Engineering Structures 172 (2018) 956–966

Contents lists available at ScienceDirect

Engineering Structures

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

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

ARTICLE INFO

Keywords:
Structure in fire
Thermo-mechanical analysis
OpenSees
Multiple scale modelling

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.

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

* Corresponding author.
E-mail address: liming.jiang@polyu.edu.hk (L. Jiang).

https://doi.org/10.1016/j.engstruct.2018.06.048
Received 13 January 2018; Received in revised form 6 May 2018; Accepted 13 June 2018
0141-0296/

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