THE UNIVERSITY of EDINBURGH

Edinburgh Research Explorer

OpenSees software architecture for the analysis of structures in fire

Citation for published version:
Jiang, J, Jiang, L, Kotsovinos, P, Zhang, J, Usmani, A, McKenna, F & Li, G-Q 2015, 'OpenSees software architecture for the analysis of structures in fire', *Journal of Computing in Civil Engineering, ASCE*, vol. 29. https://doi.org/10.1061/(ASCE)CP.1943-5487.0000305

Digital Object Identifier (DOI):
10.1061/(ASCE)CP.1943-5487.0000305

Link:
Link to publication record in Edinburgh Research Explorer

Document Version:
Publisher's PDF, also known as Version of record

Published In:
Journal of Computing in Civil Engineering, ASCE

General rights
Copyright for the publications made accessible via the Edinburgh Research Explorer is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy
The University of Edinburgh has made every reasonable effort to ensure that Edinburgh Research Explorer content complies with UK legislation. If you believe that the public display of this file breaches copyright please contact openaccess@ed.ac.uk providing details, and we will remove access to the work immediately and investigate your claim.

OPEN ACCESS

Download date: 26. Apr. 2024

OpenSees Software Architecture for the Analysis of Structures in Fire

Jian Jiang[1]; Liming Jiang[2]; Panagiotis Kotsovinos[3]; Jian Zhang[4]; Asif Usmani[5]; Frank McKenna[6]; and Guo-Qiang Li[7]

Abstract: Computational modeling of structures subjected to extreme static and dynamic loads (such as snow, wind, impact, and earthquake) using finite-element software are part of mainstream structural engineering curricula in universities (at least at graduate level), and many experts can be found in industry who routinely undertake such analyses. However, only a handful or institutions around the world teach structural response to fire (at any level) and only a few of the top consulting engineers in the world truly specialize in this niche area. Among the reasons for this are the lack of cheap and easily accessible software to carry out such analyses and the highly tedious nature of modeling the full (often coupled) sequence of a realistic fire scenario, heat transfer to structure and structural response (currently impossible using a single software). The authors in this paper describe how finite-element software can be extended to include the modeling of structures under fire load. The added advantage of extending existing finite-element codes, as opposed to creating fire-specific applications, is due to ability to perform multihazard type analysis, e.g., fire following earthquake. Due to its open source nature and object-oriented design, the *OpenSees* software framework is used for this purpose. In this work, the *OpenSees* framework, which was initially designed for the earthquake analysis of structures, is extended by the addition of new concrete classes for thermal loads, temperature distributions across element cross sections, and material laws based on Eurocodes. Through class and sequence diagrams, this paper shows the interaction of these classes with the existing classes in the *OpenSees* framework. The performance of this development is tested using benchmark solutions of a single beam with finite stiffness boundary conditions and a steel frame test. The results from *OpenSees* agree well with analytical solutions for the benchmark problem chosen and provide reasonable agreement with the test. The experience with *OpenSees* so far suggests that it has excellent potential to be the basis of a unified software framework for enabling computational modeling of realistic fires, and further work is continuing towards the achievement of this goal. The extensions made to *OpenSees* described in this work, in keeping with the open source ideals of the framework, have been included in the current *OpenSees* code and are available for researchers and practicing engineers to test, develop, and use for their own purposes. **DOI: 10.1061/(ASCE)CP.1943-5487.0000305.** © *2014 American Society of Civil Engineers.*

Author keywords: Computational modeling; *OpenSees*; Thermomechanical analysis; Software architecture; Class diagram; Sequence diagram.

Introduction

The traditional approach of evaluating the fire resistance of structures (based on prescriptive building codes) is by testing individual structural members under a standard fire [such as ISO-834 (ISO 1992) and ASTM-E119 (ASTM 2007)], where member capacity is associated with a limiting temperature. This approach does not consider natural fire scenarios and the enormous associated uncertainties, and furthermore the behavior of structural members in isolation entirely ignores the structural interactions a member would experience as part of the whole structure. The unscientific nature of prescriptive approaches has led to gradual and accelerating adoption of so-called *performance-based design* or, more accurately, *performance-based structural engineering* (PBSE) approaches, characterized by much greater reliance on scientific understanding and numerical modeling technologies. Admirable research advances have been made towards applying these methodologies in the field of earthquake engineering, most notably the Pacific Earthquake Engineering Research Center—performance-based earthquake engineering (PEER-PBEE) methodology (Deierlein et al. 2003). However, to enable the application of an equivalent PBSE methodology for engineering structural fire resistance, considerable further development of modeling technologies is required. This is because modeling tools for simulating fire, heat transfer to structural components and structural response are typically separate and unconnected due to the significantly different

[1]Postdoctor, College of Civil Engineering, Tongji Univ., No. 1239 Siping Rd., Shanghai 200092, China (corresponding author). E-mail: jiangjian_0131@163.com

[2]Ph.D. Candidate, Institute for Infrastructure and Environment, Univ. of Edinburgh, The King's Buildings, West Mains Rd., Edinburgh EH9 3JL, U.K. E-mail: s1134128@sms.ed.ac.uk

[3]Ph.D. Candidate, Institute for Infrastructure and Environment, Univ. of Edinburgh, The King's Buildings, West Mains Rd., Edinburgh EH9 3JL, U.K. E-mail: p.kotsovinos@ed.ac.uk

[4]Ph.D. Candidate, Institute for Infrastructure and Environment, Univ. of Edinburgh, The King's Buildings, West Mains Rd., Edinburgh EH9 3JL, U.K. E-mail: j.zhang@ed.ac.uk

[5]Professor, Institute for Infrastructure and Environment, Univ. of Edinburgh, The King's Buildings, West Mains Rd., Edinburgh EH9 3JL, U.K. E-mail: asif.usmani@ed.ac.uk

[6]Research Engineer, Dept. of Civil and Environmental Engineering, Univ. of California, Berkeley, CA 94720. E-mail: fmckenna@ce.berkeley.edu

[7]Professor, China State Key Laboratory for Disaster Reduction in Civil Engineering, Tongji Univ., No. 1239 Siping Rd., Shanghai 200092, China. E-mail: gqli@tongji.edu.cn

Note. This manuscript was submitted on October 4, 2012; approved on March 25, 2013; published online on April 1, 2013. Discussion period open until September 2, 2014; separate discussions must be submitted for individual papers. This paper is part of the ***Journal of Computing in Civil Engineering***, © ASCE, ISSN 0887-3801/04014030(13)/$25.00.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

physics and length and time scales, making it impossibly tedious to simulate realistic hazard scenarios and unfit to meet the challenging demand for future computational tools in this branch of science and engineering. Considerable effort has gone into the development of software for individual components of modeling structural fire resistance, e.g., computational fluid dynamics (CFD) software such as *FDS* (*Fire Dynamics Simulator*) (McGrattan 2004; McGrattan and Bouldin 2004), *ANSYS Fluent* (*FLUENT* 2003), *KFX* (*Kameleon FireEX*) (Vembe et al. 1998), and *CFAST* (*FAST* 2000). Software such as *ANSYS*, *ABAQUS*, and *FAHTS* (1995) can be used to conduct the heat transfer analysis. Many finite-element simulations of structural analysis have been published and agree well with experiments, such as the Cardington tests (Wang et al. 1995; Bailey and Moore 1999; Sanad et al. 2000; Elghazouli and Izzuddin 2001). These have mainly used specialist programs such as *Vulcan* (Huang et al. 1999, 2000), *ADAPTIC* (Elghazouli and Izzuddin 2000; Izzuddin et al. 2004), *SAFIR* (Franssen 2003; Vila Real et al. 2004), and commercial packages such as *ABAQUS* (Gillie et al. 2001, 2002) and *ANSYS* (Kodur and Dwaikat 2009; Cai et al. 2012). The computer program *Vulcan* has recently been extended to include a two-dimensional nonlinear finite-element procedure to predict the temperature distributions within the cross sections of structural members subject to given fire time-temperature regimes (Huang 2010a, b). *SAFIR* implements an uncoupled two-phase analysis to model fire-exposed structures (Quiel and Garlock 2008).

As mentioned earlier, the norm is to use separate software for fire, heat transfer, and structural response, typically without considering coupling effects (Panahshahi et al. 2006; Audebert et al. 2011; Quiel and Marjanishvili 2012). Even where the same software is used, for example in the case of a three-dimensional (3D) thermomechanical analysis in *ABAQUS*, things are not easy. A heat transfer analysis is first carried out (based on available heat flux boundary conditions from a separate fire model or from experimental data) on a mesh of continuum solid elements to establish the temperature evolution on sufficient points in the structure. The same solid-element mesh can be used for simulating the subsequent mechanical response. This, however, is orders of magnitude more expensive computationally because of the much higher mesh resolution required for the same accuracy compared to using much more efficient beam-column or frame elements. This would require the modeler to manually assign the highly variable temperature field (based on the heat transfer output) to the structural frame model. In addition to the extraordinarily tediousness and time-consuming nature of this task, an accurate heat transfer analysis is rendered meaningless as the temperature resolution obtained is not usable in the structural frame model (currently, *ABAQUS* only allows five temperature points over the cross section of a 3D beam-column element).

The need for a more automated software framework is also being voiced in other quarters. The National Construction Safety Team (NCST) recommended that, based on the investigation of the collapse of the World Trade Center towers (NIST 2005), efforts should be made to enhance the capabilities of computational methods to study the effect of realistic fire on buildings, all the way from the outbreak of fire to collapse. The FireGrid concept proposed by researchers from the University of Edinburgh (Han et al. 2010) aimed to improve the information available under emergency in a timely manner to firefighters. This required a platform on which the data collection and interpretation was run in super-real time. The enormous disparities in spatial and temporal length scales, numerical techniques, and complexity of the computer programs make the development of an efficient coupled fire-structure analysis a challenging task.

