Analysis of restrained composite beams exposed to fire using a hybrid simulation approach

Mustesin Ali Khan[a], Liming Jiang[b,*], Katherine A. Cashell[a], Asif Usmani[b]

[a] *Department of Civil Engineering, Brunel University London, UK*
[b] Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong

Abstract

Obtaining an accurate simulation of the boundary conditions is very challenging but it is essential in order to represent the true behaviour of the whole structure in fire. In recent years, hybrid simulation has been emerging as an efficient and economical method for simulating realistic boundary conditions in the field of earthquake engineering. This technique can be used to study the load redistribution that may occur in a structural system as a result of locally elevated temperatures. In this paper, the fire-exposed element will be modelled in one analysis (a 3D model) and the rest of the structure in another analysis (a 2D model). This kind of sub-structuring enables the behaviour of the structural system as a whole to be studied. A hybrid simulation (HS) approach is presented and successfully implemented using the OpenFresco and OpenSees software. This approach enables the simulation of the correct restraint provided by the cold structure to the fire affected structural element. The HS analysis of a composite beam is compared with an unrestrained or simply supported version to highlight the difference in behaviour. Finally, the Cardington restrained beam test is modelled to demonstrate the potential of HS technique. Good agreement with the test results highlights that HS approach can be an effective method for studying the behaviour of the whole structural system.

Keywords: Structure in fire, thermo-mechanical analysis, OpenSees, multiple scale modelling

1. Introduction

There are many different finite element software packages available for commercial and research purposes and most used by the structural engineering community provide an efficient and inexpensive method for analysing the behaviour of structures exposed to extreme events, such as fire, earthquake, etc. However, most of the commonly used software lacks the required features for more customised applications and where they do, they do not allow developers to implement them in the source code.

In recent years, many researchers have focussed on modelling structural system response during extreme events, such as in the context of progressive collapse, or other such behaviour that is not achievable by simulating individual components. Simulating the whole structure in three dimensions (3D) is a complex and more computationally demanding task than simulating a single component, i.e. a beam or column, owing to the interactions present. Two or more appropriate finite element analyses can be coupled for each portion of the structure to achieve more flexible and inexpensive simulation of large engineering systems, compared with simulating the whole structure in a single 3D finite element analysis. Although the study presented in the current paper is specifically related to the finite element analysis of structures exposed to fire, a similar approach can be applied to structures exposed to other types of severe loading conditions, such as earthquake, flooding and blast.

In this paper, the behaviour of a composite beam exposed to fire is studied with the application of a new simulation method that is capable of coupling two or more finite element analyses together in order to create an accurate yet efficient simulation. Accuracy is measured by comparing against solutions obtained without sub-structuring and through validation against real experiments. Efficiency is achieved by coupling a high resolution model of the structural element exposed to fire with a dimensionally reduced model of the rest of the structure, without loss of accuracy. Section 5 provides the results that corroborate this claim. A hybrid simulation (HS) approach has been applied that involves the coupling of multiple instances of the same finite element program but modelled in different dimensions. The beam which is exposed to fire may experience large deformations and needs to be analysed in greater detail than other parts of the structure. So, it is modelled using 3D elements but the rest of the structure, which remains at ambient temperature, is modelled using 2D elements. OpenSees is used to model all the sub-structures to be coupled. Open-source software framework (OpenFresco) [1,2] is employed as the middleware software to enable the coupling between the codes. The primary motivation of this work is to create a tool that would enable system level simulation of the response of structures subjected to fire with the added feature of multi-scale analysis by exploiting the hybrid simulation approach. While the idea is simple, it has not been attempted previously in the context of pure simulation. The advantage of this approach is to enable the analyst to focus on the structural element of interest and modelling it at a higher resolution (such as the ones exposed to fire or other extreme loading) while modelling the rest of the structure at a lower resolution that is adequate to simulate the correct boundary restraint conditions. This approach produces an extremely powerful and versatile tool for efficient and accurate simulations of large structural systems subjected to complex fire scenarios in the context of performance based engineering. In this paper the tool developed is used to simulate a composite steel and concrete beam subjected to fire where the correct representation of boundary restraint conditions is critical to obtaining an accurate simulation of the behaviour. Most composite beams are axially and rotationally restrained in a composite steel-framed structure, and their behaviour in fire depends significantly on the nature and magnitude of the restraints. In the majority of the studies in this area, the behaviour of composite beams exposed to fire has been investigated by performing isolated fire tests or numerically modelling single elements [3–5]. Limited tests have been performed on composite beams exposed to fire as part of a structural frame [6–8]. The HS approach is used to model restrained composite beam behaviour in fire and is validated against a well referenced full scale test (Cardington Restrained Beam Test) showing excellent agreement with the experiment both in terms of the beam response and restraint simulation measured using the horizontal displacement of the beam end against the restraint provided by the frame. An unrestrained (simply supported) composite beam test in fire is also simulated to highlight the difference in behaviour. In general the unrestrained beam experiences runaway failure well before a restrained beam [9].

2. Hybrid Simulation Framework

In the context of this paper the phrase 'hybrid simulation' refers to modelling a structure using different sub-assemblies, some of which may be represented in 2D or using standard FE elements in one assembly whilst the areas requiring more focussed attention are modelled using more complex elements (3D elements) in another assembly. Both assemblies interact using a middleware software, such as OpenFresco. The various sub-assemblies interact at every time step of the finite element analysis solution procedure. In hybrid simulation, one of the assemblies is generally selected to act as the master assembly which solves the complete structure, while the other assemblies are selected to act as slave assemblies. Each slave assembly can be modelled as a super element in the master program and the master assembly can be modelled as an adapter element in the slave program so that both assemblies are connected at the interface degrees of freedom using the middleware software.

In the general hybrid simulation set-up, the master assembly implements the boundary conditions on the slave assembly and the slave program (or programs) return the reaction forces to the master program [10]. The boundary conditions that are transferred at the interface degrees-of-freedom from the master to the slave assembly can be defined as displacements and rotations. A middleware software is required to connect the master and slave programs through the super and adapter elements. Such software solves the issues such as data storage, communication methods, system control, optimisation and data transformations. The middleware used in this work is OpenFresco [1,2]. OpenFresco software was originally developed to perform hybrid testing (also referred to as hybrid simulation), in which the physical specimen in the laboratory is linked to the FE software to execute the tests but in the study presented here, it is utilised to simply link two FE models.

Various other methods have been used by other researchers to exchange data between the master and slave assemblies. Most researchers have used a file exchange system [11,12] between coupled codes, which works in the following manner. In the first step, the master code is initiated to calculate the trial displacements and rotations at connecting nodes and these responses are stored in data files. In the next step, the slave program is started by feeding the required parameters i.e. displacements and rotations to calculate the reaction forces. Once the analysis of the current step is complete, the results from the slave program are also stored in the data files. Finally, the reaction forces are fed back into the master code to calculate the new displacements and rotations.

In the approach presented here, the data is exchanged with the help of super and adapter elements. These elements are implemented once into their respective assemblies and the data communication is managed by OpenFresco. Using this method, both the master and the slave assembly codes can run together, without the need to stop and restart resulting in increased computational efficiency.

The sub-structuring technique was also utilised in the past to decrease the size of discretization [13–16]. There are two basic differences between the previous practices and the approach presented here. The first difference is that the approach in this paper uses two different element types in both the sub-assemblies i.e. 2D beam-column elements in one assembly, which is expected to behave linearly and

3D shell elements for another assembly, which is expected to behave non-linearly. Whereas in previous sub-structuring processes, all the sub-assemblies were modelled using the same type of elements (3D elements) and combined together at interface nodes. The second difference is that the element matrix for sub-structures does not get updated after each integration step and the element matrix calculated at the beginning of the analysis is used for the whole duration of analysis. However, in HS, the matrix for each sub-assembly is calculated and updated at each integration step. HS approach also enables the user to couple two different software i.e. Abaqus and OpenSees to use the specialised features of both the software.

