Khan, M.A., Cashell, K.A., Usmani, Analysis of restrained composite perforated beams during fire using a hybrid simulation approach. Journal of Structural Engineering (United States), 146 (3). pp. 04020002 - 04020002. ISSN: 0733-944

ANALYSIS OF RESTRAINED COMPOSITE PERFORATED BEAMS DURING FIRE USING A HYBRID SIMULATION APPROACH

Mustesin Ali Khan[1], Katherine A Cashell[2], Asif S. Usmani[3]

Abstract

This paper is concerned with the behaviour of restrained perforated beams acting compositely with a profiled slab during a fire. These members are increasingly popular in the construction of long-span floor systems as they provide a structurally and materially efficient design solution and provide space for placement of building services. However, their response during a fire has received little attention from the research community until recently. In the current work, a hybrid simulation-type numerical approach is adopted using a combination of the OpenSEES, ABAQUS and OpenFresco software. The accuracy of the model is validated using available fire test data whereby the temperatures measured during the experiments are directly applied in the numerical model at various locations. The effect of axial and rotational restraint due to the connections between the beams and columns is also investigated following validation of the model. Furthermore, the hybrid simulation approach is employed to study a number of salient parameters, including load ratios, material grade and the location of the openings. The variation in axial force during the fire is also examined. Various failure modes are observed during the analysis including flexural and shear failure, failure of the web-post, concrete crushing and also a Vierendeel mechanism. The fire resistance of the analysed beams is compared with the values obtained from the most common design codes. Due to the consideration of restraint forces, which are not included in the design codes, the resistances predicted by the finite element simulations are more favourable. It is found that the location of the openings along the span and also the boundary conditions have a considerable effect on the time-displacement behaviour, axial reactions, web-post buckling behaviour as well as the fire performance of the perforated beam.

[1] PhD Candidate. Department of Civil and Environmental Engineering, Brunel University London, London, UK. e-mail: mustesin.khan@brunel.ac.uk

[2] Senior Lecturer. Department of Civil and Environmental Engineering, Brunel University London, London, UK. e-mail: katherine.cashell@brunel.ac.uk

[3] Professor. Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong. e-mail: asif.usmani@polyu.edu.hk

INTRODUCTION

In modern construction of steel framed buildings, there is an increasing tendency towards the specification of long-span floor systems which enable larger open plan spaces to be achieved. Perforated or cellular beams composite with a reinforced concrete floor slab are a popular choice for such floor systems. Perforated beams can be made either by cutting and welding hot-rolled steel sections to provide the desired shape or by fabricating the section from steel plates. The openings may be designed in any shape but the most popular ones are circular (giving a cellular beam), rectangular, elongated and sinusoidal openings. These are separated by solid web-posts and the web-post size varies according to the dimensions of the openings. Generally, perforated beams are preferred in multi-storey buildings to regular I sections as they allow for longer spans to be achieved which, in turn, leads to more flexible column-free space and shorter erection times. In addition, they can reduce the overall height of the building owing to the integration of services within the structural frame. It has been shown that using these types of beam can be very economical and reduce the total weight of steel work significantly (Nadjai et al. 2017).

The behaviour of structures in fire has been the subject of intensive research in recent decades (British Steel Plc 1999; Dwaikat and Kodur 2011; Li and Guo 2008; Liu et al. 2002) and is particularly topical at the current time following the building fire at Grenfell Tower in London in 2017. Due to the many complexities involved, most research focusses on the response of isolated structural elements, without necessarily including the whole structure in the analysis, usually idealising the effect of surrounding elements. There have been considerably more analytical studies and fire tests on simply supported cellular beams (Ellobody and Young 2015; Nadjai et al. 2007, 2016; Wong et al. 2009), compared with those that are axially and/or rotationally restrained. This is despite the fact that the majority of composite perforated beams in practice experience some degree of both axial and rotational restraint, and their behaviour is very dependent on the type and magnitude of the restraint provided by the surrounding structure.

Large-scale structural fire tests are rare because they are very costly and require specialized experimental facilities. Most of the experimental research into the behaviour of structures in fire has been carried out on isolated structural components, exposed to standard fire curves, in order to compare the fire performance under similar testing conditions. However, they do not represent real fires and building elements such as beams, floors, walls and columns are usually examined without taking into account the effect of the surrounding structural components. This is particularly important when a statically indeterminate structural assembly is subjected to fire because it experiences indirect loadings due to the restrained thermal deformations, i.e., a compressive force is induced in the member when it begins to expand but is restrained from doing so. Upon failure of the fire exposed member in an indeterminate structure, the load is

transferred to other members; this load redistribution may save the structure from collapse. In light of the benefits and challenges of structural fire testing, hybrid simulation has emerged as a promising technique in that it combines testing of isolated components with simultaneous coupled simulation of the global response.

In this context, the current paper is focused on the fire behaviour of restrained perforated steel beams which are acting compositely with a profiled slab. One of the largest studies into restrained perforated beams under fire conditions was done by researchers at the University of Ulster and the Czech Technical University (Wald et al. 2011; Nadjai et al. 2011, 2017). This work included two full-scale fire tests. These were very informative tests which monitored many of the variables which affect the response. However, end displacements at the ends of the beams were not measured during the tests although these are important parameters in terms of the overall building behaviour as well as the ultimate fire resistance of these members. Najafi and Wang (Najafi and Wang 2017a; b) developed both a numerical and analytical model to investigate the behaviour of restrained cellular beams at elevated temperature. In this study, the level of axial restraint provided was assessed and considered, however the influence of a composite slab was not included nor was the effects of rotational restraint.

A simplified analysis of a frame comprising cellular beams exposed to fire was performed by Abu (Abu et al. 2009). In this study, the effect of the openings was incorporated by considering an equivalent web thickness, resulting in a solid beam. Although this was a useful investigation in many ways, this approach cannot predict the failure modes associated specifically with perforated beams such as web-post buckling and the Vierendeel mechanism. In general, there is a lack of data on the behaviour of restrained composite perforated beams exposed to fire. Accordingly, the work presented in the current paper aims to bridge this gap and study the behaviour using a hybrid simulation-type numerical approach. This modelling method is employed because it is capable of analysing the whole structure in an accurate yet computationally efficient manner. The structure is divided into two sub-sections, or assemblies, and the area which is expected to behave nonlinearly and undergo large deformations is modelled in fine numerical detail in one assembly whilst the rest of the structure which behaves elastically is modelled in another assembly at a much lower computational cost. A middleware or interaction software such as UI-SIMCOR or OpenFresco (Kwon et al. 2007; Takahashi and Fenves 2006) is used at the interface to transfer data from one assembly to the other and vice versa. To date, hybrid simulation is most commonly used in earthquake engineering although it is very useful for studying the effect of other extreme events, such as a fire, as will be presented in the current paper.

VIRTUAL HYBRID SIMULATION

In this section, hybrid simulation and virtual hybrid simulation are described, with a particular emphasis on their use for analysing structural response in fire. This is followed by a description of the virtual hybrid simulation approach that has been developed in the current work.

Background to Hybrid Simulation and Virtual Hybrid Simulation

Hybrid simulation (HS) is most commonly employed by earthquake engineers to study the seismic behaviour of structures. In this computational method, the structure is divided into two assemblies or substructures. The assembly which is expected to experience large deformations or whose seismic performance needs to be evaluated in fine detail is tested physically in the laboratory and is known as the physical substructure (PS). The rest of the structure is modelled using a standard FE software in the other assembly which is referred to as the numerical substructure (NS). Both assemblies interact with each other at each time step of the response using a communication software e.g., OpenFresco. In summary, performing a hybrid simulation involves the following steps:

1. The NS is started and calculates the displacements and rotations at the interface for the first time-step.
2. These displacements and rotations are applied to the PS in the laboratory through the actuators, which are directly connected to the FE model (NS).
3. The reaction forces and moments as a result of the applied displacement and rotations in the physical test are recorded at the interface of the PS and then fed back to the NS to perform the next integration step and determine the new set of displacements and rotations. These steps are repeated until the end of the test.

Using this technique, it is possible to apply real boundary conditions owing to the surrounding structure to the PS in the laboratory and obtain an accurate depiction of the whole system behaviour. Although this approach is relatively common in seismic engineering, there are few examples in the literature of hybrid simulation being applied to analyse structures exposed to fire. Mostafaei (Mostafaei 2013) conducted a hybrid analysis of a reinforced concrete frame, including a fire test of the first floor central column. The rest of the structure was modelled in the non-linear finite element software SAFIR (Franssen and Gernay 2017). Utilising the symmetry of the structure, the substructures interacted with each other through manual control of the axial force at the column ends. The interaction between the physical and numerical substructures was not automatic but was user-controlled, meaning that the user paused the physical test every five minutes to log the numerical data and then the simulation was re-started. The accuracy of this approach was compromised due to the manual nature of the test.

More recently, Whyte et al. (2016) performed the first fully-automated hybrid simulation in fire. The experiment was carried out in the elastic range on a scaled specimen which was exposed to temperatures up to 200°C. While it was the first fully-automated hybrid fire simulation, the methodology requires further validation for large-scale specimens subjected to

realistic fires. Another hybrid fire testing method was proposed by Sauca et al. (2018), in which a hybrid simulation is performed entirely in a numerical environment. This method is fully automated and displacement-controlled. In these numerical simulations, the integration model is defined as a stiffness matrix and the substructure model is simulated in the FE software, SAFIR.

In further developments, Wang et al. (2018) conducted a hybrid fire simulation by testing a central ground-floor column in the laboratory whilst simultaneously modelling the surrounding structure in ABAQUS. To transfer the data between two substructures, a network interface program called NICON was employed (Zhan and Kwon 2015). Similar to other arrangements, this test also utilised the symmetry of the structure and the substructures interacted with each other through the control of axial forces and displacements at the column interface.

The stability of hybrid simulation in a numerical environment is rarely compromised during the data exchange between the two substructures. However, the accuracy of the model can be affected by the size of the time step and also the mesh size and structure. The time step used in both assemblies should be sufficiently small to avoid instability without compromising the efficiency of the model. A large time step may also lead to noise in the solution. The mesh employed in the slave assembly is also important. An irregular mesh may result in development of stress concentrations and lead to unrealistic results. Moreover, the mesh size should be small enough to capture all important behavioural features of the model with reasonable accuracy while minimising the computational effort.

In a real hybrid fire test with a physical specimen, many factors can introduce instabilities during the test. Since the fire exposure is time dependent, the time interval required to perform the calculations for the NS and to apply the interfacial actions and displacements to the PS may become a source of error, because while the interface data is calculated, the PS continues to be exposed to fire loading. Therefore the calculated interface data may not be in agreement with altered state of the PS and the equilibrium and compatibility requirements at the interface may not be satisfied. The time interval required to apply the target values on the PS therefore need to be carefully estimated and implemented to avoid instability in real HFT. The other most common factor affecting the results of hybrid simulation are the experimental errors associated with the data transfer system. A poorly tuned transfer system may induce errors by not being able to impose the calculated response quantities accurately, which is not an issue for VHFT.

Hybrid simulation in a real fire testing scenario is very complex, particularly if it is necessary to apply manual control to more than one interface. There are up to six unknowns (i.e. degrees of freedom in the NS and forces and moments in the PS) requiring control and communication at each interface. Manual control of these quantities increases the complexity of the tests and can introduce error into the process. So, before conducting a hybrid simulation in fire it is necessary to establish a hybrid simulation framework in a numerical environment. This means

replacing the physical testing element of the hybrid simulation with another numerical model which uses high resolution elements such as 3D shell and brick elements to create a so-called virtual hybrid simulation approach. The successful implementation of a virtual hybrid simulation framework eliminates the requirement for manual involvement between the two assemblies and this system can then be employed with a physical substructure in place of the detailed FE model in future work. In this approach, a multiple number of responses can be controlled and communicated at the interface between the two assemblies.