Various methodologies and tools have been developed to study the interaction between fire, thermal, and structural models. Ghojel (1998) proposed a simple heat transfer model to simulate temperature profiles of steel structures under real fire conditions accounting for the convective and radiative properties of the main products of combustion. It did not consider the geometrical shape of the enclosure and assumed uniform temperature distribution across or along the elements. A gap radiation model was proposed by Ali et al. (2004) to simulate radiative heat transfer between the gas and the structure surface. It assumed the exposed portions of the structure were totally enveloped by the hot gas. Three-dimensional heat transfer analysis and subsequent 2D structural analysis were performed using the *ABAQUS* software. Prasad and Baum (2005) proposed an *FDS*-interface-*ANSYS* analysis procedure. The interface employed a *zone model* to manage the data generated by *FDS*. The zone model divided the compartment into a hot upper and cool lower layer. The properties of the two layers are taken from suitably chosen temporal and spatial average of output generated by FDS. A concept of adiabatic surface temperature (AST) (Wickstrom and McGrattan 2007) was introduced as an efficient interface between the fire model and the structural model. The AST was calculated from the heat flux and gas temperature obtained from the fire mode and then translated back to a net heat flux in the structural model. The advantage was that only one quantity (AST) was transferred, instead of heat flux and gas temperature from the fire model, which was computationally convenient and cost-effective. The method was tested by and *FDS*-AST-*ANSYS* simulation of series of compartment fire experiments.

Liew et al. (1998) performed a transient heat transfer analysis using *FAHTS* (1995) which forms a link between the fire simulation model *KFX* and the structural analysis program *USFOS*. The gas temperature at each time step was prescribed in space grids that envelop the structures. The temperature distribution across the element section was calculated by subdividing line beam-column element into a four-node quadrilateral element that can be retrieved by the structural analysis program. Shi et al. (2008) developed an integrated simulation system, *BFireSAS*, to simulate the overall fire safety performance of large buildings supported by several software tools such as *AutoCAD*, *FD*S, and *ANSYS*. Additional model transformers were created to transfer the gas temperature from *FDS* to structural analysis solved by *ANSYS*. *AutoCAD* was used to construct the geometry of the structural model, *FDS* to simulate a fire field, and *ANSYS* for structural analysis. A core database was developed to support the data store and exchange of integrated system and bridge the connection between the different modules. Duthinh et al. (2008) presented two interfaces in fire-thermal-structural analysis. A macroscopic finite-element model was developed by Kodur et al. (2009) for predicting the entire fire response of reinforced concrete structures from fire analysis to structural collapse analysis. Lee et al. (2011) proposed an *FDS*-interface-*ABAQUS* analysis. A *Matlab* subroutine was created as a tool for transferring the *FDS* temperatures to *ABAQUS* input. The transfer was conducted by tracing the same coordinate of heat transfer model with the *FDS* model. A novel fiber-element approach was developed by Jeffers and Sotelino (2012) to evaluate the thermostructural response of nonuniformly heated structural frames. The same fiber discretization in the structural model was used as in the heat transfer model.

Previous studies focused on the development of interfaces between specialist software or commercial packages. Although specialist programs are cost-effective to purchase and easy to use, they lack generality and versatility. In addition, more tellingly continuous development, quality, robustness, and long-term sustainability of such research group–based software must remain in perpetual doubt because of a relatively small number of users

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

and developers. The commercial packages have a large library of finite elements and excellent graphical user interfaces (GUIs) to enable efficient and detailed modeling of structural responses to fire and also to allow user subroutines for modeling special features of behavior. Despite obvious advantages, commercial packages require substantial recurring investment for purchase and maintenance that often make them unaffordable for researchers and deter new entrants to the field. Furthermore, the development of commercial codes is not in the hands of the user and users have little control over the direction the development takes. This is usually dictated by the needs of the largest commercial subscribers and rarely addresses the needs of discounted subscription–paying researchers.

An alternative to commercial software is open-source software, where the source code for the software is made available for anyone to download, modify, and use (mostly for free). In successful open-source projects, many outside developers contribute new developments and bug fixes back to the project to further its capabilities. Examples of successful open-source projects include *Mozilla Firefox*, *GNU Linux*, and the *Apache HTTP server software. In the structural engineering field, OpenSees* (McKenna 1997) is an open-source object-oriented software framework developed at the University of California–Berkeley, and supported by PEER and Nees. *OpenSees* has so far been focused on providing an advanced finite-element computational tool for analysing the nonlinear response of structural and geotechnical systems subjected to seismic excitations. In contrast to algorithm-based programs, object-oriented programs are composed of objects, each with a number of attributes and methods, and can be viewed as the interaction between objects by the sending of messages due to the support of abstraction, encapsulation, modularity, and inheritance (Booch 1994). These features of object-oriented programs make *OpenSees* computationally efficient, flexible, extensible, and portable (McKenna 1997; Scott et al. 2008). This means a developer can combine and reuse the existing classes in *OpenSees* to create an application to solve one's own specific problem. Given that *OpenSees* is open-source and has been available for best part of this decade, it has spawned a rapidly growing community of users as well as developers who have added to its capabilities over this period. For the analysis of structural and geotechnical systems, it now has capabilities developed by researchers that have yet to appear in commercial software. *OpenSees* offers the potential of a common community owned research program with large and growing modeling capability in many areas of structural engineering. It will enable researchers to collaborate freely across geographical boundaries with a much greater potential longevity of research and development efforts.

The research team at the University of Edinburgh has been working to add a *structures in fire* modeling capability in *OpenSees*. Eventually, this capability will involve a heat transfer model, a structural model and an interface between them to map the temperature data automatically from the heat transfer analysis to the structural analysis, without losing the spatial and temporal resolution of the temperatures when applied to the structural elements. Further work is planned to link *OpenSees* to the open-source CFD model *OpenFOAM* (capable of modeling compartment fires) leading to a fully automated software framework for modeling fire, heat transfer, and structural response (Fig. 1).

This paper presents the extensions to *OpenSees* to enable 2D thermomechanical analysis. This involved creating a new thermal load pattern, modifying existing material classes to include temperature-dependent properties and modifying methods in element and section classes in *OpenSees*. The algorithm used for thermomechanical analysis of structures is given first, followed by class diagrams describing the hierarchy and architecture of the development in *OpenSees*. Based on the algorithm and class hierarchy, sequence diagrams are presented to illustrate the interaction between thermal load classes and the material, section, and element classes. The sequence diagrams provide an overview of important aspects of how to apply thermal load and obtain element forces.

Thermomechanical Algorithm

In an incremental-iterative nonlinear analysis, three phases can be identified: predictor, corrector, and convergence check (Yang and Kuo 1994). The predictor needs to predict an initial out-of-balance force and calculate the displacement increment due to this unbalanced force, given the stiffness matrix at the previous step. For thermomechanical analysis, in addition to the general external load increment, the unbalanced force should include the equivalent fixed-end force due to thermal load and material softening.

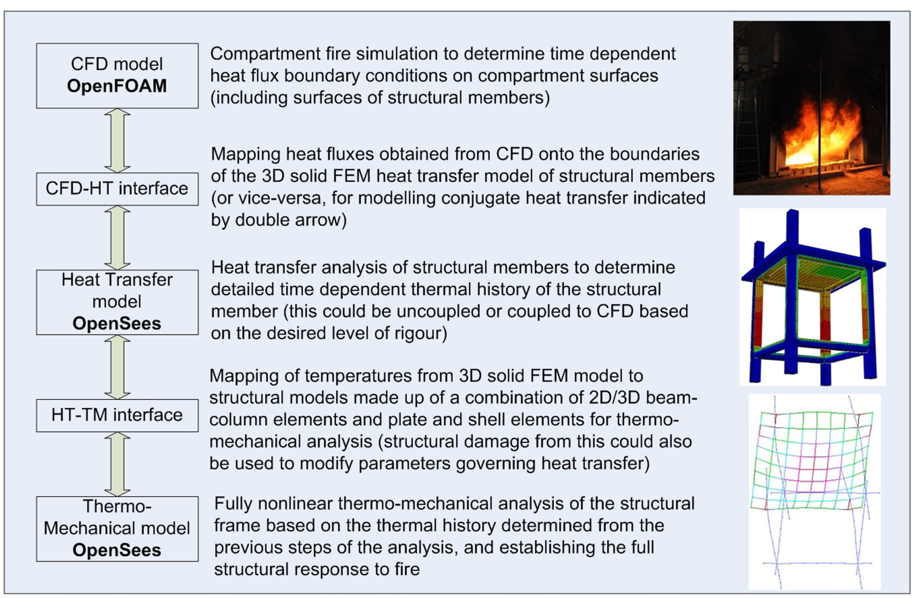

Fig. 1. An open software framework for modeling structures in fire

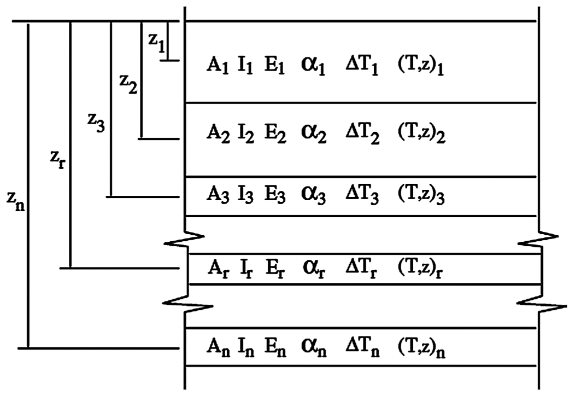

Fig. 2. A general section divided into n fibers

The corrector is concerned with the recovery of element force increment from the displacement increment obtained in the predictor phase. Equilibrium of the structure is checked at the end of each iteration to ensure that convergence is achieved in the new deformed configuration.

