STRUCTURAL RESPONSE UNDER NATURAL FIRE OF BARREL SHAPE SHELL CONSTRUCTION

Michal Malendowski [a], Adam Gglema [a], Zdzislaw Kurzawa [a], Lukasz Polus [a]

[a] Institute of Structural Engineering, Faculty of Civil and Enviromental Engineering, Poznan University of Technology, ul. Piotrowo 5, 60-965 Poznan, Poland

Abstract

In the paper thermo-mechanical analysis of the structural shell construction with natural fire scenario is presented. The barrel shape coverlet with dimensions 40m x 80m is the roof system for the shopping arcade. The braced shell structure is made of steel rectangular hollow sections. The steel construction is directly covered by special glass system. The interior fire case is considered as the main goal of research. Fire is simulated with the use of FDS software based on computer fluid dynamics. Then the temperatures and/or heat fluxes are transferred to the non-linear Abaqus finite element software system. The 3D geometry FE computational model is prepared using 3D beam finite elements with mechanical and thermal degrees of freedom. CFD and FE analyses are sequentially coupled using special external own scripts. The analyses show influence of the natural fire onto the structural behaviour of the roof.

Keywords: fire safety, fire engineering, CFD, FEA, natural fire, coupled problems.

INTRODUCTION

Nowadays a big attention is paid to provide certain level of people's safety in case of fire. So that, especially in large buildings, several active and passive protection systems are installed simultaneously, to assure assumed safety. Number and types of fire protection measures usually came from prescriptive regulations, what leads often to overestimation of needs, and subsequently to excessive costs. From this reason, following paper refers to performance based approach for designing of structures in accidental situation of fire. Particularly, the thermo-mechanical analysis of the braced barrel vault made from rectangular hollow sections in natural fire is provided. Thanks to such formulation, structural response of that construction is approximated more accurately and because of these informations, it is possible to find the golden mean between costs of protection and necessary fire resistance of analysed structure.

1 OBJECT OF ANALYSIS

Analysed structure is the barrel shape vault having dimensions of 40 m x 80 m in plane and about 15 m height. In project, it is a part of bigger leisure and shopping centre, which connect two other parts of this complex of buildings. In this case fully triangulated system has been used in the construction of single-layered braced vault in steel. The steel bar triangles are made from the hollow section which are fixed to each other using special connectors. These connectors allows to make the assembly process satisfactory simple, and after erection they work as rigid connections between bars. Every 25 meters long special spatial truss diaphragms support the coverlet and ensure the spatial stability of whole vault. Fig. 1 shows the isometric view of analysed construction.

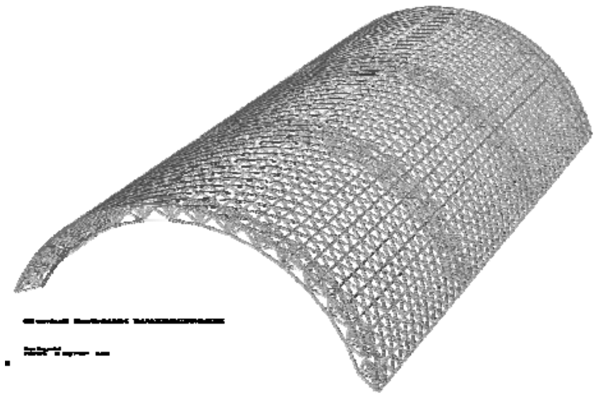


Fig. 1 Isometric view of FEM model

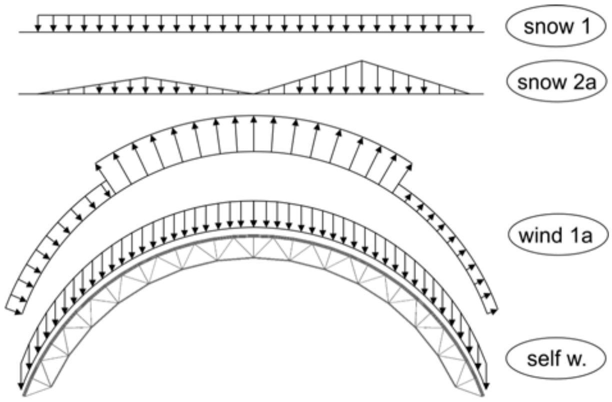


Fig. 2 Types of loads

2. ACTIONS

According to PN-EN 1990 (Polish version of Eurocode) the following combination of loads are distinguished:

Tab. 1 Combination of actions

No.	Permanent	Leading accidental action	Variable actions
1.	1.0 · self_weight	Net heat fluxes comes from natural fire analysis	0.2 · snow_1
2.			0.2 · snow_2a
3.			0.2 · snow_2b
4.			0.2 · snow_1 + 0.2 · wind_1a
5.			0.2 · snow_2a + 0.2 · wind_1a
6.			0.2 · snow_2b + 0.2 · wind_1a

Types of variable actions are presented in Fig. 2. Loads having subscript “a” having their symmetric equivalent “b”. The net heat flux to all members of structure are calculated using formulae (3.1) with (3.2) and (3.3) from PN-EN 1991-1-2. To catch whole heat coming into structure, adiabatic surface temperature from natural fire analysis is substituted into those equations. Then temperatures of members are calculated in accordance to equation (4.24) from PN-EN 1993-1-2 like for unprotected internal steelwork.

3. CFD ANALYSIS

Computational fluid dynamics analyses were made in FDS (Fire Dynamics Simulator). These analyses are the topic of another paper. For mechanical analysis one fire scenario was chosen and data resulted from analysis of that particular fire scenario are transferred to mechanical model. To do that, in FDS, at surfaces creating analysed vault, boundary data were collected. The quantity of particular interest was adiabatic surface temperature. Thanks to that, both convective and radiative heat flux could be described by one quantity (Wickstrom et.al 2007).
The important property of FDS is possibility to use just rectangular geometry. Therefore it is worth to mention that data got from FDS must be treat as some estimation, especially when in this case real geometry of structure is barrel-shape.
FDS is collecting boundary data in so called boundary files. Special scripts are used to read data from these files.

4. FEM MODEL

Finite element method calculations are made in Abaqus. Model is composed of 3D beam elements B31 which are able to carry on both large deformations and large strains. Thermal response of these elements is included by thermal strains caused by additional internal forces resulted from temperature gradient along restrained elements. Time step in analysis is set up

to max. 50 sec, so implicit integration is adopted. During analyses, equilibrium at each time increment is found using the Newton-Raphson method. Both material and geometrical non-linearities are taken into account.
Full model is prepared using external scripting interface in Scilab, which is free equivalent for Matlab. Thanks to such process of model preparation, authors have got full control on model properties, and all necessary operations such as: applying both static and thermal boundary conditions, setting the constrains, applying loads, setting the direction of cross-sections, etc..

5. TRANSFEREING DATA FROM CFD INTO FEM ANALYSES

Temperature history is transferred from FDS results, collected in FDS boundary files, first into Scilab where necessary operations on these data are carried on. At start, comparison of CFD and FEM models have to be done. So some actions on coordinate systems in both models are executed to place FEM model mesh in right place of bigger FDS model. After that's, scripts, already having right coordinates of FEM model, automatically search the nearest nodes from FDS boundary files, and from those, maximum adiabatic surface temperature is chosen. So at the end, temperature history for each node is distinguished.
Because adiabatic surface temperatures got from CFD analysis have got significant noise, it is approximated by multi-linear function. To do it, for each temperature-time curve, minimization problem is solved in the least square sense (Malendowski, 2012).
Next, section temperatures are calculated using Eurocode procedure (formula (4.24) from PN-EN 1993-1-2), with substitution of adiabatic temperature-time curve resulted from above. Such prepared section temperatures are transferred into Abaqus model as boundary conditions at each node. The whole procedure of transferring data between CFD and FEM is presented in Fig. 3.