There have been less than a handful of previous studies on the application of Hybrid Simulation to the study of structures in fire, which have established a sub-structuring process [17–20]. They have used 2D beam-column elements for both assemblies to establish the sub-structuring process. The process was intended to conduct hybrid tests with the real physical specimen. In all previous studies, the physical specimen was idealised as 2D beam column element, and it was connected to the rest of the 2D structure through sub-structuring process. To verify sub-structuring process, same results were achieved by modelling the whole structure in single analysis without using the sub-structuring process. In the study presented here, the physical specimen is replaced by a detailed 3D model, which is modelled using shell elements. The 3D model is connected to the rest of the structure (modelled using 2D beam column elements) to analyse whole structural behaviour in fire using the HS technique. This type of modelling, where two different dimension elements are used, is not possible in a single analysis without coupling the two models.

Therefore, the coupling technique is customised by simulating the slave assembly in 3D and the master assembly in 2D. This kind of coupling results in a dimensionally reduced hybrid simulation. The idea of dimensionally reduced analysis is useful where it is beneficial to model a portion of the structure in greater detail than the rest, such as in multi-scale simulations. In multi-scale modelling, one part of the system is modelled in detail using a finer mesh and the other parts are modelled using a coarser mesh with all the parts interacting simultaneously to describe the system. In this paper, a portion of the structure is expected to behave in a non-linear manner because of exposure to fire loading and therefore needs to be investigated in more detail (i.e. in 3D) while the rest of the structure which is not expected to experience high-intensity loads can be modelled in 2D. This optimises the efficiency of the model without compromising the accuracy of the analysis for the portion undergoing the extreme loading and its effect on the overall structural response.

2.1. OpenSees and OpenFresco

OpenSees, the Open System for Earthquake Engineering Simulation, is a software framework for simulation applications, originally for earthquake engineering applications, using finite element methods. It was originally developed at the University of California, Berkeley [21], and was later extended to perform structural fire analysis by researchers at the University of Edinburgh [22].

OpenSees is an object-oriented software implemented in the C++ language, through an open-source development process and uses the 'Tcl' scripting language as the platform. It is a collaborative program which is constantly being developed by numerous researchers [23–25]. Globally, a collaborative framework such as OpenSees can be a valuable tool as it offers a standard program-developing environment and thereby optimizes the structural engineering problem-solving strategies. The main advantage of using this particular finite element software for hybrid simulation, besides the fast computation capabilities, comes from the possibility of directly linking OpenSees to any hybrid simulation setup through the middleware software OpenFresco. Moreover, the object-oriented and open-source approach allows any developer to add components to fit their particular needs in specific areas of engineering research, and simultaneously disseminate the development to potential users. One-way or sequential coupling method has been adopted in OpenSees to simulate the response of structures exposed to fire, beginning with analysing the heat transfer to structural members due to the prescribed fire exposure, and applying the temperature history data to the structural analysis as a loading considering thermally induced strains and accounting for temperature-dependent material properties. In the study presented here, temperatures obtained from tests are applied directly to perform the structural analysis.

2.1.1. Fire models and heat transfer analysis in OpenSees

A growing library of fire models has been made available in OpenSees Thermal. For fire scenarios where a uniform compartment gas-temperature is assumed to exist at an instant of time, models such as the standard fire and parametric fire [26], etc., can be employed to define the gas temperature evolution, which is usually considered to be reasonable for small compartments. In recent years, localized burning and travelling fire behaviour in large compartments have attracted greater attention. Localized fire models have been provided in the Eurocode [26] and Structural Fire Protection Engineering (SFPE) handbook, and travelling fire models are beginning to appear in the technical literature in this field [27–29]. These advanced fire models are based upon applying a time history of heat flux at all spatial coordinates of the exposed surfaces of structural members resulting in fully characterizing the thermal loading demand on the structure corresponding to any given fire scenario.

The thermal impact on structural members caused by the fire exposure can be calculated in heat transfer analyses. The 'Heat Transfer' module in OpenSees Thermal can be deployed to run one-dimensional (1D) to three-dimensional (3D) heat transfer analyses. The basic architecture of heat transfer module follows the original OpenSees convention, which stores the modelling objects (heat transfer nodes, elements, and boundary conditions) in a HeatTransferDomain and performs the calculation in the HeatTransferAnalysis object. Tcl scripting commands have been provided for the heat transfer analysis as well, which utilise a mesh tool to discretise the structural members or sections into heat transfer nodes and elements. Commonly used structural materials such as steel and concrete have been added to the material library.

2.1.2. Thermo-mechanical analysis in OpenSees

When temperatures obtained from heat transfer analysis are applied at various locations in the structure and a stress analysis is performed, this type of analysis is termed as thermo-mechanical analysis. The thermo-mechanical analysis of structures subjected to fire is performed using beam-column (or frame) elements and shell elements in a 3D structural model or using both types of elements (a multi-scale model). The class hierarchy of the thermo-mechanical element implementation is illustrated in Fig.1, where Material, SectionForce Deformation, Element, and ElementalLoad are all abstract classes (base classes). The Fig.1 also shows the dependencies that these have between each other. The frame element can be formulated based on displacement interpolation or force interpolation, while its cross section is discretised into a number of fibres associated with uniaxial material models. A range of models for structural materials are added to the UniaxialMaterial collection which adopts the temperature dependent material properties from the Eurocodes. Thermo-mechanical shell elements (ShellMITC4Thermal and ShellNLDKGQThermal) are developed to model a thin plate such as beam flanges and concrete slabs. Both shell elements adopt a layered plate section, which can be either simply defined as a MembranePlateFiberSectionThermal of five layers of consistent material, or an advanced LayeredShellFiberSectionThermal which accepts various number of layers and different material type for each layer. Currently, the thermo-mechanical versions of multi-axial materials (NDMaterial) for shell elements are available as elastic models with stiffness degradation and thermal elongation (ElasticIstropic3dThermal), steel models defined as rebar meshes (PlateRebarMaterialThermal) or plane stress layers (J2PlasticityThermal) and a plane stress form of concrete damaged plasticity model (CDPPlaneStressThermal).
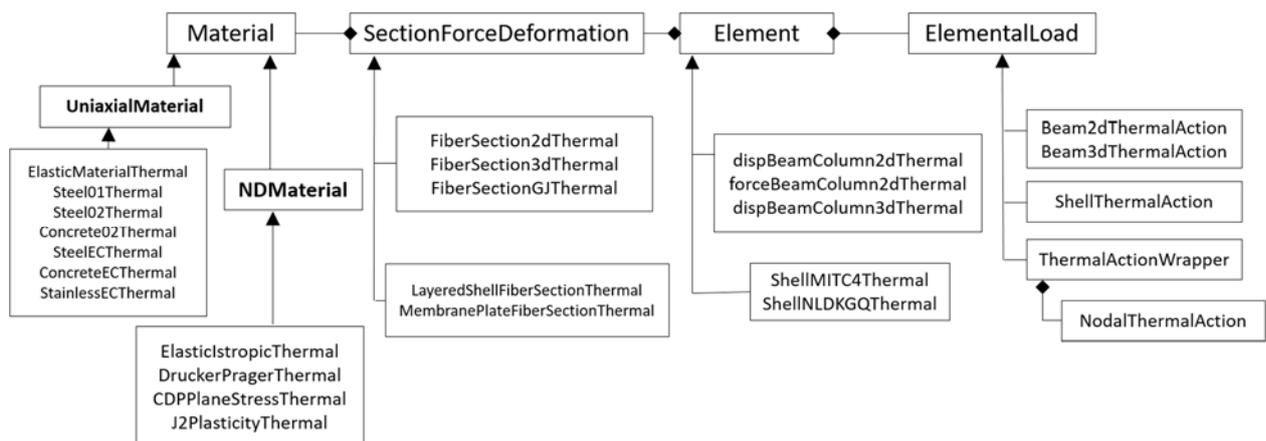

Fig. 1. Classes developed in OpenSees for thermo-mechanical analysis

2.1.3. OpenFresco

The hybrid simulation (HS) architecture used in the OpenFresco program is based on a master–slave architecture. The master is represented by an FE analysis, which is implemented in OpenSees in this

case. OpenFresco comprises four different software classes (with abbreviations in parenthesis): experimental element (ExpElement), experimental site (ExpSite), experimental setup (ExpSetup), and experimental control (ExpControl). OpenFresco facilitates the storage, transformation and transfer of data between the master and slave assemblies. The slave is also represented by an OpenSees analysis in this work. The OpenFresco ExpElement is a subclass of the Element class in OpenSees, and is a communication link between OpenSees and OpenFresco. The ExpElement class then communicates with the ExpSite class. In a hybrid simulation, described herein, ExpSite simply performs the structural data transfer between the ExpElement and the ExpSetup. The ExpSetup transforms the structural data between the experimental element degrees of freedom (DOF) in OpenFresco and the DOFs in the slave assembly, while the ExpControl stores and transfers the structural data between the slave and the master assembly.

