Numerical modeling of building structures in fire conditions

Mô hình số kết cấu công trình khi cháy

Chu Thi Binh[1*], J.C. Dotreppe[2] , J.M. Franssen[2]
(1. Department of Civil Engineering, HAU, Hanoi, Vietnam;
2. Department of Architecture, Geology, Environment and Construction (ArGEnCo), University of Liège, Belgium)

Abstract: *Numerical modelling is a very useful tool to predict the fire resistance of a structure, or to reproduce artificially what has happened during a real fire. SAFIR, a non linear computer code developed at the university of Liege-Belgium, is specifically written for modeling the behavior of structures subjected to fire. It allows a determination of the temperatures in the structure resulting from the fire and, in a subsequent analysis, determination of the successive positions of equilibrium of the structure until collapse. SAFIR program accommodates various elements for different idealizations, different calculation procedures and several material models incorporating stress-strain behaviour. The elements include 2-D SOLID, 3-D SOLID, BEAM, SHELL and TRUSS elements. The stress-strain material laws are generally linear-elliptic for steel and non-linear for concrete. This paper introduces the main hypotheses, the possibilities and the validity of this program. Comparisons of test results and simulated results by SAFIR program for concrete structures, steel structures and concrete-steel composite structures are presented.*

Keywords: *Numerical modeling; Fire conditions; Steel; Concrete; Composite*

Tóm tắt: *Mô hình số là một công cụ hữu ích trong dự tính sự chống cháy của kết cấu, hoặc để mô phỏng ảo những gì xảy ra trong đám cháy thực. SAFIR, một chương trình máy tính được phát triển tại đại học Liege-Bỉ, được viết một cách đặc biệt cho mô hình hóa ứng xử của kết cấu chịu lửa. Nó cho phép xác định nhiệt độ trong kết cấu do cháy và trong phân tích tiếp theo, xác định trạng thái cân bằng của kết cấu cho tới khi bị sụp đổ. Chương trình SAFIR cung cấp đa dạng các phần tử cho các mô phỏng khác nhau. Các phần tử bao gồm: khối 2-D, khối 3-D, dầm, tấm và dàn. Quan hệ ứng suất biến dạng của vật liệu là tuyến tính elíp chung cho thép và phi tuyến cho bê tông. Bài báo này giới thiệu những giả thuyết chính, khả năng và sự chính xác của chương trình. So sánh giữa kết quả thí nghiệm và kết quả tính toán bằng SAFIR cho kết cấu bê tông, kết cấu thép và kết cấu bê tông thép liên hợp được trình bày.*

Từ khóa: *Mô hình số; điều kiện cháy; thép; bê tông; vật liệu tổng hợp.*

1. Introduction

In evaluating the behavior of building structures subjected to fire, numerical modeling is known to present several advantages compared to experimental testing such as: much lower cost and time required, ability to analyze the behavior of complete structures with complex load redistribution and to perform parametric analyses which cannot be carried out in the tests. Experimental tests are in fact not only time consuming and very expensive, but there is a limitation on the size of the furnaces which makes it impossible to test complex and large structures.
SAFIR, a nonlinear computer code, developed at the university of Liege-Belgium, is specifically

[*] Chu Thi Binh, Lecturer
Correspondence to Chu Thi Binh; E-mail: bvdkientruc@yahoo.com

written for analyzing structures subjected to fire. It allows determining the temperatures in the structure working in fire conditions and, in subsequent analyses, determining the successive positions of equilibrium of the structure until being collapsed. This paper introduces the main hypotheses, the possibilities and the validity of this program. Some comparisons of test and simulated results by SAFIR program for concrete, steel and concrete-steel composite structures are also presented.

2. The SAFIR finite element program

SAFIR is a non-linear finite element software developed at the University of Liege. It is a special purpose computer program for the analysis of structures under elevated temperature conditions; ambient temperature condition can also be considered as a special case. The program, which is based on the Finite Element Method (FEM), can be used to study the behaviour of two and three-dimensional structures. SAFIR accommodates various elements for different idealizations, several calculation procedures and various material. The elements include the 2-D SOLID elements, 3-D SOLID elements, BEAM elements, SHELL elements and TRUSS elements. The stress-strain material laws are generally linear-elliptic for steel and non-linear for concrete.

Using the program, the analysis of a structure exposed to fire consists of two steps. The first step involves predicting the temperature distribution inside the structural members, referred to as "thermal analysis". The second step, named "structural analysis", is carried out for the main purpose of determining the mechanical response of the structure due to static and thermal loading. More information is given by Franssen [1].

2.1 Thermal analysis

The temperature fields within a given network are established by an integration method for time steps. It is assumed that conduction is the main heat transfer mechanism. Convection and radiation act essentially as heat transfer from the fire environment to the structures.

Heat transfer by conduction in solid materials is described by the Fourier Eq. (1) that is solved in SAFIR according to the standard finite element procedure.

$$k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+Q-C\rho\frac{\partial T}{\partial t} = 0, \quad (1)$$

where k thermal conductivity, W/mK;
T temperature, K;
x, y, z coordinates, m ;
Q internally generated heat, W/m³;
C specific heat, J/kgK;
ρ specific mass density, kg/m³;
t time, s.

The main hypotheses are:

- The materials are isotropic, not submitted to movements, not compressible and have no mechanical dissipation;
- No contact thermal resistance exist at the interface between adjacent materials.

