Fire Safety Journal 140 (2023) 103896

Contents lists available at ScienceDirect

Fire Safety Journal

journal homepage: www.elsevier.com/locate/firesaf

An open-source software framework for the integrated simulation of structures in fire

Aatif Ali Khan [a,*], Mustesin Ali Khan [b,**], Katherine A. Cashell [c], Asif Usmani [d]

[a] *University of Canterbury, Christchurch, New Zealand*
[b] *University of Central Lancashire, Qatar*
[c] *University College London, UK*
[d] *The Hong Kong Polytechnic University, Kowloon, Hong Kong*

ARTICLE INFO

Keywords:
CFD
FEM
Coupling
Open-source
Structure fire

ABSTRACT

The traditional methods to understand the development of elevated temperature in a structure, and also the associated structural response, are not representative of realistic fire scenarios. To provide a more accurate and realistic reflection of the fire development, the current paper develops a generic middleware which interfaces between the computational fluid dynamics (CFD) software Fire Dynamics Simulator (FDS) and the finite element (FE) analysis software OpenSees. This framework enables a fully integrated simulation of a realistic fire scenario including the heat transfer through the structure and the resulting thermo-mechanical response. The proposed framework is open-source and freely available and therefore can be used and further developed by researchers and practicing engineers and customised to their requirements. This paper shows validation against two sets of experimental results and one real fire incident. A number of different types of thermal boundary conditions such as gas temperatures and heat fluxes, are obtained from the CFD analysis and are then used in the subsequent heat transfer and thermo-mechanical analysis. The primary advantage of this computational tool is that it provides consultants and designers with the means to undertake large-scale projects requiring performance-based fire engineering solutions.

1. Introduction

Over the past century, there has been considerable attention given by researchers and engineers towards understanding the response of structures during different fire scenarios. This has included studying the influence of elevated temperature at material, element and system level, as well as trying to understand the development of fire and fire spread in different scenarios. In terms of the escalation of elevated temperature with time, various different fire models have been proposed, such as the standard time-temperature curve, highest temperatures in a compartment, and localised fires to define the fire scenario in a large open compartment [1]. Khan et al. [2] reviewed and discussed the limitations and applicability of the most widely-adopted fire models to understand their adequacy for structural fire assessment.

The recent trend towards performance-based engineering (PBE) approaches for fire design has led to greater demands for more accurate and realistic representations of fires in analysis compared with traditional models. Consequently, the need for realistic thermal boundary conditions for fire-exposed structural components has resulted in greater utilisation of advanced computational models. Travelling fire scenarios, which are not generally depicted by the standard fire models, have been simulated using either sophisticated computational fluid dynamics (CFD)-based approaches, or through analytical modelling (e.g. [2–5]). CFD can provide an accurate approach for simulating a realistic fire scenario, especially when these techniques are steadily improving owing to increased research focus, more advanced computational capabilities and also following more validation against experiments.

It is widely accepted that CFD provides a better resolution (both spatial and temporal) of the thermal boundary conditions over structural surfaces compared with parametric fires or standard fire curves [5]. However, to date, it remains very challenging to couple a CFD thermal model with a finite element (FE) analysis structural model for either simultaneous or sequential simulations. The utilisation of CFD together with FEM accelerated following the World Trade Centre (WTC) disaster

* Corresponding author.
** Corresponding author.
E-mail addresses: aatif.khan@canterbury.ac.nz (A.A. Khan), makhan21@uclan.ac.uk (M.A. Khan).

https://doi.org/10.1016/j.firesaf.2023.103896
Received 7 June 2023; Received in revised form 31 July 2023; Accepted 5 August 2023
Available online 6 August 2023

in 2001, to better understand the complex structural fire response during this very high-profile incident. Since then, several researchers have attempted to couple CFD with FE simulations and have recognized the potential for producing a realistic fire scenario for structural analysis [6]. In all approaches, thermal properties such as the gas temperatures, radiative heat fluxes, and heat transfer coefficients were directly calculated from CFD calculations. Although more approaches have been proposed to couple CFD with FE analysis, in most cases the source codes are not made freely available as open-source and are restricted to the host institutions. Furthermore, the proposed methodologies are generally compatible with commercial analysis packages only. Therefore, to understand the structural response to real fires, engineers generally need to develop their own codes and then solve the difficulties presented at the fire-structure interface by determining the most suitable boundary conditions [2], which can be very challenging.

In response to the challenges described, the current paper provides a complete open-source framework for the coupling of CFD and FE analysis that provides engineers with a freely-available solution and also allows them to learn from, implement, and improve the original source codes. It focuses on how the most important prerequisite information for a reliable structural analysis is achieved i.e., representing the thermal boundary conditions in a realistic manner for structural analysis and then automatically providing these to the FE software for the heat transfer and structural response analysis. This is the only open-source and free-to-use computational framework that allows the structural fire community to customise and modify the framework according to client requirements and also provides the rigorous and accurate analysis desired in the context of PBE.

2. Complexity in fire-structure interaction

The input parameters describing the fire scenario must be reliable and able to represent the physical characteristics of the fire in order to enable an accurate simulation of the thermo-mechanical response of structures. Some of the major challenges that need to be resolved when performing structural analysis for a given fire scenario are given as:

- **Appropriate boundary conditions**: The boundary conditions for conducting the heat transfer from the gas phase temperature to the solid boundary must be reasonable and represent the fire scenario accurately.
- **Spatio-temporal scale**: The thermal properties of the structural material influence the thermal response of a structure to the exposed environment. The variation in the gas phase temperatures in the fluid domain nearby to a structural surface is significantly faster than the temperature variation in the solid phase. The difference in spatial and temporal scales plays a critical role in the selection of an appropriate time step for coupled modelling.
- **Fire duration and severity:** The severity of the fire in terms of how it affects the structure broadly depends on the duration of the fire. Therefore, the boundary conditions need to be carefully assessed throughout the whole duration of the fire including the decay period (cooling phase).
- **Geometry:** The compartment geometry such as the ventilation characteristics, energy distribution, rate of fuel consumption, etc., affects the burning conditions. Accordingly, the fire may follow a pattern of growth, steady burning, and decay or may never achieve a steady phase.
- **Fuel distribution**: Generally, codes and standards do not provide information about the fuel distribution or its influence on the fire development [5,7], even though it is known to be very influential [8].

2.1. CFD-FEM coupling

Despite the aforementioned shortcomings, the most prevalent fire protection engineering practices are still prescriptive and code-based. The response of complex structures to real fires cannot be determined using the widely accepted prescriptive fire models for structural designs. The most accurate way to simulate the structural response to real fires is using CFD modelling and then coupling it with the FE analysis software to depict the structural response. Currently available CFD tools are capable of generating realistic and high-fidelity fire scenarios. Therefore, it is imperative to exploit the potential of CFD to overcome the inaccuracies of standard fire models. In this paper, a robust extendable open-source framework is proposed which utilises the capability of the CFD-based fire simulations and performs a sequential thermo-mechanical analysis of structures in fire.

In order to develop an accurate simulation of the development of fire, and the consequent thermo-mechanical response of the structure to that fire over time, in an integrated framework, three different models are required, namely (i) a fire model (e.g., using CFD), (ii) a thermal or heat transfer model, and (iii) thermo-mechanical (structural) model. It is then necessary to couple these models in a sequential manner, which is a challenging task [2]. The proposed methodologies for coupling CFD-generated fires with structural FE models are limited to commercial FE software because of the compatibility of scripts and elements used in the thermal and structural analysis. Moreover, these tools are generally limited in use to the research team responsible for their development and are not capable of further advancements or modification by the general research community. Therefore, the challenge around coupling has become like a so-called black box for researchers and practicing engineers.