2.2. Implementation details

The sequence of steps in exchanging the data between master and slave assembly to perform the hybrid simulation is shown in the Fig. 2. The steps required to couple the two analyses using OpenFresco are as follows:

1) Run the analysis for the master program and, as a result, the super element receives a displacement vector of global trial displacements (u_{super}) for all of its degrees of freedom from the master integration program.

2) The displacement vector obtained in the previous step is sent to OpenFresco using a TCP/IP socket (where TCP/IP means a Transmission Control Protocol/Internet Protocol which is the basic communication language or protocol of the Internet) as can be seen in Fig. 2. Here the SimAppSiteServer class is used to start the simulation server process.

3) The storage and transformation tasks for the displacement vector are performed by the LocalExpSite and ExperimentalSetup objects (See Fig. 2). Transformation of the data is not required in this instance because no physical specimen (i.e. laboratory test specimen) is involved. So, the NoTransformation object as ExpSetup is utilised.

4) The trial displacement vector 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. The adapter element then forms a resultant displacement vector by combining the trial displacements (u_{super}) with its own elemental displacements. Subsequently, corrosponding to the resultant displacement vector, a resultant force vector (P_{adpt}) for the adapter element is calculated and returned to the slave assembly.

5) After the solution convergence from the slave program, the negative resultant force vector (-P_{adpt}) is sent to the ExperimentalControl object through the TCP/IP socket. Again, the storage and transformation of the force vector is carried out by the LocalExpSite and ExperimentalSetup objects (see Fig. 2).

6) The SimAppSiteServer then sends the force vector to the super element in the master program through the TCP/IP socket.

7) The super element saves these values as element forces and returns them to the master integration method for the next step. The master program then determines the new trial displacements and Step 1 to Step 7 are repeated until the analysis is complete.
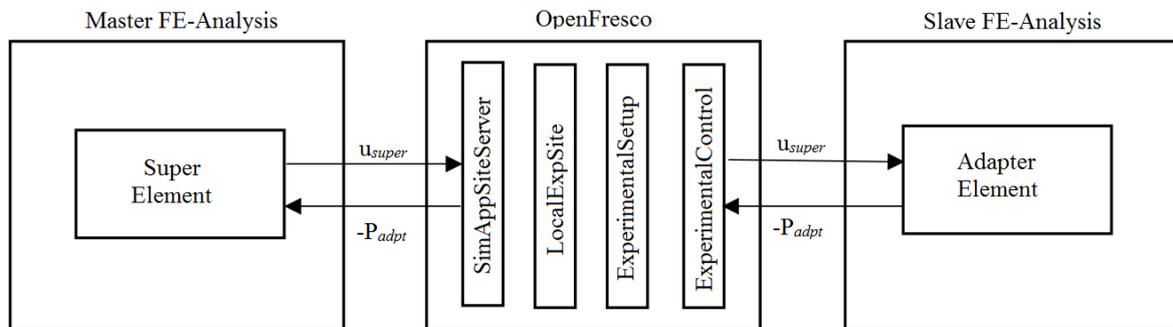

Fig. 2. Sequence of operations and data exchange

3. Hybrid simulation of a 2D frame with a single column heated

In this section, an example frame is modelled in OpenSees in order to illustrate the process described in the previous section. The example consists of two instances, one slave program and a master program, as shown in Fig. 3. The whole structure comprises a three-storey, three-bay steel moment-resisting frame, with one column which is exposed to fire. The bay widths and story heights of the structure are 8 m and 3.5 m, respectively. The size of beams and columns, used in both assemblies and the thermal expansion coefficient are identical to frame analysed in Section 5. The maser assembly is modelled using 20 dispBeamColumn elements and the slave column is modelled using one dispBeamColumnThermal element. Fixed boundary conditions are assigned to all the columns at the base. For this simple example, all the beams and columns are modelled with elastic beam-column elements with Young's modulus of 2.1×10^5 N/mm^2 for steel. The total load applied to the beams is assumed to be 4.5 kN/m. As shown in Fig. 3, the master program performs the analysis of the majority of the structure (which does not experience fire loading) and the slave program analyses the ground floor column which is exposed to fire.

Master and slave FE-software are connected at 1 interface node using super and adapter element. Horizontal displacement, vertical displacement and rotation are the three degrees of freedoms at the interface node. A super element in the master assembly is required to communicate the above 3 degrees of freedom to adapter element in slave assembly. Therefore a super element is defined using a 3×3 stiffness matrix corresponding to the interface degrees of freedom. For the initial stiffness values in the matrix of the super element, a unit displacement is applied at one interface degree of freedom in the slave assembly while keeping the other degrees of freedom restrained. However, the stiffness matrix for the adapter element is defined by assigning a high stiffness value of 1×10^{10} N/mm^2 to the diagonal elements, a very high stiffness value can cause convergence problems while a low stiffness value can

lead to inaccurate results [10]. Accurate results are obtained and convergence problems are avoided by using the above stiffness values.

In this example, the column in the slave assembly is subjected to a uniform temperature of 800 °C while the rest of the structure in the master assembly remains at ambient temperature. Vertical displacements at the interface nodes are traced between the master and slave assemblies, as presented in Fig. 4. As the temperature increases, there is an elongation in the length of the column due to thermal expansion which is indicated by the upward movement of the interface node. As soon as the material softens, the interface nodes start moving back downwards due to the increased load from the upper stories. For validation of the sub-structuring process, the whole building has also been analysed in a single assembly without sub-structuring into master and slave assemblies. Results obtained from the single analysis (single analysis node) are compared with the partitioned analysis and presented in Fig. 4. It is evident from Fig. 4 that vertical deflection at the interface is very similar between the master assembly, slave assembly and whole building analysis.
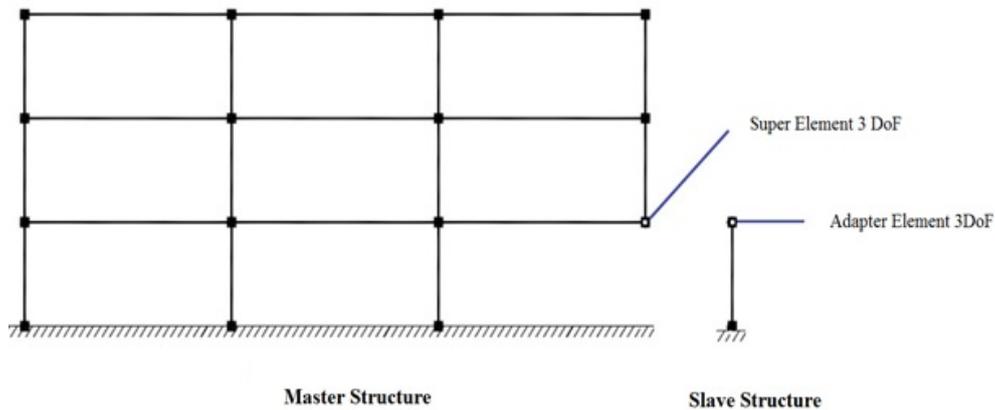

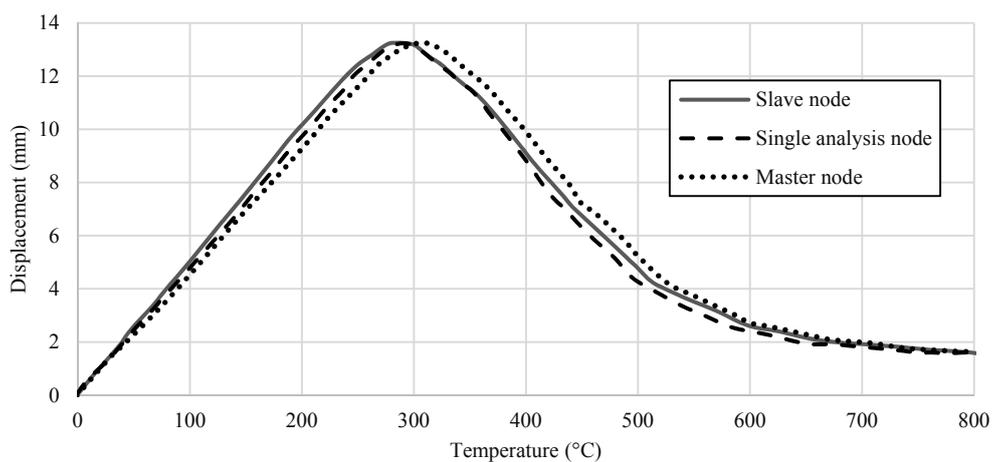**Fig. 3.** Sub-structuring for the 2D building example

Fig. 4. Vertical displacement comparison at interface node

This agreement indicates that the master and slave assemblies are communicating successfully with each other at each time step. The agreement of the master and slave node displacement with the whole

building analysis shows the accuracy of the coupling method and the HS approach. The HS approach produces practically identical results as those obtained by analysing the whole structure in a single analysis. The slight difference in the vertical deflections at the master and slave nodes results from computational noise and can be reduced by choosing more stringent convergence criteria, however in practice this is not necessary for ordinary structural engineering simulations.