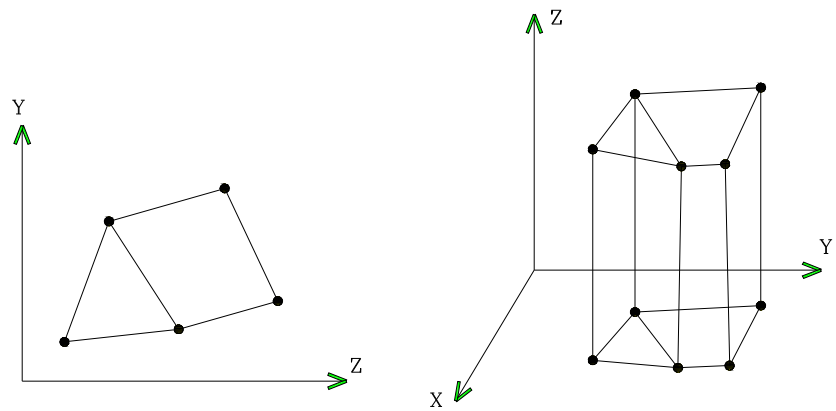

Fig.1 Finite elements for the thermal analyses [1]

Linear isoparametric finite elements are used in order to represent the geometry, based on the coordinates of the nodes, and the temperature distribution, based on the temperature at the nodes. In a 2D analysis with SAFIR, triangles and quadrangles can be used whereas 6 noded and 8 noded elements can be used in a 3D analysis (see Fig.1). Heat exchanges by radiation and convection in internal cavities can also be taken into account.

The discretisation for plane sections of different shapes is possible by using triangular and/or quadrilateral elements. For each element the material can be defined separately. Any material can be analysed provided its physical properties at elevated temperatures are known. The variation of material properties with temperature can be considered.

Fig.2 shows a rather crude disretisation for one-quarter of a cross-section of a circular steel tube column filled with reinforced concrete. The steel tube is introduced by the material named STEELEC3, the material of reinforcing bars is STEELEC2, the concrete is SILCONC_EN. The temperature around the tube named FISO follows standard temperature/time curve ISO 834. The temperatures determined in this section could be used, for example, in a mechanical analysis using 2D beam finite elements.

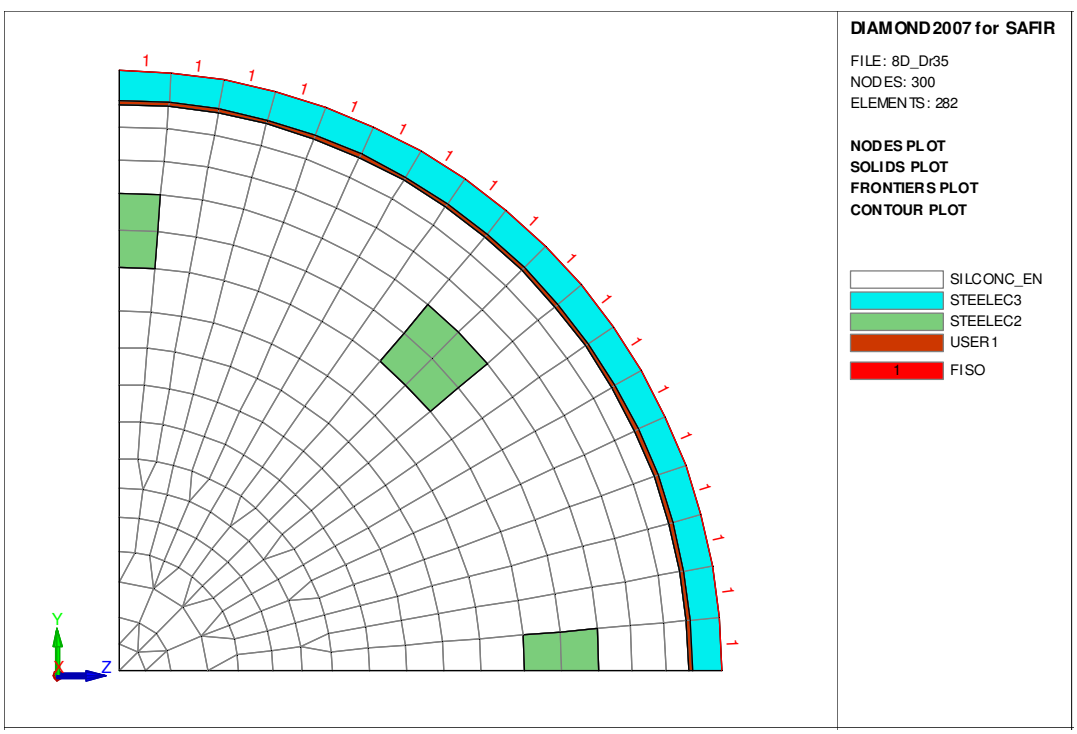

Fig.2 Discretization of one-quarter of a circular steel tube filled with reinforced concrete [2]

Fig.3 shows the discretisation of a prefabricated system made of hot rolled steel sections; one can be seen in the center of the picture. Prefabricated short terracotta elements are placed on the lower flange of adjacent steel sections and a concrete layer is added on top. Finally, a layer of gypsum is applied underneath the system.