3. Computational analysis of fire development and structural behaviour

3.1. OpenSees and FDS

OpenSees, originally developed at the University of California, Berkeley for the simulation of structural response to earthquakes, has expanded into a rapidly-growing community of developers who have advanced its capabilities over the past two decades [9,10]. In recent years, many researchers exploited OpenSees to simulate the thermo-mechanical response of structures in fire [e.g. 11,12]. Fire Dynamics Simulator (FDS) is a free and open-source CFD software package which was developed by the National Institute of Standards and Technology (NIST) of the United States Department of Commerce [13].

As stated previously, providing an accurate, realistic and efficient analysis by any means of the spread of fire through a structure is complex. Independent of this, conducting a nonlinear analysis of the structural behaviour of a system, even under normal conditions, is also challenging. Combining these two elements, and their interaction, with the many salient parameters involved, provides yet more complexity but is essential for the development of improved, more useable and realistic structural fire analysis procedures than currently exist. In the following sub-sections, the simulation tools for open-source analysis of the structural behaviour, fire development, and their interaction, are described. Each of these software uses particular terms to describe functions or inputs required, and these can differ between software, even when referring to similar parameters. Some of the most common terms used in this paper (e.g., Device, Entity) are explained in detail in the *supplement material.*

3.2. Coupling of FDS-OpenSees

The current paper describes how the output determined from the FDS can be employed in OpenSees to provide more realistic boundary conditions for heat transfer analysis and subsequent structural analysis.

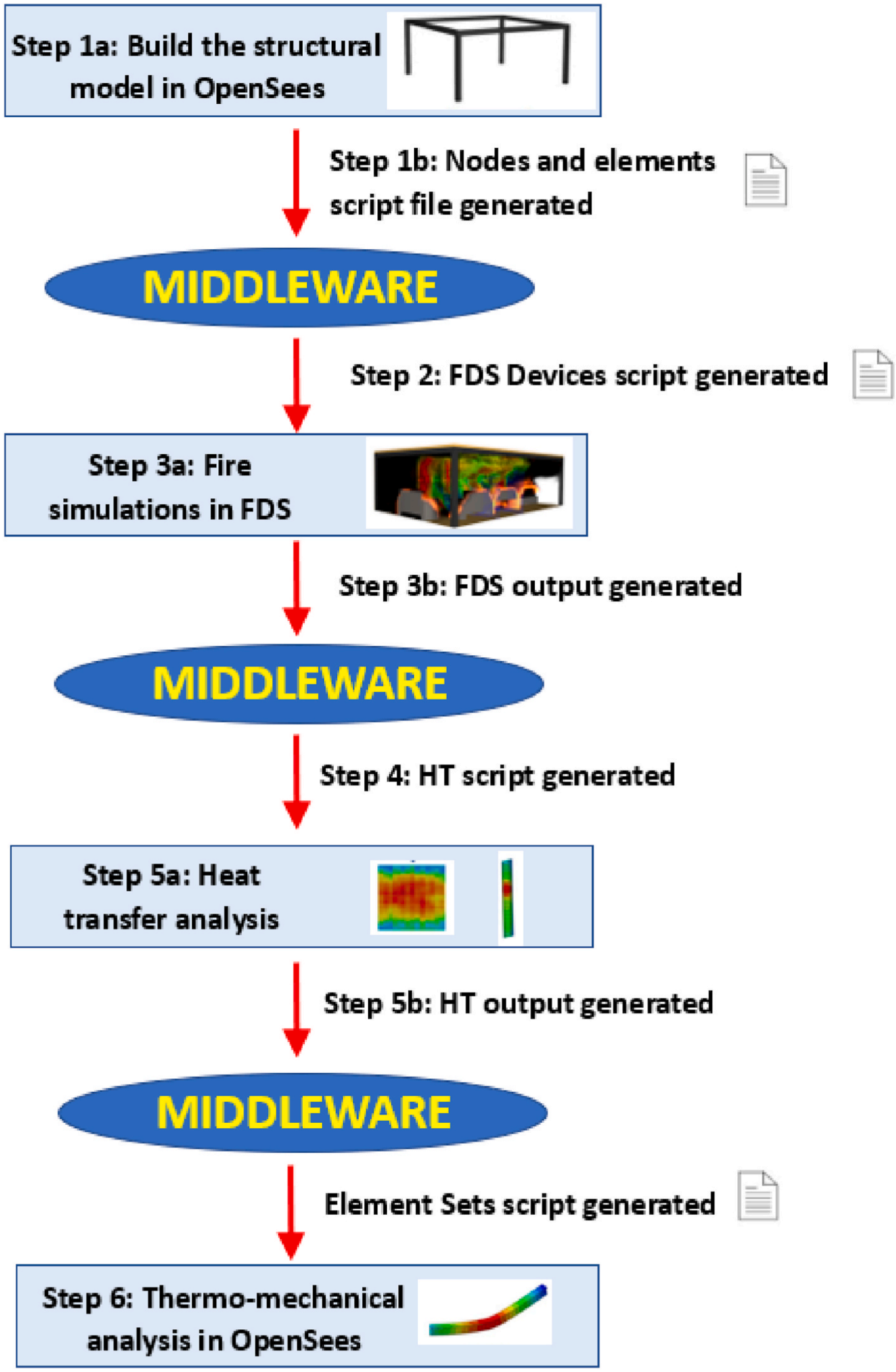


Fig. 1. Graphical illustration of the FDS-OpenSees sequential coupling framework.

Fig. 1 illustrates the process of the proposed coupling and generation of all scripts to conduct all three types of analysis. A middleware is developed to streamline the process of sequential coupling of FDS and OpenSees. The first step in the process is building the structural model in OpenSees (Step 1a in Fig. 1), which produces a script containing the geometrical details (nodes and elements) of the structural model (Step 1b in Fig. 1). This geometrical information from the structural model is used by the middleware to generate the data recording devices in the FDS model at specific locations (i.e. thermocouples, heat flux, AST (adiabatic surface temperature) devices [2], gas temperature devices, etc.), as shown in Fig. 1 (Step 2). The FDS Device script is added to the FDS model and fire simulations are carried out (Step 3a), result in an output file from these simulations (Step 3b). While generating the thermocouples or any other data recording devices, the middleware generates scripts for conducting the heat transfer and thermo-mechanical analysis (HT Script and Element Sets), as shown in Fig. 1 (Step 4) (described in more detail in Section 4.1). A module of the middleware (refer Section 4.2 for more detail) converts the FDS output in the format required by OpenSees (Step 4) to conduct the heat transfer analysis (Step 5a). The output of the heat transfer analysis (Step 5b) is then employed as a boundary condition for the thermo-mechanical analysis (Step 6), conducted in OpenSees using the script (Element Sets) generated by the middleware. The whole process of performing all three analyses using the proposed middleware is explained in the next section.

4. Modules of the middleware

This section describes the development of the middleware which is designed to integrate the FDS with OpenSees to perform all three analyses (fire modelling, heat transfer analysis, and thermo-mechanical analysis) in order to obtain the fire response of the structure. The middleware generates scripts for all three independent analyses (named FDS Devices script, HT script, and Element Sets script in Fig. 1) which are required to establish a seamless interface between the three models with minimal requirements for user-intervention. The middleware includes a number of modules that are written in the Python programming language. The middleware comprises two major modules, which are (i) the *Devices and Elements* module, which generates part of the script for all three analyses such as the part related to the location of the devices for the FDS script, and heat transfer script with elements set corresponding to each device where the output from FDS is applied as thermal boundary condition and elements sets for the structural model script which will receive thermal load from the heat transfer output and (ii) the *FDS2OpenSees* module, which converts the FDS output data in to the format required for OpenSees. A graphical user interface (GUI) is also developed for each module to facilitate the implementation for the user (download from Ref. [14]).

