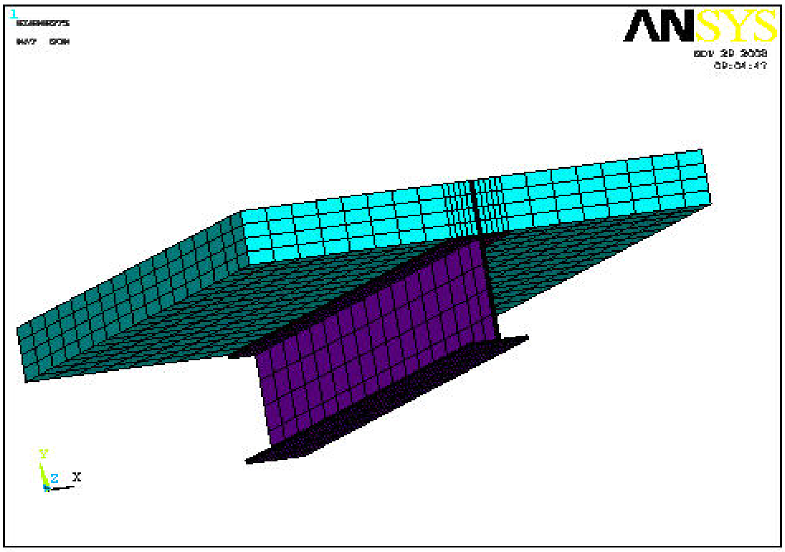


Fig. 1. Typical composite beam FE mesh.

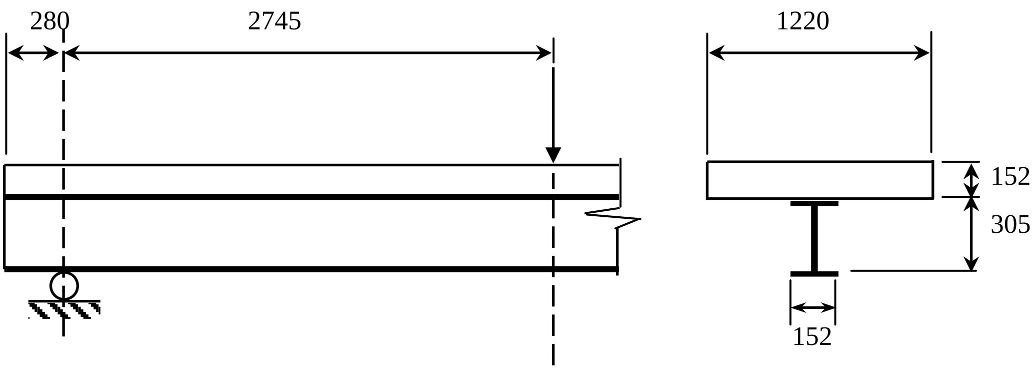


Fig.2. Simply supported beam layout (dimensions in mm).