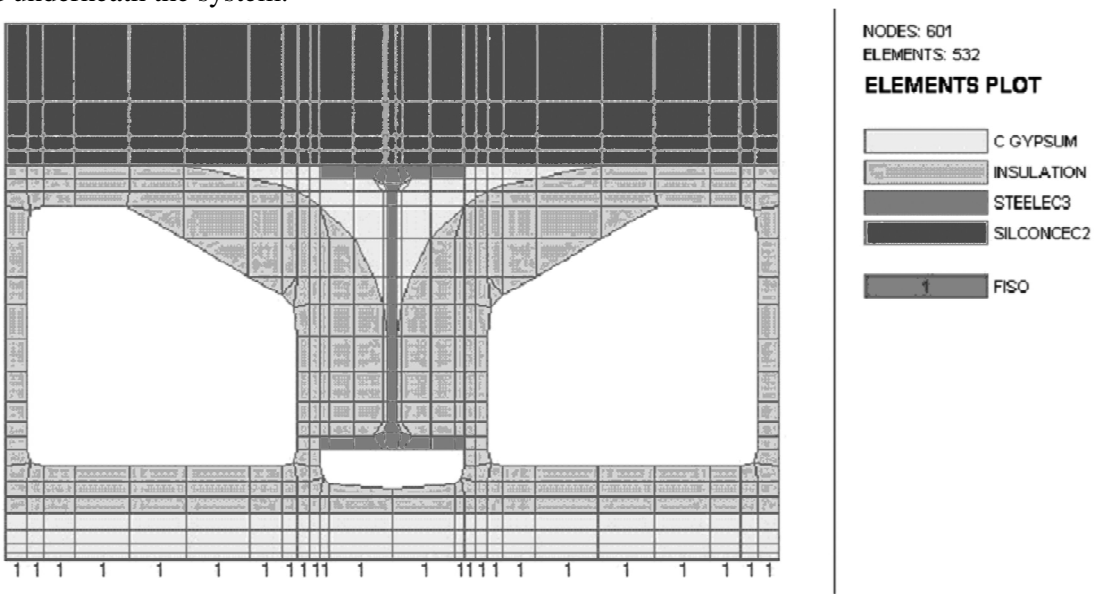

Fig.3 Part of an old prefabricated flooring system [1]

Fig.4 shows the temperature distribution after 90 minutes of ISO fire in a steel beam connected with a steel column (3 planes of symmetry have been used to allow modeling of only 1/8 of the assembly).

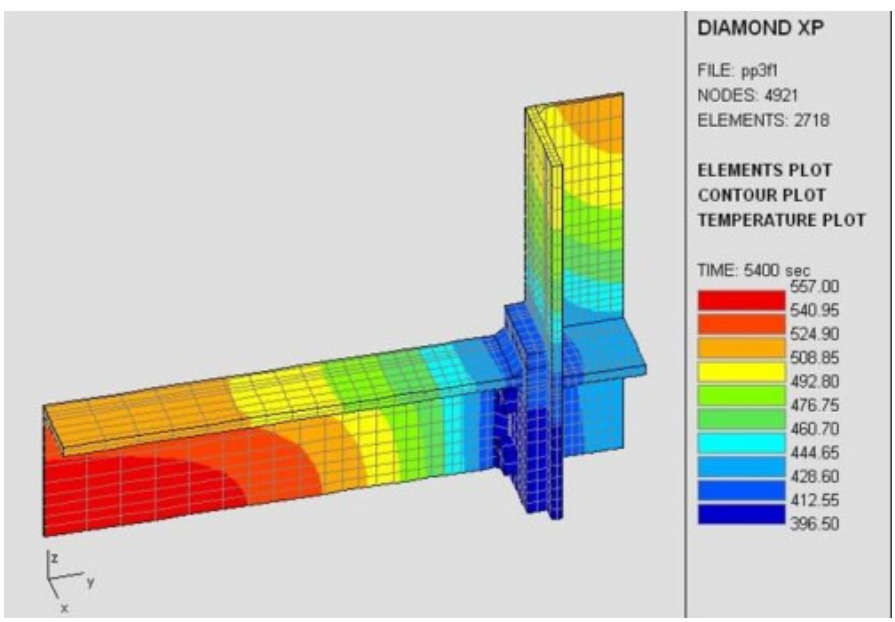

Fig.4 Temperatures in a steel beam connects with a steel column after 90 minutes of fire

2.2 Structural analysis

The basis of the mechanical analysis of structures undergoing large displacements is the incremental form of the principal of virtual work. A total co-rotational configuration is used with Eq. (2) in which the forces applied on the surface of the structure have not been considered.

$$\int_V \left(\overline{D}_{ijkl}\, d\overline{E}_{kl}\, \delta\overline{E}_{ij} + S_{ij}\, \delta d\overline{E}_{ij} \right) dV = \int_V \left(d\overline{f}_i\, \delta\overline{u}_i + \overline{f}_i\, \delta d\overline{u}_i \right) dV , \qquad (2)$$

where $\overline{x}$ means the quantity x evaluated not from the initial position of the element but from a position obtained by a rigid body movement that translates the undeformed element as close as possible to the deformed position,

$V = \overline{V}$ the undeformed volume of the element;

S_{ij} tensor of the Piola-Kirchoff stress n°2;

$\overline{D}_{ijkl} = D_{ijkl}$ tensor defining the incremental constitutive law of the material, see Eq. (3);

$\delta\overline{E}_{ij}$ tensor of the Green virtual field of displacement, see Eq. (4);

$\overline{f}_i$ volume forces;

$\delta\overline{u}_i$ virtual field of displacements from the deformed position of the element.

In a material where the temperatures change during the simulation, the constitutive law is given by Eq. (3).

$$dS_{ij} = D_{ijkl}\left(dE_{kl} - dE_{kl}^{th} \right) = D_{ijkl}\, dE_{kl}^{m}\,, \tag{3}$$

where dE_{kl} tensor of total strains included thermal strain and mechanical strains;

dE_{kl}^{th} tensor of incremental thermal strain;

dE_{kl}^{m} tensor of mechanical, or stress-related, strains.