4.1. FDS Devices and other basic inputs

Although it is not required to define the structural components in the FDS model, it is necessary to define the '*quantity*' of data recording devices at the specific locations where data is required to be recorded for input as boundary conditions for the heat transfer and FE models. The FDS Devices script is written using the FORTRAN language. On the other hand, the input files for OpenSees are written in the **.tcl* format. The middleware is capable of converting the scripts in the corresponding formats for both FDS and OpenSees. The solid domain of the FE model contains only the structural components, excluding the fluid domain. Therefore, the data recording device location is the primary link between the FDS fire model and the FE model to perform a coupled heat transfer analysis. To obtain thermal data from the FDS model such as the AST, heat flux (HF) and convective heat transfer coefficients (HTC), it may be required to define the location and orientation of the data recording devices. This module writes a part of the FDS model script defining the device locations based on the geometry of the structural model (Fig. 2). Each device is defined for specific nodes and elements (Fig. 2). The middleware can generate scripts for four *quantities* (i.e., AST, HF, HTC and gas temperatures) to obtain time-variant thermal data of each quantity for conducting the heat transfer analysis in OpenSees.

A typical structure is composed of various members (columns, beams, trusses, slabs, etc.) and the direction of the structural members can be provided in the middleware's GUI. The location of the data recording devices in the selected direction is governed by the number of devices along a direction required for an adequate level of accuracy in terms of the thermal gradients for structural analysis, which allows the user to select an appropriate mesh size for CFD simulations. It is worth

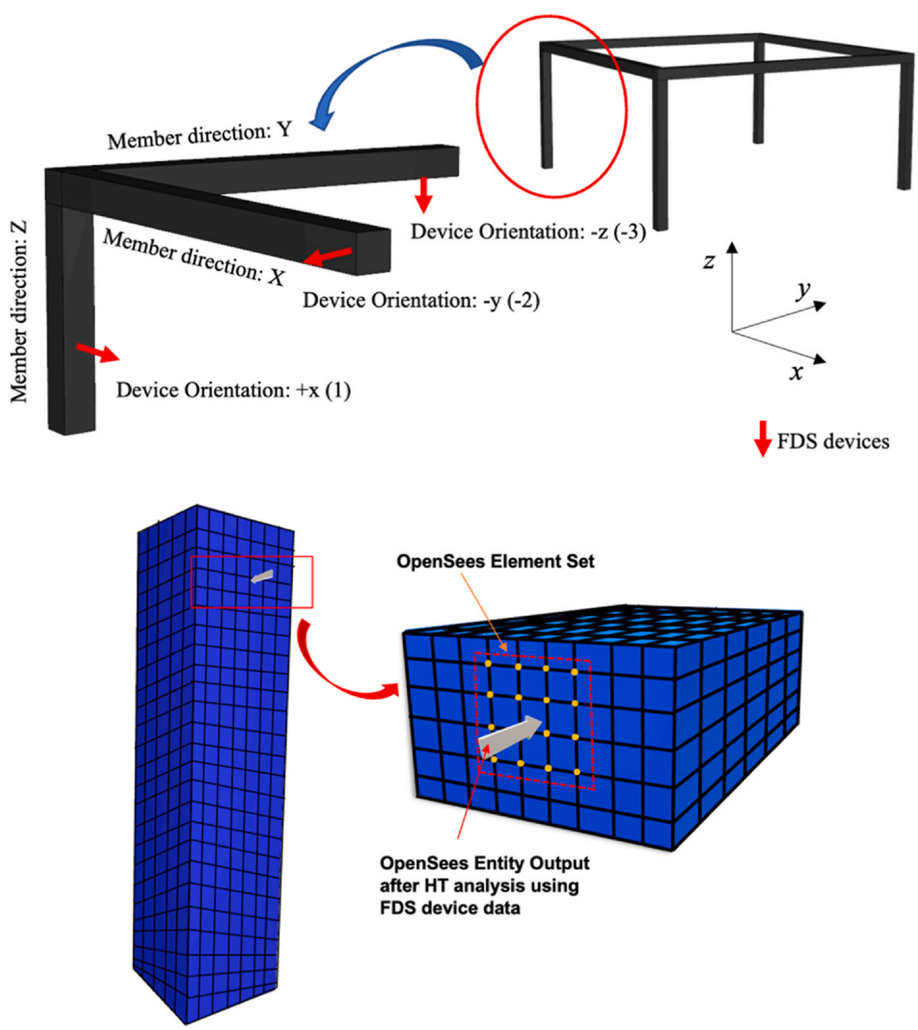


Fig. 2. Orientation of the devices in the FDS and the nodes and elements of structural member.

noting that while generating the model in FDS, by default, the Z-axis is taken as being in the vertical direction (gravity acts along the Z-axis) therefore, it is recommended to follow the same global coordinate system in structural model. The orientation of the devices is also required for the AST, HF, and HTC, based on the outward position of the installed devices (−1, −2, −3, 1, 2, and 3 for -X, -Y, -Z, +X, +Y, and +Z, axis, respectively) [15] (Fig. 2). Once all of the input data is provided, the *Devices and Elements* module produces script containing the device information (the images of the GUI and more details can be found in the supplement material of this paper).

4.2. OpenSees heat transfer entities

To perform the heat transfer analysis in OpenSees, the structural components are defined as *entities* corresponding to the section type, such as I-section, block, and brick entities [16]. The user needs to provide the heat input parameters for the heat transfer analysis, including some fundamental information such as the type of material, heat transfer constants, time scale, faces where thermal boundary conditions are applied, etc.; more detailed information on these is available elsewhere [16]. To determine the time scale, in his PhD thesis, Jowsey [17] recommended using the Biot number of a material, defined as the ratio of the thermal resistance for conduction inside a body to the resistance for convection at the surface of the body, to estimate the characteristic time scale for structural analysis. This approach allows the user to estimate the time scale required in structural analysis for accurate and efficient assessments. A comprehensive list of the input information required to generate the script for the heat transfer analysis in OpenSees can be found in the supplement material.

To perform the heat transfer analysis in OpenSees, six different types of boundary conditions are available for the user to select from, with the combinations of the *quantities* obtained from the FDS, as shown in Table 1. It is important to note that these *quantities* have certain limitations and may not be universally applicable to reflect every fire scenario. For a more comprehensive understanding and specific application of these *quantities*, it is advisable for readers to look for relevant literature [2]. For example, in the AST method, the configuration factor and the emissivity of gases are assumed to have values equal to unity, which provides a reasonable approximation in scenarios involving flaming regions or rooms filled with soot. Similarly, when selecting the gas temperatures for the boundary condition, these are also considered as radiation temperatures, which may differ from the gas temperatures. A full discussion on the effect of these limitations and assumptions is provided in the review paper published by Khan et al. [2]. This paper also discusses the applicability of several methods used as boundary conditions for structural analysis.

In the first method, only the time-variant AST with a fixed value of the convective heat transfer coefficient (HTC) (h_c) are used for the heat transfer analysis. By applying the AST as a boundary condition, the time-temperature history for the structure can be obtained (similarly for other inputs presented in Table 1). The module creates an entity in the heat transfer model corresponding to each data recording device in the FDS model. The AST obtained from individual devices in the FDS is then applied to the corresponding entity in the heat transfer model as thermal boundary conditions, to obtain the time-temperature history. This time-temperature output, which is generated by the heat transfer model, can be used as the thermal load to conduct the thermo-mechanical analysis of the structure in OpenSees.

Table 1
Boundary conditions for heat transfer analysis in the current module.