Development of the Virtual Hybrid Simulation Method

The current paper is concerned with the development of a virtual hybrid simulation framework which has been developed using both the ABAQUS (ABAQUS 2010) and OpenSEES (McKenna et al. 2009) finite element (FE) analysis software's. There are many different FE packages available for commercial and research purposes which can provide an efficient and inexpensive method for analysing the behaviour of structures exposed to extreme events, such as fire. However, most commercial software does not allow developers to implement their own source code and therefore open-source software such as OpenSEES is advantageous for developing novel simulation techniques such as virtual hybrid simulation.

In the current work, the detailed numerical modelling (replicating the PS in hybrid simulation) is simulated in the ABAQUS software (ABAQUS 2010) in what is termed the slave assembly. ABAQUS is employed as it provides specific capabilities for the high-fidelity material modelling, including capturing nonlinear local failures such as local buckling of the steel section and crushing or cracking of the concrete. The other assembly, referred to as the master assembly, represents the surrounding structure and is modelled in the OpenSEES software using simpler beam-column elements. Traditionally, OpenSEES has been selected as the preferred software for hybrid simulations in earthquake engineering. Building on this experience, OpenSEES has been developed for fire engineering, such that it can accurately model the response of framed structures subjected to realistic fire scenarios (such as travelling fires, for example), which is currently not possible in other FE softwares. Moreover, OpenSEES capability in earthquake response simulation also provides the capability of multi-hazard simulations in future advancements. Therefore, OpenSEES is selected in the current work because it an open-source software and is capable of providing an accurate depiction of the global or system-level nonlinear structural response in extreme conditions such as a fire.

OpenSEES

OpenSEES is an open-source finite element analysis software which was originally developed for earthquake engineering applications at the University of California, Berkeley (McKenna et al. 2009; McKenna 1997). Owing to its open-source framework, OpenSEES is a collaborative programme which allows developers to add components to the original source code depending on their particular needs, and then disseminate these developments to other potential users. It

is constantly being developed by researchers for different types of application (Kolozvari et al. 2018; Zhu et al. 2018), including structural fire engineering (Jiang et al. 2015; Jiang and Usmani 2018a; b). OpenSEES is an object-oriented software implemented in the C++ programming language, using the 'Tcl' scripting language.

In the virtual hybrid simulation approach, both the master and slave assemblies interact at the interface nodes at each integration step. This is facilitated using the middleware OpenFresco which transfers the boundary conditions and reaction forces between the master and slave assemblies. A super element and an adapter element are defined in the slave and master assemblies, respectively, to couple the two models (if using the same software for PS and NS) or two software packages. The displacements and rotations that are calculated in the master programme (i.e. representing the whole structure) are implemented as boundary conditions in the slave assembly when investigating the detailed response of the perforated beam under fire conditions. Subsequently, the slave programme estimates the corresponding reaction forces and returns them to the master. This approach presents an alternative to modelling the whole structure in 3D using high resolution elements which is both complex and computationally expensive.

OpenFresco

There are a number of different approaches for enabling the two assemblies to communicate with each other. The most traditional method is to use a file exchange system, which has been implemented in a number studies (Kolozvari et al. 2018; Kwon et al. 2007). In this system, the trial displacements and rotations are firstly estimated by the master FE code at the interface and saved in the form of data files. Then, the saved quantities are applied as boundary conditions in the slave assembly and the reaction forces at the interface node are estimated. These responses are also stored in a similar way in data files. Finally, the stored reaction forces are applied at the interface node in the master assembly to determine the new displacements and rotations for the next time step. The above steps are repeated for each integration time step until the end of the analysis. Although the file exchange system provides an effective means for transferring data between the two assemblies, the main disadvantage is that the two FE models cannot run concurrently and therefore it is not very computationally efficient.

In the current study, the data is transferred between the two codes using a middleware software called OpenFresco. Using this method for data communication between the two programmes enables both FE codes to run simultaneously, without restarting the analysis at every time step. This reduces the complexity and enhances the computational efficiency of the process, compared with the file exchange system. Accordingly, OpenFresco facilitates the storage, transformation and transfer of data between the master and slave assemblies.

Thermomechanical Analysis in OpenSEES

Elevated temperature finite element analysis is conducted in OpenSEES in two stages. First, a heat transfer analysis is conducted in which a fire is applied to the structure in the form of a time-temperature relationship and then the resulting temperatures at various points in the structure are determined. Secondly, the temperature history data from the heat transfer analysis is used in the thermo-mechanical stress analysis to assess how the structure responds to the applied loads during a fire. Both the thermal strains and the temperature-dependent material properties are considered in the thermo-mechanical analysis. In the current study, there is no requirement to conduct a heat transfer analysis as the temperatures at various points in the structure are taken directly from the test data and input into the thermo-mechanical model. The class hierarchy in the implementation of the thermo-mechanical element is illustrated in Fig. 1, where 'Material', 'SectionForce Deformation', 'Element' and 'ElementalLoad' are all abstract (or base) classes and the others such as 'UniaxialMaterial' and 'NDMaterial' are the derived classes. The beam-column element used in this study to model the structural frame is formulated based on interpolation of the displacement or force. A range of uniaxial material classes are available in the OpenSEES and the mechanical and thermal properties employed in the current study are taken from the Eurocodes (EN-1994-1-2 2005; EN 1992-1-2 2004; EN 1993-1-2 2005).

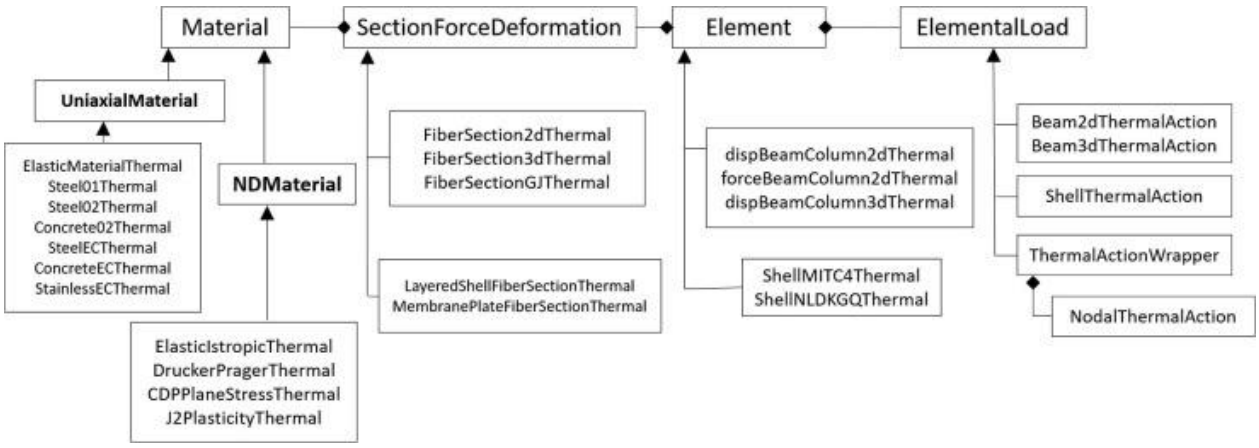
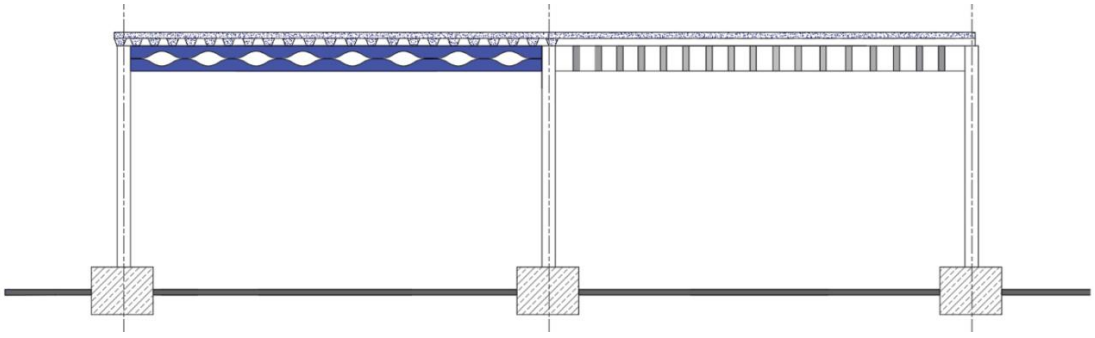


Fig. 1. Classes developed in OpenSEES for thermo-mechanical analysis

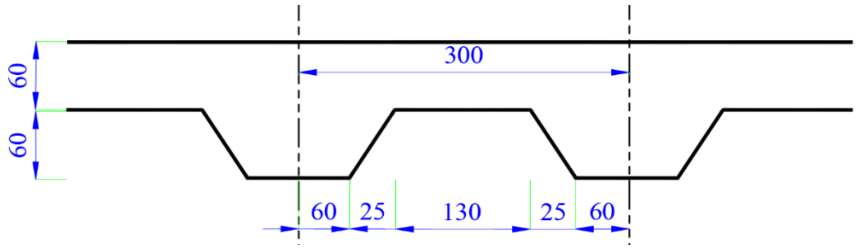
IMPLEMENTATION OF THE VIRTUAL HYBRID SIMULATION APPROACH

The virtual hybrid simulation approach is validated using the fire test which was conducted on an administrative building in Mokrsko by the Czech Technical University (Wald et al. 2011). The experimental structure represented one floor of an administrative building which was 12 × 18 m in floor area, with a height of 4 m. This included a composite cellular beam which was subjected to fire, known as the AS4 composite cellular beam. The schematic for this test is shown in Fig. 2 (a). Beam AS4 was made using an IPE270 I-beam in grade S235 steel which was cut to create sinusoidal openings giving an overall depth of 395 mm. The concrete slab had an overall depth of 120 mm, comprising a flat portion and ribs which were 60 mm each in

depth, as shown in Fig. 2 (b). The compressive strength of the concrete was 32.5 MPa with a mass density of 2400 kg/m^3. The slab reinforcement was located at the mid-depth of the flat portion of the slab, was 5 mm in diameter, and was positioned at 100 mm centres in both directions. It had a yield a strength of 500 N/mm^2 and the cover distance to the edge of the concrete was 20 mm. IPE 400 sections were used for the edge beams, which were also in grade S235 steel, whilst the columns were made using HEB 180 sections. The adjacent beams were made using a WTB 500 section made from grade S320 steel. The test was performed in two stages. Firstly, a mechanical load of 5.6 kN/m^2 was applied uniformly to the floor, representing a load ratio of 0.26. Then, the load was maintained at a constant level whilst the elevated temperature was applied as shown in Fig. 3 (Wald et al. 2011).



(a)



(b)

Fig. 2. (a) Schematic for Mokrosko fire test (b) Profiled deck slab (all dimensions are in mm)

In order to represent this test arrangement in the virtual hybrid simulation, the fire exposed perforated beam is modelled as slave structure in ABAQUS using a high resolution mesh of 3D shell elements for perforated steel beam and 3D solid elements for the concrete slab whilst beam-column elements are utilised for the rest of the structure in the master assembly in OpenSEES. The schematic for both the master and slave assemblies is shown in Fig. 4. This approach investigates the behaviour of the fire-exposed perforated beam while including the effect of remaining structure without modelling the whole system in 3D. So, it provides a computationally efficient methodology for analysing the whole structural system in fire. It is noteworthy that in the Mokrosko fire test, the slab lost its resistance in compression and no two-way tensile membrane action was reported to have developed. Although it is intuitively advantageous to model the whole structure in as much detail as possible, to trace the real

behaviour of structures in fire, it can add unnecessary complexity. In this case, modelling the entire slab up to its perimeter accounting for two-way action is not required as no two-way action developed, and therefore one-way composite beam action is assumed in the model for the sake of simplicity.