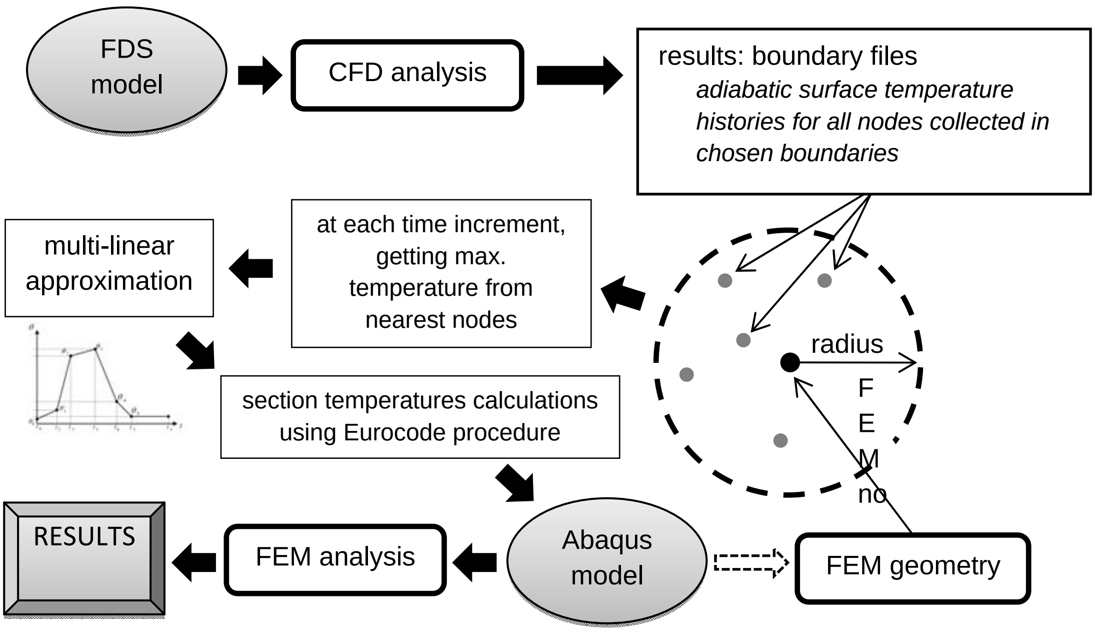


Fig. 3 Data transfer procedure

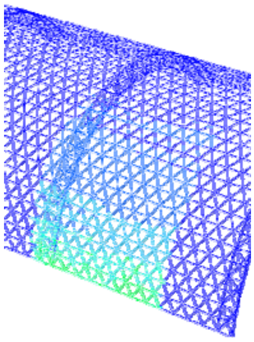
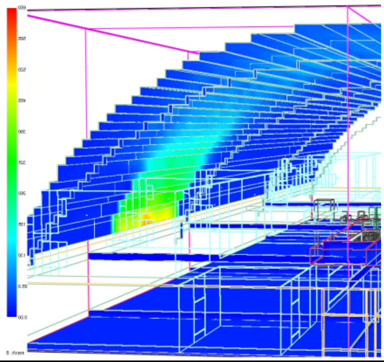
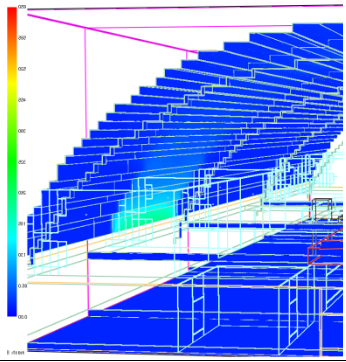
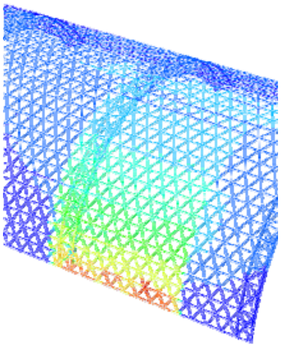
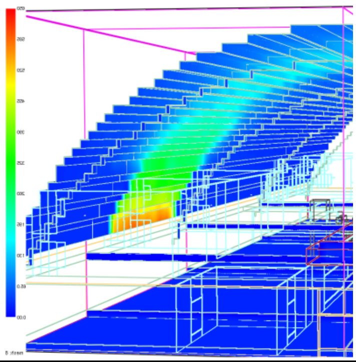
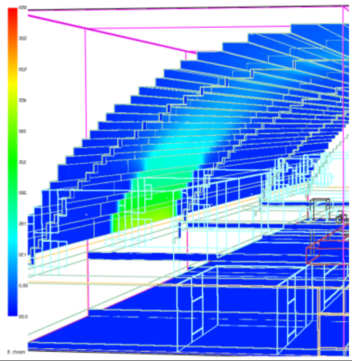
6. RESULTS

Results from analyses are collected and distinguished between temperatures of element's nodes, nodal displacements and plastic strains in elements.

6.1 Temperatures

In Tab. 2 colour-maps of three comparable quantities are shown. It can be seen that members temperatures calculated according to Eurocode procedure, using adiabatic surface temperature is higher than temperature of surfaces (wall temperatures) from FDS results.

Tab. 2 Comparison between temperatures given in FDS results and calculated in accordance to Eurocode's procedure.

time	Section temperature (EC calc. with AST)	Adiabatic surface temperature (FDS)	Wall temperature (FDS)
10 min			
40 min			

How scripts calculating and transferring temperature work is shown in Fig. 4 at the example of node with maximum reached temperature.