Method	Boundary condition	Value of convective heat transfer Coefficient	Type of fire in OpenSees HT file
1	AST	Fixed	1
2	HF	Fixed	3
3	AST + HTC	Varying	2
4	HF + HTC	Varying	4
5	GAS	Fixed	1
6	GAS	Varying	2

4.3. Element Sets

After performing the heat transfer analysis, the output data must be transferred to the OpenSees structural model for the thermo-mechanical analysis. In OpenSees, there is no coupling feature to map the data from the heat transfer analysis to the thermo-mechanical analysis, in contrast to commercial FE packages such as Abaqus and ANSYS. The middleware allows for a seamless mapping of the thermal data from the heat transfer (HT) analysis to the thermo-mechanical FE model. The output data from the HT analysis is applied as the thermal loads to the structural elements. Once the HT analysis is complete, output files are generated based on the locations where the temperature history is required to perform the thermo-mechanical analysis. The thermal data in each file is based on the number of nodes ('*nodesets*') required along the depth of the entity. The output files containing the time-temperature history for each entity containing the '*nodeset*' data is applied to the structural elements at the exact geometrical location as in the heat transfer model (and in the FDS model). The middleware is capable to identify elements associated with a particular geometrical location where the entity output is then be assigned as the thermal loads. While generating the devices and OpenSees section entities, the middleware assigns unique identification numbers to the output files of the HT analysis based on the device location and elements of the structural FE model. Finally, the module generates a part of the script for the thermo-mechanical model which contains the information associated with the application of thermal loads to the corresponding structural elements. To apply the thermal loads to beam-column or shell elements, the temperature field should be defined at specific data points across the depth of the section to capture thermal gradients.

Once the script files generated from the middleware, the CFD file can be updated with other details such as the geometry of the fire compartment, fire load, fire size, reaction, etc, [15]. Similarly, the structural analysis script can be updated with other necessary details. This framework streamlines the process of mapping the data from the CFD to the FEM models. Now, the scripts for all three analyses are generated for sequential coupling. The CFD simulation in FDS using the script generates the output for the heat transfer analysis. The output from the FDS is in the form of time-varying heat fluxes or a time-temperature history.

4.4. FDS2OpenSees module

This module converts the FDS output data into the format required for OpenSees. It is required to pre-process the output data and convert it into a suitable format for the HT analysis. For the HT analysis in OpenSees, a separate file containing the time-varying data for properties such as temperature or heat flux for each HT entity (in "*.*dat*" format), is required. On the other hand, the FDS provides a single "*.*csv*" file containing all data for the selected data recording devices. This module generates the necessary input file for each entity.

After the input boundary files are created using the module, OpenSees is run to conduct the HT analysis [16]. The output from the HT analysis provides input files for subsequent thermo-mechanical analysis. These files are used as the thermal load acting on the elements. The name of each file is already specified in the file generated using the middleware.

5. Validation and analysis

The proposed framework for the fully integrated analysis of structures under realistic fire conditions is validated and further examined for three different scenarios. The first two are experimental studies, whilst the third case employs the data from a real accidental fire. With reference to the latter case, this is included to assess the capability of the tool for forensic examinations.

5.1. Case 1: Square hollow section steel column

Kamikawa et al. [18] carried out a series of four experiments on square hollow section (SHS) steel columns, where the members were exposed to a pool fire (fuelled by propane). All of the test specimens were made of STKR400 SHS sections which had a cross-section of 0.1 m × 0.1 m, a thickness of 3.2 mm and an overall height of 1.6 m. Each specimen had different loading and boundary conditions although in all cases, the base of the columns was fixed. The specimens were exposed to fire loading as shown in Fig. 3. Although all of the tests were examined in the current study, just one set of data is presented herein (Test 4) for illustration. In this experiment, the column was fixed at the base and lateral (horizontal) restraint was provided towards the top of the column, 1400 mm from the base. During the test, thermal expansion and bending behaviour were observed when the column was exposed to 1 h of fire loading. Once the temperature in the steel column reached a steady state (Fig. 4(a)) which was around 52 min after ignition, a vertical force was applied to the top of the column and increased until failure occurred. The observed failure mode was local buckling in the cross-section.

An FDS model (computational domain, i.e. simulated compartment size, of 0.75 × 0.45 × 1.8 m^3) was generated using the data obtained from the experiments as shown in Fig. 4(b). A propane fuelled burner which was 0.25 m in height was placed near one face of the column as in the experiment. The heat release rate (HRR) was set to 52.5 kW in the FDS simulation. Using the middleware, the "*brick*" heat transfer entities were used to model the SHS column (0.1 m × 0.1 m × 3.2 mm and 1.6 m in height) in OpenSees [20]. The output of the FDS simulations was used to conduct a 3D heat transfer analysis in which the conduction of heat along the length of the structural member and across the width and depth was considered. The results obtained after the heat transfer analysis was then compared with the experimental data, as shown in Fig. 4(a). It is clear from these results that the temperature distributions were highly non-uniform across the section and along the length of the column in the test, and this is well represented by the numerical analysis. Only the front face was in direct contact with the flame, and the temperature increased on the other faces of the square hollow section, due to conduction, radiation, and cavity radiation. It is noteworthy that the temperature development at the back and side faces of the section slightly lag in the numerical results compared to the experimental temperatures. This is because the cavity radiation effect was not included while conducting the heat transfer analysis; this is a current limitation of OpenSees). Nevertheless, this has a relatively minor influence on the accuracy of the simulation, as evident from the data in Fig. 4.

To obtain the structural response using the proposed framework, a thermal stress analysis was conducted by importing the time-temperature history obtained after heat transfer analysis to the thermo-mechanical model in OpenSees. In order to trace the yielding and global buckling behaviours of the columns as observed in the experiment, the SHS column was modelled using beam-column elements, with a total of 160 elements, each of which was 10 mm in length. In Test 4, which was laterally restrained at the top of the column, a vertical concentrated load was applied at the top of the column when temperatures reached a steady-state, and this load was gradually increased until the column failed. Fig. 4(b) presents the vertical deflection versus time from both the experiment [18] and the proposal numerical model in OpenSees. The maximum vertical deflection obtained from OpenSees was 5.4 mm, which compares very favourably with the corresponding experimental value of 5.6 mm. In addition, the OpenSees model presented in this paper predicted that failure would occur at an applied load equal to 380 kN, which was within 1.5% of the experimental failure load of 375 kN).To demonstrate the capability of the proposed FDS-OpenSees framework in comparison with other

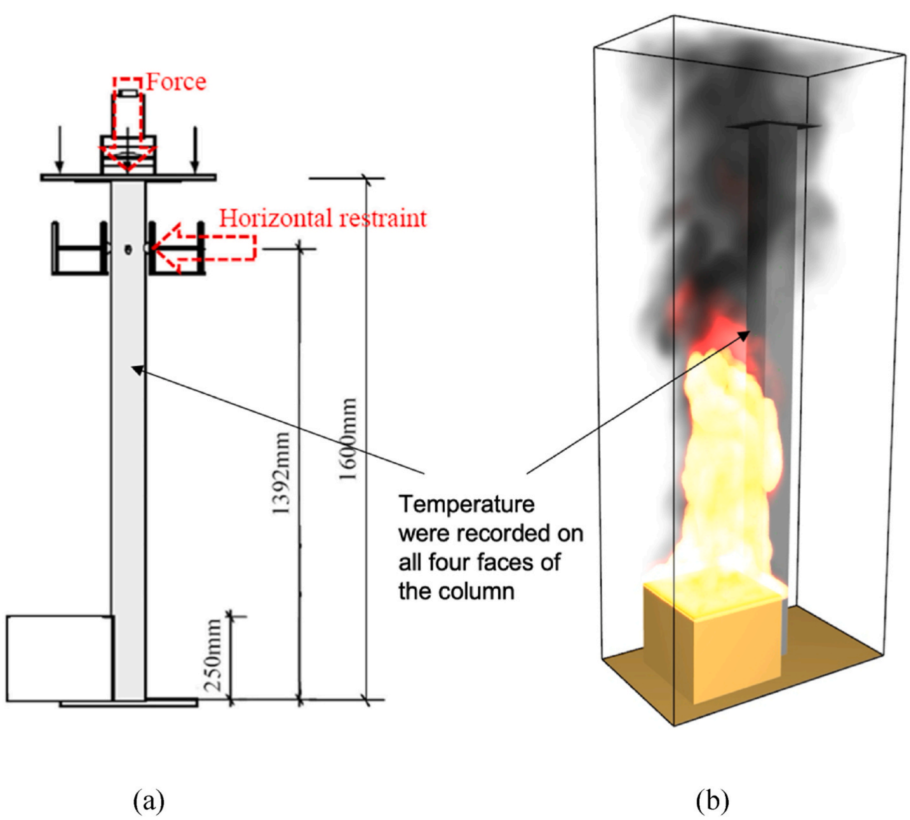


Fig. 3. Images of (a) the experimental set up of Kamikawa et al. [18] and (b) the computational domain employed in the simulation.