4. 3D Thermo-mechanical model in OpenSees

In the previous section, the hybrid simulation technique was applied to a simple 2D structure with one element exposed to a uniform fire. In the next section, the method is advanced in order to represent a structure using 3D elements and a more realistic fire exposure. In order to apply the aforementioned solution algorithm in OpenSees, it is necessary to develop and modify the existing material classes to include the temperature-dependent properties. This development is described in the current section. Most of the previous thermo-mechanical analyses performed in OpenSees by other researchers have used 2D displacement beam-column elements [23]. However, in this study, 3D thermal elements and material models are employed and therefore the existing 'J2Plasticity' material class must be modified to the 'J2PlasticityThermal' material class, to incorporate the thermal response of the structure. The J2Plasticity material class in OpenSees uses the von Mises yield criterion. In the following section, the modifications to the material model to account for thermal effects are described.

4.1. Modified material class

There are many types of material models available in OpenSees for steel, each of which defines the mechanical constitutive relationships. However, some of these require modifications to include temperature-dependent properties. In the current analysis, the effects of temperature on the properties of steel such as yield strength and elastic stiffness are taken from the Eurocode design rules [30]. A temperature-dependent steel material model (J2PlasticityThermal) has been created based on the existing steel material model known as 'J2Plasticity', which represents the ambient temperature stress-strain relationship. For a J2Plasticity material model, the yield function follows the idealised von Mises yield criterion and is given as:

$$\emptyset(\boldsymbol{\sigma}, \boldsymbol{q}) = \|\boldsymbol{dev}(\boldsymbol{\sigma})\| - \sqrt{\boldsymbol{2/3}} * \boldsymbol{q} \tag{1}$$

where q is the yield strength with hardening and $dev(\sigma)$ is the deviatoric stress.

It is necessary to mention that thermo-mechanical concrete materials are available in OpenSees to enable analyses performed for concrete or composite structures in fire. As this paper only uses the steel material to demonstrate HS technique, users can refer to other papers [31,32] and the website http://openseesforfire.github.io for further information about thermo-mechanical concrete materials.

4.1.1. Temperature dependent mechanical properties

The temperature dependent mechanical properties of the steel are determined as defined in Eurocode 3 for carbon steel at elevated temperatures. Reduction factors are defined for effective yield strength

fy,T, proportional limit stress fp,T and the modulus of elasticity ET. The variation in reduction factors for the mechanical properties of steel at elevated temperature is shown in Fig.5.
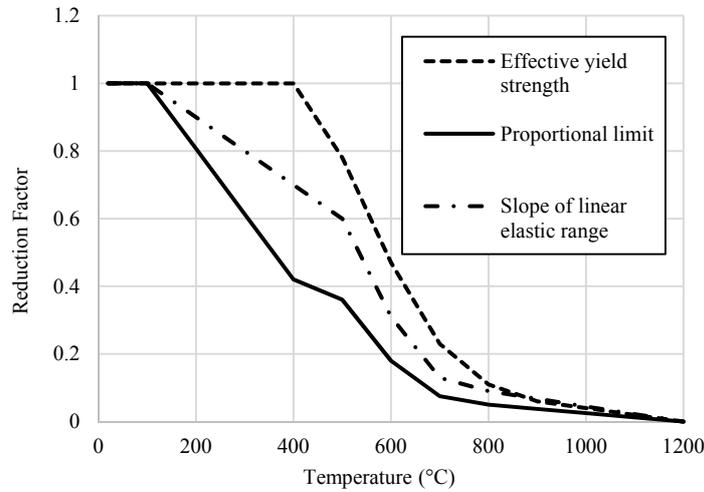

Fig. 5. Reduction factors for carbon steel at elevated temperatures [30]

4.1.2. Thermal elongation strain

The values of thermal elongation strain are calculated in accordance with Eurocode 3 [30]. The thermal elongation strain of steel (ε_{sth}) can be determined according to different temperature range as follows:

$$\varepsilon_{sth} = -2.416\times10^{-4} + 1.2\times10^{-5}\ T + 0.4\times10^{-8}\ T^2; \qquad \text{for } 20°C \leqslant T \leqslant 750°C \qquad (2.1)$$

$$\varepsilon_{sth} = 11\times10^{-3}; \qquad \text{for } 750°C \leqslant T \leqslant 860°C \qquad (2.2)$$

$$\varepsilon_{sth} = -6.2\times10^{-3} + 2\times10^{-5}\ T; \qquad \text{for } 860°C \leqslant T \leqslant 1200°C \qquad (2.3)$$

where T is the temperature in °C.

4.2. Material validation of J2PlasticityThermal at elevated temperature

A number of simply supported composite beams were subjected to an ISO834 standard fire by Wainman and Kirby [33] in a series of experiments conducted at the Swinden Laboratories. The structural configuration of one of these beams (Test 16 is selected for illustrative purposes herein) is shown in Fig. 6 and this is used in the current analysis to validate the newly developed 'J2PlasticityThermal' material. The same beam was analysed using 2D beam-column elements in OpenSees [34]. The results from the tests and also the numerical analysis are used to validate the results obtained by 3D thermomechanical analysis performed in this paper. The steel beam and slab are modelled using shell elements (ShellMITC4Thermal) with element size (35 × 30 mm). The 'J2PlasticityThermal' and 'DruckerPragerThermal' material models in OpenSees are selected to represent the steel and concrete material in the composite beam, respectively. Fig. 7 shows the temperature distribution in different components of the tested composite beams, during the experiment. No concrete slab temperature profiles were reported in the literature and therefore the temperature distributions through the thickness of the slabs are modelled based on the recommendations in Eurocode

4 [35]. Fig. 8 shows the mid-span deflections that were measured during the test together with the predicted results using the OpenSees 3D model, as well as the results from the Jiang et al. analysis [34]. The OpenSees predictions show reasonable agreement with both the test results and the predictions made by Jiang et al. This agreement validates the behaviour of newly developed 'J2PlasticityThermal' material model, as well as the 3D modelling capabilities of OpenSees under thermomechanical loading.
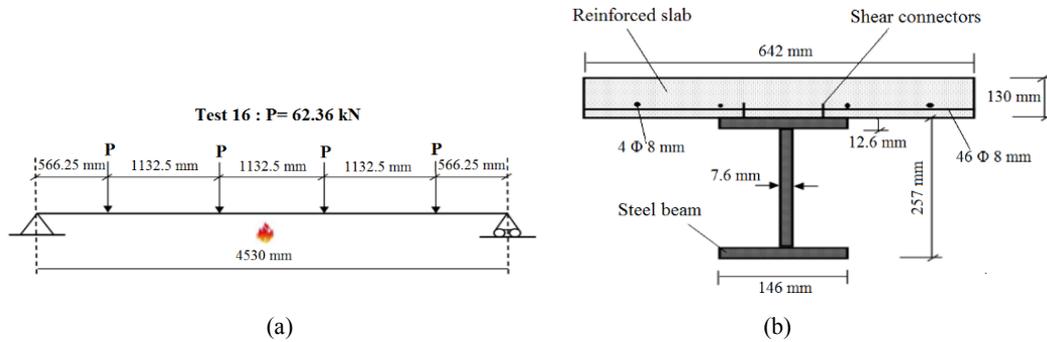