The steel beam is represented using S4R shell elements which are available in the ABAQUS library, and each element is 25 × 25 mm in size, based on a mesh sensitivity study. The concrete slab is modelled using C3D8R solid elements while the reinforcement is modelled using T3D2 truss elements. The steel decking is not included in the model as it is not expected to have a significant influence on the fire behaviour. The connection between the perforated beam and the deck slab is represented using the tie constraint in ABAQUS. In terms of material modelling, it is assumed that the steel behaves in an elastic-plastic manner whilst the concrete damaged plasticity model is employed to represent the concrete. The stress-strain relationship for both concrete and steel at elevated temperature is adopted from the Eurocode (EN 1992-1-2 2004; EN 1993-1-2 2005). For the thermal properties, constant values of 0.000014 /°C and 0.000009 /°C are used for the coefficient of thermal expansion of steel and concrete, respectively. The remainder of the frame comprising the adjacent beam (WTB 500) which is modelled as section IPE 400 (section used for edge beams) and columns are modelled as section HEB 180 using 3D beam-column elements in the master assembly in OpenSEES.

At the interface between the master and slave assemblies, OpenFresco requires that an adapter element and a super element are defined in the slave and master assembles, respectively. For the 2-node adapter element in the slave assembly, the nodes at the left and right of the perforated beam are constrained as shown in Fig. 4. This element effectively applies the boundary conditions at the interface degrees of freedom of the perforated beam. A very high stiffness value of 1×10^{12} N/mm is employed for the diagonal element of the stiffness matrix to ensure that the boundary conditions are accurately represented (Schellenberg et al. 2008). The kinematic compatibility of the approach has been described in more detail elsewhere (Khan et al. 2018). Similarly, a 2-node super element is defined at the interface nodes in the master assembly. The initial stiffness matrix for the super element is determined by applying a unit displacement to the perforated beam model in ABAQUS and measuring the resulting force. As in the test, the analysis is performed in two stages, with a mechanical load applied first, followed by the fire loading.

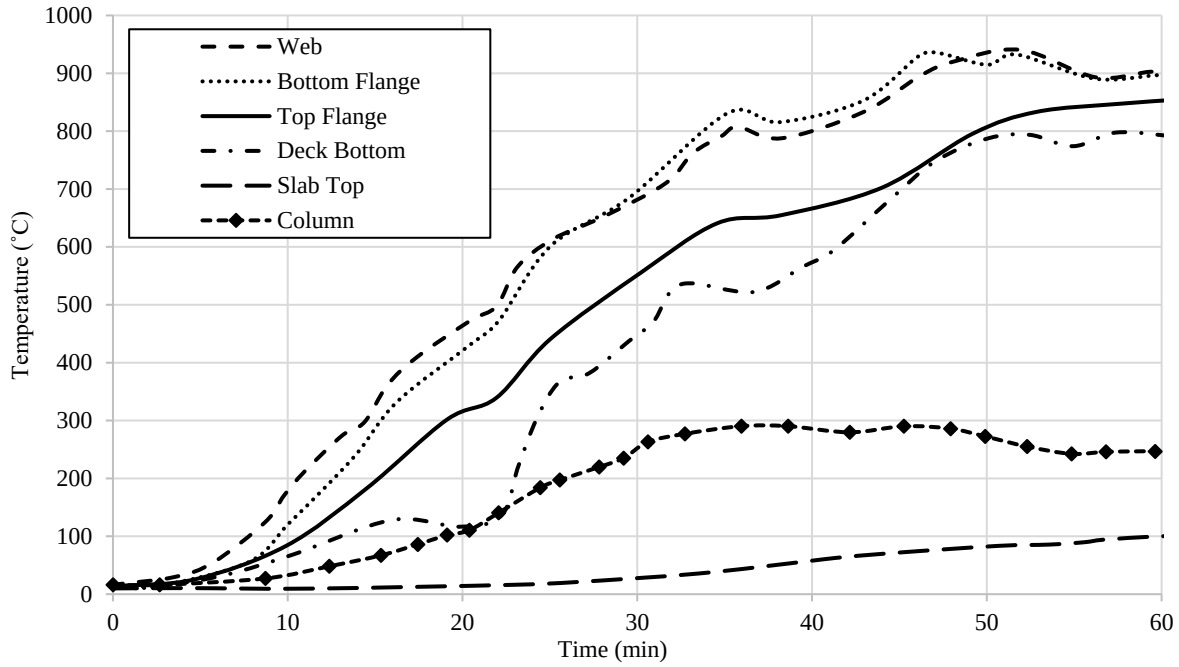


Fig. 3. Temperature profile at the various locations (Wald et al. 2011)

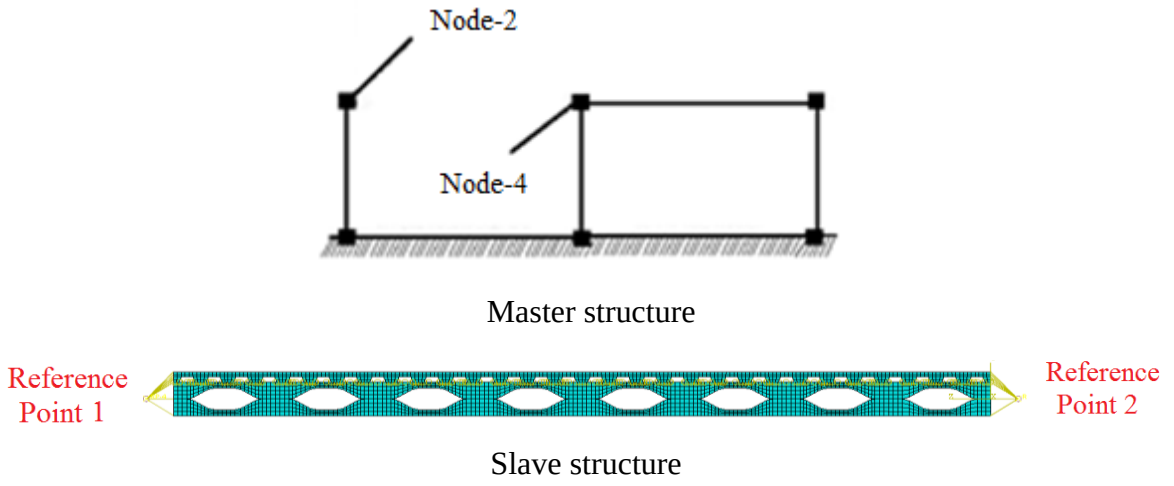


Fig. 4. Sub-structuring for hybrid simulation

During the simulation, firstly the displacements at the interface nodes are determined in the master assembly. Then, these displacements are communicated and applied at the interface nodes in the slave assembly. The reaction forces are calculated in the slave assembly and applied at the interface nodes of the master assembly for the determination of the new displacement. The communication between the two codes and the development of the model is presented in Fig. 5 and outlined as:

Step 1: The master programme analysis commences and generates a displacement vector for the global trial displacements (u_{super}) at the interface degrees of freedom. u_{super} is then sent to the super element by the master assembly.

Step 2. The TCP/IP socket (where TCP/IP means a Transmission Control Protocol/Internet Protocol which is the basic communication language or protocol for the internet) is used to send u_{super} to OpenFresco and the simulation server process begins.

Step 3. Other objects i.e., the 'LocalExpSite' and 'ExperimentalSetup' classes (see Fig. 5) are used to store and transform $u_{super.}$ Data transformation is not required in this study because no physical specimen (i.e. laboratory test specimen) is used. So, the 'NoTransformation' object as 'ExpSetup' is utilised.

Step 4. The u_{super} element is then transferred to the 'ExperimentalControl' object which feeds the trial displacement vector to the adapter element in the slave assembly, using a TCP/IP socket. A resultant displacement vector is formed by the adapter element by combining u_{super} with its own elemental displacements. Subsequently, the resultant element force vector (P_{adpt}) is also updated by the adapter element.

Step 5. Upon convergence in the slave programme, the negative resultant force vector ($-P_{adpt}$) is communicated to the 'ExperimentalControl' object through the TCP/IP socket. 'LocalExpSite' and 'ExperimentalSetup' objects are again utilised for the storage and transformation of the force vector (see Fig. 5).

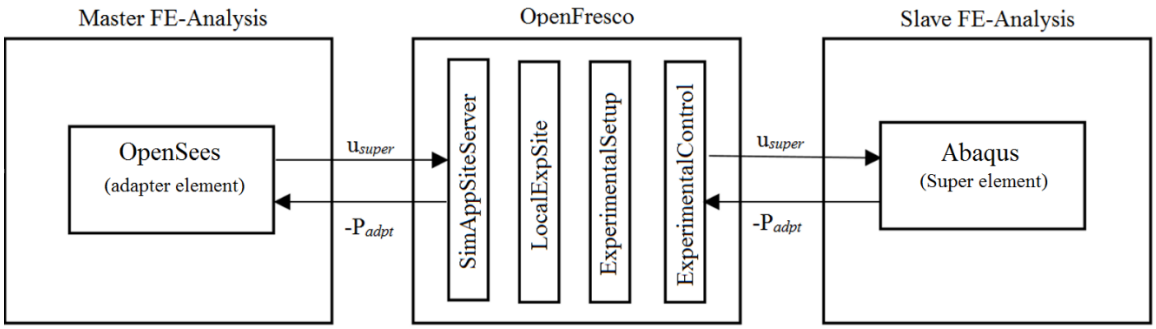


Fig. 5. Sequence of operations and data exchange

Step 6. The force vector is transferred to the super element in the master programme by 'SimAppSiteServer' object through the TCP/IP socket.

Step 7. $-P_{adpt}$ is saved as element forces by the super element and returned to the master integration method for the next step. The new trial displacement vector is calculated by the master programme and Step 1 to Step 7 are repeated until the analysis is complete.

VALIDATION AND PARAMETRIC STUDY

In this section, the virtual hybrid simulation approach is applied to the experiment conducted at Mokrsko and described previously (Wald et al. 2011). Fig. 6 presents the deformed shape predicted by the finite element simulation. The images show that during the fire, the bottom flange of the beam displaced laterally which caused the steel web to bend. Meanwhile, the top flange was restrained by the composite slab. The midspan vertical deflection of the restrained

composite perforated beam (AS4) as predicted by the hybrid simulation is plotted against time and compared with the test results in Fig. 7 and it is shown that a good agreement is obtained. Fig. 8 presents the variation in axial reaction during the fire, as predicted by the virtual hybrid simulation (measurements from the test are not available). Initially, it is shown that the beam experiences axial compression (indicated by a positive value in Fig. 8) due to restrained thermal expansion which increases until the onset of local buckling in the steel section. Subsequently, the compressive forces reduce due to the significant changes in material stiffness at high temperature, and the axial forces change from compression to tension as the full cross-section goes into tension and behaves like a tensile catenary.