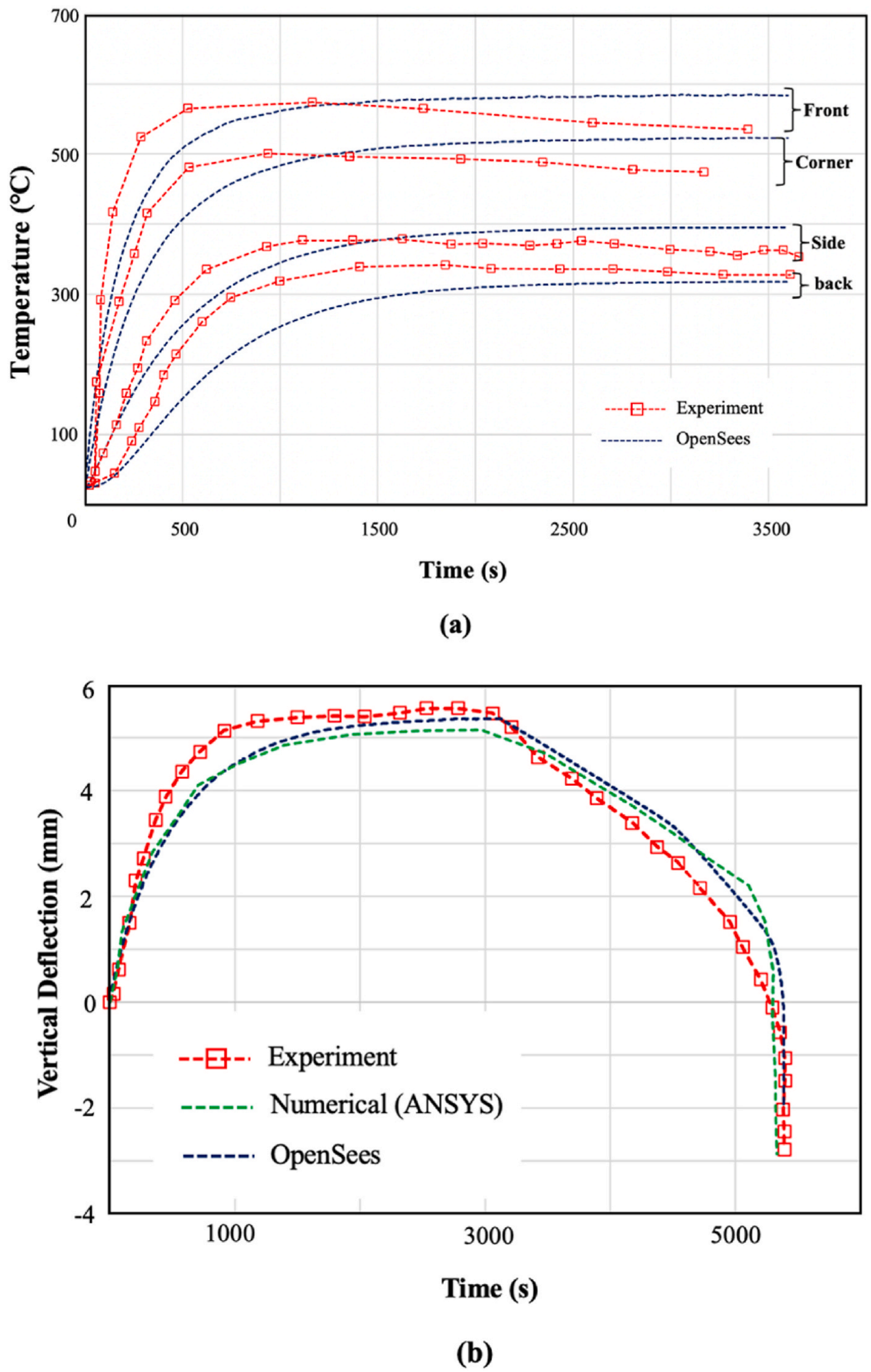


Fig. 4. Comparison between the experimental results [18] and the numerical analysis [19] using the proposed framework, including (a) the temperature development at various locations in the column versus time, and (b) the vertical deflection versus time.

commercially available software, Fig. 4(b) includes the results from another study conducted by Zhang et al. [19], who coupled FDS with ANSYS in their analysis. As shown in Fig. 4(b), it is clear that OpenSees produces similar results as ANSYS, with the significant added advantage of being open-source and freely available software.

5.2. *Case 2:* Steel beam in a localised fire

Wakamatsu et al. [21] experimentally investigated the thermal impact of localised fires on structural members to understand the effects of flame and smoke on the structure, and the set-up is presented in Fig. 5(a). Three different HRRs were examined in the experiments, namely 569 kW, 848 kW, and 1127 kW. In these tests, temperatures and heat fluxes were recorded over a 6 m long steel beam made from an H-section, which was 400 mm and 15 mm in web depth and thickness, respectively, and 200 mm and 13 mm in flange width and thickness, respectively. Smoke soffits with a depth of 1 m deep were employed at every edge of the slab to investigate the effects of the smoke layer; more details can be found elsewhere [21]. The dimensions used in the FDS model are shown in Fig. 5(b).

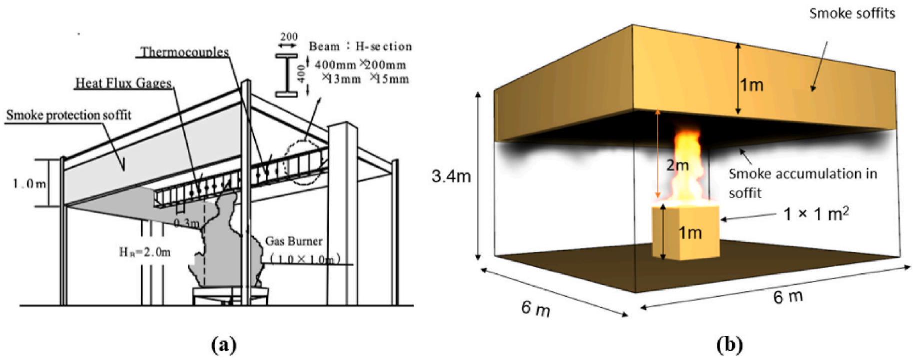


Fig. 5. (a) Experimental set up [21], and (b) computational domain.