Predictor

The unbalanced force resulting from thermal load and material softening should be calculated in the predictor phase. The thermal load can be considered as elemental load derived from the temperature distribution along the section. In the finite-element analysis, the elemental load should be transformed into equivalent nodal load. Fig. 2 shows a general fiber section, which is subdivided into longitudinal fibers, with the geometric properties and temperature conditions as defined by a uniform temperature increment, ΔT_r, and a through-depth thermal gradient, $(T_{,z})_r$, for a given fiber, r. Thermal gradient has not been implemented in *OpenSees*; only mean temperature is used for simplicity. However, this can conceivably be implemented in future to model very steep thermal gradients with fewer fibers. If the beam that the section belongs to is fully restrained, each fiber will have a force and moment associated with it. Integrating the forces in each fiber gives section force $F_{\text{sec}} = [\bar{F}\,\bar{M}]$, defined as (Usmani et al. 2001)

$$\bar{F} = \sum_r E_r A_r \alpha_r \Delta T_r \tag{1}$$

$$\bar{M} = \sum_r F_r (z_r - \bar{z}) + \sum_r E_r I_r \alpha_r (T_{,z})_r \tag{2}$$

where the subscript r represents the rth fiber; E_r and A_r are the Young's modulus and area of the fiber; I_r = second moment of area; $\bar{F}$ and $\bar{M}$ are the axial force and moment of the section; F_r = axial force; α_r = thermal elongation coefficient; z_r = location of fiber r through the thickness of the section; and $\bar{z}$ is the centroid of the section given by

$$\bar{z} = \frac{\sum_r A_r E_r \times z_r}{\sum_r A_r E_r} \tag{3}$$

Another source of unbalanced force is the material softening or material degradation due to the increment of the temperature. The imbalance between the applied external load and reduced resisting force leads to further deformation of the structure. Therefore, at the beginning of each thermal load step, the temperature-dependent material properties should be updated, given current temperature, and then the resisting force should be calculated again, given the converged deformation at the previous step using the updated material properties.

The out of balance force F_u^1 at the beginning of each load step is determined by

$$F_u^1 = F_{ex} + F_{th} - F_{re} \tag{4}$$

where F_{ex} = external load; F_{th} = elemental thermal force by integration of section force F_{sec} along the element; and F_{re} = updated resisting force due to material softening.

Corrector

Once the initial displacement increment is obtained due to the updated out-of-balance force F_u^1, iterations are needed to determine the converged displacements for the nonlinear problem. In this case, when forming the out of balance force, there is no need to consider thermal force F_{th}, i.e.,

$$F_u = F_{ex} - F_{re} \tag{5}$$

where F_u = out-out-balance force calculated for the iterations after the first iteration [i.e., F_u^1 in Eq. (4)] where the temperature-induced elemental resisting force F_{th} is not considered. Remember that the stress state depends only on the mechanical strain

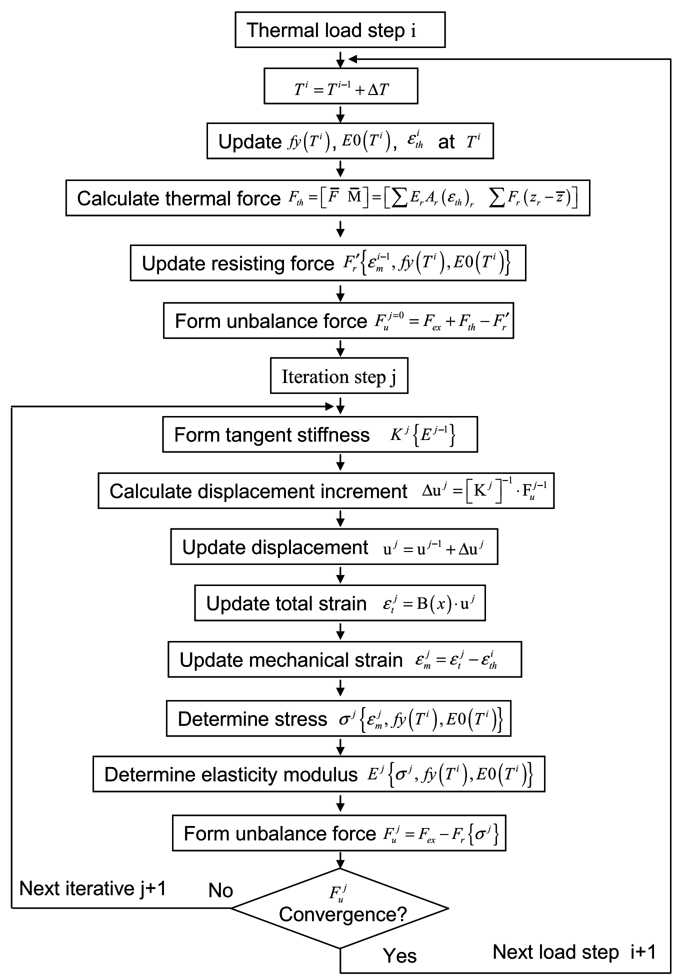

Fig. 3. Flow chart for thermal-mechanical analysis

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

$$\varepsilon_{\text{mechanical}} = \varepsilon_{\text{total}} - \varepsilon_{\text{thermal}} \tag{6}$$

where $\varepsilon_{\text{mechanical}}$, $\varepsilon_{\text{total}}$, and $\varepsilon_{\text{thermal}}$ are the mechanical strain, total strain, and thermal strain, respectively. The total strain can be obtained from the strain-displacement relation and the thermal strain can be calculated as $\varepsilon_{\text{thermal}} = \alpha \Delta T$.

With these two modifications, the corrector phase of thermomechanical analysis can follow the general procedure of mechanical analysis of structures (Spacone and Filippou 1992). Fig. 3 shows the flowchart of element state determination for thermomechanical analysis.

Class Diagrams for Thermomechanical Analysis in OpenSees

In order to implement the aforementioned solution algorithm in *OpenSees*, new subclasses were implemented and new methods were developed that derive behavior from existing components in *OpenSees*. These involved creating a new thermal load pattern class, and modifying existing material classes to include temperature dependent properties. Fig. 4 shows the class hierarchy of new classes added in *OpenSees* using the graphical unified modeling language notation (Booch et al. 1998). The class ThermalLoadPattern was created to store the temperature distribution in the structure and can be used as an interface. It paralleled other load patterns such as earthquake. The temperature distribution stored can be either retrieved from the output of the heat transfer analysis or directly input by the user according to standard codes and experimental data. The data transfer between heat transfer and structural model was designed to account for the disparity in spatial and temporal scales and different element types. One of the functions of ThermalLoadPattern was to call the class Beam2dThermalAction to pass the temperature distribution across the section. It can then be retrieved by the element class such as DispBeamColumn2dThermal and be passed to material classes (e.g., Steel01Thermal) through section classes such as FiberSection2dThermal. The material properties at elevated temperature will be updated corresponding to the temperature input. Beam2dThermalAction can also be used independently to define simple temperature profiles, such as uniform and linearly distributed temperature distribution. The temperature distribution in the structural element can be considered as elemental load. Therefore, the class Beam2dThermalAction defining the temperature distribution in the element was created as a subclass of ElementalLoad. The detailed attributes and implementation of these classes will be presented in the following sections.

Thermal Load Class

Fig. 5 shows the class diagram of thermal load classes created in *OpenSees*, and their implementations are shown in Fig. 6.

Thermal load class Beam2dThermalAction was defined as a subclass of ElementalLoad ranked with point load and uniform load. Beam2dThermalAction was created to store the temperature distribution through the depth of the beam section defined by coordinate (*LocY*) and corresponding temperature (T). The temperature of each fiber located along the depth of beam section will be determined by interpolating the temperature at the nearest coordinate point according to its location. At this stage, three kinds of constructors were defined in Beam2dThermalAction to deal with the input of two, five, and nine temperature points through the height of beam section, respectively. Uniform and linearly distributed temperatures can be defined using two temperature points defined at the top and bottom of the section, respectively.

Thermal load pattern ThermalLoadPattern was created to define detailed and highly varying time-dependent temperature distributions in structural members. It can be used as an interface to transfer the temperature distribution from the heat transfer model to the structural model where the structural responses will be predicted. So far, the thermal analysis and structural analysis has been uncoupled in *OpenSees*, which means that temperature distribution along the element should be provided as input before the structural analysis. Parallel work is under progressing on automatically generating time-varying structural temperature data from a heat transfer analysis within *OpenSees* (Usmani et al. 2012), however, direct inputs will always be required for modeling of experiments. A series of parameters containing time points and corresponding temperature for the nine temperature points along the height of the section respectively are defined as the input of ThermalLoadPattern. The maximum temperature at each temperature point through the whole fire duration will be defined first and the temperature can then be defined as a ratio of its absolute value to the corresponding maximum temperature. This scheme can accommodate both heating and cooling scenarios.