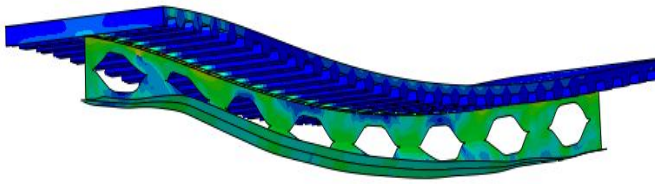


Fig. 6. Deformed shape of the perforated test beam

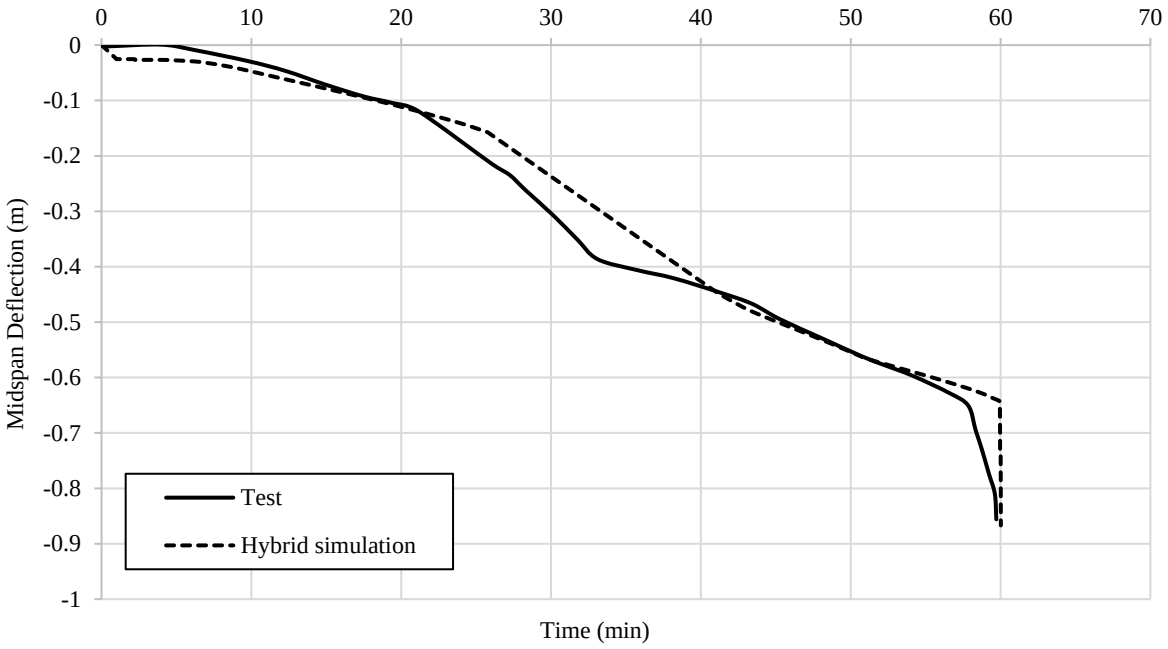


Fig. 7. Vertical deflection of the AS4 beam during the fire

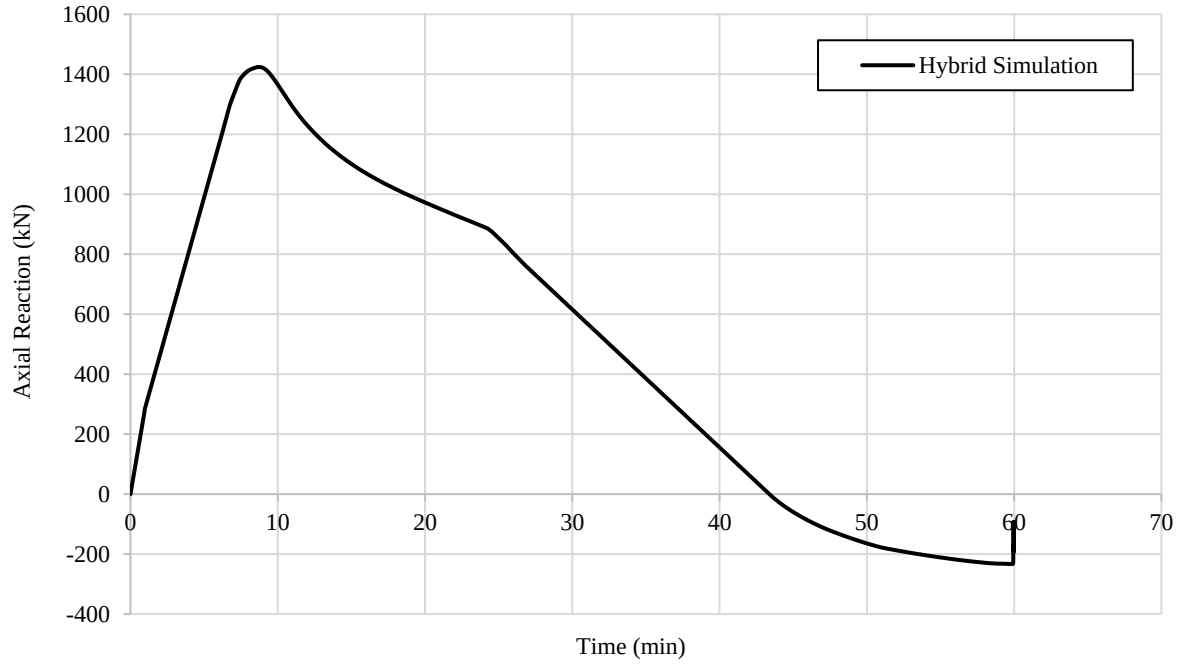


Fig. 8. Axial reaction at the beam support, with time, for beam AS4

Following this validation, the model is employed to conduct an extensive parametric study to analyse the response of restrained composite perforated beams in fire. The effect of load ratio, steel strength and opening position on the behaviour is studied. The various parameters considered in the study are summarised in Table 1, including three different load levels (namely LL1=3.1 kN/m^2, LL2=4.3 kN/m^2 and LL3=5.6 kN/m^2, respectively) and three grades of steel (with yield strengths of 275, 450 and 550 N/mm^2, respectively). Four opening layouts are considered in the parametric study which are presented in Fig. 9, and described as follows:

Layout 1: Openings in the centre of the beam (bending zone);

Layout 2: Openings at 250 mm from the one end (one shear zone);

Layout 3: Openings at 250 mm from both the ends (both shear zones); and

Layout 4: Combination of layout 1 and layout 3 (bending and shear zone).

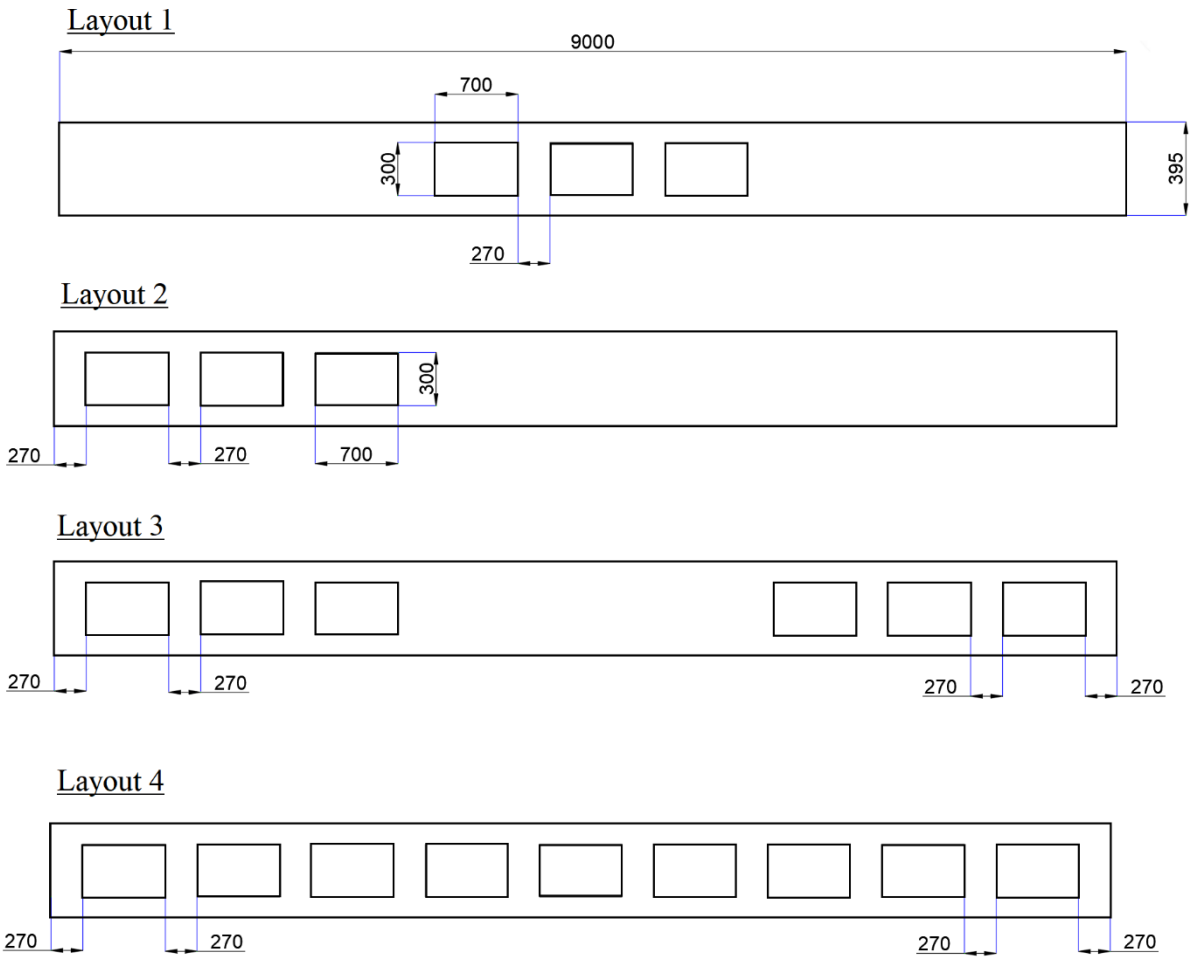


Fig. 9. Schematic of opening layout (all dimensions are in mm)

It is noteworthy that the load levels mentioned above result in different load ratios for each beam in the parametric study as the member capacity is related to the opening arrangement, steel grade, etc. The parametric study is divided into twelve different groups for ease of analysis. Each group comprises 3 simulations, all of which have the same opening layout and steel grade but a different applied load level (i.e. LL1, LL2 or LL3). Accordingly, groups 1, 5 and 9 correspond to beams with opening layout 1 and steel strength of 275, 450 and 550 N/mm^2, respectively. Similarly, groups 2, 6 and 10 investigate opening layout 2 whilst groups 3, 7 and 11 study layout 3 and groups 4, 8 and 12 contain beams with opening layout 4. All other material and geometric properties of the beam and surrounding structure are identical to that described earlier in the model validation and the Mokrsko AS4 test.

Table 1. *Composite frames investigated in the parametric study*

Group No.	Beam	Opening Layout	Steel Grade	LL
Group 1	Beam-1	Layout 1	275	LL1
	Beam-2	Layout 1	275	LL2
	Beam-3	Layout 1	275	LL3
Group 2	Beam-4	Layout 2	275	LL1
	Beam-5	Layout 2	275	LL2
	Beam-6	Layout 2	275	LL3
Group 3	Beam-7	Layout 3	275	LL1
	Beam-8	Layout 3	275	LL2
	Beam-9	Layout 3	275	LL3
Group 4	Beam-10	Layout 4	275	LL1
	Beam-11	Layout 4	275	LL2
	Beam-12	Layout 4	275	LL3
Group 5	Beam-13	Layout 1	450	LL1
	Beam-14	Layout 1	450	LL2
	Beam-15	Layout 1	450	LL3
Group 6	Beam-16	Layout 2	450	LL1
	Beam-17	Layout 2	450	LL2
	Beam-18	Layout 2	450	LL3
Group 7	Beam-19	Layout 3	450	LL1
	Beam-20	Layout 3	450	LL2
	Beam-21	Layout 3	450	LL3
Group 8	Beam-22	Layout 4	450	LL1
	Beam-23	Layout 4	450	LL2
	Beam-24	Layout 4	450	LL3
Group 9	Beam-25	Layout 1	550	LL1
	Beam-26	Layout 1	550	LL2
	Beam-27	Layout 1	550	LL3
Group 10	Beam-28	Layout 2	550	LL1
	Beam-29	Layout 2	550	LL2
	Beam-30	Layout 2	550	LL3
Group 11	Beam-31	Layout 3	550	LL1
	Beam-32	Layout 3	550	LL2
	Beam-33	Layout 3	550	LL3
Group 12	Beam-34	Layout 4	550	LL1
	Beam-35	Layout 4	550	LL2
	Beam-36	Layout 4	550	LL3

All of these analyses are conducted in two stages whereby the mechanical load is first applied to the whole structure in both the OpenSEES and ABAQUS models and this is then followed by the application of the thermal load in the detailed ABAQUS model of the perforated beam. The same fire model (i.e. time-temperature relationship) as used in the model validation previously described, is applied in all cases. Accordingly, the temperatures obtained from the AS4 beam test are applied to all the groups as shown in the Fig. 3.

RESULTS AND DISCUSSION

In this section, the results of the parametric study using virtual hybrid simulation are presented and discussed. Particular attention is given to analysing the fire resistance, vertical displacement, axial restraint, local buckling and deflected shape in order to understand the behaviour and assess the influence of the most critical parameters.