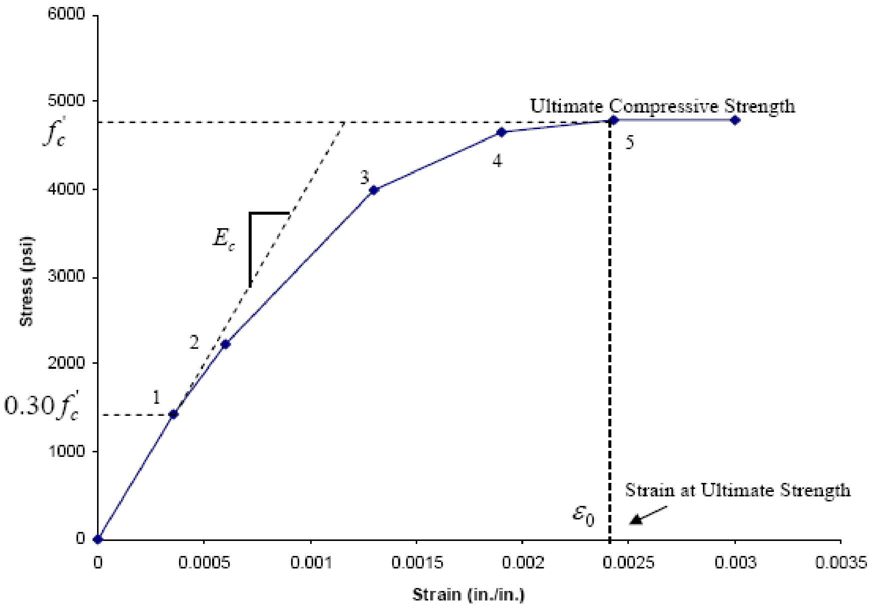


Figure 3. – Uniaxial Stress-Strain Curve[9]

3. Analysis Procedure

The analysis procedure consisted of two main parts.

3.1Thermal Analysis

A nonlinear finite-element analysis in connection with a time-step integration is used to calculate the temperature distribution in the section. The chosen time steps have to be quite small (At = 5 min), because the characteristic values of the thermal conductivity k, specific density p, and specific heat capacity *Cp* are dependent on temperature, as shown in Fig. 4[10] . The fire load is used by following the standard time-versus-temperature relationship outlined in ISO-834 *(International* 1975) [11], as shown in Fig. 5. Depending on the experimental measurements, a regression analysis was done to determine the fire temperature of the exposed surface of the investigated specimens during the whole test period according to CEB 208 (Comit6 1991b) [12].

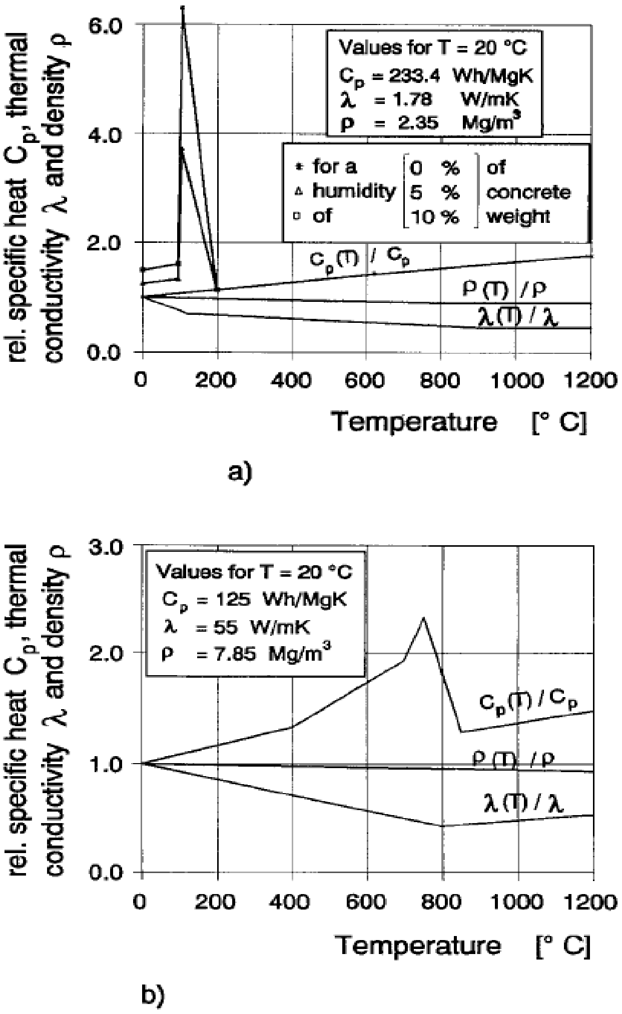


Fig. 4. Thermal Material Properties: (a) Siliceous Concrete; (b) Structural and Reinforcing Steel [1 Mg/m^3 = 1,000 Kg/m^3; Wh/(Mg. K) = 3.6 J/(kg. K)] [10]

The conservation of energy in a differential form can be written as:

$$\rho \frac{\partial \mathbf{c}\,\mathbf{T}}{\partial t} = \mathbf{Q} + \frac{\partial}{\partial \mathbf{x}}\left(\mathbf{k}_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial \mathbf{y}}\left(\mathbf{k}_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial \mathbf{z}}\left(\mathbf{k}_z \frac{\partial T}{\partial z}\right) \qquad \text{……..6}$$

Where,

ρ is the density of the material, **c** is specific heat,**Q** is heat generated inside the element, and **t** is time.