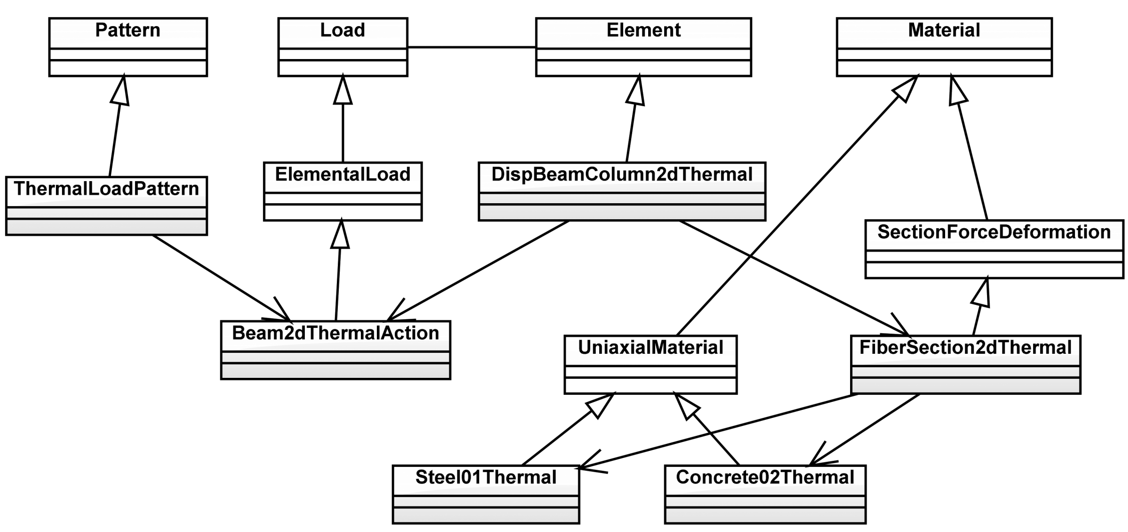

Fig. 4. Class diagram for thermomechanical analysis in *OpenSees*

J. Comput. Civ. Eng. 2015.29.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

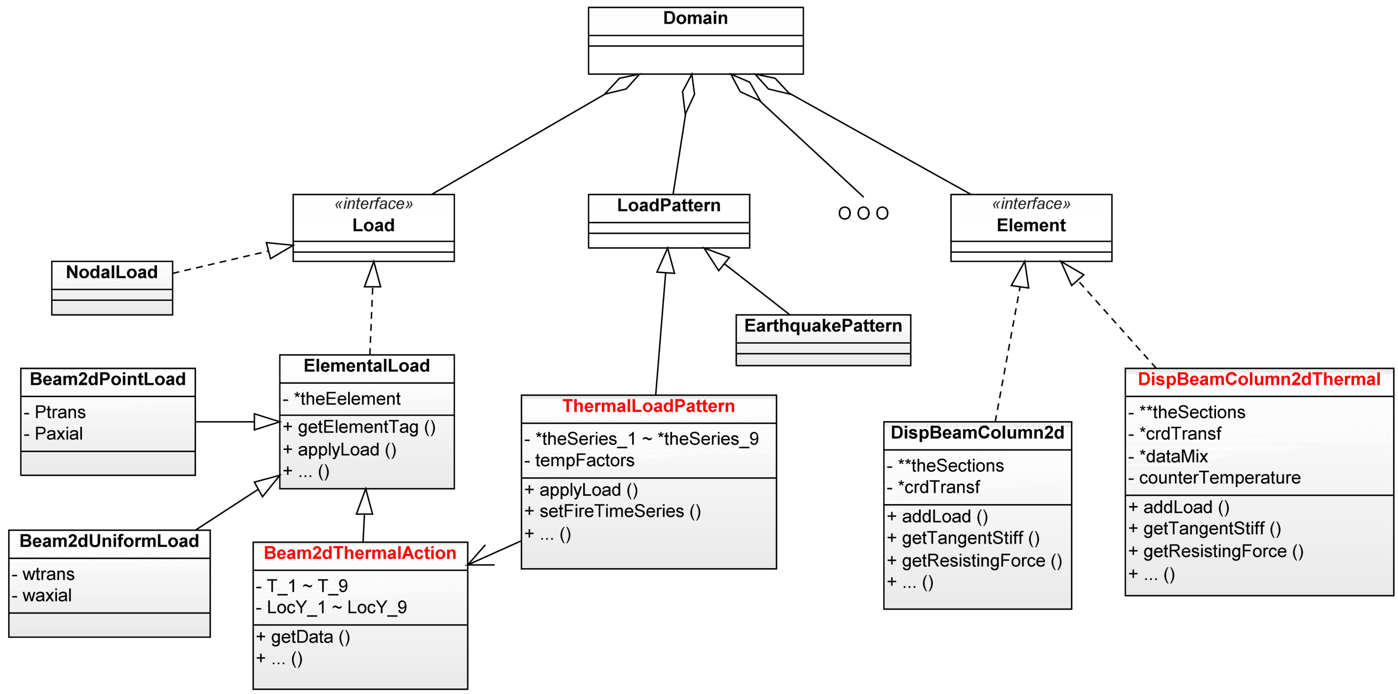

Fig. 5. Class diagram of thermal load classes in *OpenSees*

Modified Material Class

There are many types of material models available in *OpenSees* for steel and concrete, defining their mechanical constitutive relationships, however, some of these needed to be modified to include temperature dependent properties. New temperature-dependent material classes Steel01Thermal (for steel) and Concrete02Thermal (for concrete) were created by modifying the existing material class steel01 and Concrete02. While these new classes share the same stress-strain relations in absence of thermal effects (thus our choice of names), they are not derived from the existing classes, as no reuse of any of the existing class methods was possible. The temperature-dependence added in these two material classes were based on Eurocode stipulations [ENV 1992-1 -2 (Eurocode 2 2004); ENV 1993-1 -2 (Eurocode 3 2004)]. Fig. 7 shows the class

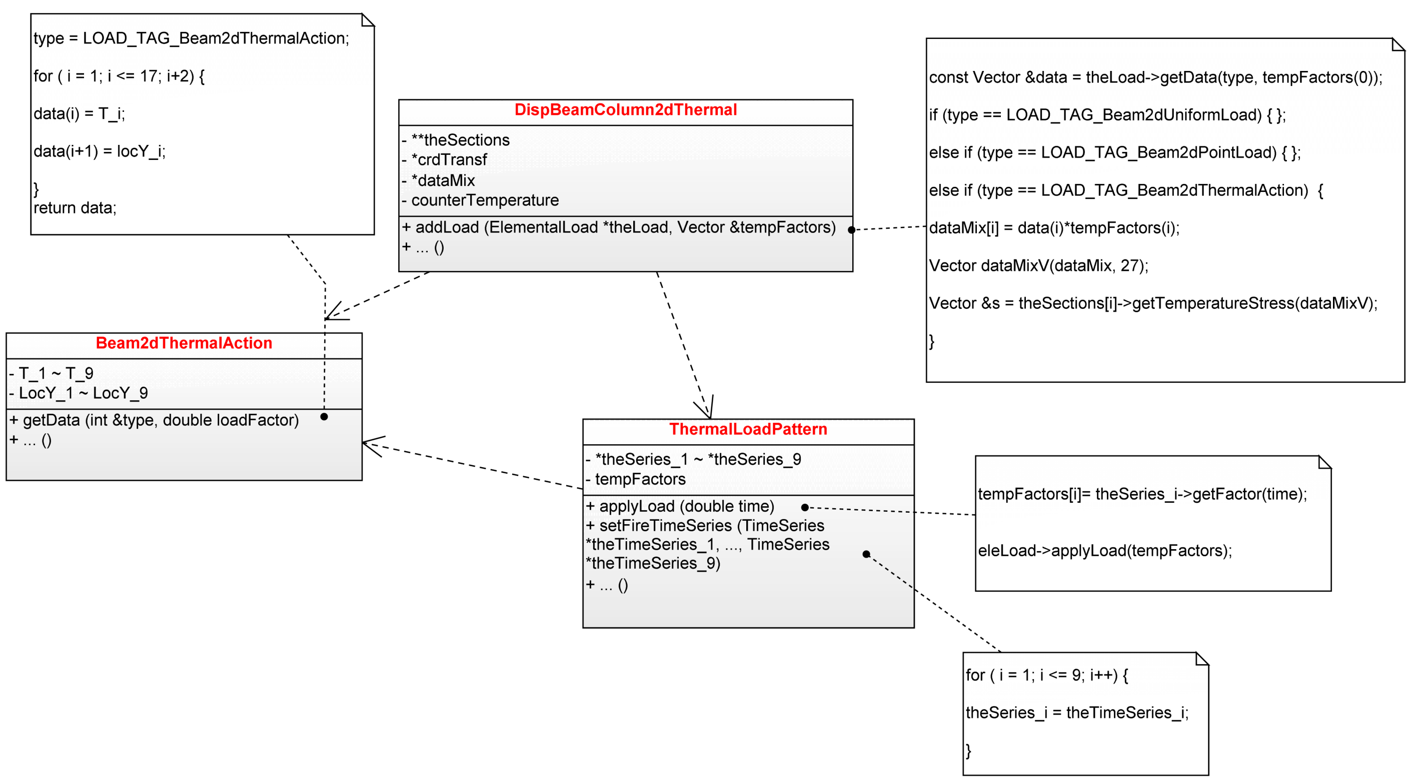

Fig. 6. Implementation of functions defined in thermal load classes in *OpenSees*