Fig. 6. *Schematic view of tested beam.* (a) beam setup; (b) beam section
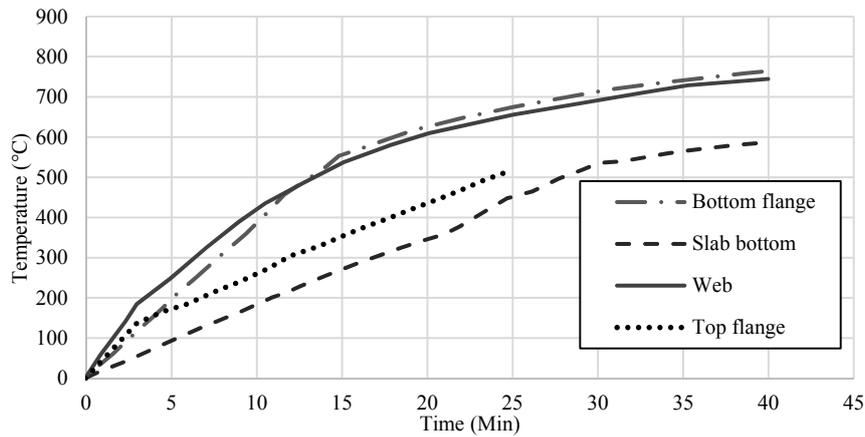
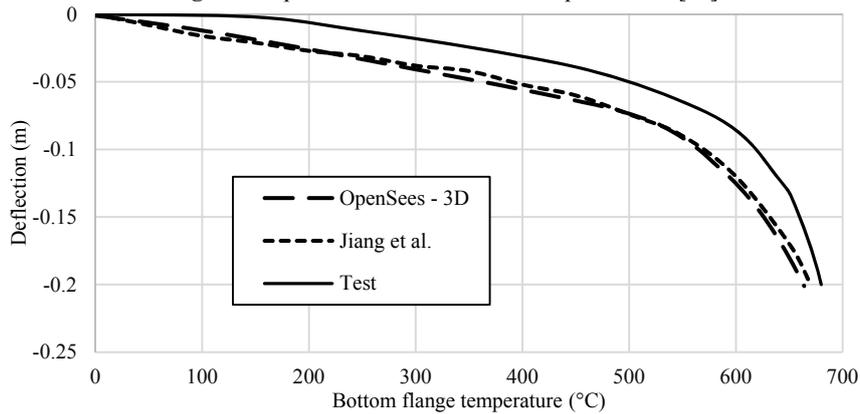**Fig. 7.** Temperature distribution at mid-span in Test [33]

Fig. 8. Vertical deflection comparison at mid-span

5. Hybrid simulation of a whole frame in fire

In this section, the restrained beam tested during the Cardington experiments is simulated and validated to establish the hybrid simulation approach in 3D. These fire tests were performed by British Steel [36] on the 7th floor of a composite steel framed structure at Cardington, as shown in Fig. 9. The

new 3D thermomechanical material class described in the previous section is employed. The restrained beam (which was a 305×165×40 universal beam) was heated over the middle 8.0 m of its 9.0 m length ensuring that the beam-column connection was at ambient temperature. In this section, a three-dimensional model of the Cardington restrained beam on the 7th floor is built in the slave assembly using OpenSees and the rest of the frame is modelled in a master assembly in a 2D OpenSees model as shown in Fig. 10. The composite restrained beam is modelled in the slave assembly and is connected to the rest of the structure at interface degrees of freedom through the adapter and super elements. The structure consists of a moment-resisting frame in the master assembly that is connected at interface nodes with the composite restrained beam modelled in the slave assembly. All beams in the master assembly are of length 9 meter with section (305×165×40 universal beam) and columns are of height 4 meter with section (254×254×89 universal column).
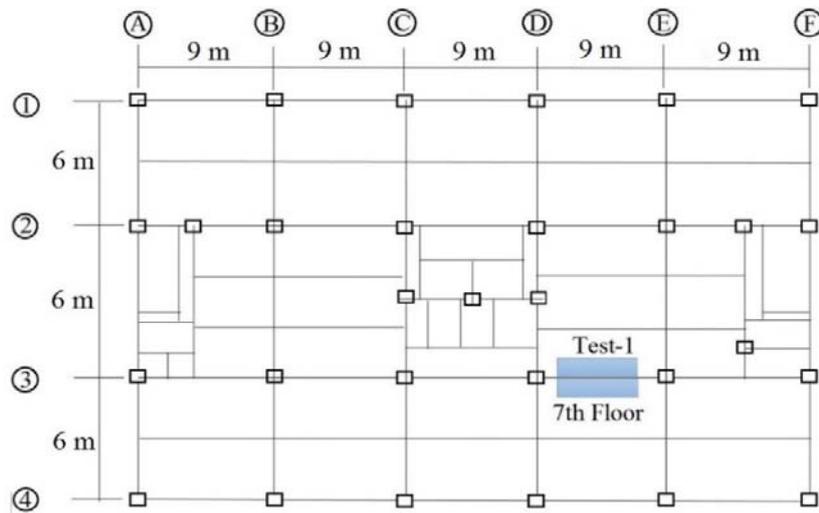

Fig. 9. Location of the restrained beam test

The profiled slab is modelled separately using shell elements (ShellMITC4Thermal) with an element size of 50 × 40 mm, for the flat part of the reinforced concrete slab and 3D beam-column elements (DispBeamColumn3DThermal) with an element length of 40 mm for the concrete ribs. The total number of shell elements used to model the flat portion of slab and 3D beam column elements to model the ribs are 13500 and 2325 respectively. The ShellMITC4Thermal elements with an element size of 50 × 40 mm for flange and 50 × 60 mm for web are used to model the steel beam. The steel beam is modelled using 2340 shell elements. The slab, ribs and beams are connected using the rigid link element (rigidLink beam). The rest of the columns and beams in the master assembly are modelled using 2D beam elements (DispBeamColumn2DThermal). The compressive strength of concrete is 48 MPa and the yield stress of the steel is 280 MPa. The DruckerPragerThermal material class is used to model the concrete in the slab (modelled using shell elements, which require a biaxial material model) and Concrete02Thermal [37] is used for the concrete in the ribs (modelled using beam-column elements, therefore a uniaxial material model is used). The 'J2PlasticityThermal' material model which was

developed and described in the previous section is used to model the steel beam. The slab reinforcement is modelled using a smeared layer distribution in the shell elements.
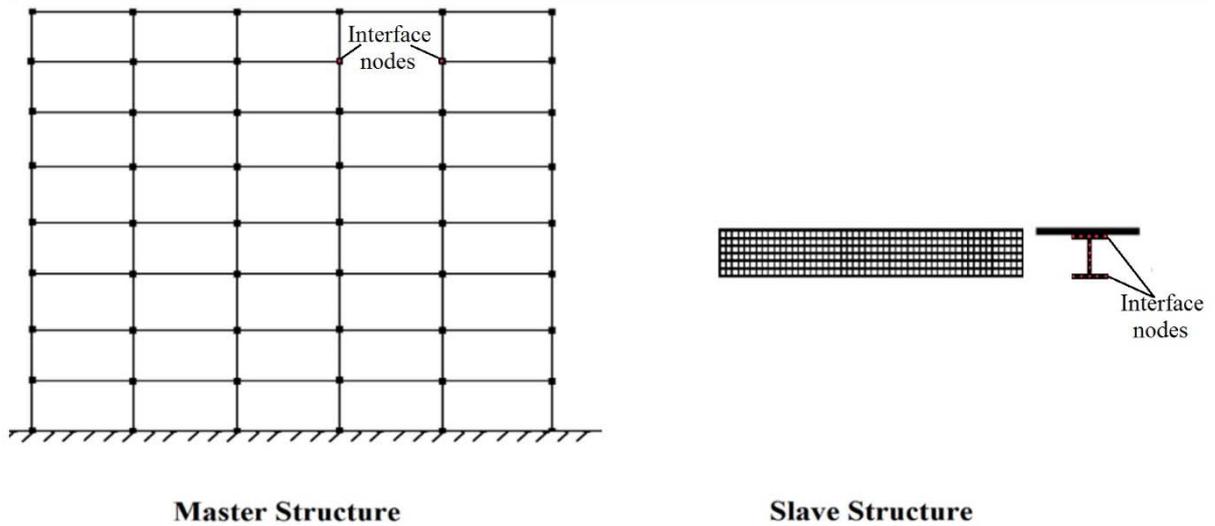