In the finite element formulation, this equation can be written for each element as follows;

$$[C(T)]\left\{\dot{T}\right\} + [K(T)]\{T\} = \{Q(T)\} \qquad \ldots\ldots\ldots 7$$

where,

[C(T)] is specific heat matrix,

[K(T)] is conductivity matrix

{T} is vector of nodal temperatures

$\left\{\dot{T}\right\}$ is vector of time derivative of {T}

{Q(T)} is nodal temperature vector

If the thermal conductivity of the material, k is assumed constant, then Eq.6. becomes:

$$\rho c\frac{\partial T}{\partial t} = Q + K\left(\frac{\partial}{\partial x}(\frac{\partial T}{\partial x}) + \frac{\partial}{\partial y}(\frac{\partial T}{\partial y}) + \frac{\partial}{\partial z}(\frac{\partial T}{\partial z})\right) \qquad \ldots\ldots\ldots.8$$

In our analysis, temperature-dependent thermal properties were assumed, therefore non-linear equations were solved, with all complexity related to their solutions. This assumption was made observing that the temperature changes (gradients) encountered in the analysis are so large that the change of thermal properties could not be neglected. Using the finite element analysis the thermal and stress analysis are uncoupled while in reality thermal effect and structural deformation occur at the same time. The de-coupling of the analyses becomes acceptable if one assumes that dimensional changes (structural deformation) during heating process are negligible because thermal energy change is predominant over mechanical work done during heating, and the internal energy dissipation effect on the temperature distribution is negligible.

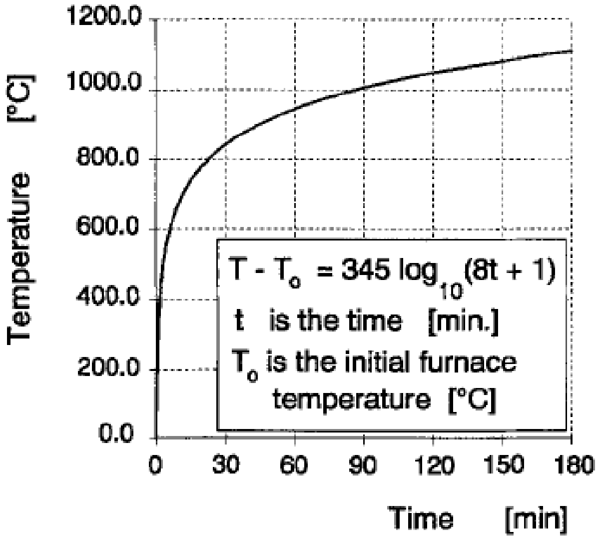


Fig.5. ISO-834 Time-Temperature Curve (1 min = 60 sec; 0~ = 273.15 K)

3.2 Structural Analysis

To determine the displacements, stresses, strains, and forces under the action of the applied loads, a finite-element analysis was executed on the same network used in the calculation of the temperature distribution. The mechanical properties used for steel in this study is shown in table [1].