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

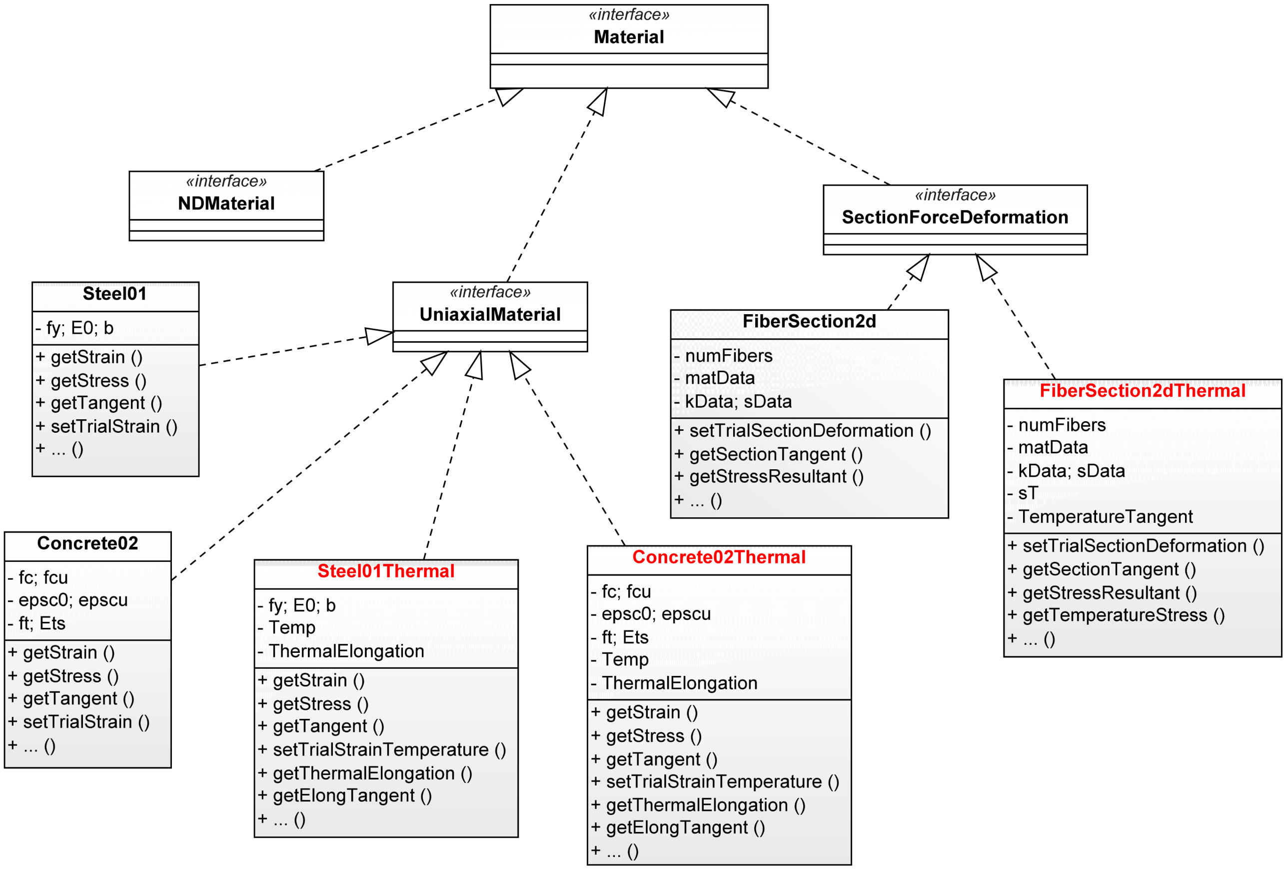

Fig. 7. Class diagram of temperature-dependent material classes in *OpenSees*

diagram of temperature-dependent classes created in *OpenSees* with implementations shown in Fig. 8.

Sequence Diagram for Thermomechanical Analysis in OpenSees

The previous section presented a static view of the new classes contributed to OpenSees. To describe how these objects interoperate to conduct thermomechanical analysis, this section presents sequence diagrams showing how to apply thermal load and obtain element resisting force.

Figs. 9 and 10 show the sequence diagrams for applying thermal load to beam element through thermal load classes. The thermal load is applied by invoking method *applyLoad*() in class ThermalLoadPattern. This method is primarily responsible for two operations. The first responsibility is to retrieve temperature ratio of each temperature point according to current time point from object LinearSeries by calling method *getFactor*(). The second step is to invoke the method *addLoad*() in the associated DispBeamColumn2dThermal object to add thermal load to the beam element as shown in Fig. 9. The temperatures and their distributions (dataMix) at current time are calculated and then passed to the section class by invoking the method *getTemperatureStress*(), which in turn will invoke the method *getElongTangent*() in the materials. The method *getElongTangent*() has two operations. One is to update the material properties according to the current temperature. The other function is to send back the temperature dependent elastic modulus (*tangent*) and thermal elongation (*ThermalElongation*) of each fiber material to the section class. These temperature-dependent properties are then used to calculate the force of each fiber through which the section thermal force (sT) can be calculated by integration. The thermally induced resisting force of the element can be calculated by integration through sections.

Fig. 11 shows the procedure for obtaining elemental resisting force. As mentioned in Section 2, the out-of-balance force of an element at the beginning of each load step comes from three sources including mechanical load, thermal load and reduced resisting force due to material degradation. In the method *getResistingForce*(), a parameter (*counterTemperature*) is set to determine whether it is the first iteration of each load step. If *counterTemperature* = 0 (means first iteration), the method *update*() is invoked to update the element state due to the material degradation and the thermally induced resisting force is considered to calculate the total out-of-balance force ($s + sT$). For the next iteration, only mechanically induced out-of-balance force is considered.

Validation

In order to test the performance of the thermomechanical analysis capability in *OpenSees*, two benchmark cases were carried out, including bending of a single beam with finite-boundary conditions and a steel frame test.

Single Beam Benchmark

Fig. 12 shows the schematic of the single-beam model which is extracted from a framed structure. The beam in the middle

J. Comput. Civ. Eng. 2015.29.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

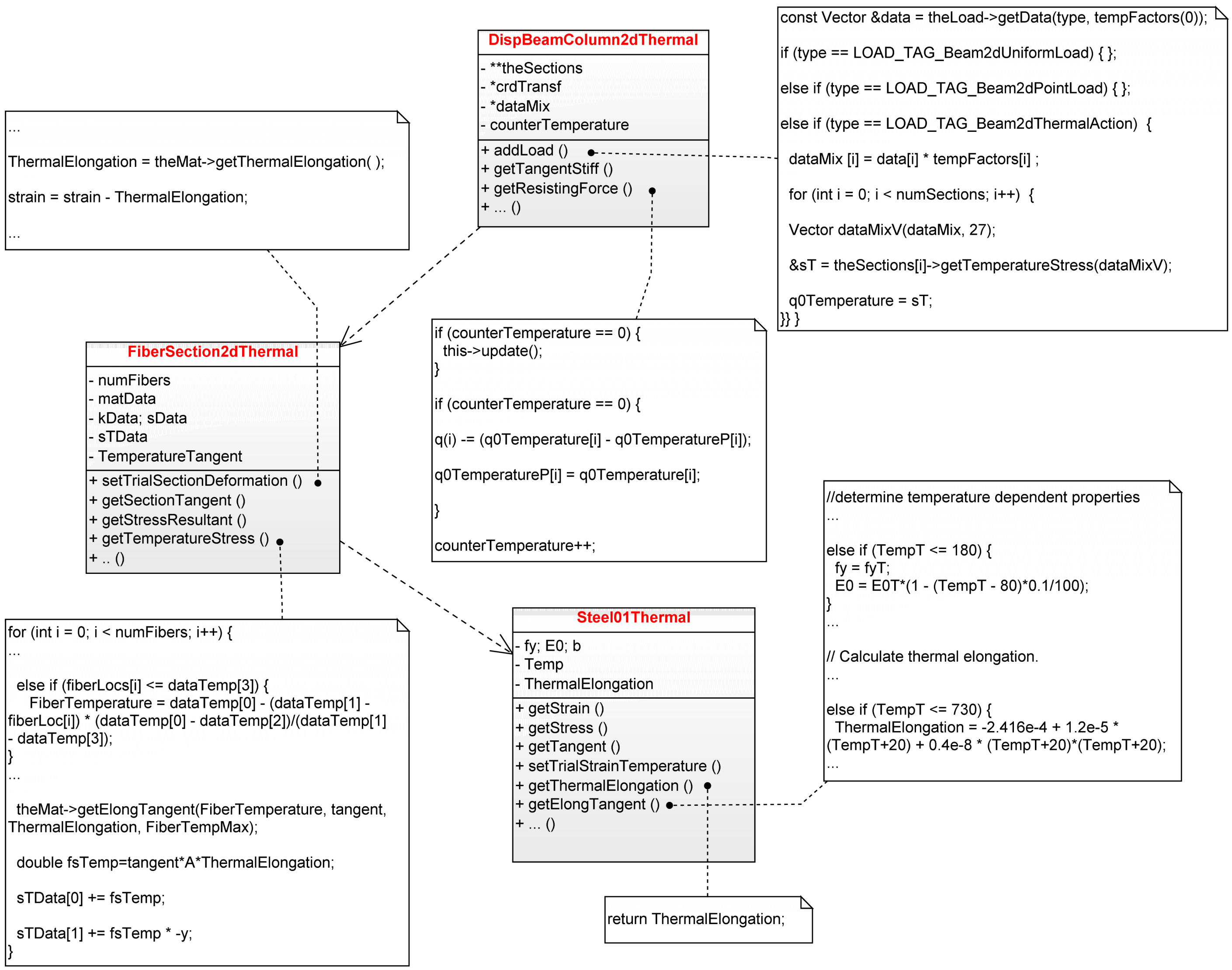

Fig. 8. Implementation of functions defined in temperature-dependent material classes in *OpenSees*