Transition Time and Temperature

The transition time refers to the point in the response at which the axial force in the beam changes from compression to tension, as observed in Fig. 8. For a typical restrained perforated beam in bending, as load is applied at ambient temperature, the top tee-section and slab

experience compression forces whereas the lower part of the beam goes into tension. With the addition of fire loading, initially the restraint against thermal expansion results in an increase in the compressive forces in the top of the beam and a reduction of the tensile forces in the lower portion. Then, as the deflection increases and due to the thermal gradient across the depth of the section, compressive arching occurs and the whole cross-section is in compression. Local failure may occur either if the bottom tee-section buckles or yields under this compressive stress or through the formation of a Vierendeel mechanism, depending on the position of the opening; both of these local mechanisms are presented in Fig. 10 for the 4 opening layouts. As deflections increase further, the compressive forces begin to reduce in the section. Web-post buckling and web buckling due to lateral movement of the lower flange may develop and the axial force in the beam changes from compression to tension. The time at which this occurs is known as the transition time and the temperature at which this change takes place is the transition temperature.

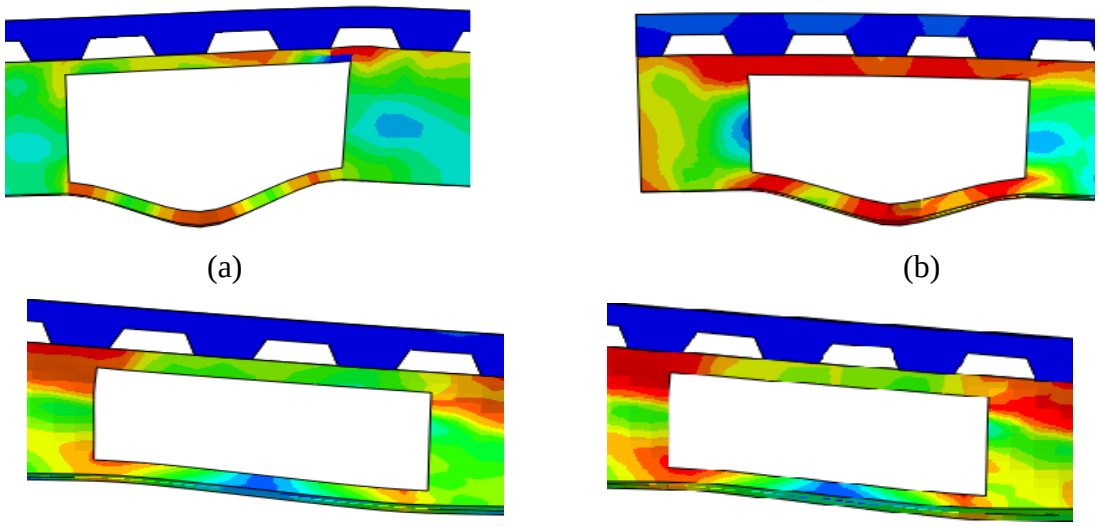


Fig. 10. Local failure modes for beams made with steel S275 for opening layout (a) 1, (b) 2, (c) 3 and (d) 4

Load Level

Fig.11 presents the mid-span deflection versus time responses for beams in Group 1 to Group 4 with different opening layouts and load levels whilst Fig. 12 illustrates the axial force response against time for the same beams. With reference firstly to Fig. 11, it is observed that in general the overall behaviour remains unchanged irrespective of the load level. The response is quite different depending on the opening layout and each of the figures in Fig. 11(a) to (d) presents differently shaped curves (this will be discussed further in Section Opening Layout). As expected, an increase in the load level results in greater mid-span deflections for a given time, as well as an earlier transition time and ultimate failure. It is observed that for layout 2, 3 and 4, presented in Fig. 11(b) to (d), respectively, the beam deflects in a downward direction

from the beginning of the analysis. However, for layout 1 (Fig. 11(a)), the beam initially deflects upwards for all load levels. As shown previously in Fig. 3, the temperature in the bottom flange increases much more rapidly compared with the temperature of the web, top flange and the slab.

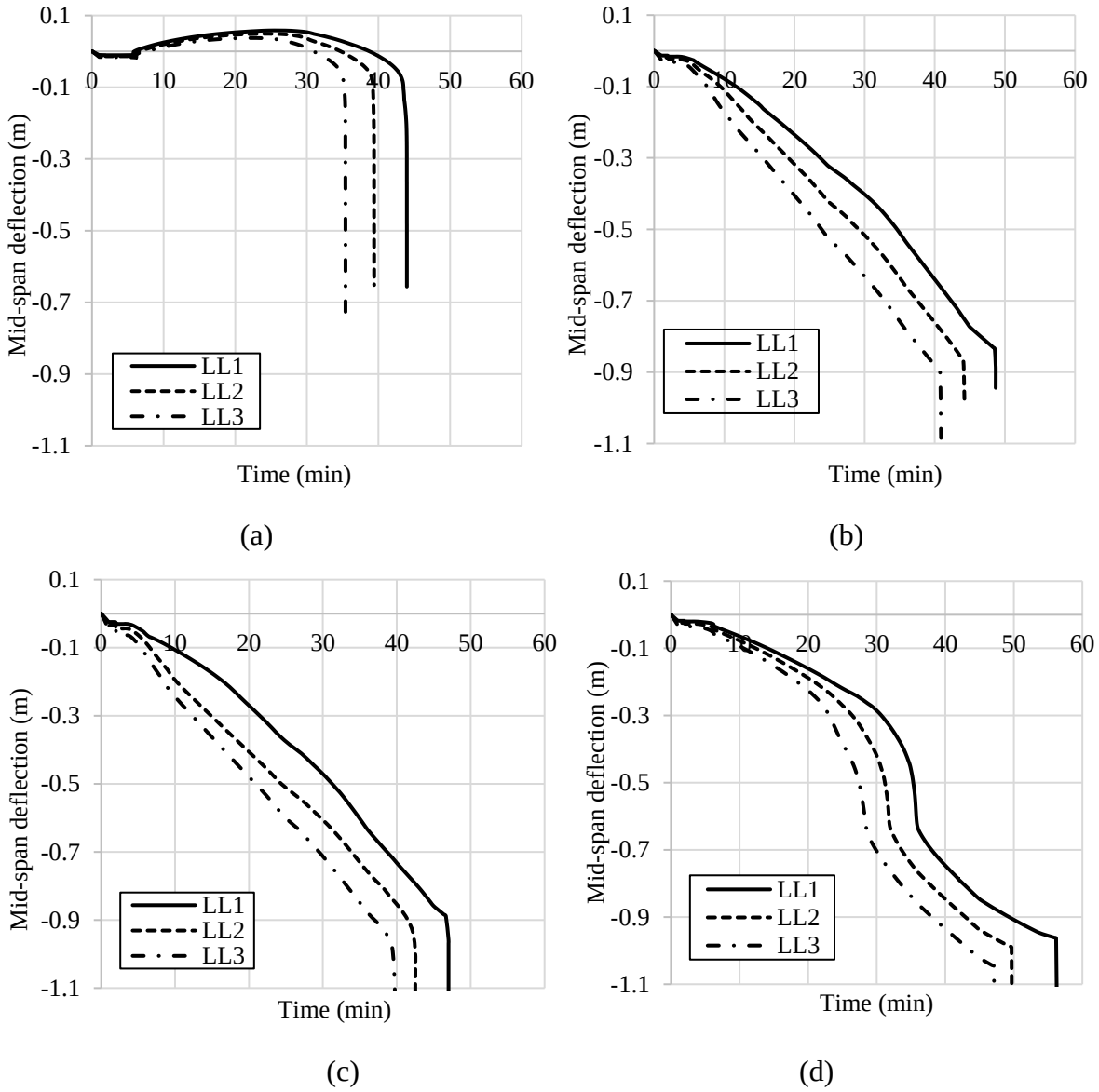


Fig. 11. Time-deflection behaviour for beams made with steel strength S275 and opening layout (a) 1, (b) 2, (c) 3 and (d) 4.

As a result, initially, the compressive force in the bottom tee due to the restrained thermal expansion is greater than that in the top tee and slab causing a hogging moment to develop with corresponding upward deflections. For layout 1, the opening is at the centre of the beam and therefore the section is weaker at this location and the beam tends to deflect in an upward direction as it initially behaves somewhat like two separate cantilever beams under hogging moments. Arching action also develops in the beam and the applied load induces more

compressive forces in the section. Owing to this, the bottom tee section yields and the web-post begins to buckle causing the beam to eventually deflect in a downward direction.

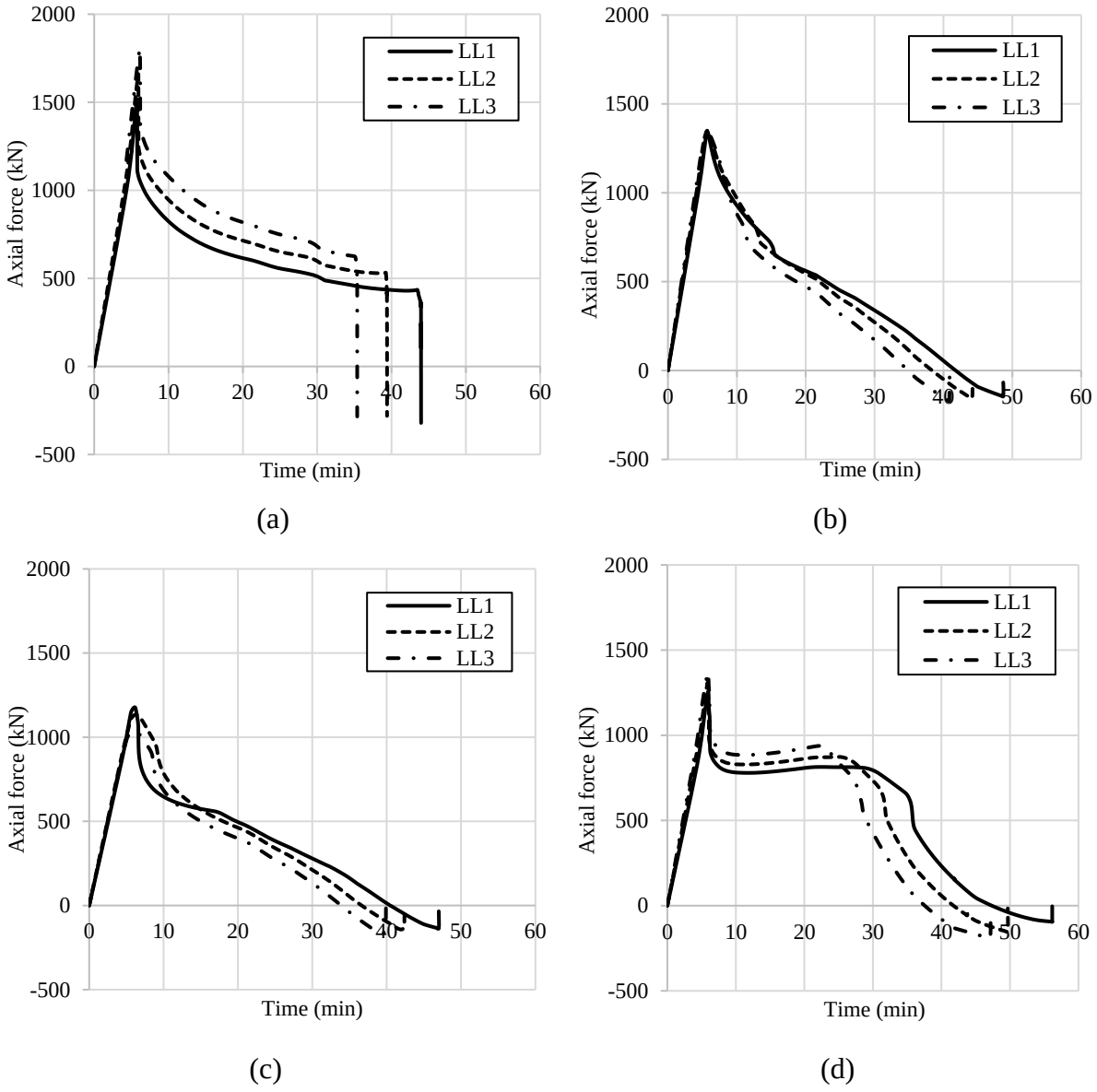


Fig. 12. Development of axial force for beams made with S275 at different load level and opening layout (a) 1, (b) 2, (c) 3 and (d) 4.