Eq. (4) gives the tensor of Green of the virtual field of displacement.

$$\delta\overline{E}_{ij} = \frac{1}{2}\left(\delta\overline{u}_{i,j} + \delta\overline{u}_{j,i} + \overline{u}_{k,i}\,\delta\overline{u}_{k,j} + \delta\overline{u}_{k,i}\,\overline{u}_{k,j} \right) \tag{4}$$

In order to solve Eq. (2) in a displacement based finite element formulation, the field of displacements is represented in an approximate manner by the discretized field u that depends on the displacements of the nodes p via appropriately chosen shape functions **N**, see Eq. (5) written in a matrix form.

$$\mathbf{u} = \mathbf{N}\,\mathbf{p} \tag{5}$$

This allows the incremental tensor of strains to be derived also as a function of the nodal displacements, see Eq. (6) in which the matrix **B** contains not only spatial derivatives of the shape functions as in a small deformations formulation, but also the nodal displacements that are not identically equal to 0 in the corrotated configuration.

$$d\mathbf{e} = \mathbf{B}\, d\mathbf{p} \tag{6}$$

The matrix equation that governs the iteration from one position to the next position of equilibrium is Eq. (7).

$$\int_v \mathbf{B}^T\, \mathbf{D}\, \mathbf{B}\, dV\, d\mathbf{p} + \int_v \mathbf{S}^T\, \delta d\mathbf{e}\, dV\, d\mathbf{p} \quad = \quad \left(\mathbf{K}_u + \mathbf{K}_S\right) d\mathbf{p} \quad = \quad \mathbf{f}^{ext} - \mathbf{f}^{\text{int}}\,, \tag{7}$$

where K_u comprises the linear elastic and the geometric stiffness matrices;

K_s is the stress generated stiffness matrix.

The nodal forces energetically equivalent to the applied forces, f^{ext}, and the nodal forces obtained from integration of the internal stresses, f^{int}, are also obtained from the principle of virtual work via similar considerations that are not given here for reasons of space, see [3].

The SAFIR truss element

The element has a constant section of area A along the longitudinal axis that is a straight line extending between the two end nodes. The axial displacement along the element is thus linear and the Green strain E_{xx} is constant, given by Eq. (8).

$$E_{xx} = \frac{L^2 - L_0^2}{2L_0^2}\,, \tag{8}$$

where L length of the deformed element,

L_0 initial length of the element.

The internal nodal force f_x produced by the axial stress S_{xx} is given by Eq. (9).

$$f_x^{\text{int}} = -A\, S_{xx} \frac{L}{L_0} \tag{9}$$

The SAFIR beam element

The beam elements are based on the following hypotheses:

- Displacement type element in a total corotational description;
- The displacement of the node line is described by the displacements of three nodes, two nodes at each end of the element supporting, for a 2D element, two translations and one rotation plus one node at mid-length supporting the non-linear part of the longitudinal displacement. For a 3D element, each of the two end nodes supports three translations, three rotations plus the warping, thus allowing non uniform torsion to be considered. The longitudinal displacement of the node line is a second-order power function of the longitudinal co-ordinate. The transversal displacement of the node line is third-order power function of the longitudinal co-ordinate;
- The Bernoulli hypothesis is considered, i.e., the cross section remains plane under bending moment;
- The hypothesis of Von Karman is used: the strains are small;
- The rotations are assumed to be small (note that they are evaluated in the co-rotated configuration);
- The longitudinal integrations are numerically calculated using Gauss' method;
- The integration of the longitudinal stresses and stiffness on the section is based on the fibre model; the section is supposed to be made of a certain number of parallel fibres. In fact, the same discretisation as the one used for the thermal analysis is used. Each finite element of the thermal analysis, with its known material type and temperature, is considered as a fibre.

The SAFIR shell element

The SAFIR shell element is a four-node quadrilateral element. The shell element is defined by four corner nodes and has a constant thickness, *h*. The middle points of the edges of the element are a, b, c and d while the centre of the local system of coordinates lies at the intersection, o, between a-c and b-d. The *z* axis is perpendicular to the d-b and a-c plane.

There are 4 Gauss integration points on the surface of the shell element. There are also integration points distributed across the depth of the shell at the positions of the surface integration points. The number of Gauss integration points across the thickness is defined by the user, ranging from 2 to 10.

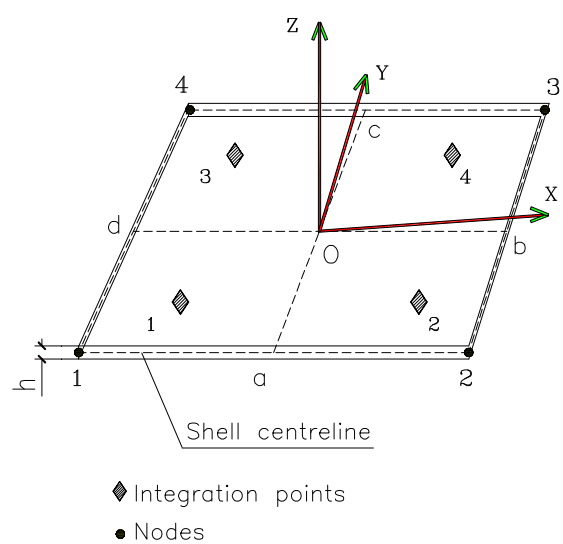

Fig.5 Geometry of the shell element

The shell element combines the membrane properties of a membrane element and the flexural properties of a plate element.
The properties of the plate element are based on the Discrete Kirchhoff Quadrilateral (DKQ) which was originally formulated by Batoz et al. Some of the properties of the DKQ plate element are:

- The out-of-plane displacements and rotations are parabolic along each side;
- The contribution of the shear strain energy is neglected;
- The shear strains at selected points on each side are set to zero;
- The rotations along the edges vary linearly.

Some of the membrane properties of the shell element are:

- The element has a cubic membrane displacement field;
- The shear strains over the element are assumed to be constant.

More information is given in [1,4].

Example models of structural analysis

Using SAFIR program, plane or 3-D structures can be analyzed. The structure is discretized by means of a combination of the three different element types above: truss elements, made of one single material with one uniform temperature per element; beam elements, either pure steel, reinforced concrete or composite-steel sections; and shell elements.
Here are some examples of structural models.