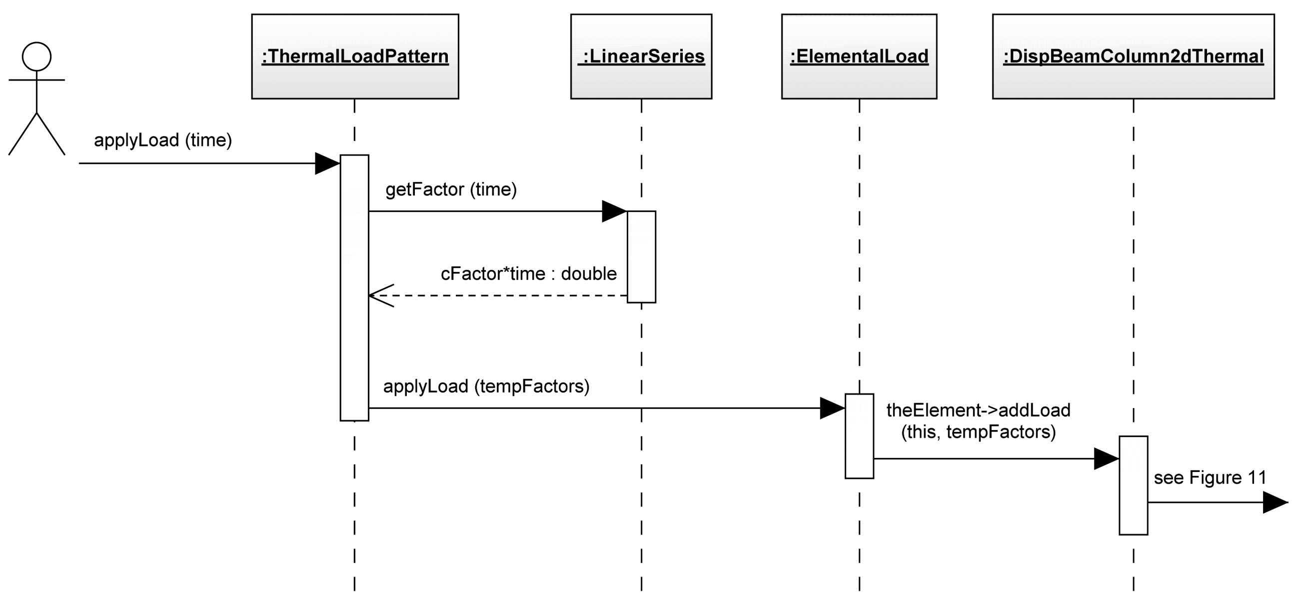

Fig. 9. Sequence diagram for applying thermal load in thermal load classes

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

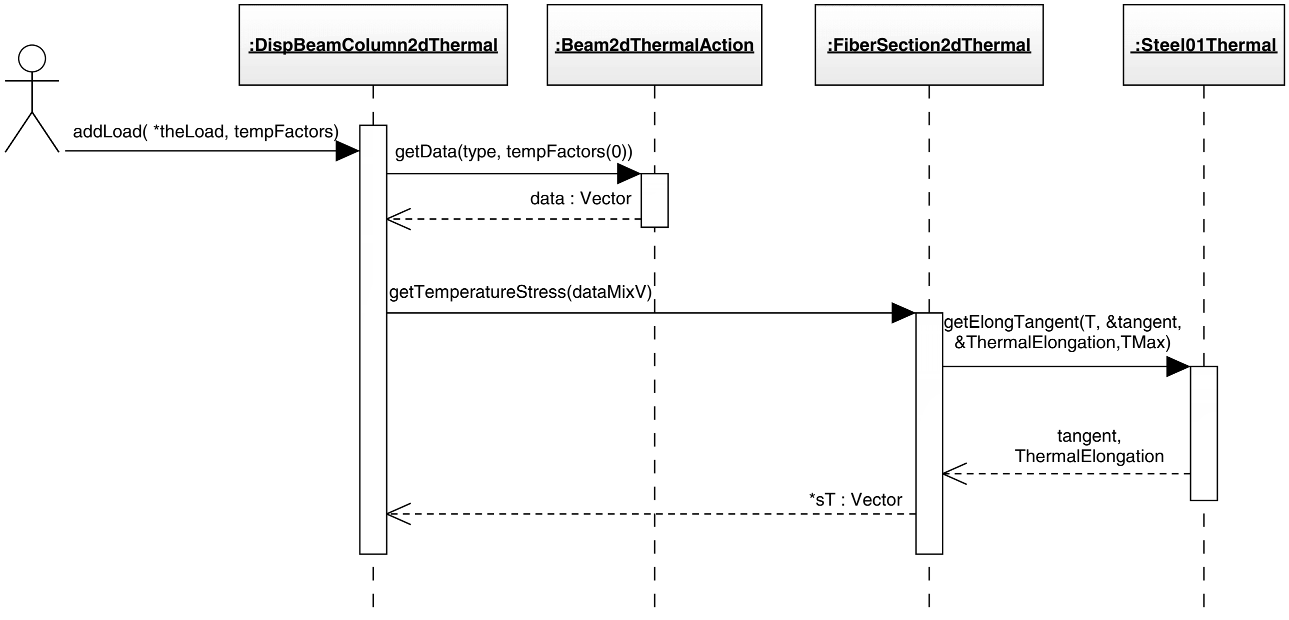

Fig. 10. Sequence diagram for adding thermal load in beam element

of the frame is subjected to uniformly distributed load (UDL) and is restrained by one beam and two columns at both ends. The restraining capability offered by these surrounding elements can be represented by equivalent rotational and translational springs. Therefore, the framed structure can be transformed to an equivalent single-beam with finite end restraints. The dimensions of beams and columns in the Cardington restrained beam test (Kirby 1997) are used (i.e., 305 × 165 × 40UB for beam and 254 × 254 × 89UC for column). The corresponding second moment of area of column (I_c) and beam (I_b) cross section can be calculated as

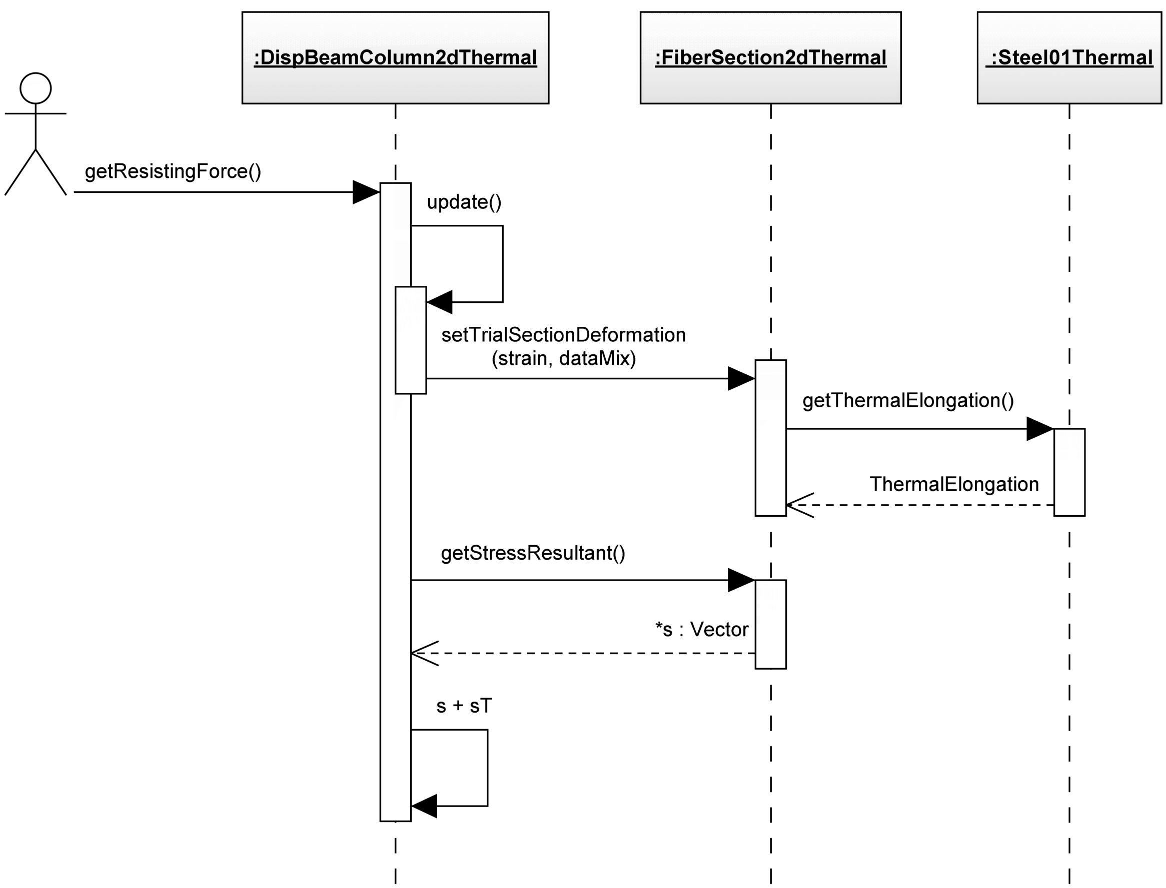

Fig. 11. Sequence diagram for obtaining element resisting force

J. Comput. Civ. Eng. 2015.29.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

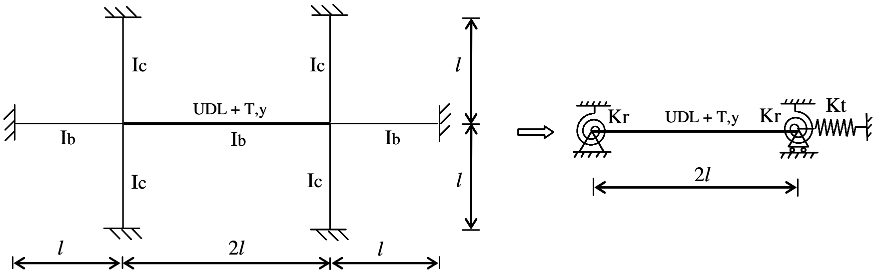

Fig. 12. Beam with translational and rotational springs at the ends

$I_c = 1.4 \times 10^{-4}$ m^4, $I_b = 0.8 \times 10^{-4}$ m^4. The equivalent stiffness of rotational spring (K_r) and translational spring (K_t) can be calculated as $K_r = 8EI_c/l + 4EI_b/l$ and $K_t = 2EA_b/l + 48EI_c/l^3$ with values of $K_r = 4.8 \times 10^4$ kN · m/rad and $K_t = 3.8 \times 10^5$ kN/m.

Based on fundamental structural mechanics, the analytical solution of the response of the beam subjected to UDL and thermal gradient $T_{,y}$ can be given as

$$\delta = \frac{5ql^4}{384EI}(1 - 0.8\varphi_r) + \frac{\alpha T_{,y} l^2}{8}(1 - \varphi_r) \tag{7}$$

$$\theta = \left(\frac{ql^2}{12} + EI\alpha T_{,y}\right)\frac{1}{K_r + 2EI/l} \tag{8}$$

$$u = \alpha \Delta T l(1 - \varphi_t) \tag{9}$$

where δ, θ and u are the mid-span deflection, end rotation and horizontal displacement of the beam respectively; $\varphi_r = 1/(1 + 2EI/K_r l)$ and $\varphi_t = 1/[1 + (EA/l)/K_t]$ is a factor due to the rotational and translational end restraint, respectively.