Master Structure   **Slave Structure**

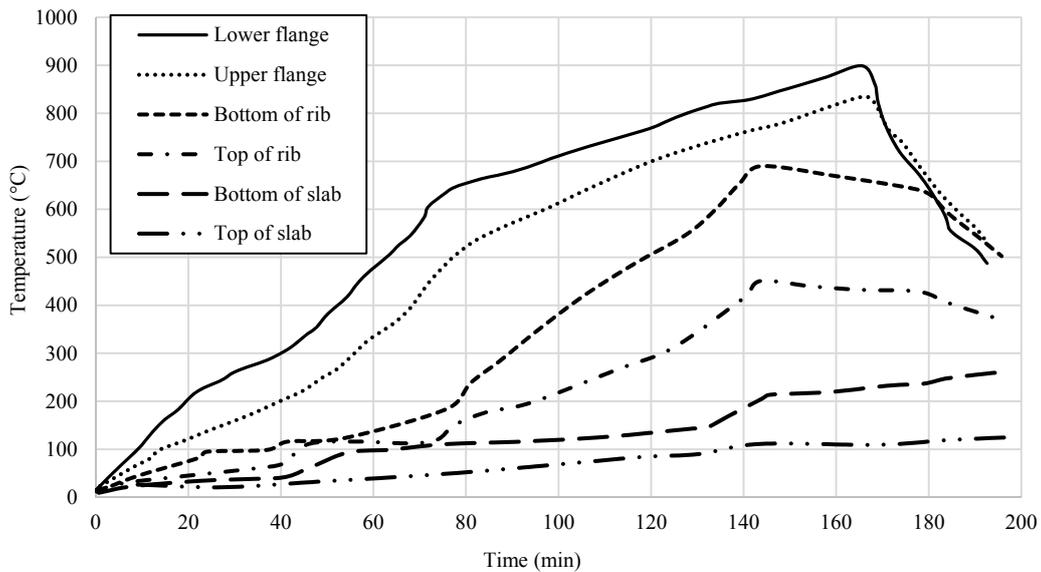**Fig. 10.** Sub-structuring for Cardington restrained beam hybrid simulation

Fig. 11. Temperature distribution during restrained beam test [36]

The slab is subjected to a uniformly distributed load of 5.48 kN/m^2 as reported in the literature [9] and the temperature profile obtained from the test data (see Fig. 11) is applied as a thermal load to the beam and the slab. The structure is loaded in two steps. In the first step, the static load is applied while the rest of the structure is at ambient temperature. In the second step, the thermal load is applied to the restrained beam while the remaining structure at ambient temperature and constant static load. A nonlinear dynamic analysis is performed in OpenSees to investigate the behaviour of the restrained beam under fire.

This structure is an ideal candidate for analysis using reduced dimensional hybrid simulation, where the frame is modelled using 2D displacement beam-column elements and the composite beam assembly is modelled with 3D elements. The steel moment-resisting frame assembly is analysed in the master FE software and the beam exposed to fire is analysed in the slave FE software. Both these assemblies are shown in Fig. 10. The moment-resisting frame is connected to the beam through 14 interface nodes at each end so, a 28-noded super element is added to the master program and a 28-noded adapter element is added to the slave program. Hence, the adapter element in the slave model connects to the interface node of the frame through the super-element in the master program. OpenFresco is used to transfer displacements and forces between slave and master assembly.

5.1. Vertical deflection

Fig. 12 shows the comparison of the mid-span deflection of the restrained beam. It is evident that reasonable agreement with the test results is achieved by the hybrid simulation. The vertical deflection increases at a constant rate during most of the fire test and no runaway deflection is observed. This is accurately depicted by the numerical model, as shown in Fig. 12.
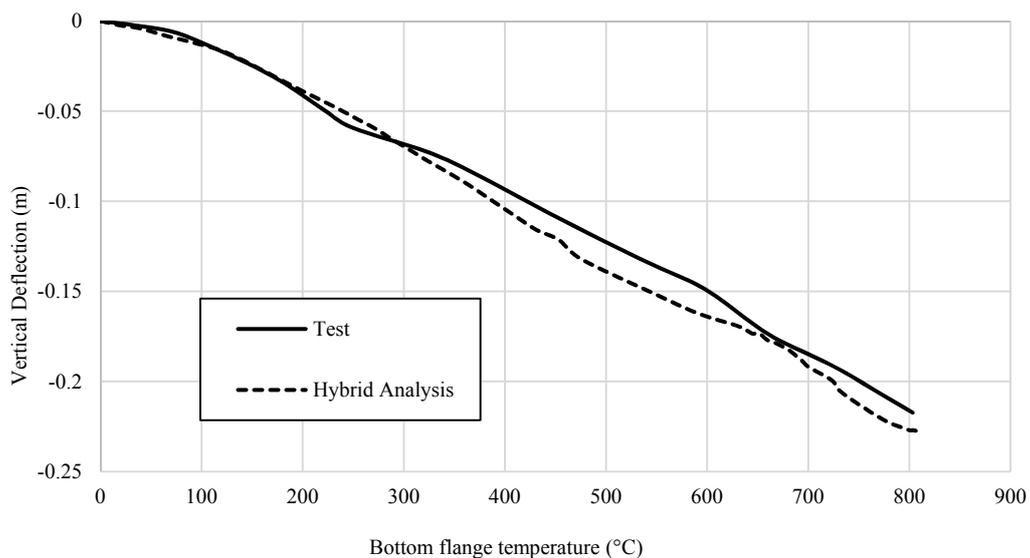

Fig. 12. Vertical deflection of the restrained beam at mid-span

5.2. Horizontal displacement

The horizontal displacement of the column at floor level is also traced and compared with the test results. The plateau in the horizontal displacement of the column demonstrates a very interesting aspect of the behaviour during the test, as shown in Fig. 13. In the initial stages of the test, the horizontal displacement of the column increases with temperature, until about 250 °C. Then, it plateaus until around 600 °C after which it begins to increase again. The initial increase in horizontal displacement is dominated by thermal expansion of the steel beam. At about 250 °C, the bottom flange of the steel beam yields and a reduction in the restraint to thermal expansion is observed which results in no further increase in horizontal displacement and is evidenced by the plateau in Fig. 13, which is also shown in

the OpenSees predictions. The increase in the column horizontal displacement in the second phase is due to the thermal expansion of the concrete slab because the temperature in the slab rises at a relatively slower rate than the steel beam and expands later than the steel beam.
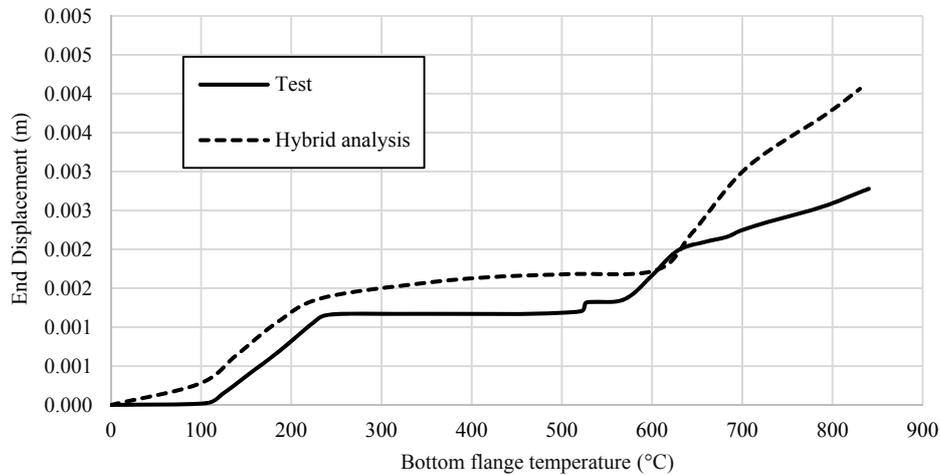

Fig. 13. Horizontal displacement at end of the restrained beam

In accordance with the general behaviour expected of restrained beams under fire, when the beam reaches runaway, it behaves as a catenary and the ends of the beam exert an inward "pull" force on the restraints, which results in a reduction in the horizontal displacement. However, in this case, it can be clearly seen in Fig. 12 that the test beam does not experience runaway and the horizontal displacement continues to increase without experiencing any reduction.