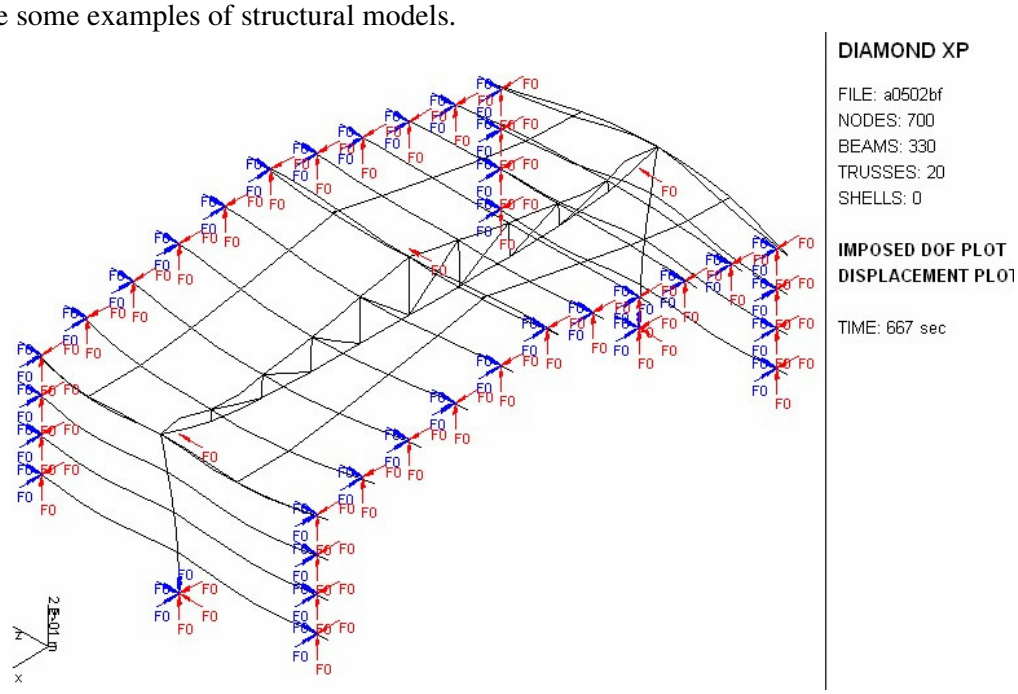

Fig.6 Model of an industrial steel building with 3D beam elements [1]

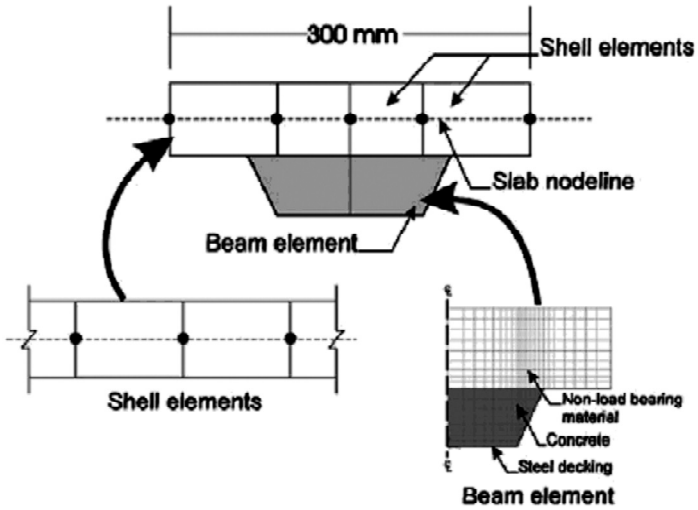

Fig.7 Discretisation of a Hi-bond composite slab for structural analysis [5]

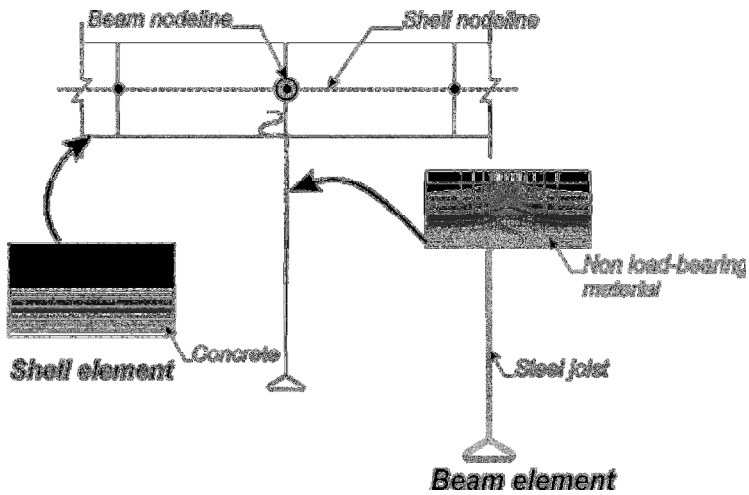

Fig.8 Discretisation of the Speedfloor slab cross section with the shell and beam elements [5].

3. Validation of SAFIR program

3.1 Validation of the thermal model

Collections of test results from some researches, comparison between calculated and measured temperatures at some point of the tests are laid out.

Fig.9 shows the temperatures at steel tube (1), reinforcing bar (2) and concrete center (3) in a steel tube column filled with reinforced concrete.