OpenSees was used to analyse the response of a 6-m beam ($l = 3$ m) with finite end restraints ($K_r = 4.8 \times 10^4$ kN · m/rad and $K_t = 3.8 \times 10^5$ kN/m) subjected to UDL and thermal gradient. The UDL is assumed to be 30 kN/m; the temperature at top of the beam was assumed to be 0°C and it varied linearly over the depth of the beam from temperatures of 100°C to 1,000°C. Steel01Thermal was used to model the steel material. A finite large value was assigned to yield stress in order to make the material behavior largely elastic. The elastic modulus at ambient temperature is 200 GPa, and a constant coefficient of thermal elongation $\alpha = 1.2 \times 10^{-5}$/°C was assumed. Three different analyses were conducted, including materially and geometrically linear analysis and materially nonlinear but geometrically linear analysis, as well as both materially and geometrically nonlinear analysis. Material nonlinearity is limited to elastic modulus being dependent on temperature. Fig. 13 shows good agreement

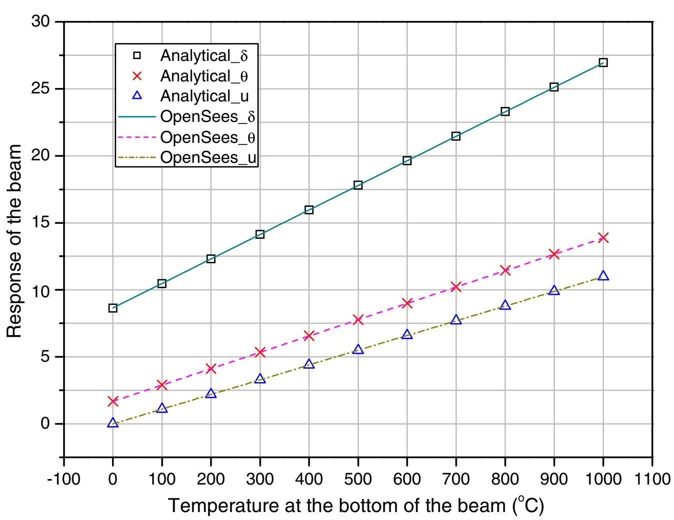

Fig. 13. Responses of the beam from *OpenSees* compared with analytical solutions (δ, u:mm; θ:10^{-3} rad)

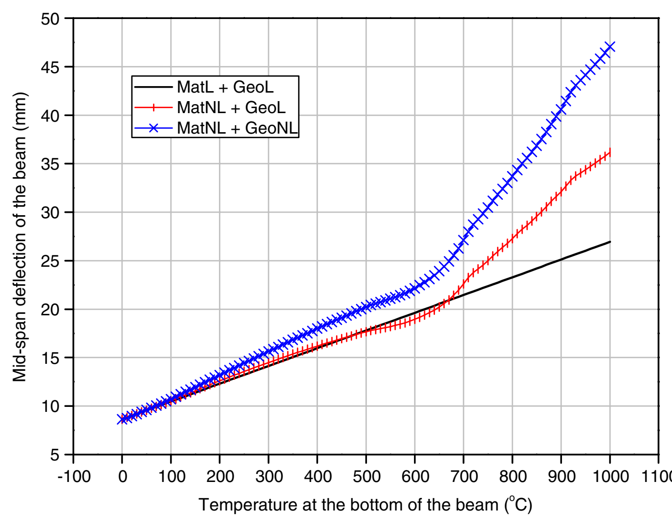

Fig. 14. Midspan deflection of the beam against temperature

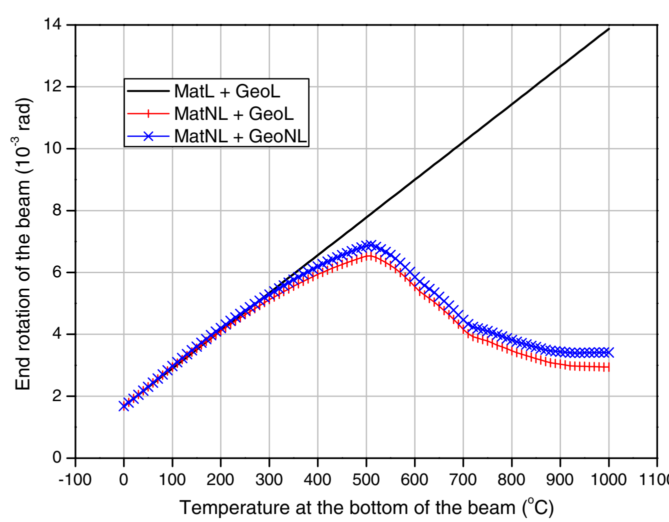

Fig. 15. End rotation of the beam against temperature

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

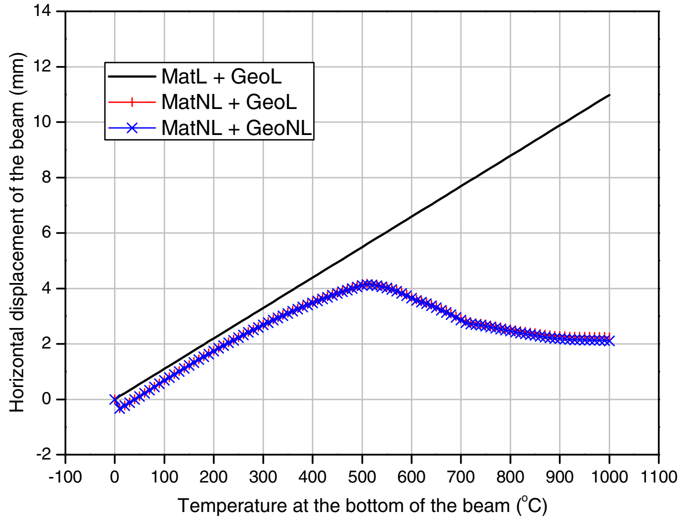

Fig. 16. Horizontal displacement of movable end of the beam against temperature

between *OpenSees* and analytical solutions using Eqs. (7)–(9) for the linear case.

The responses of the beam subjected to UDL and thermal load for the three analyses in *OpenSees* are shown in Figs. 14–16. The symbol MatL represents materially linear analysis and MatNL for materially nonlinear analysis. Similarly, GeoL represents geometrically linear analysis and GeoNL for geometrically nonlinear analysis. The material nonlinearity has an obvious effect on the end rotation and horizontal movement of the beam end. In contrast, the effect of the geometrical nonlinearity is obvious on the mid-pan deflection but negligible on the rotation and horizontal displacement of the beam. The midspan deflection of the beam continued to increase and experienced a larger slope after about 600°C for nonlinear analysis. This is because the beam deflection, as temperature increases, is dominated by the thermal bowing effect and this downward bending is accelerated by material degradation at high temperature. As shown in Figs. 15 and 16, the rotation and horizontal displacement of the beam increased first driven by the thermal elongation (and thermal gradient) until 500°C and then began to decrease as the decreasing of modulus of elasticity of the beam is unable to resist the stored strain energy and elastic rebound of the unheated rotational and translational spring respectively.

Three-dimensional frame models can also be transformed into an equivalent single-beam model with finite end restraints as long as that the stiffness of rotational spring includes the torsional stiffness from the out-of-plane beams connected at the ends.

Steel Frame Test

A series of tests on plane steel frames at elevated temperatures were performed in Germany (Rubert and Schaumann 1986). A schematic diagram of two steel frames EHR3 and ZSR1 are shown

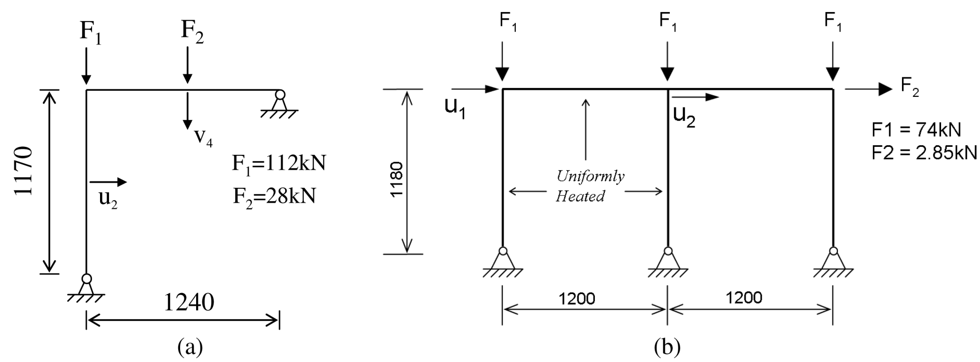

Fig. 17. Schematic of the tested steel frames (mm): (a) frame EHR3; (b) frame ZSR1

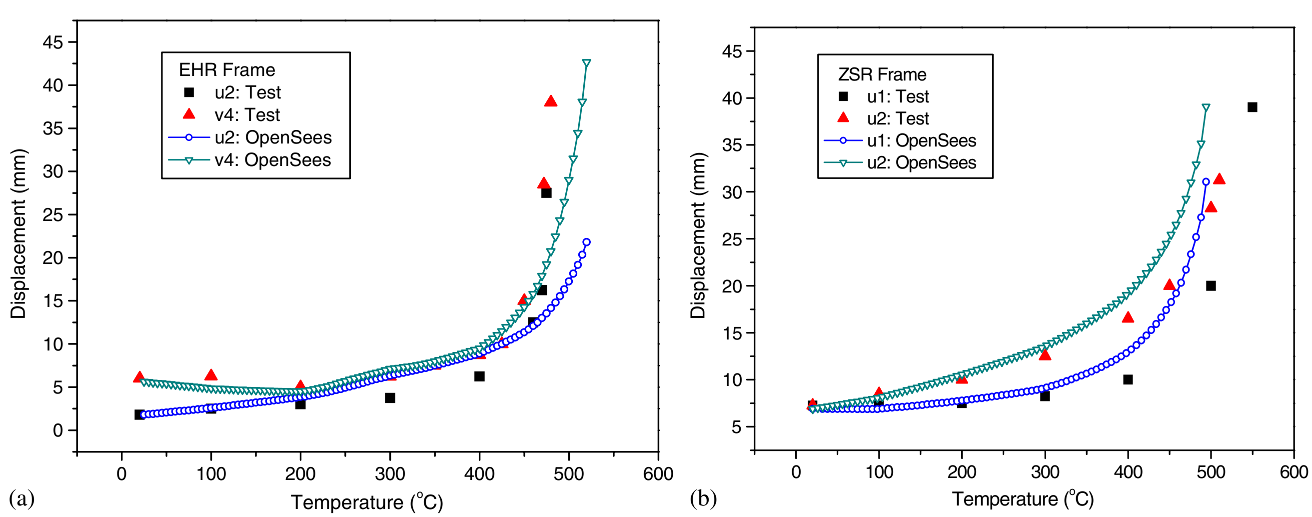

Fig. 18. Comparison between predicted and test deflection results: (a) frame EHR3; (b) frame ZSR1

J. Comput. Civ. Eng. 2015.29.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

in Fig. 17. The braced two-bar frame (HER3) was subjected to a uniform temperature rise and only one bay of the two-portal frames (ZSR1) was uniformly heated. All structural elements were made of IPE80 I-shaped steel. The yield stresses and modulus of elasticity are 382 $\mathrm{N/mm^2}$ and 210 $\mathrm{N/mm^2}$ at ambient temperature for EHR3 and 355 $\mathrm{N/mm^2}$ and 210 $\mathrm{N/mm^2}$ for ZSR1, respectively. Steel01 material class was used to model the properties of the Steel 37 material and nonlinear static analysis was conducted in *OpenSees*. Comparisons between the predicted deflections and the test results illustrated in Fig. 18 show satisfactory agreement. For this steel frame example, a bilinear material was used to model the realistic steel material in the experiment and reasonable qualitative agreement was achieved. The author consider this to be adequate validation considering that actual test conditions (such as restraint, temperature distributions, and material behavior at elevated temperature) in large-scale thermal testing (as this was) cannot really be fully or accurately represented in models.