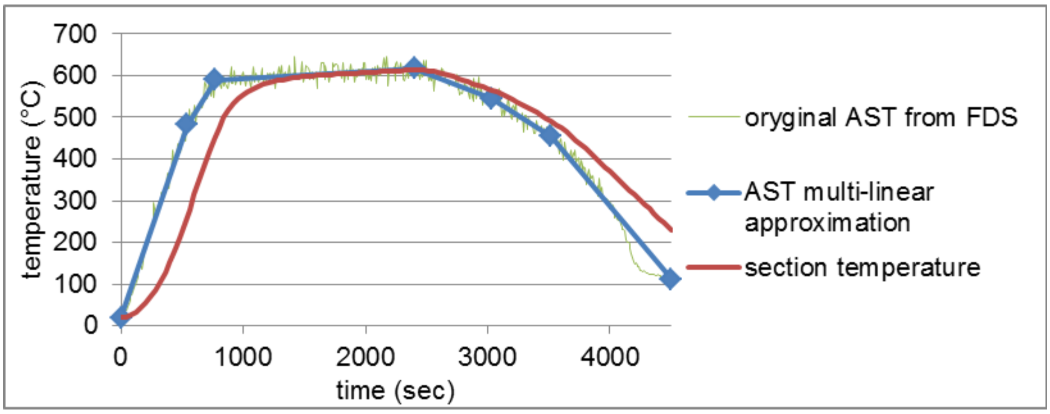


Fig. 4 Temperature history of node with max. temperature

6.2 Displacements

Displacement field at time when maximum temperature is reached, show no observable differences with reference to combinations of actions. The magnitude of total deflection is about 15 cm in all cases. Also shape of deformation is similar (Fig. 5).

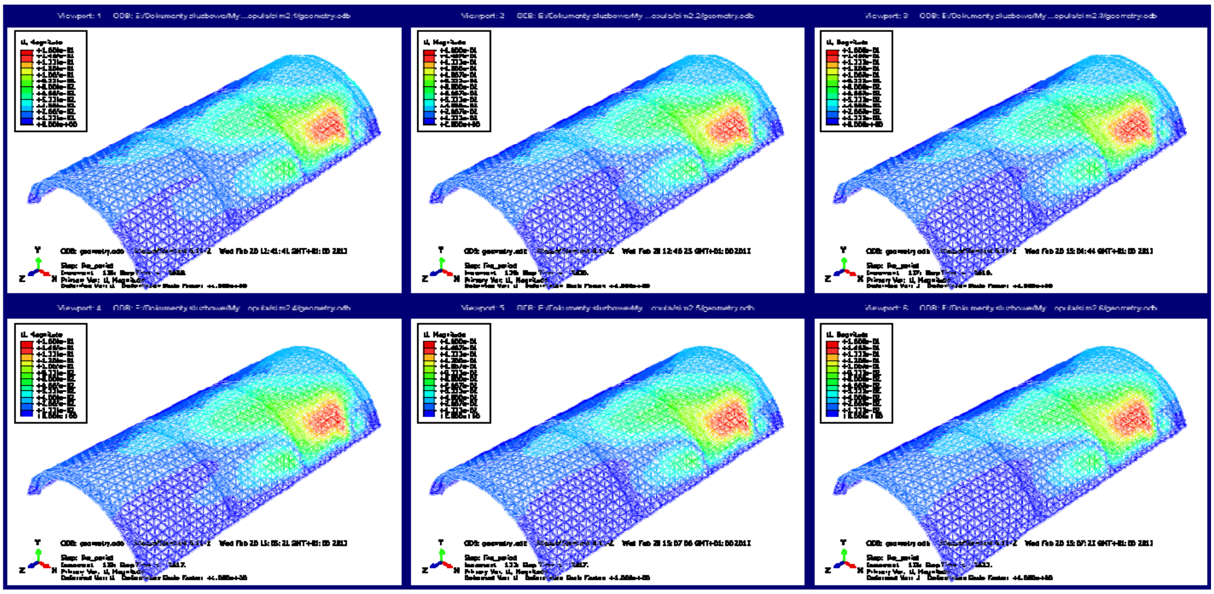


Fig. 5 Shape of deformation for all combinations of actions

6.3 Plastic zones

Because it was found, that there are not significant differences between results for different load combinations, just one map of plastic zones is shown in Fig. 6, particularly for load combination number 1 and for time about 40 minutes.. It can be seen from this picture, that only on longitudinal bars stresses reach yield stress, and plastic strains appear.