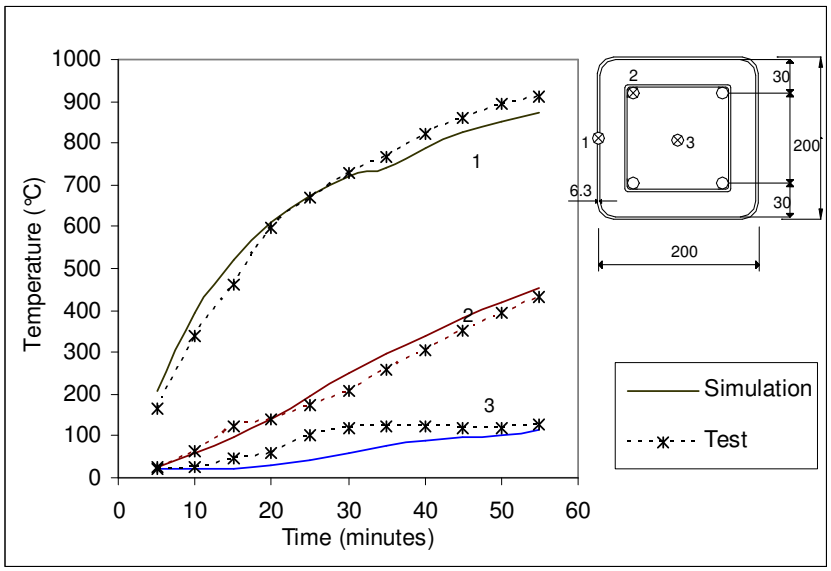

Fig.9 Comparison between calculated and measured temperatures in the cross-section of a steel tube filled with reinforced concrete [2]

Fig.10 shows the temperatures of a steel tube (Nodes 1- 6), embeded steel profile (Nodes 2-7) and concrete center (Node 3) in a steel tube column filled with reinforced concrete.

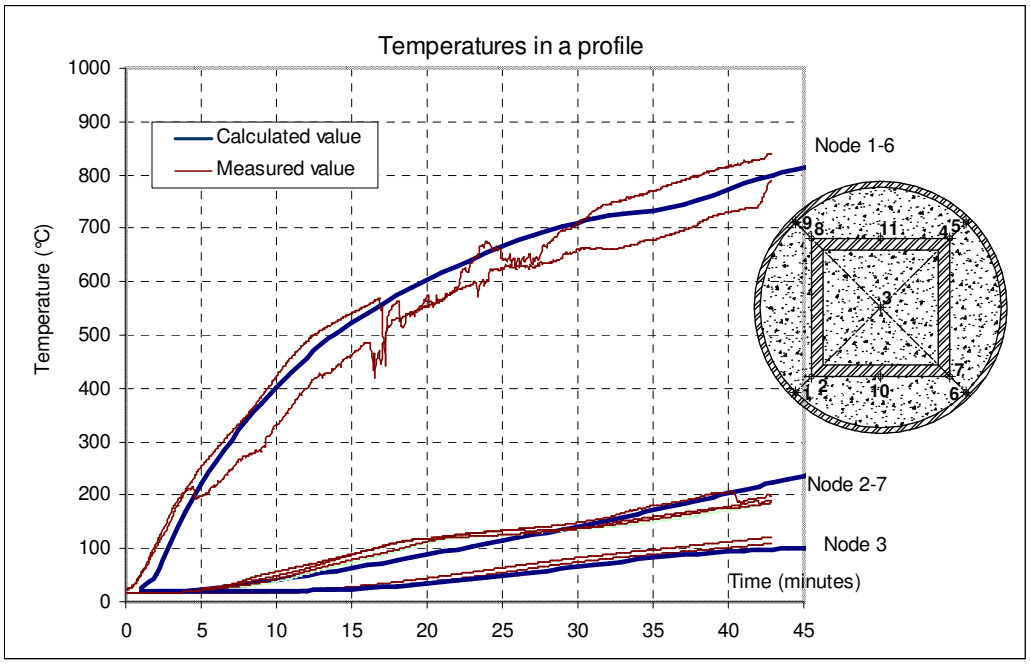

Fig.10 Temperatures in the section of a composite steel concrete column [2]

Fig.11 compares the SAFIR predictions of the temperatures across a 100 mm thick concrete slab during the fire with the experimentally measured temperatures. The variation of the temperatures with time in the fire was plotted at the heated surface, mid-depth and the unheated surface. The graph shows that apart from some slight variation, the SAFIR predictions of the temperatures across the depth of the slab are very close to the experimentally measured temperatures.

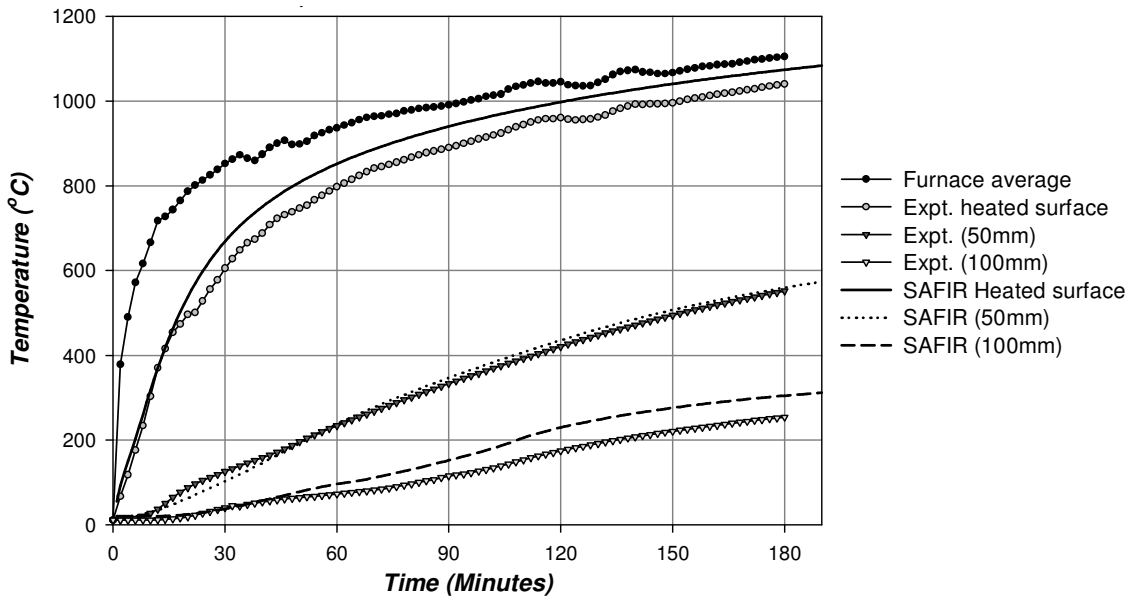

Fig.11 Comparison of temperatures from fire test and SAFIR thermal analysis [6].