With reference to Fig. 12, similar behaviour is observed for all perforated composite beams with the same opening layout, regardless of steel strength and load level applied. In general, it is noteworthy that the peak axial force value in the beams is much greater for layout 1 than for the other opening arrangements. These beams have openings in the bending zone, which results in the development of arching action and a greater axial force. For layouts 2 and 3 which have openings in the shear zone only, arching action does not develop and therefore the levels of axial forces in the beam are lower. For beams with opening layout 4, some arching action does

develop but because the relative stiffness of the section is similar along the whole length of the beam, only a small portion of the hogging moment is transferred to the mid-section, which leads to the development of partial arching action. The local failure modes including web-post buckling, buckling of the bottom flange and the lateral displacement of the web is presented in Fig. 13 for each of the layouts. Buckling of the web-post and lower flange is observed for all the opening layouts due to the narrow width of the web-post. On the other hand, the top flange remains straight and buckling is prevented by the slab. Of the four layouts studied, only layout 4 results in lateral displacement of the web which is attributed to the reduced stiffness of the web in this arrangement owing to the presence of openings along the whole span.

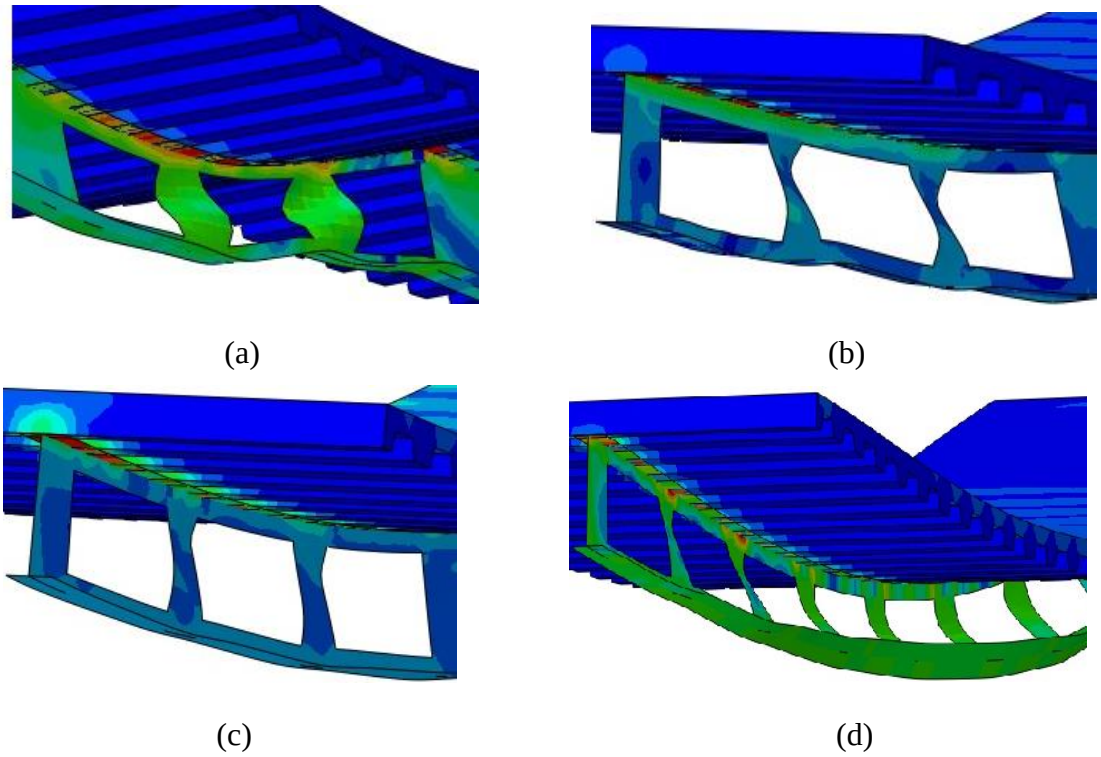


Fig. 13. Web-post buckling and lateral torsional displacement for beams made with S275 and opening layout (a) 1, (b) 2, (c) 3 and (d) 4.

Steel Grade

In this section, the influence of different grades of steel on the overall behaviour is assessed. As given in Table 1, three different grades are included in the study, with yield strengths of 275, 450 and 550 N/mm^2, respectively. Fig. 14 presents the development of axial force in the beam for each steel type, in the 4 different layout arrangements. It is shown that greater axial force develops in the beams made from relatively stronger steel, as expected. Moreover, the beams made from S275 steel start behaving as a tensile catenary earlier than the beams with higher steel strength. The magnitude of the axial reaction provides an indication of the stiffness and strength of the beam relative to the surrounding structure. An increase in steel strength has a positive effect on the web-post buckling and lateral torsional buckling behaviour and the out-

of-plane displacements of the web-post are reduced. It is noteworthy that although there is a difference in the axial force between beams made from S450 and S550, the transition time is almost identical. Therefore, any further increase in the steel strength may not give any improvement in terms of fire resistance for perforated beams.

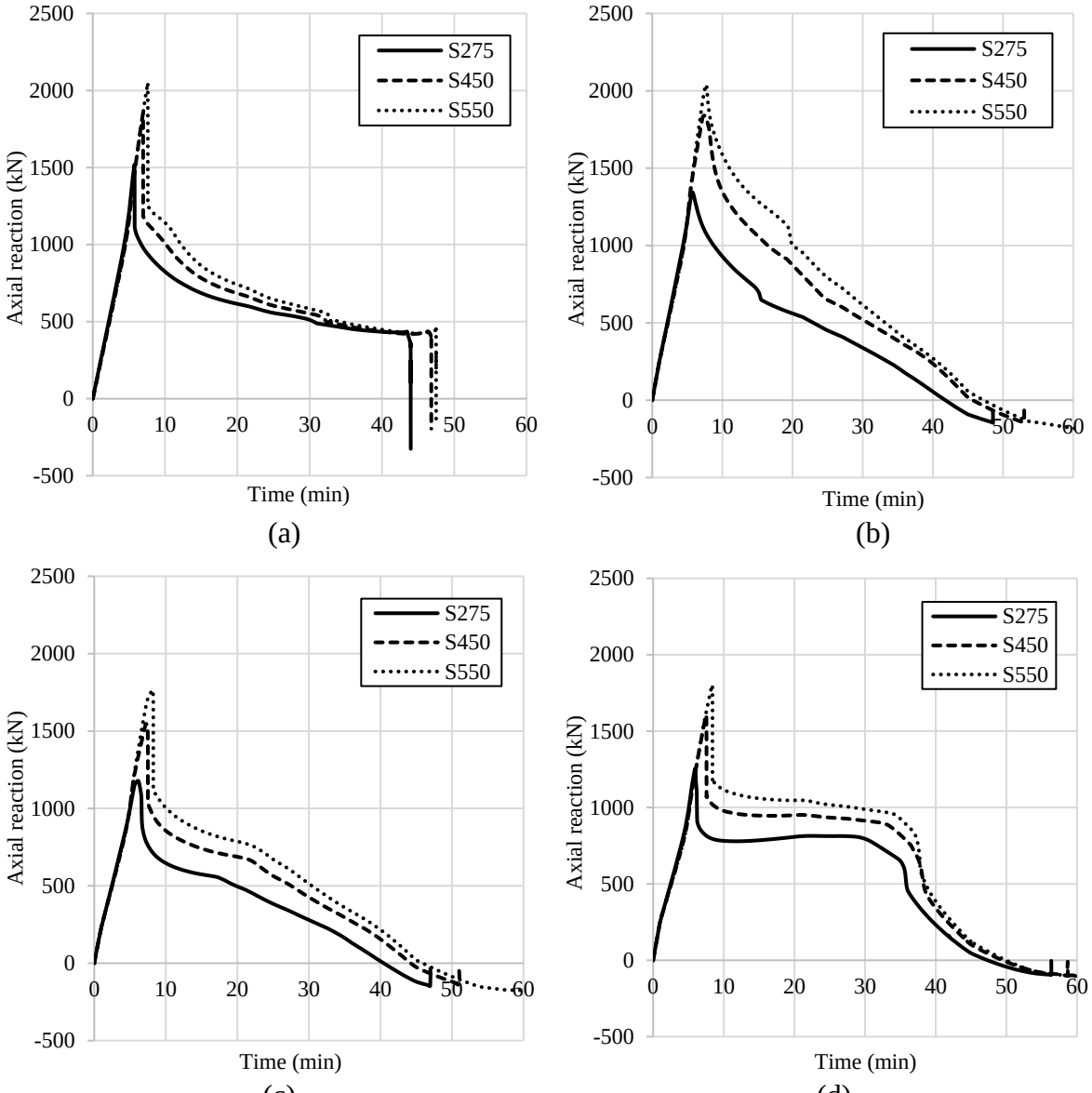


Fig. 14. Development of axial force for beams made with different strength steels and opening layout (a) 1, (b) 2, (c) 3 and (d) 4.

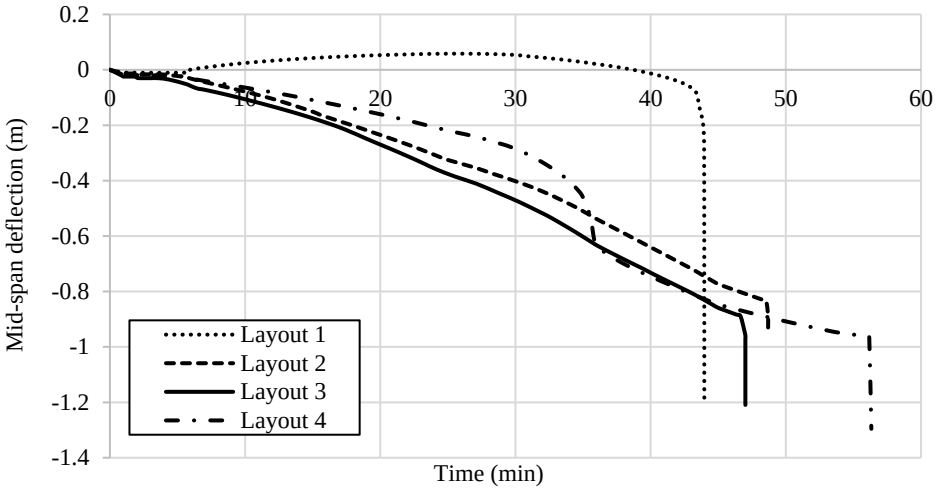
Opening Layout

It has already been presented that the arrangement of openings in the perforated beam can be highly influential to the overall performance in fire and this will be discussed further in this section. Towards this end, Figs. 15 and 16 presents the development of both vertical mid-span deflection and axial force, with time, for perforated composite beams made from S275, S450