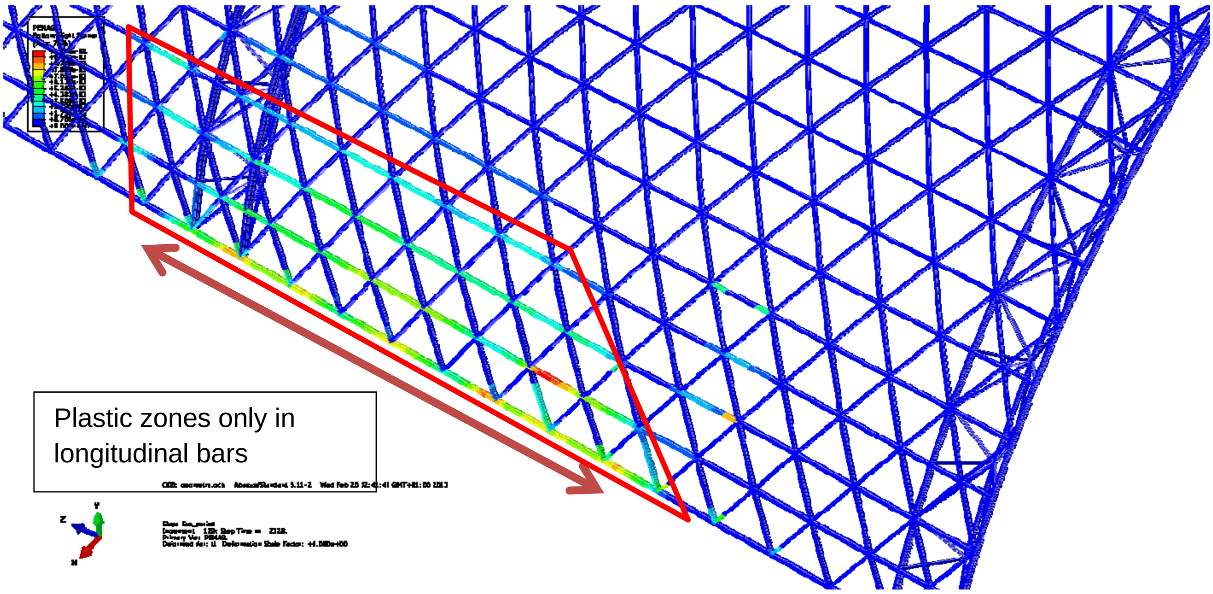


Fig. 6 Plastic zones in bars after 40 minutes of fire

Places where plastic zones propagate coincide with areas having the biggest temperatures. It is worth to mention, that in areas with the biggest temperatures, the largest plastic deformations are near to the support, where structure is most constrained. Supports constrains the structure in longitudinal direction and together with big temperature cause this described phenomena.

7. SUMMARY

In this paper, thermo-mechanical analysis of braced barrel vault structure in fire was presented. Analyses were done by solving the coupled CFD-FEM problem, with use of external scripts, which help with model preparation and transferring data between those two different computational approaches.

Proposed procedure shows how complicated is process of satisfactorily accurate calculation of construction in natural fire. The biggest problem comes from fact, that it needs multi-physical approach, which up to now must be done by coupling different codes (here: FDS and Abaqus).

Results from these analyses show clearly that during fire, thermal action have to be necessarily taken into account, because in situations where construction is several times statically undetermined, thermal elongation of restrained members cause huge damage.

It was shown, that the most suffering members are not those where the biggest deflection was observed, but those which were the most constrained.

Nevertheless, the whole fire course did not threat the overall structural safety of construction. Even if in analysed case always the maximum temperature was chosen and members were threaten as unprotected, global redistribution of forces and stiffness of construction result in safe design from structural point of view. Analysed vault experienced significant thermal deformations, but only a finite number of bars were unable to carry on loads.

Additional finding from these work came from analyses of temperatures. In designing of such big open complexes like upper one, it is unreasonable to use standard ISO curve to describe temperature history. In none of points temperatures reach those from ISO curve.

REFERENCES

Eurocode 0, PN-EN 1990, Basic of structural design. European Committee for Standardization, 2004 (in Polish).

Eurocode 1, PN-EN 1991-1-2, Actions on Structures Part 1-2 General Actions – Actions on structures exposed to fire, European Committee for Standardization, 2002 (in Polish).

Eurocode 3, PN-EN 1993-1-2, Design of steel structures Part 1-2 Structural fire design, European Committee for Standardization, 2005 (in Polish).

Wickstrom U., Duthing D., McGrattan K., Adiabatic surface temperature for calculating heat transfer to fire exposed structures, International Interflam Conference XI, 2007.

Kevin McGrattan et al., Fire Dynamics Simulator (Version 5) User’s Guide, NIST, 2007.

ABAQUS, User's Manual.

Scilab, Scilab Consortium - Digiteo Foundation (www.scilab.org)

Malendowski M., Approximation of the noisy temperature data got from CFD analysis and its influence onto FE analyses, Fire Safety of Construction Works Conference, ITB, Warsaw 2012.