Table [1] Variation of mechanical properties of steel with temperature

Temperature, C^0	Elastic Modulus, GPa	Thermal Expansion, $1/C^0*10^{-6}$	Yield Stress, MPa	Poisson's Ratio
20	210	2.92	290	0.3
100	207	2.92	262	0.3
200	202	3.1	241	0.33
300	200	3.2	196	0.35
400	168	3.3	151	0.37
500	105	3.5	132	0.385
600	60	3.6	110	0.4
700	45	3.7	20	0.42
800	31	3.7	20	0.44
900	22	3.7	20	0.46
1000	20	3.7	20	0.48
1100	18	3.7	20	0.48
1200	15	3.7	20	0.48

The concrete slab behaviour is modelled by a multilinear isotropic hardening relationship with elastic modulus (27.2 GPa and poissson's ratio of 0 .15).
The general scheme in Figure 6. shows the way the analysis can be carried out from start to finish. The main parts are the thermal analysis and the structural analysis and these can generally be considered to be uncoupled. This means that it is practical to first model the development of temperatures and then implement the results to the structural model which is used to obtain information about the behavior of the structure subject to the thermal and other loads.

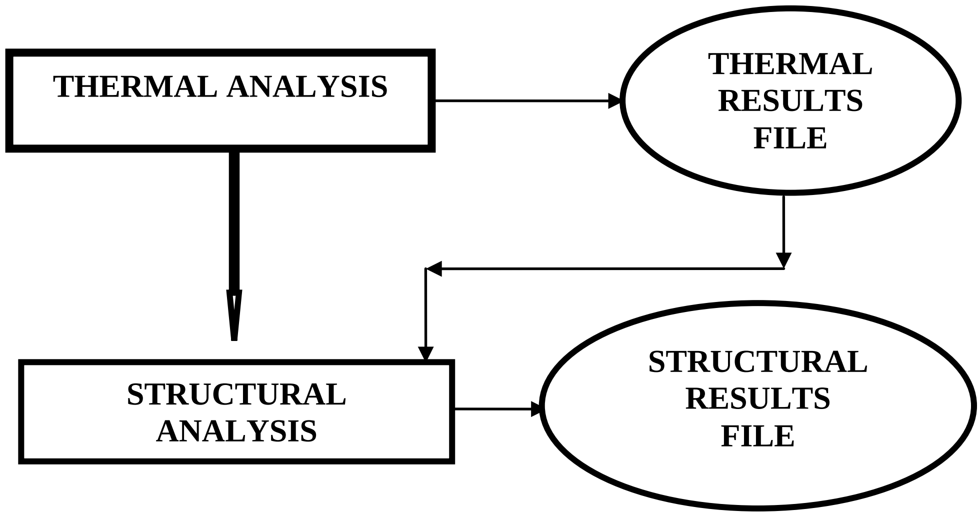


Fig.6. Data flow for a sequential thermal-structural analysis

4.Numerical results and comment

4.1Temperature distribution during fire

The non-linear transient heat transfer FE analysis is performed with automatic time stepping to cover the heating process time, the heat was input in ramped load steps. The nodal temperature solutions obtained from thermal analysis were read as loading into the stress analysis. In order to capture the stresses induced due to the heating, the temperature history needed to be read at a sufficiently large number of time points, especially where the temperature gradient is large. However, the greater the number the computational time and the larger the store space required. Fig. 7 shows the temperature field of a composite beam exposed to fire after 30, 60, and 90 min.