3.2 Validation of the structural model

To examine the validity of the structural model, comparisons between simulated results and experiments have been done for composite columns [2], steel columns [6], and concrete and composite slab [5].

In reference [2], fire resistance and displacements of numbers of composite columns were compared between simulated results by SAFIR code and measured results by tests. They show good agreement. Fig.12 is one example. The transversal displacement at mid-height of the column is calculated by SAFIR with two values of initial out of straightness: y=0 and y=L/500.

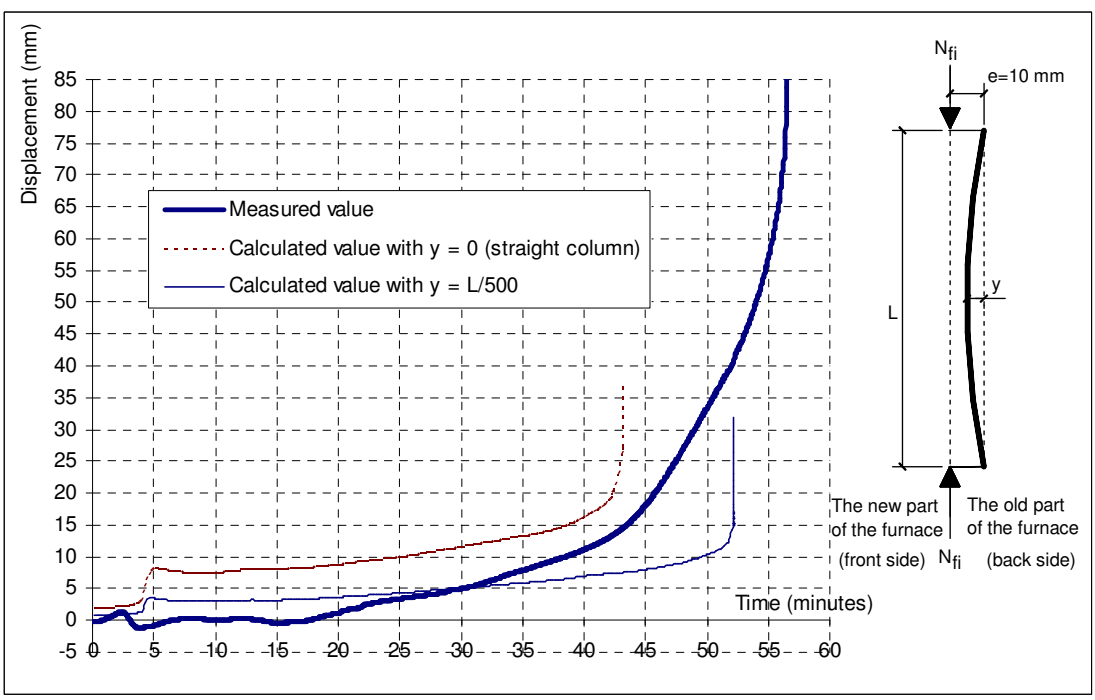

Fig.12 Transversal displacement of a composite column [7]

Fig.13 shows the deformation of a H rolled profile. As can be seen in the figure, the local buckling obtained by calculation has the same shape as the one obtained in a test made in the U.K..

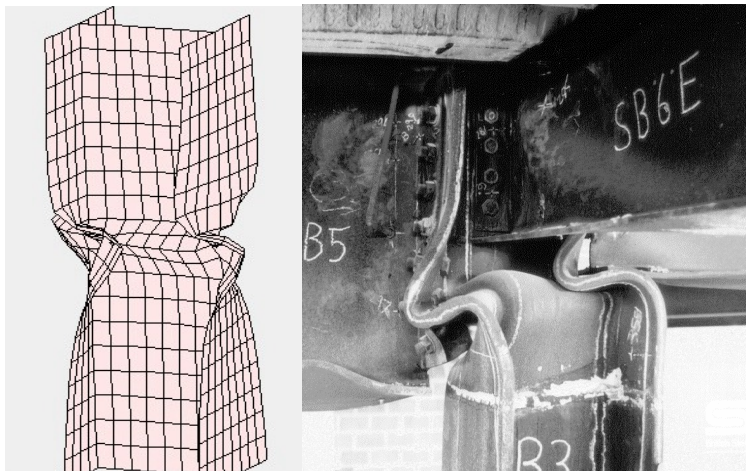

Fig.13 Deformation of the H profile at the last step where convergence can be reached and local buckling of a real structure subjected to fire [6]

Fig.14 compares the experimental results with the SAFIR predictions. The slab was analysed with concrete tensile strengths of 0MPa, 1.5MPa and 3MPa. Zero concrete tensile strength corresponds to a slab that is fully cracked and 3.0MPa corresponds to a slab with full concrete tensile strength, based on the Eq. $0.5\sqrt{f'_c}$, where f'_c is 36MPa. An intermediate value of 1.5MPa ($0.25\sqrt{f'_c}$) was also used.