5.3. Rotation

In addition to the vertical and horizontal displacements, hybrid simulation also predicts the end rotations of the beam during the fire. These values are compared with the corresponding test data [36] in Fig. 14, and it is evident that a good agreement has been achieved by the numerical analysis. It is noteworthy, with reference to Figs 12-14, that the mid-span deflection (Fig. 12) and end rotations (Fig. 14) continue to increase even during the plateau stage in the horizontal displacements (Fig. 13). This is because of thermal bowing which develops due to the steep thermal gradient in the composite beam floor system.
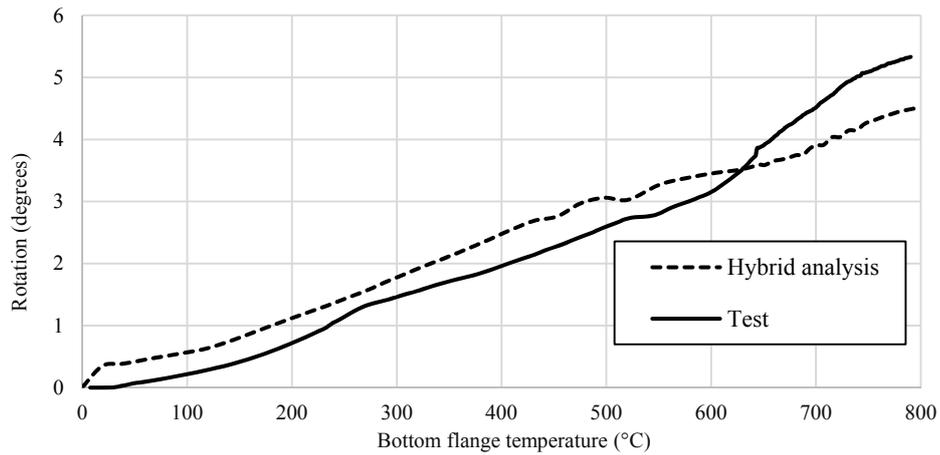
Fig. 14. End rotation of the restrained beam

5.4. Analysis of the boundary conditions

A comparative analysis has been performed between the restrained beam analysed in the previous section and a simply-supported beam to show the significance of using the hybrid simulation technique to simulate the actual boundary conditions in a structure. A beam with the same material and geometric properties as the restrained beam but with simply-supported boundary conditions is analysed to make the comparison. This beam is also exposed to the same mechanical and thermal loading conditions. Fig. 15 compares the deformed shape for both of these arrangements after 70 minutes of fire exposure. The simply-supported beam has no axial or rotational restraint at the ends and therefore allows greater vertical deflection. As shown in Fig. 15, after 70 minutes, the simply-supported beam has nearly double the vertical deflection as the restrained beam.
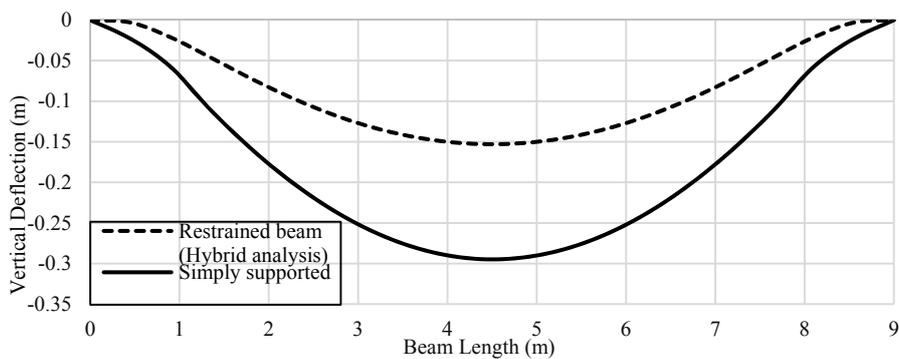

Fig. 15. Comparison of the vertical deflections along the beam length for a restrained and simply-supported beam
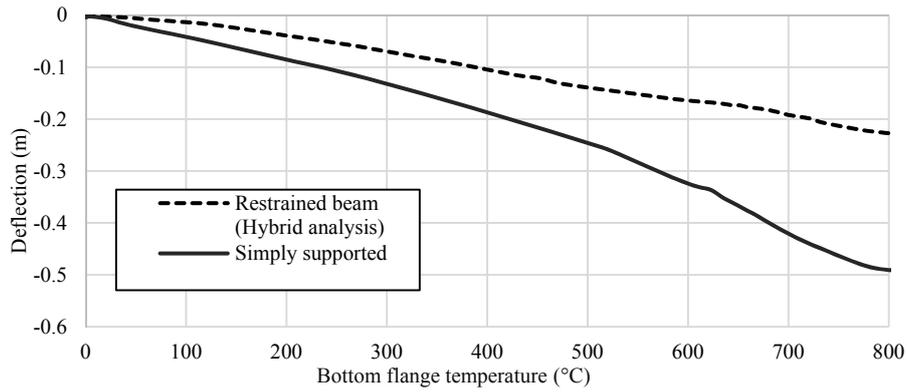
Fig. 16. Comparison of the vertical deflections of a restrained and simply-supported beam at mid-span
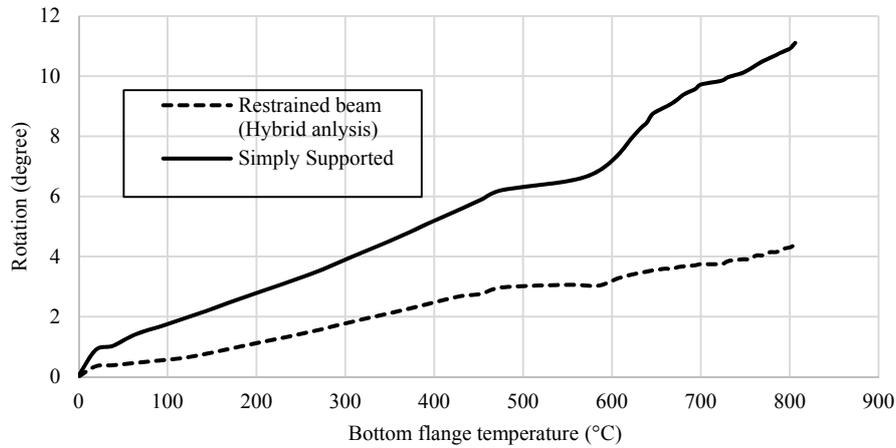

Fig. 17. Comparison of the end rotations of a restrained and simply-supported beam at support

The vertical deflections and end rotations for the two beams are compared as a function of temperature as presented in Fig. 16 and 17, respectively. A comparison of the vertical deflection in Fig. 16 shows that the deflection in the simply-supported beam at 800 °C is almost 2.5 times that of the restrained beam. The same pattern can clearly be seen in Fig. 17 for the end rotations, as the values for the simply-supported beam are about 2.5 times that of the restrained beam after 130 minutes. This is because there is no rotational restraint in the simply-supported beams.

Simply-supported, pinned and fixed end conditions are a hypothetical idealised scenario, which do not really exist in practical situations. The hybrid simulation technique is advantageous in that it simulates the actual boundary conditions as in case of the Cardington restrained beam test. The hybrid simulation method has been shown to provide an excellent method for analysing the true behaviour of the structures by modelling the specific part of the structure in 3D and rest of the structure in 2D. It enables the true behaviour of the whole structure to be analysed in an efficient yet accurate way and eliminates the need for modelling the whole structure in 3D.