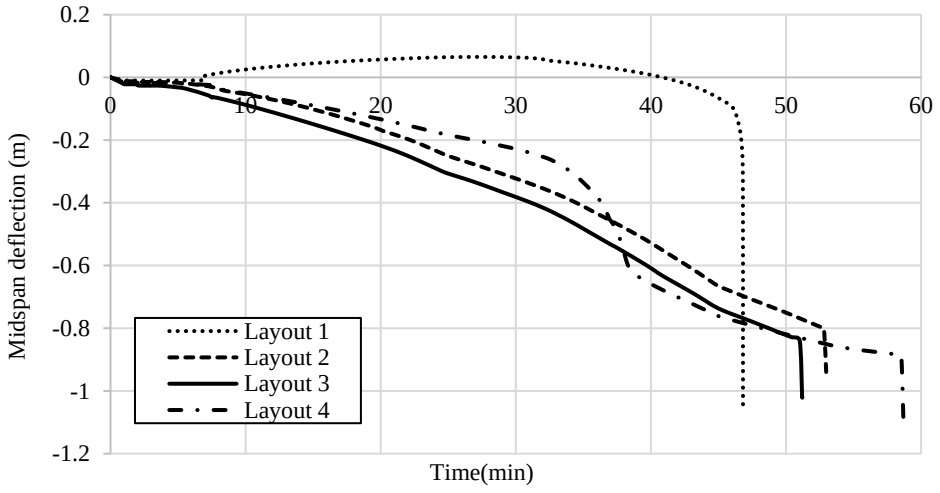
and S550 steel, respectively. From these figures, it is clear that the presence of openings is very influential to the behaviour and a number of observations are made, as follows:

- All beams with layout 1 (which has an opening in the bending zone only) experience an initial upward deflection initially before the beam changes direction and starts moving downwards. This then results in the development of less arching action and a later transition time as well as a lower ultimate displacement, compared with beams that do not have openings in the bending area.
- The area of the openings in layout 1 is identical to that in layout 2 as shown in Fig. 9, but the mid-span deflections are much lower for the former arrangement, irrespective of load ratio or steel strength which is illustrated in Fig. 15. This is because the beam with openings in layout 1 experiences both bending and axial compression and the behaviour is influenced by buckling of the bottom tee as well as web-post buckling. On the other hand, layout 2 only has openings in the shear zone and develops relatively less axial compression in the absence of arching action. The development of an arching action in the beams with layout 1 reduces the midspan deflection considerably.
- For beams that have openings in the bending zone (i.e. layout 1 and 4), the behaviour is governed by the buckling resistance of the bottom tee at the openings. On the other hand, for beams with openings in the shear zone, these must be able to resist Vierendeel bending in addition to bending and axial compression. In the Vierendeel mechanism, transverse shear is transferred across the opening leading to the formation of plastic hinges at the corners of the opening. In the later stages of a fire (i.e. after 15 minutes), the reduction in compression forces in layout 1 and 4 is gradual and the transition time is relatively high compared to Layout 2 and 3, as shown in Fig.16.
- The degree of web-post buckling in layout 1 and lateral displacement of the web for beams with layout 4 is greater than the beams with opening layout 2 or 3 due to the presence of the openings in the bending zone.
- Transition time is an important concept in structural fire engineering as it indicates the fire resistance of the beam in a given fire scenario. For the cases considered herein, at the transition time, the beam is no longer resisting the applied loads through bending action but by acting as a tensile catenary. The transition time is greatest for opening layout 4, followed by layout 1, 2 and 3, respectively. It is clear that beams with openings in layout 1 fail suddenly as shown in Fig. 15 and Fig. 16, whereas all other cases considered in this study continue to carry load through the development of catenary action. For layout 4, the presence of openings along the whole length of the beam helps in distributing the load uniformly across the span, which delays failure, compared with the other layouts. The development of arching action reduces the mid-span deflections and keeps the beam in compression for longer, which delays the transition time and therefore improves the fire resistance. It is

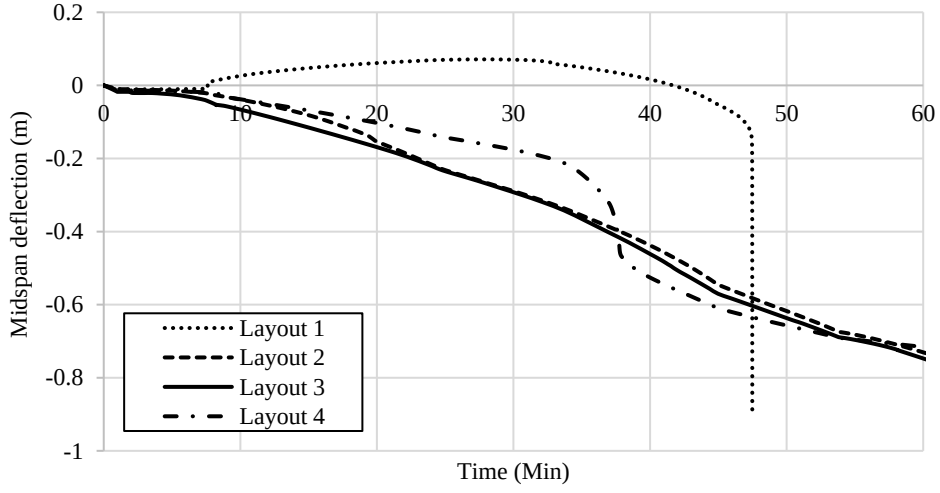
noteworthy that layouts 2 and 3 behave very similarly with similar transitions times, despite layout 3 having an additional set of openings.



(a)

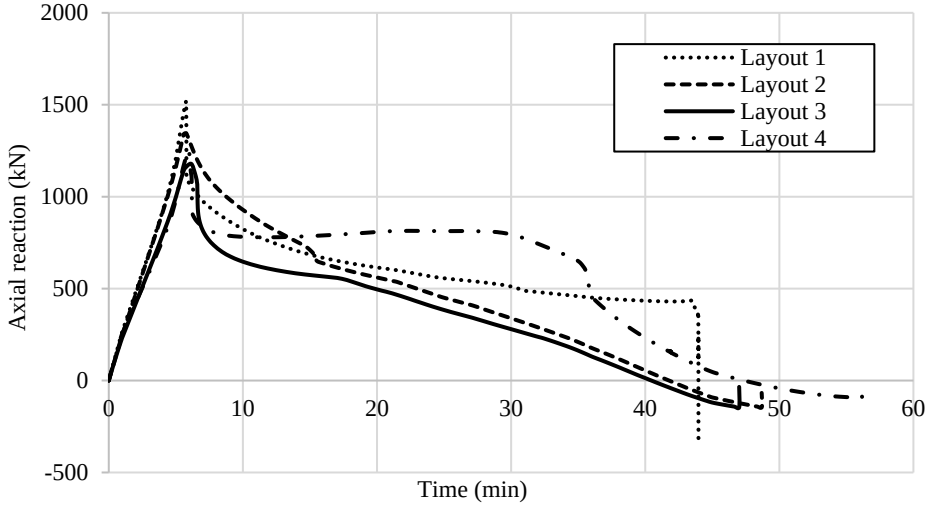


(b)

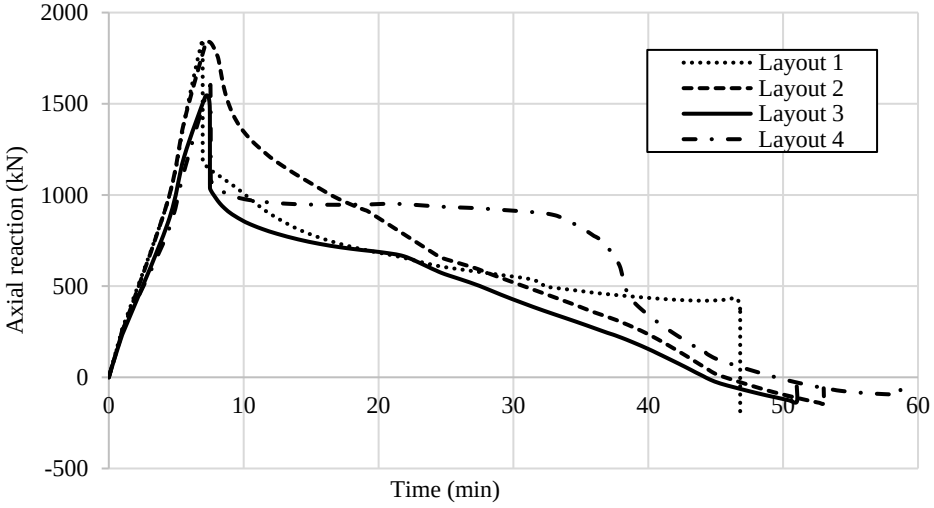


(c)

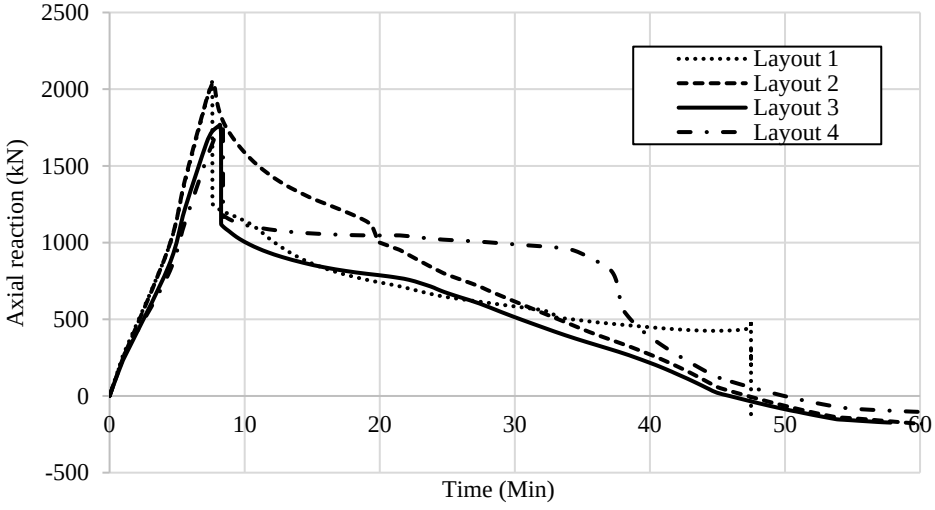
Fig. 15. Midspan deflection variation for all layouts, a) Steel grade 275 N/mm^2 b) Steel grade 450 N/mm^2 c) Steel grade 550 N/mm^2



(a)



(b)



(c)

Fig. 16. Axial reaction variation for all layouts, a) Steel grade 275 N/mm^2 b) Steel grade 450 N/mm^2 c) Steel grade 550 N/mm^2

COMPARISON WITH DESIGN CODES

In this section, the fire resistance of the 36 perforated composite beams presented in Table 1 is compared with the design values obtained using the SCI P355 design guide (Lawson and Hicks 1998) by applying reduction factors from Eurocode 4 (EN-1994-1-2 2005), which covers the design of composite beams with large web openings. As P355 only covers the ambient temperature response, the design equations are modified by applying the elevated temperature reduction factors given in Eurocode 4. The critical temperature is defined as the point at which the beam fails and the failure modes considered are bending, shear, Vierendeel bending and web-post buckling. Once the critical temperature is established, the fire resistance can be estimated.

Table 2 presents the results including the fire resistance predicted by the finite element simulation (Rfi,FEM), SCI P355 (Rfi,P355) and also the ratio of these two values. It is shown that in almost all cases, the finite element analysis predicts that the beams will last for a longer period of time before failure occurs, compared with the design code. It is important to note that the design equations do not account for axial restraint and assume simply-supported boundary conditions. It has been shown in previous studies that restrained beams offer greater fire resistance compared with unrestrained beams (British Steel Plc 1999; Izzuddin and Moore 2002). With reference to Table 2, it can be seen that beams with opening layout 2, 3 or 4 have identical fire resistance according to the design equations because the behaviour is governed by the Vierendeel mechanism and buckling of the web-post in the shear zone. Therefore, these equations do not account for the effect of multiple openings at various locations. On the other hand, the finite element analysis considers all of the influential aspects such as opening shape and location as well as the arching effect due to axial restraint. For opening layout 1, the fire resistance estimated using finite element simulation is lower than the fire resistance computed using design equations. This behaviour is due to the sudden failure of these beams at the onset of the axial force transition from compression to tension. For most of the cases investigated in this study, a close agreement is achieved between the design calculations and the simulations and the slight differences are due to the above-mentioned factors. However for design purposes, it is important to note that the analytical equations provide a conservative estimate of fire resistance.