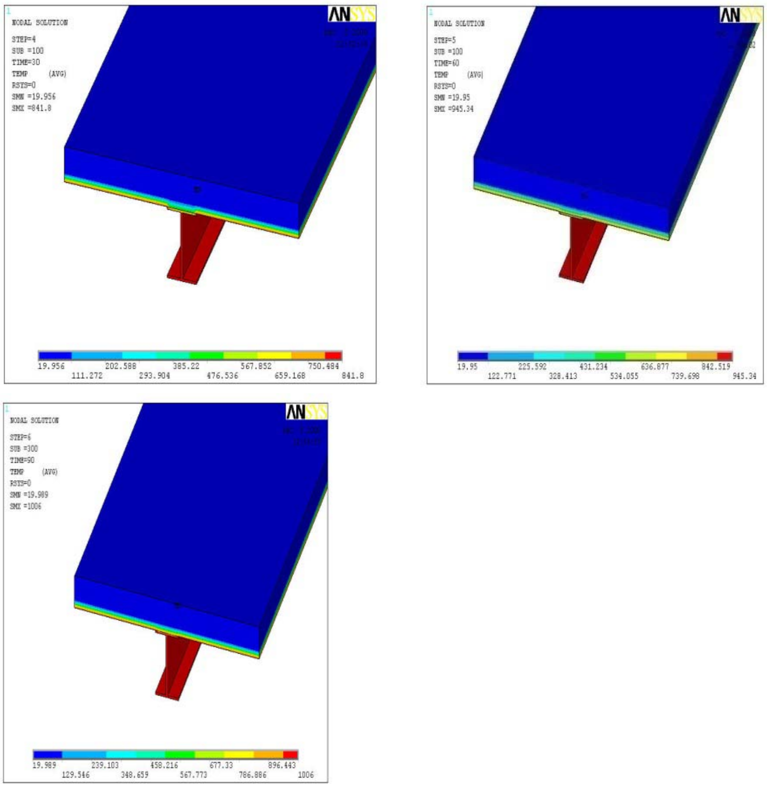
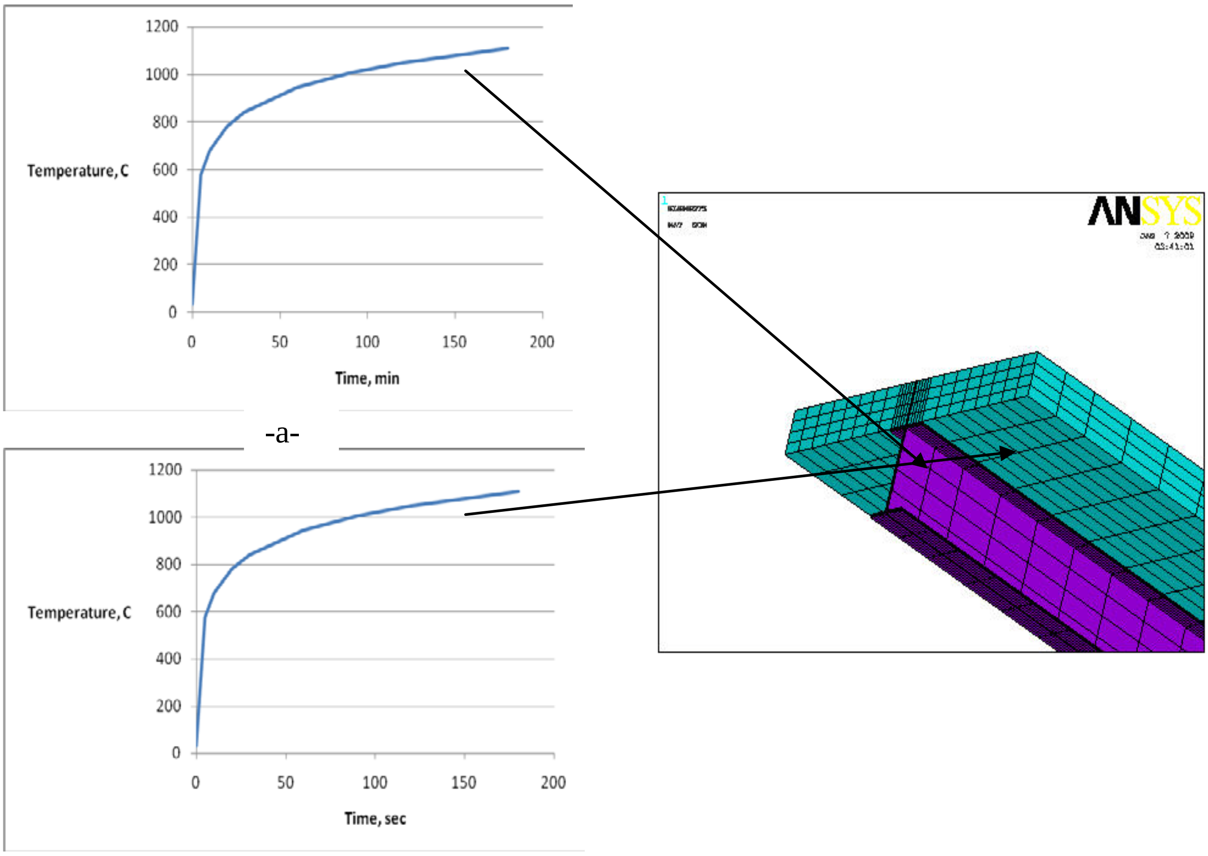