6. Conclusions

A hybrid simulation approach for thermo-mechanical analysis is established and verified by studying the behaviour of a three bay, three-storey frame under thermal and mechanical loads. The OpenSees framework has been extended to perform thermo-mechanical analysis of composite structures in the 3D

environment. A simply supported composite beam exposed to fire is also modelled and the behaviour is compared with the restrained beam, analysed using the HS approach. The HS approach simulates the actual restraint provided by the unheated structure and this behaviour is contrasted against the unrestrained (simply supported) simulation. Significantly lower mid-span deflection is observed in HS analysis due to the restraining effect. The performance of the capacity developed in OpenSees is validated by predicting the mid-span deflection of a tested composite beam under thermomechanical loading as part of the Cardington restrained beam test. An additional feature of this analysis is that the tested composite beam is modelled in 3D but taking advantage of the directional nature of the tested element the rest of the system is reduced to a 2D representation in order to save computation time. Excellent agreement is achieved between the HS OpenSees predictions and the experimental measurements of the mid-span deflection of the tested beam and horizontal displacement of the column, which is an excellent measure of the restraint provided by the cold structure. A dimensionally reduced hybrid simulation approach has therefore been established and verified.

Further work will focus on the modelling fire-exposed parts of structures in 3D and the remainder of the structure in 2D, in order to study the local buckling behaviour in the fire exposed structural components.

References

[1]   Schellenberg A, Mahin SA, Fenves GL. A Software Framework for Hybrid Simulation of Large Structural Systems. Struct Eng Res Front 2017. doi:doi:10.1061/40944(249)3.

[2]   Takahashi Y, Fenves GL. Software framework for distributed experimental-computational simulation of structural systems. Earthq Eng Struct Dyn 2006;35:267–91. doi:10.1002/eqe.518.

[3]   Łukomski M, Turkowski P, Roszkowski P, Papis B. Fire Resistance of Unprotected Steel Beams-Comparison between Fire Tests and Calculation Models. Procedia Eng 2017;172:665–72. doi:10.1016/j.proeng.2017.02.078.

[4]   Kodur VKR, Naser MZ. Effect of local instability on capacity of steel beams exposed to fire. J Constr Steel Res 2015;111:31–42. doi:10.1016/j.jcsr.2015.03.015.

[5]   Dwaikat MMS, Kodur VKR, Quiel SE, Garlock MEM. Experimental behavior of steel beam-columns subjected to fire-induced thermal gradients. J Constr Steel Res 2011;67:30–8. doi:10.1016/j.jcsr.2010.07.007.

[6]   Dong Y, Prasad K. Experimental Study on the Behavior of Full-Scale Composite Steel Frames under Furnace Loading. J Struct Eng 2009;135:1278–89. doi:10.1061/(ASCE)0733-9445(2009)135:10(1278).

[7]   Zhao JC, Shen ZY. Experimental studies of the behaviour of unprotected steel frames in fire. J Constr Steel Res 1999;50:137–50. doi:10.1016/S0143-974X(98)00246-6.

[8]   Liu TCH, Fahad MK, Davies JM. Experimental investigation of behaviour of axially restrained steel beams in fire. J Constr Steel Res 2002. doi:10.1016/S0143-974X(01)00062-1.

[9]   Gillie M, Usmani AS, Rotter JM. A structural analysis of the Cardington British steel corner test. J Constr Steel Res 2002;58:427–42. doi:10.1016/S0143-974X(01)00066-9.

[10]   A. S, Huang Y, Mahin SA. Structural FE-Software Coupling through the Experimental Software Framework, OpenFresco. 14th World Conf. Earthq. Eng., Beijing: 2008.

[11]   Wang T, Nakashima M, Pan P. On-line hybrid test combining with general-purpose finite element software. Earthq Eng Struct Dyn 2006;35:1471–88. doi:10.1002/eqe.586.

[12]   Kwon O-S, Nakata N, Park K, Elnashai A, Spencer B. 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). 2007.

[13]   Furuike T. Computerized multiple level substructuring analysis. Comput Struct 1972;2:1063–73. doi:10.1016/0045-7949(72)90056-9.

[14]   Peterssons H, Popovo EP. Substructurin ~ and Equation System Solutions in Finite Element Analysw 1976.

[15]   Gurujee CS. an Improved Method of Substructure 1978;8:147–52.

[16]   Sheu CH, De Roeck G, Van Laethem M, Geyskens P. Application of the substructuring technique to non-linear dynamic structural analysis. Comput Struct 1990;35:593–601. doi:10.1016/0045-7949(90)90387-H.

[17]   Sauca A, Gernay T, Robert F, Tondini N, Franssen J-M. Hybrid fire testing: Discussion on stability and implementation of a new method in a virtual environment. J Struct Fire Eng 2018:JSFE-01-2017-0017. doi:10.1108/JSFE-01-2017-0017.

[18]   Whyte CA, Mackie KR, M.ASCE, Stojadinovic B. Hybrid Simulation of Thermomechanical Structural Response. J Struct Eng 2016;142. doi:10.1061/(ASCE)ST.1943- 541X.0001346.

[19]   Whyte CA, Abbiati G, Stojadinovic B. Validation of OpenFresco-Based Thermomechanical Hybrid Simulation to Address an Earthquake-Fire-Earthquake Coupled Problem (PDF Download Available).pdf. VI Int. Conf. Comput. Methods Coupled Probl. Sci. Eng. COUPLED Probl. 2015, 2015.

[20]   Tondini N, Abbiati G, Possidente L, Stojadinovic B. Hybrid simulation applied to fire testing : a newly conceived numerical framework. EACS 2016 – 6th Eur. Conf. Struct. Control, Sheffield, England: 2016.

[21]   McKenna FT. Object-oriented finite element programming: Frameworks for analysis, algorithms and parallel computing. ProQuest Diss Theses 1997.

[22]   Usmani A, Zhang J, Jiang J, Jiang Y, Kotsovinos P, Zhang J, et al. Using OpenSees for Structures in Fire. Struct Fire 2010;3:919–26.

[23]   Jiang L, Usmani A. Computational performance of beam-column elements in modelling structural members subjected to localised fire. Eng Struct 2018;156:490–502.

[24]   Kolozvari K, Orakcal K, Wallace JW. New opensees models for simulating nonlinear flexural and coupled shear-flexural behavior of RC walls and columns. Comput Struct 2018;196:246–62. doi:10.1016/j.compstruc.2017.10.010.

[25]   Zhu M, McKenna F, Scott MH. OpenSeesPy: Python library for the OpenSees finite element framework. SoftwareX 2018;7:6–11. doi:10.1016/j.softx.2017.10.009.

[26]   British Standards Institution. BS EN 13501-1: 2007 Fire classification of construction products and building elements - Part 1: Classification using test data from reaction to fire tests. 2007.

[27]   Stern-Gottfried J. Travelling Fires for Structural Design. Fire Saf J 2011;54:74–85.

[28]   Stern-Gottfried J, Rein G. Travelling fires for structural design-Part II: Design methodology. Fire Saf J 2012;54:96–112.

[29]   Dai X, Welch S, Usmani A. A critical review of "travelling fire" scenarios for performance-based structural engineering. Fire Saf J 2017;91:568–78.

[30]   European committee for standardizaion. EN 1993-1-2: Eurocode 3: Design of steel structures - Part 1-2: General rules - Structural fire design. Br Stand Inst 2005;2.

[31]   Jiang L, Usmani A. Towards scenario fires – modelling structural response to fire using an integrated computational tool 2018. doi:10.1177/1369433218765832.

[32]   Gernay T, Millard A, Franssen JM. A multiaxial constitutive model for concrete in the fire situation: Theoretical formulation. Int J Solids Struct 2013;50:3659–73. doi:10.1016/j.ijsolstr.2013.07.013.

[33]   Wainman DE, Kirby BR. Compendium of UK standard fire test data unprotected structural steel-1. Swinden Laboratories, Rotherdam: 1988.

[34]   Jiang J, Li G-Q, Usmani A. Analysis of Composite Steel-concrete Beams Exposed to Fire using OpenSees. J Struct Fire Eng 2015;6:1–20.

[35]   CEN (European Commitee for Standardization) Eurocode 4 - Design of composite steel and concrete structures - Part 1-2 General rules - Structural fire design. En-1994-1-2. Eurocode 4 2005;2.

[36]   British Steel (BS) plc. The behaviour of multi-storey steel framed buildings in fire. Swinden Technology Centre, Rotherham: 1999.

[37]   Jiang J, Usmani A, Li G-Q. Modelling of Steel-Concrete Composite Structures in Fire Using OpenSees. Adv Struct Eng 2014;17:249–64. doi:10.1260/1369-4332.17.2.249.