In all three tests, propane was used as the fire source, and it was injected through a porous sand gas burner in the experiments. The chemical composition of the other products, such as carbon (C to CO fraction = 0.85) and hydrogen content (H to H_2 fraction = 0.5), were taken from the experiment conducted by NIST for validation purposes [22]. After performing the mesh sensitivity study, the element size was

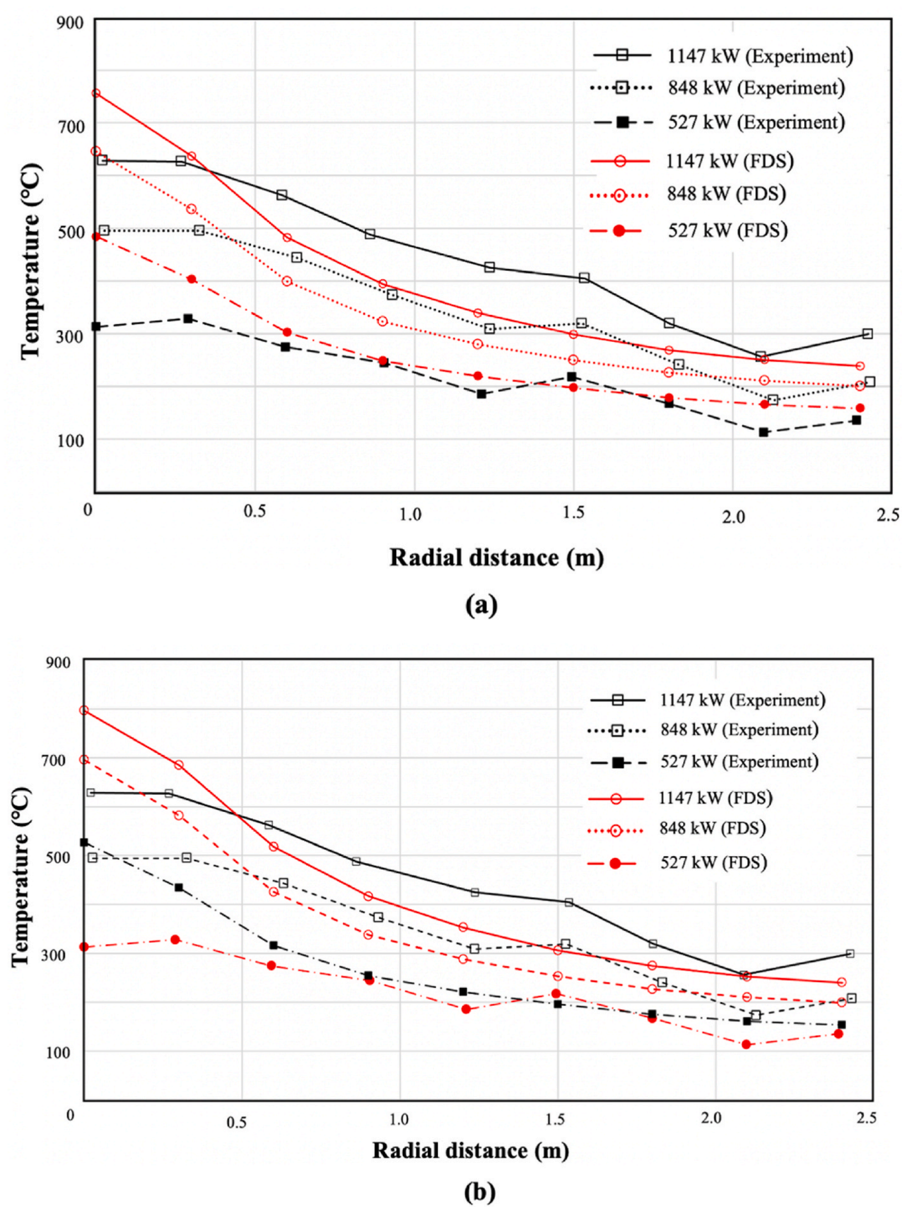


Fig. 6. Comparison between the experimental results [21] and the proposed numerical framework for the temperatures obtained for (a) AST, and (b) heat flux boundary conditions.

selected to be 0.05 m for the burning region] and a larger size of 0.2 m for all other areas external to the fire. In the FDS model, a total of 18 measurements were taken to calculate the adiabatic surface temperature T_{AST} and to gauge the heat fluxes q''_{gauge} (HF). Due to symmetry in the computational model, only nine measuring devices were required for each quantity (AST and HF).

Using the proposed middleware, FDS devices were generated which then created an OpenSees script for conducting the heat transfer analysis. Using the I-section entity, a 2D heat transfer analysis was performed. The surface temperatures of the beams which were recorded during the tests are compared with those obtained from the FDS-OpenSees analysis and are presented in Fig. 6(a) and (b) for the AST and heat flux, respectively. The temperature distributions along the length obtained for all three magnitudes of HRR from the heat transfer analysis are presented. It is observed that the temperatures obtained just above the fire source, which was at the centre of the compartment, from the FDS model are greater than those recorded during the experiments by a significant margin. This is likely associated with the fact that FDS uses semi-empirical wall functions to approximate the convective heat transfer coefficient, which is required to calculate the AST and heat fluxes. Given that the flow at the wall is not well-resolved in these simulations, it is expected that this point will be a source of error, particularly since resolving near-wall flows requires very small mesh sizes that are not consistent with the geometries of large compartments.

5.3. Case 3: *I-65 Birmingham bridge fire incident*

In 2002, a tanker truck which was carrying 37.5 m^3 of gasoline was travelling on the I-65 Birmingham bridge (Fig. 7(a)) in the USA when crashed into one of the bridge piers. Although the piers supporting the girders survived the tanker impact, the resulting fire severely damaged the bridge girders within a few minutes of elevated temperature exposure. During the following investigation, it was found that one of the girders was heavily damaged and deflected up to 2.5 m [23,24] during the fire. In the current work, this case is simulated using the proposed framework to validate the approach for an actual fire incident. A simplified geometrical FDS model with a computational domain (compartment size) of 86 × 40 × 14 m^3 was developed for the I-65 Birmingham Bridge, as shown in Fig. 7(b). The input parameters, such as the material properties and the soot yield, were obtained from an article by Also-Moya et al. [23].

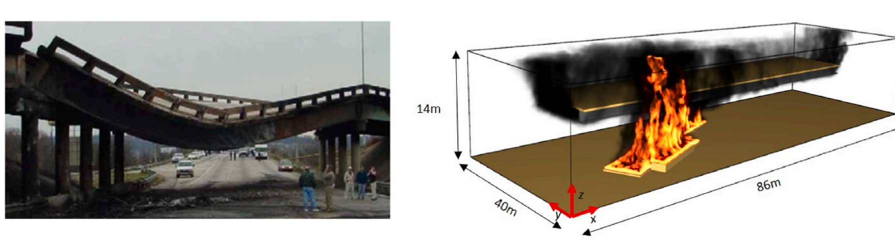
A cell size of 0.2 m was used in the fire simulations following a mesh sensitivity analysis. The tanker was considered as a 'burner' of size 12 × 2.5 × 1 m^3, where the HRR per unit area was assigned as 2500 kW/m^2 at the tanker, and 1000 kW/m^2 for the spill region [23]. To capture the temperature profiles along the bridge span, AST devices were installed at intervals of 1 m. Using the structural geometry, a total of 48 AST measuring devices were placed in the fire simulation, which created the same number of OpenSees heat transfer entities. Half of the AST devices recorded the temperatures on the steel girder, whilst the other half recorded the same data on the deck. The fire load location was assumed in accordance with the data obtained from the fire incident to validate the model in terms of failure time. The fire simulations were conducted for two different fire intensities, namely 1000 kW and 1500 kW, to analyse the effect of this parameter on the structural performance of the bridge.

Using the data obtained from the FDS fire model, the heat transfer analysis is conducted using AST values as the boundary condition. The central span of the bridge, which was 37 m in length and was mainly exposed to the fire, comprised a steel girder made from material with a yield strength of 350 N/mm^2 and a reinforced concrete deck, which was 170 mm in thickness. The concrete in the deck had a compressive strength of 40 N/mm^2 and was connected to the girder using shear studs. The other information of the structural model can be found elsewhere [23,24]. A FE model of the bridge was developed using OpenSees to conduct the heat transfer analysis, using block type entities to model the steel girder and the concrete deck. The thermal properties of the concrete and steel were defined in accordance with the values given in the Eurocodes [25,26]. The temperature profiles across the sections above the location of the vehicle after conducting the heat transfer analysis for a 1500 kW intensity fire are shown in Fig. 8(a).