Fig. 7. Snapshot of a computer model at *t* = 30,60,90 min showing temperature distribution in the I-beam and concrete slab.

The time history temperature distribution of a point in the inner surface of the steel beam is shown in Fig.8a., while Fig.8b. shows the time history temperature distribution of a point in the bottom surface of the slab. Since both surfaces is exposed to the same level of fire, then their temperature response will be the same as shown in Fig.(8a&b).



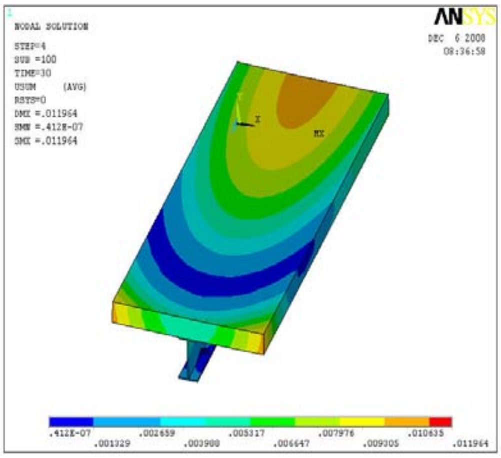
-a-

-b-

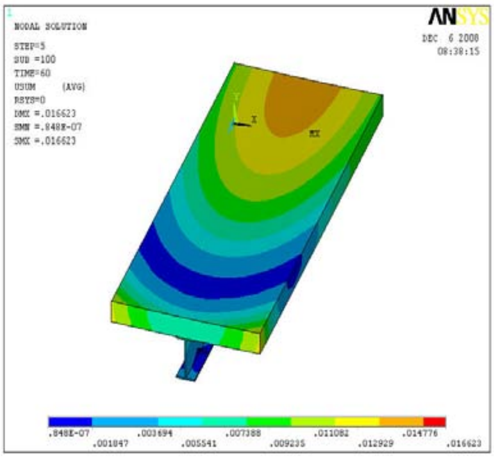
Fig.8. Time history temperature distribution of
a- a point in the inner surface of the steel beam
b- **a point in the bottom surface of the slab**.

4.2Deflection Behavior during Fire and External Loading

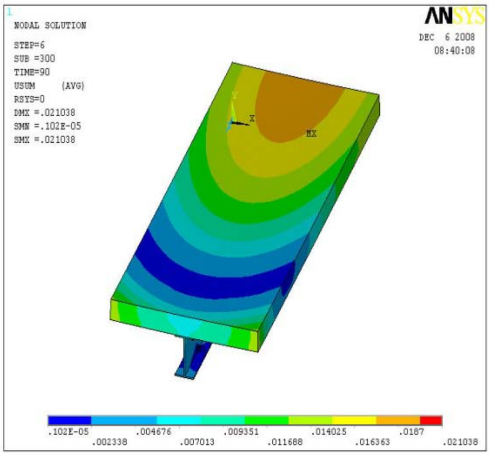
The deflection behavior of the investigated specimen under external and thermal loadings was studied using finite element method. Regarding application of load, concentrated loads are incrementally applied to the model by means of an equivalent displacement to overcome convergence problems (displacement control). For the convergence criterion, the L2-norm (square root sum of the squares) of displacements is considered. Uniformly distributed loads are represented by means of point loads (100KN) applied at all mid-section concrete nodes. Preliminary attempts to overcome the convergence problems arising from the use of the load control method included the specification of different types of equation solvers. The best
approach in terms of numerical performance was the option in which the software ANSYS selects a solver based on the physics of the problem. Fig.9 shows displacement of the studied model under both thermal and external loadings at times(30,60,90,180 min) , as shown the maximum displacement occurs on the upper surface of the slab, this indicates that the external load has more significant effect than the thermal load and may cause failure more faster than thermal load.