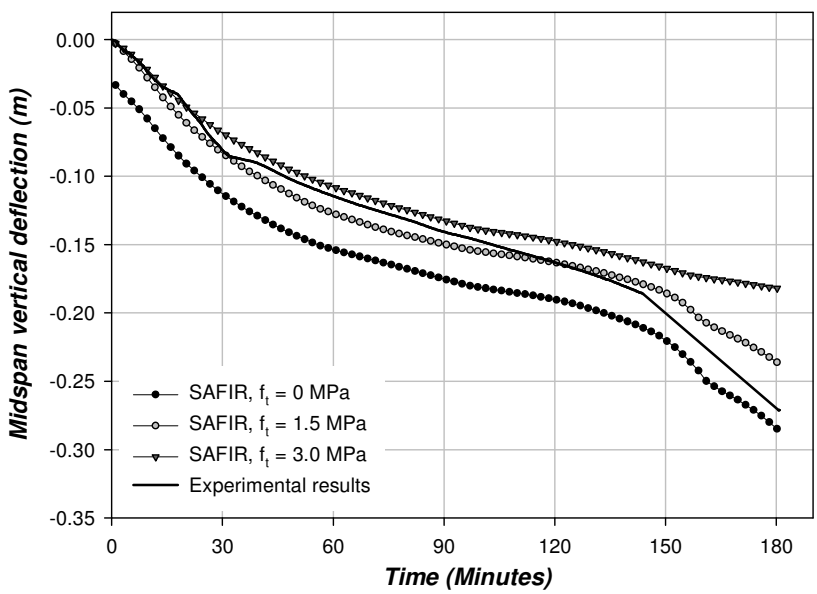

Fig.14 Midspan vertical deflections with different values of concrete tensile strength of a concrete slab [5]

Fig.15 compares the midspan vertical deflections of a composite slab (*Speedfloor*) predicted by SAFIR with the experimental results. The analysis of the *Speedfloor* slab was performed with zero and $0.25\sqrt{f'_c}$ concrete tensile strength. $0.25\sqrt{f'_c}$ concrete tensile strength was used as it had shown fairly good representation of the concrete tensile strength of the slab. The graph shows that the SAFIR analyses predicted the deflection trend of the tested slab fairly well throughout the entire fire duration.

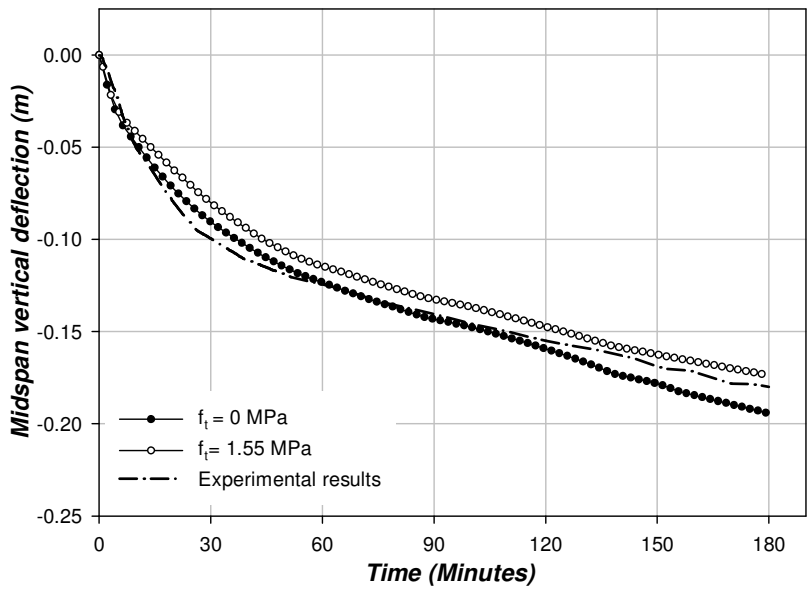

Fig.15 Midspan vertical deflections of the Speedfloor slab during the ISO fire [5].

4. Conclusion

SAFIR is a computer program specifically written for modeling the behavior of structures subjected to fire. It is a powerful program thanks to its ability not only to predict the heat transfer to a structural element, but also how the structures react to the variations of temperatures under its given load conditions.

Using SAFIR program, complex building structures under fire conditions can be analyzed. The validity of the program has been proved through various references.

References

[1] Franssen J.M. SAFIR. A Thermal/Structural Program Modelling Structures under Fire [J]. *Engineering Journal, A.I.S.C.* Vol 42. (3), 2005.

[2] Chu Thi Binh. Hollow steel section columns filled with self-compacting concrete under ordinary and fire conditions [C]. *PhD thesis*. University of Liege, Belgium, 2009.

[3] Franssen, J.-M. Contributions à la modélisation des incendies dans les bâtiments et de leurs effets sur les structures [C]. *Thèse d'agrégation*. University of Liege, Belgium, 1997.

[4] Talamona D., L. Lim L. & J.-M Franssen**.** Validation of a shell finite element for concrete and steel structures subjected to fire [C]. *4 th Int. Seminar on Fire and Explosion Hazards.* Londonderry, University of Ulster, Sep 2003, pp198-199.

[5] Lim Linus, Andrew Buchanan, Peter Moss, Jean-Marc Franssen. Numerical modelling of two-way reinforced concrete slabs in fire [J]. *Engineering Structures*. Volume 26, Issue 8, July 2004, Pages 1081-1091.

[6] Talamona D., and J.M. Franssen. A Quadrangular Shell Finite Element for Concrete and Steel Structures Subjected to Fire [J]. *Journal of Fire Protection Engineering.* Vol. 15, No. 4, 237-264 (2005).

[7] J.C. Dotreppe, Thi Binh Chu, J.M. Franssen. Steel hollow columns filled with self-compacting concrete under fire conditions [C]. *The 3rd Congress of the International Federation for Structural Concrete (fib) and PCI Conference*. Washington DC, May 2010.