It is observed that the temperatures of the steel girder are significantly greater compared to the temperatures at the bottom surface of the concrete deck throughout the fire, which is attributed to the high specific heat and lower thermal conductivity of concrete compared to steel. Therefore, a steep thermal gradient is induced in the composite section. The maximum temperatures in the steel girder and at the bottom surface of the deck were 1040 °C and 750 °C, respectively, as shown in Fig. 8. Moreover, the temperatures at the mid-depth and top surface of the deck were significantly lower compared to the bottom surface of the deck, resulting in a local thermal gradient within the concrete deck also.

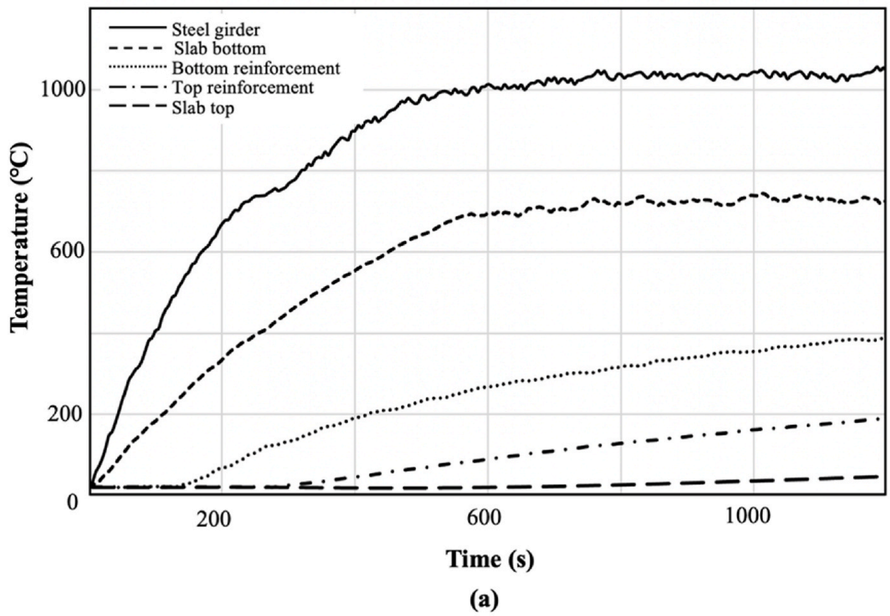
The output of the heat transfer analysis was then employed for a sequential thermo-mechanical analysis in OpenSees. In the OpenSees FE model, the mechanical properties of concrete and steel at elevated temperatures were adopted in accordance with the values given in the Eurocodes [27,28]. Shell elements (ShellMITC4Thermal) which are available in the OpenSees material library were utilised to model the steel girder and the concrete deck. The reinforcement in the concrete deck was modelled using the smeared layer approach. Shear studs are represented assuming a rigid connection between concrete deck and steel girder. The thermal history obtained from the OpenSees heat transfer analysis was imported into the thermo-mechanical model as the thermal load at all locations, using the middleware.

The maximum deflections of the span are shown in Fig. 8(b) for both a 1000 kW fire and a 1500 kW fire. In this image, it is shown that despite the simply-supported boundary conditions of the girder, the rate of increase of deflection during the initial phase of the fire was very high due to the significant thermal gradient within the overall composite bridge section as well as just within the concrete deck. It is clear that when the bridge was exposed to a 1500 kW fire, the failure occurred much sooner after around 5 min (300 s) of fire, whereas failure occurred after approximately 12 min (720 s) for the 1000 kW fire. For the 1500 kW fire, after 5 min of fire exposure, the maximum temperature in the steel girder reached around 800 °C while the deck's temperatures were



(a) (b)

Figs. 7. I-65 Birmingham bridge fire incident [23] and the computational domain employed in FDS.



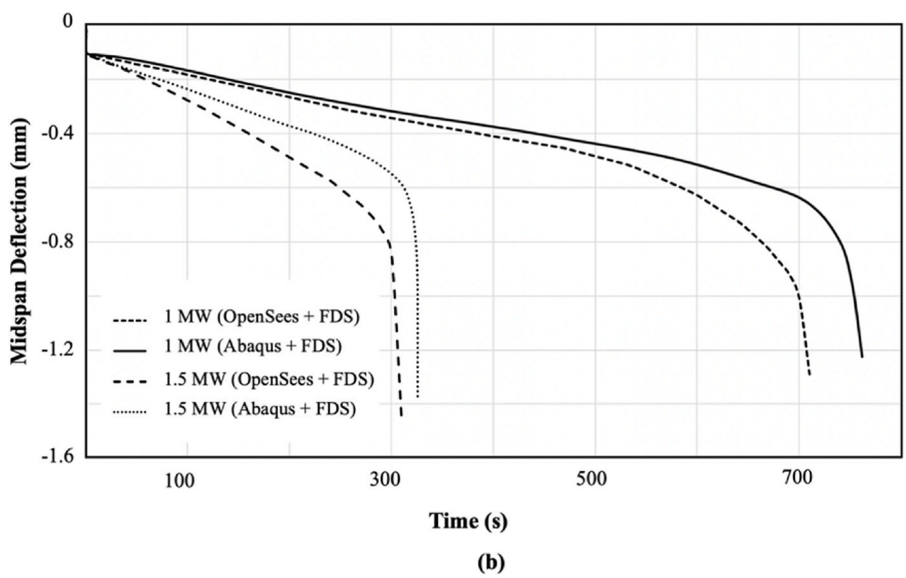


Fig. 8. (a)Temperatures obtained at various locations in the bridge section after conducting the heat transfer analysis for a 1500 kW intensity fire, (b) Midspan deflections obtained from FDS + OpenSees and FDS + Abaqus [23] for 1000 kW (1 MW) and 1500 kW (1.5 MW) fire.

significantly lower at 450 °C at the bottom surface and 30 °C at the mid-depth, as shown in Fig. 8(a). The overall deck temperature was relatively low compared to temperatures in the steel girder, therefore the strength reduction of the steel girder was much more significant compared to the concrete deck. This is the main reason, with reference to Fig. 8(b), that there was a sudden increase in deflection after 5 min of temperature exposure.

The overall behaviour for the 1000 kW fire was quite similar to that of the 1500 kW fire, as observed in Fig. 8(b). In this case, the bridge failed after around 12 min of fire exposure when steel girder and bottom of concrete deck temperature reached 800 °C and 400 °C, respectively, similar to the 1500 kW fire case. The longer survival time for the bridge exposed to a 1000 kW fire is attributed to the slower rate of increase of temperature. The bridge's failure after 12 min of 1000 kW fire exposure as predicted by the proposed OpenSees thermo-mechanical analysis also confirmed by previous analysis on this fire which is available in the literature, using FDS and the commercial FE software Abaqus [60, 61]. Therefore, it is concluded that the proposed framework is applicable for an accurate prediction of the behaviour during real fire incidents.

6. Current limitations of the proposed framework

Although the proposed analysis framework is shown in the current paper to be capable of conducting an accurate thermo-mechanical analysis even for large structures, it is important to acknowledge and note the assumptions and limitations which still remain. Some of these are related to limitations of the current numerical capabilities of OpenSees whilst others are related to the middleware. With reference to the latter, researchers can readily address these shortcomings if they are important for a given application owing to the open-source nature of the middleware. Moreover, it is expected that the middleware can remain in a continuous state of development and improvement, for different applications and computational abilities, within the open-source environment.

The current limitations of the framework, are identified as:

- To capture gradients of a quantity (thermal or any other measuring quantity such as heat transfer coefficient, heat fluxes) along the length, the mesh size of the CFD domain must be smaller than the spatial resolution required for the structural analysis. FDS computes a gas-phase quantity in a cell volume. If the cell size is bigger than the structural component, only a single value of the quantity is obtained from the CFD simulation. For example, if different quantities are needed for the flange and web of an I-beam, both the flange and the web must be present in different cell volumes.
- The current version of OpenSees is not capable of capturing the radiation through cavities (such as in a hollow section).
- To represent the structural geometry (i.e. the model used in the FE software which represents structural components such as beams, columns, etc.), currently only a few entities are available in the OpenSees software, however users can create a "*user-defined*" entity to overcome this issue.
- Currently, solid continuum elements which may be required to simulate high resolution structural problems, are not available in the fire analysis version of OpenSees.
- Due to the continuous consumption and potential alteration of the structural geometry in timber structures during a fire, the application of sequential coupling may yield less realistic results in this scenario, particularly in the later stages of the fire. To accurately represent the behaviour of a timber structure in fire may require two-way coupling which can account for these changes in the structure during and because of the fire.