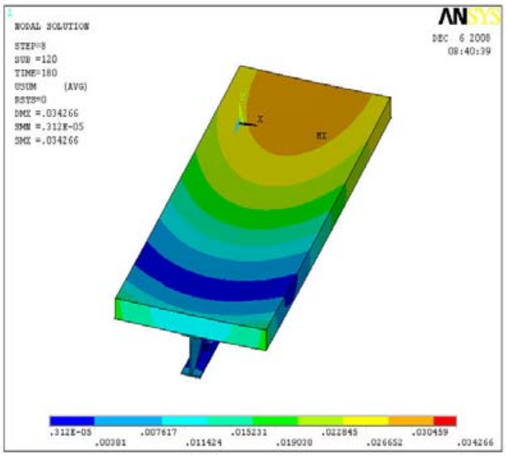
a-30 min



b- 60 min



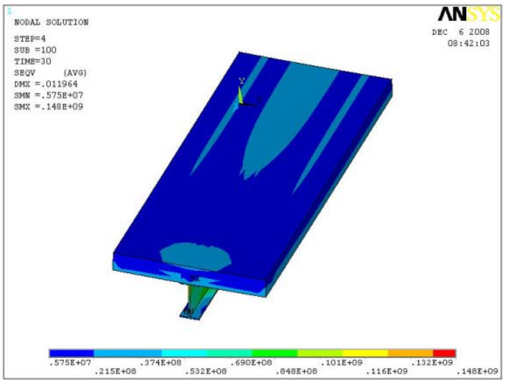
c-90 min



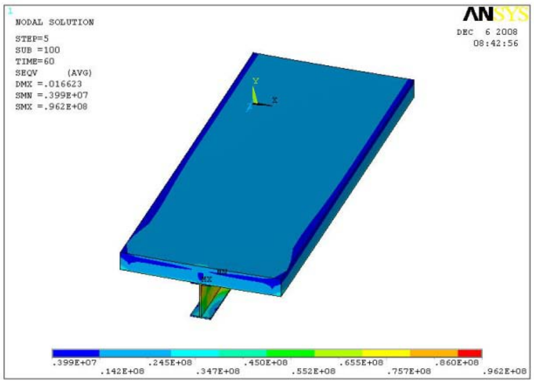
d-180 min

Fig.9. Displacement field of composite beam at different times under thermal and external loadings.

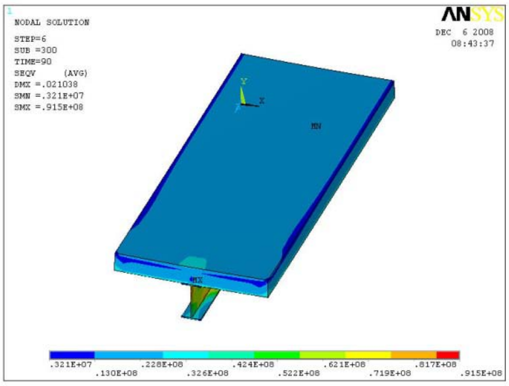
Fig.10 shows the von- mises stress distribution at times (30, 60, 90,180 min), the maximum stress is shown at the hub surface on the steel beam (78.3 MPa) and this value is much greater than the yield stress used in this study at temperature 1200 C^{O} for steel beam, while the slab encounter only small stress of (6.512 MPa). This fact ensures that the failure in steel will be prior to that of concrete.