1 ***Table 2.*** *Finite element analysis results and SCI-P355/EC-4 fire resistance*

Group	Beam	$R_{fi,FEM}$ (min)	$R_{fi,P355}$ (min)	$R_{fi,FEM}$ / $R_{fi,P355}$
Group 1	Beam-1	43	**43**	1
	Beam-2	39	**41**	*0.95*
	Beam-3	35	**37**	*0.95*
Group 2	Beam-4	48	**43**	1.12
	Beam-5	44	**39**	1.13
	Beam-6	41	**33**	1.24
Group 3	Beam-7	47	**43**	1.09
	Beam-8	42	**39**	1.08
	Beam-9	39	**33**	1.18
Group 4	Beam-10	56	**43**	1.3
	Beam-11	49	**39**	1.26
	Beam-12	47	**33**	1.42
Group 5	Beam-13	47	**50**	*0.94*
	Beam-14	43	**47**	*0.91*
	Beam-15	39	**44**	*0.89*
Group 6	Beam-16	52	**50**	1.04
	Beam-17	49	**43**	1.14
	Beam-18	46	**39**	1.18
Group 7	Beam-19	51	**50**	1.02
	Beam-20	47	**43**	1.09
	Beam-21	45	**39**	1.15
Group 8	Beam-22	58	**50**	1.16
	Beam-23	52	**43**	1.21
	Beam-24	48	**39**	1.23
Group 9	Beam-25	50	**60**	*0.83*
	Beam-26	46	**50**	*0.92*
	Beam-27	43	**46**	*0.93*
Group 10	Beam-28	60	**60**	1
	Beam-29	60	**46**	1.3
	Beam-30	56	**42**	1.33
Group 11	Beam-31	60	**60**	1
	Beam-32	55	**46**	1.2
	Beam-33	52	**42**	1.24
Group 12	Beam-34	60	**60**	1
	Beam-35	60	**46**	1.3
	Beam-36	57	**42**	1.36

CONCLUSIONS

This paper has presented the framework for performing a hybrid simulation type approach for the analysis of restrained perforated beams subjected to fire, in a real building. Firstly, a detailed description of the novel virtual hybrid simulation method was presented. This includes the slave assembly, in ABAQUS, and the master assembly, implemented using the open-source software OpenSEES. This virtual hybrid simulation approach offers both accuracy and excellent computational efficiency, particularly for complex problems such as modelling a whole building which is subjected to fire. Hybrid simulation, which includes physical testing, has been extensively applied to seismic engineering applications. However, in order to develop a framework for examining other extreme conditions such as fire, it is convenient to replace the physical testing element with a detailed numerical assembly, as is developed herein.

The virtual hybrid simulation approach is validated for perforated beams subjected to fire using available test data. Thereafter, the model is employed to conduct a parametric study. The key variables included in the study are the opening position, steel strength and load level. During the analysis of the results, it is noted that for axially restrained perforated composite beams, there are a number of distinct stages during the response. Initially, at ambient temperature, the section is in pure bending. Then, as the temperature increases, the whole section starts experiencing axial compression owing to the restraint provided. As soon as initial yielding occurs either by buckling of the bottom tee and web-post or by the Vierendeel mechanism, these compression forces reduce. Beams with openings in the bending zone experience a more rapid unloading compared with beams that have openings in the shear zone only. Thereafter, with increasing deflection, the axial force in the section transitions from compression to tension as the beam behaves as a tensile catenary. It is noted that compressive arching action develops for all beams that have openings in the bending zone which delays the transition time of the beam giving greater fire resistance.

The effect of load level on the fire behaviour of perforated beams has also been studied and it is found that an increase in the load applied to the beam results in a relative reduction in the transition time i.e., a reduction in the fire resistance of the beam. The overall behaviour remains the same with a change in the load level and steel strength. As expected, beams made using a higher strength steel grade demonstrate greater capacity to develop axial forces. Interestingly, beams which have openings throughout their length are shown to offer the greatest fire resistance of those examined in this study. This is due to the fact that the stresses and loads are more evenly distributed across the length and the development of arching action provides a significant mechanism for resisting the applied loads, supplementing the bending resistance. On the other hand, beams which have openings in the bending zone only are shown to offer the lowest fire resistance.

In terms of design, the current design equations which are available for the ambient temperature response have been modified to include the effects of elevated temperature. It is shown that this approach provides a good estimation of the fire resistance for most cases, apart from layout 1 of the studied arrangements. In all other cases, the fire resistance predicted by the virtual hybrid simulation is greater compared with the fire resistance calculated using design equations. This is mainly because the effects of axial and rotational restraint are accounted for in the model. On the other hand, for design purposes, ignoring the effect of the support restraints is shown to provide a conservative estimate of the fire resistance for the range of parameters included in this study.

REFERENCES

ABAQUS. (2010). "ABAQUS Documentation, Dassault Systèmes, U.S.A." Dassault Systèmes.

Abu, A., Block, F., Butterworth, N., and Burgess, I. (2009). "Structural fire engineering assessments of the Fracof and Moksko fire tests: An Engineering prediction." *Application of Structural Fire Engineering*, Prague, Czech Republic.

British Steel Plc. (1999). *The behaviour of multi-storey steel framed buildings in fire*. Swinden Technology centre, Rotherham.

Dwaikat, M. M. S., and Kodur, V. K. R. (2011). "A performance based methodology for fire design of restrained steel beams." *Journal of Constructional Steel Research*, 67, 510–524.

Ellobody, E., and Young, B. (2015). "Nonlinear analysis of composite castellated beams with profiled steel sheeting exposed to different fire conditions." *Journal of Constructional Steel Research*, 113, 247–260.

EN-1994-1-2. (2005). "Eurocode 4: Design of composite steel and concrete structures - Part 1-2 General rules - Structural fire design." *European Commitee for Standardization*, Brussels.

EN 1992-1-2. (2004). "Eurocode 2: Design of concrete structures - Part 1-2: General rules - Structural fire design." *European Commitee for Standardization*, Brussels.

EN 1993-1-2. (2005). "Eurocode 3: Design of steel structures - Part 1-2: General rules - Structural fire design." *European Commitee for Standardization*, Brussels.

Franssen, J. M., and Gernay, T. (2017). "Modeling structures in fire with SAFIR®: Theoretical background and capabilities." *Journal of Structural Fire Engineering*, 8(3), 300–323.

Izzuddin, B. A., and Moore, D. B. (2002). "Lessons from a full-scale fire test." *Proceedings of the ICE - Structures and Buildings*, 152(4), 319–329.

Jiang, J., Jiang, L., Kotsovinos, P., Zhang, J., and Usmani, A. (2015). "OpenSEES Software Architecture for the Analysis of Structures in Fire." *Journal of Computing in Civil Engineering*, 29(1), 04014030.

Jiang, L., and Usmani, A. (2018a). "Towards scenario fires – modelling structural response to fire using an integrated computational tool." *Advances in Structural Engineering*, 21(13), 2056–2067.

Jiang, L., and Usmani, A. (2018b). "Computational performance of beam-column elements in

modelling structural members subjected to localised fi re." *Engineering Structures*, Elsevier, 156(June 2017), 490–502.

Khan, M. A., Jiang, L., Cashell, K. A., and Usmani, A. (2018). "Analysis of restrained composite beams exposed to fire using a hybrid simulation approach." *Engineering Structures*, Elsevier, 172(May), 956–966.

Kolozvari, K., Orakcal, K., and Wallace, J. W. (2018). "New OpenSEES models for simulating nonlinear flexural and coupled shear-flexural behavior of RC walls and columns." *Computers and Structures*, 196, 246–262.

Kwon, O.-S., Nakata, N., Park, K., Elnashai, A., and Spencer, B. (2007). *User Manual and Examples for UI-SIMCOR v2.6 (Multi-Site Substructure Pseudo-Dynamic Simulation Coordinator) and NEES-SAM v2.0 (Static Analysis Module for NEESgrid)*.

Lawson, R. M., and Hicks, S. J. (1998). "Design of beams with large web openings." *Steel Construction Institute*, Berkshire.

Li, G. Q., and Guo, S. X. (2008). "Experiment on restrained steel beams subjected to heating and cooling." *Journal of Constructional Steel Research*, 64, 268–274.

Liu, T. C. H., Fahad, M. K., and Davies, J. M. (2002). "Experimental investigation of behaviour of axially restrained steel beams in fire." *Journal of Constructional Steel Research*, 58, 1211–1230.

McKenna, F., Scott, M. H., and Fenves, G. L. (2009). "Nonlinear Finite-Element Analysis Software Architecture Using Object Composition." *Journal of Computing in Civil Engineering*, 24(1), 95–107.

McKenna, F. T. (1997). "Object-oriented finite element programming: Frameworks for analysis, algorithms and parallel computing." *ProQuest Dissertations and Theses*.

Mostafaei, H. (2013). "Hybrid fire testing for assessing performance of structures in fire - Methodology." *Fire Safety Journal*, Elsevier, 58, 170–179.

Nadjai, A., Bailey, C. G., Vassart, O., Han, S., Zhao, B., Hawes, M., Franssen, J. M., and Simms, I. (2011). "Full-scale fire test on a composite floor slab incorporating long span cellular steel beams." *Structural Engineer*, 89(21), 18–25.

Nadjai, A., Han, S., Ali, F., Alam, N., and Allam, A. (2017). "Fire resistance of axial restraint composite floor steel cellular beams." *Journal of Constructional Steel Research*, 136, 229–237.

Nadjai, A., Petrou, K., Han, S., and Ali, F. (2016). "Performance of unprotected and protected cellular beams in fire conditions." *Construction and Building Materials*, Elsevier Ltd, 105, 579–588.

Nadjai, A., Vassart, O., Ali, F., Talamona, D., Allam, A., and Hawes, M. (2007). "Performance of cellular composite floor beams at elevated temperatures." *Fire Safety Journal*, 42(6–7), 489–497.

Najafi, M., and Wang, Y. C. (2017a). "Axially restrained steel beams with web openings at elevated temperatures, Part 1: Behaviour and numerical simulation results." *Journal of Constructional Steel Research*, 128, 745–761.

Najafi, M., and Wang, Y. C. (2017b). "Axially restrained steel beams with web openings at elevated temperatures, Part 2: Development of an analytical method." *Journal of Constructional Steel Research*, 128, 687–705.

Sauca, A., Gernay, T., Robert, F., Tondini, N., and Franssen, J.-M. (2018). "Hybrid fire testing: Discussion on stability and implementation of a new method in a virtual environment." *Journal of Structural Fire Engineering*, JSFE-01-2017-0017.

Schellenberg, A., Huang, Y., and Mahin, S. A. (2008). "Structural FE-software coupling through the experimental software framework , openfresco." *The 14th World Conference on Earthquake Engineering*, 1–8.

Takahashi, Y., and Fenves, G. L. (2006). "Software framework for distributed experimental-computational simulation of structural systems." *Earthquake Engineering and Structural Dynamics*, 35(3), 267–291.

Wald, F., Kallerová, P., Chlouba, P., and Sokol, Z. (2011). *Fire test on administrative building*. *Steel and Aluminium Structures*, Mokrsko.

Wang, X., Kim, R. E., Kwon, O.-S., and Yeo, I. (2018). "Hybrid Simulation Method for a Structure Subjected to Fire and Its Application to a Steel Frame." *Journal of Structural Engineering*, 144(8), 04018118.

Whyte, C. A., Mackie, K. R., and Stojadinovic, B. (2016). "Hybrid Simulation of Thermomechanical Structural Response." *Journal of Structural Engineering*, 142(2), 04015107.

Wong, V. B., Brugess, I., and Plank, R. (2009). "Behaviour of composite cellular steel - concrete beams at elevated temperatures." *Steel Structures*, 9, 29–37.

Zhan, H., and Kwon, O. (2015). "Actuator controller inteface program for pseudo-dynamic hybrid simulation." *Advances in Structural Engineering and Mechanics (ASEM15)*.

Zhu, M., McKenna, F., and Scott, M. H. (2018). "OpenSEESPy: Python library for the OpenSEES finite element framework." *SoftwareX*, Elsevier B.V., 7, 6–11.