7. Conclusions

This paper has presented a comprehensive, integrated and sophisticated framework for the analysis of structures due to realistic fire events. The key contribution of this paper, over existing work, is that the framework is developed entirely using open-source software, and hence is not limited to what is available in commercial packages, and can be used by anyone, freely. To date, the more complex existing packages for analysing structural response to fire required access to expensive software as well as knowledge of the complex software, and their limitations. The work presented herein is largely motivated by a desire to provide a workable solution for all structures and engineers. Due to the uniqueness of each building and environment, as well as the recent trend towards performance-based designs, it is imperative for engineers to be able to implement a realistic fire scenario during their analysis. CFD has emerged as the most appropriate tool for capturing a realistic fire scenario. The proposed framework presented herein achieves the following key outcomes:

- The package presented in this paper is the first and only open-source and free of cost package for CFD-FEM coupled analysis for a structure in fire. All of the links to source codes and instructions which are required for the middleware are provided in the paper. It makes it an ideal tool for both researchers and practising engineers.
- Data mapping from the CFD to FEM is simplified compared with existing methods and also reduces the chances of human error while defining the boundary conditions for the heat transfer and the thermo-mechanical analysis.
- GUIs are created for the whole process, making it easier to use the middleware for users who have no programming experience.
- Different types of boundary conditions can be provided to conduct the heat transfer analysis such as AST, heat fluxes and gas temperatures.
- Three different cases are used to validate the software package, including an experiment studying the fire behaviour of SHS column exposed to a pool fire, an experiment to evaluate the impact of localised fires on structural members and also a real bridge fire incident.
- Excellent agreement is achieved in the validation cases between the experiments/real fire and the simulated predictions from the proposed framework. In addition, the real fire incident highlights the capability of the proposed framework in terms of generating realistic fire scenarios for a variety of structural forms.
- The tool can also be used for conducting forensic investigations of a structural failure due to fire such as occurred at the World Trade Centre and the Plasco Building.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

No data was used for the research described in the article.

Acknowledgement

The work reported in this paper has formed part of the SureFire project (T22-505/19-N) funded by the Research Grants Council Hong Kong under its Theme-based Research Scheme. This research is also funded by the RGC Hong Kong GRF Scheme and HKPolyU.

Appendix A. Supplementary data

Supplementary data to this article can be found online at https://doi.org/10.1016/j.firesaf.2023.103896.

References

[1] X. Dai, S. Welch, O. Vassart, K. Cábová, L. Jiang, J. Maclean, et al., An extended travelling fire method framework for performance-based structural design, Fire Mater. 44 (2020) 437–457.
[2] A.A. Khan, A.S. Usmani, J.L. Torero, Evolution of fire models for estimating structural fire-resistance, Fire Saf. J. 124 (2021) 103367.
[3] C. Clifton, Fire Models for Large Firecells, 1996. Auckland, New Zealand.
[4] J. Stern-Gottfried, G. Rein, Travelling fires for structural design-Part II: design methodology, Fire Saf. J. 54 (2012) 96–112.
[5] EN-1992-1-1, Eurocode 1 : Actions on Structures Exposed to Fire, vol. 2, 2011.
[6] S. Welch, S. Miles, S. Kumar, T. Lemaire, A. Chan, FIRESTRUC - integrating advanced three-dimensional modelling methodologies for predicting thermo-mechanical behaviour of steel and composite structures subjected to natural fires, Fire Saf. Sci. (2008) 1315–1326.
[7] NFPA, NFPA 557 : Standard for the Determination of Fire Loads for Use in Structural Fire Protection Design, 2020.
[8] M.A. Khan, A.A. Khan, A.S. Usmani, X. Huang, Can fire cause the collapse of Plasco Building: a numerical investigation, Fire Mater. 46 (3) (2021) 560–575.
[9] N. Elhami Khorasani, M.E.M. Garlock, S.E. Quiel, Modeling steel structures in Open-Sees: enhancements for fire and multi-hazard probabilistic analyses, Comput. Struct. 157 (2015) 218–231.
[10] M.H. Scott, G.L. Fenves, F. McKenna, F.C. Filippou, Software patterns for non-linear beam-column models, J. Struct. Eng. 134 (2008) 562–571.
[11] M.A. Khan, L. Jiang, K.A. Cashell, A. Usmani, Analysis of restrained composite beams exposed to fire using a hybrid simulation approach, Eng. Struct. 172 (2018) 956–966.
[12] M.A. Khan, L. Jiang, K.A. Cashell, A. Usmani, Virtual hybrid simulation of beams with web openings in fire, J. Struct. Fire Eng. 11 (1) (2019) 118–134.
[13] K. Mcgrattan, R. McDermott, S. Hostikka, J. Floyd, M. Vanella, Fire Dynamics Simulator Technical Reference Guide, vol. 1, Mathematical Model, 2019.
[14] OpenFIRE, OpenFIRE. https://github.com/aatif85?tab=repositories 2021.
[15] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, K. Overhold, Sixth Edition Fire Dynamics Simulator User 's Guide (FDS), NIST Special Publication, 2016, p. 1019 (Sixth Edit).
[16] OpenSEES, OpenSEES for fire. http://openseesforfire.github.io/user.html n.d.
[17] A. Jowsey, FIRE IMPOSED HEAT FLUXES FOR STRUCTURAL ANALYSIS, The University of Edinburgh, 2006.
[18] D. Kamikawa, Y. Hasemi, K. Yamada, Nakamura, Mechanical response of a steel column exposed to a localized fire, Proc. fourth Int. Work. stuctures fire, Aveiro, Port. (2006) 225–234.
[19] C. Zhang, J.G. Silva, C. Weinschenk, D. Kamikawa, Y. Hasemi, Simulation methodology for coupled fire-structure analysis: modeling localized fire tests on a steel column, Fire Technol. 52 (2016) 239–262.
[20] OpenSEES, OpenSEES for fire. http://openseesforfire.github.io/user.html n.d.

[21] T. Wakamatsu, Y. Hasemi, A.V. Ptchelintsev, Heating mechanism of building components exposed to a localized fire - FEM thermal and structural analysis of a steel beam under ceiling -, Proc. Int. Conf. Offshore Mech. Arctic Eng. - OMAE 2 (1997) 51–58.
[22] K. Mcgrattan, FDS Validation: NIST Pool Fires, Add Propane Results, 2013.
[23] J. Alos-Moya, I. Paya-Zaforteza, M.E.M. Garlock, E. Loma-Ossorio, D. Schiffner, A. Hospitaler, Analysis of a bridge failure due to fire using computational fluid dynamics and finite element models, Eng. Struct. 68 (2014) 96–110.
[24] M.A. Khan, A.A. Khan, R. Domada, A. Usmani, Fire hazard assessment, performance evaluation, and fire resistance enhancement of bridges, Structures 34 (2021) 4704–4714.
[25] CEN. Eurocode 2, Design of Concrete Structures Part 1-1: General Rules and Rules for Buildings, 2015.
[26] EUROCODE. Eurocode 3. Design of Steel Structures. General rules. Structural fire design. (n.d).
[27] EN1992, Eurocode 2: design of concrete structures - Part 1-1, General rules and rules for build. Eurocode (2011) 1.
[28] EN 1993-1-1, Eurocode 3: Design of Steel Structures - Part 1-1: General Rules and Rules for Buildings, European Commitee for Standardization, 2005.