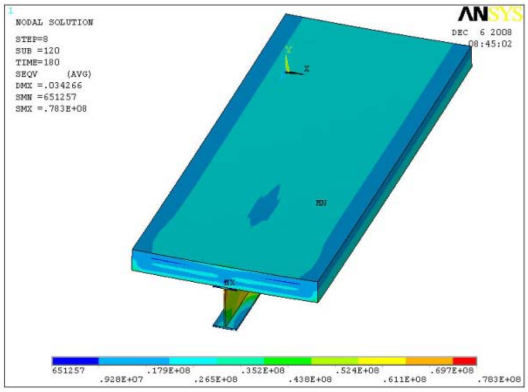
a-30 min



b-60 min



c-90 min



d-180 min

Fig.10. Von-mises stress distribution of composite beam at different times under thermal and external loadings

CONCLUSIONS

By using the finite-element analysis, the internal temperature field for any cross section can be established and the global deformation of structures can be given in order to show either its evolution as a function of elapsed time or the situation just before failure. The present numerical analysis is sufficiently reliable in accuracy and practice to study a wide range of structural elements under fire conditions. The main conclusion of this paper is that under such circumstances of fire and external applied loads steel may fail prior to the concrete.

References:

[1] Gianluca Ranzi and Mark Andrew Bradford. (2007). “Analytical Solutions for Elevated-Temperature Behavior of Composite Beams with Partial Interaction.” Journal of Structural Eng., Vol.133, No. 6, (788-799).

[2] Nader Panahshahi, Majid Molki, Mark Rossow, Khalil Rabiei and Kia Gharib. (2006). “The Response of Framed Steel Structures to Fire.” ASCE Journal of Structures.

[3] F.D. Queiroz, P.C.G.S. Vellasco and D.A. Nethercot. (2007). "Finite element modelling of composite beams with full and partial shear connection." Journal of Constructional Steel Research, Vol.6, No. 3, (505-521).

[4] Gattesco N. (1999) "Analytical modelling of nonlinear behaviour of composite beams with deformable connection." Journal of Constructional Steel Research ,52,(195–218).

[5] Pi YL, Bradford MA and Uy B. (2006). " Second order nonlinear inelastic analysis of composite steel–concrete members. II: Applications. Journal of Structural Engineering, ASCE ,132(5),(762–71).

[6] Swanson Analysis Systems, ANSYS. Online manual, version 11.0 and Theory Reference. 11th ed. Swanson Analysis Systems, s.1., s.d.

[7] Anthony J. Wolanski, B.S. (2004). " Flexural Behavior of Reinforced and Prestressed Concrete Beams using Finite Element Analysis" MSc Thesis, Marquette University.

[8] MacGregor, J.G. (1992), Reinforced Concrete Mechanics and Design, Prentice-Hall, Inc., Englewood Cliffs, NJ.

[9] Kachlakev, D.I.; Miller, T.; Yim, S.; Chansawat, K.; Potisuk, T. (2001), "Finite Element Modeling of Reinforced Concrete Structures Strengthened With FRP Laminates," California Polytechnic State University, San Luis Obispo, CA and Oregon State University, Corvallis, OR for Oregon Department of Transportation, May.

[10] Hosser, D., Dorn, T., and El-Nesr, O. (1993). "Entwicklung und Absicherung Praxisgerechter N~iherungsverfahren ffir die Brandschutztechnische Bemessung von Verbundbauteilen." *Abschlussbericht des Forschungsprojekts*, Institut f(ir Baustoffe, Massivbau und Brandschutz, TU Braunschweig, Germany (in German).

[11] International standard IS0-834; fire-resistance tests--elements of building construction. (I975). 1st Ed., 1101.

[12] ComitE Euro-International du B6ton (CEB). (1991b). "Fire design of concrete structures." *Bull. d'Information No. 208*, Lausanne, Switzerland.