Conclusions

The open-source, object-oriented, finite-element-based structural engineering framework *OpenSees* was extended to perform thermomechanical analysis. The class and sequence diagrams presented provide a logical overview of the hierarchy and relationship between the newly created and modified classes in *OpenSees*. The thermomechanical analysis capability was tested using two cases including bending of a beam and a steel frame test. Good agreements were achieved between *OpenSees* and analytical solutions of the beam benchmark test. Reasonable agreement was found against test data. The verification of the extended *OpenSees* framework is currently limited to two-dimensional cases. The classes described in this work have been added to the *OpenSees* framework and are available for others to review, download, and use. Further work is being done to extend *OpenSees* for large deflection of 3D frames including plate and shell elements in fire.

Acknowledgments

Jian Jiang and Liming Jiang were supported by China Scholarship Council and the University of Edinburgh. Jian Zhang is supported by Heriot-Watt University. The work was finalized by Jian Jiang under supervision of Professor Guo-Qiang Li with the financial support of National Natural Science Foundation of China under Grant No. 51120185001.

References

Ali, H. M., Senseny, P. E., and Alpert, R. L. (2004). "Lateral displacement and collapse of single-story steel frames in uncontrolled fires." *Eng. Struct.*, 26(5), 593–607.

ASTM. (2007). "Standard test methods for fire tests of building construction and materials." *E119-07*, West Conshohocken, PA.

Audebert, M., Dhima, D., Taazount, M., and Bouchair, A. (2011). "Numerical investigation on the thermo-mechanical behaviour of steel-to-timber joints exposed to fire." *Eng. Struct.*, 33(12), 3257–3268.

Bailey, C. G., and Moore, D. B. (1999). "The behaviour of full-scale steel framed buildings subject to compartment fires." *Struct. Eng.*, 77(8), 15–21.

Booch, G. (1994). *Object-oriented analysis and design with applications*, Addison-Wesley, Reading, MA.

Booch, G., Rumbaugh, J., and Jacobson, I. (1998). *The unified modelling language user's guide*, Addison-Wesley, Reading, MA.

Cai, J. G., Feng, J., and Zhang, J. (2012). "Thermoelastic buckling of steel columns with load-dependent supports." *Int. J. Non-linear Mech.*, 47(4), 8–15.

Deierlein, G. G., Krawinkler, H., and Cornell, C. A. (2003). "A framework for performance-based earthquake engineering." *Proc., Pacific Conf. on Earthquake Engineering, New Zealand Society for Earthquake Engineering*, Wellington, New Zealand, 1–8.

Duthinh, D., McGrattan, K., and Khaskia, A. (2008). "Recent advances in fire-structure analysis." *Fire Saf. J.*, 43(2), 161–167.

Elghazouli, A. Y., and Izzuddin, B. A. (2000). "Response of idealised composite beam-slab systems under fire conditions." *J. Constr. Steel Res.*, 56(3), 199–224.

Elghazouli, A. Y., and Izzuddin, B. A. (2001). "Analytical assessment of the structural performance of composite floors subject to compartment fires." *Fire Saf. J.*, 36(8), 769–793.

European Committee for Standardization (CEN). (2004b). "Design of steel structures, part 1.2: General rules, structural fire design." *Eurocode 3 ENV 1993-1-2*, Brussels.

FAHTS. (1995). *Fire and heat transfer simulations of frame structures, theory and user's manual, SINTEF Structures and Concrete*, Trondheim, Norway.

FAST. (2000). "A user's guide for: Engineering tools for estimating fire growth and smoke transport." *NIST Special Publication 921*, U.S. Government Printing Office, Washington, DC.

FLUENT 6.1. (2003). *User's guide, Fluent Inc.*, Lebanon, NH.

Franssen, J. M. (2003). "SAFIR: a thermal/structural program modelling structures under fire." *Proc., North American Steel Construction Conf.*, A.I.S.C., Baltimore.

Ghojel, J. I. (1998). "A new approach to modelling heat transfer in compartment fires." *Fire Safety J.*, 31(3), 227–237.

Gillie, M., Usmani, A. S., and Rotter, J. M. (2001). "Structural analysis of the first cardington test." *J. Constr. Steel Res.*, 57(6), 581–601.

Gillie, M., Usmani, A. S., and Rotter, J. M. (2002). "A structural analysis of the cardington british steel corner test." *J. Constr. Steel Res.*, 58(4), 427–443.

Han, L. X., et al. (2010). "FireGrid: An e-infrastructure for next-generation emergency response support." *J. Parallel Distrib. Comput.*, 70(11), 1128–1141.

Huang, Z. (2010a). "Modelling the bond between concrete and reinforcing steel in a fire." *Eng. Struct.*, 32(11), 3660–3669.

Huang, Z. (2010b). "The behaviour of reinforced concrete slabs in fire." *Fire Saf. J.*, 45(5), 271–282.

Huang, Z., Burgess, I. W., and Plank, R. J. (1999). "Nonlinear analysis of reinforced concrete slabs subjected to fire." *ACI Struct. J.*, 96(1), 127–135.

Huang, Z., Burgess, I. W., and Plank, R. J. (2000). "Effective stiffness modelling of composite concrete slabs in fire." *Eng. Struct.*, 22(9), 1133–1144.

International Organisation for Standardisation (ISO). (1992). "Part 1: Fire resistance tests—elements of building construction." *ISO-834*, Geneva, Switzerland.

Izzuddin, B. A., Tao, X. Y., and Elghazouli, A. Y. (2004). "Realistic modelling of composite and R/C floor slabs under extreme loading – Part I: Analytical method." *J. Struct. Eng.*, 10.1061/(ASCE)0733-9445(2004)130:12(1972), 1972–1984.

Jeffers, A. E., and Sotelino, E. D. (2012). "An efficient fiber element approach for the thermo-structural simulation of non-uniformly heated frames." *Fire Saf. J.*, 51(3), 18–26.

Kirby, B. R. (1997). "British steel technical European fire test programme design, construction and results." *Report of fire, static and dynamic tests of building structures*, British Steel, London.

Kodur, V., Dwaikat, M., and Raut, N. (2009). "Macroscopic FE model for tracing the fire response of reinforced concrete structures." *Eng. Struct.*, 31(10), 2368–2379.

Kodur, V., and Dwaikat, M. M. S. (2009). "Response of steel beam-column exposed to fire." *Eng. Struct.*, 31(2), 369–379.

Lee, C. H., Chiou, Y. J., Chung, H. Y., and Chen, C. J. (2011). "Numerical modelling of the fire-structure behaviour of steel beam-to-column connections." *J. Constr. Steel Res.*, 67(9), 1386–1400.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.

Liew, J. Y. R., Tang, L. K., Holmaas, T., and Choo, Y. S. (1998). "Advanced analysis for the assessment of steel frames in fire." *J. Constr. Steel Res.*, 47(1–2), 19–45.

McGrattan, K. B. (2004). "Fire dynamics simulator (version 4), technical reference guide." *NIST Special Publication 1018*, National Institute of Standards and Technology, Gaithersburg, MD.

McGrattan, K. B., and Bouldin, C. (2004). "Simulating the fires in the World Trade Center." *Proc., 10th Int. Conf.*, Interscience, London.

McKenna, F. T. (1997). "Object-oriented finite element programming: Frameworks for analysis, algorithms and parallel computing." Ph.D. thesis, Univ. of California, Berkeley, CA.

NIST NCSTAR. (2005). *Final report on the collapse of the World Trade Center Towers*, National Institute of Standards and Technology, Gaithersburg, MD.

Panahshahi, N., Molki, M., Rossow, M., Rabiei, K., and Gharib, K. (2006). "The response of framed steel structures to fire." *Structures Congress*, ASCE, Reston, VA.

Prasad, K., and Baum, H. (2005). "Coupled fire dynamics and thermal response of complex building structures." *Proc. Combust. Inst.*, 30(2), 2255–2262.

Quiel, S. E., and Garlock, M. E. M. (2008). "A closed-form analysis of perimeter member behaviour in a steel building frame subject to fire." *Eng. Struct.*, 30(11), 3276–3284.

Quiel, S. E., and Marjanishvili, S. M. (2012). "Fire resistance of a damaged steel building frame designed to resist progressive collapse." *J. Perform. Constr. Facil.*, 10.1061/(ASCE)CF.1943-5509.0000248, 402–409.

Rubert, A., and Schaumann, P. (1986). "Structural steel and plane frame assemblies under fire action." *Fire Saf. J.*, 10(3), 173–184.

Sanad, A. M., Rotter, J. M., Usmani, A. S., and O'Connor, M. A. (2000). "Composite beams in large buildings under fire—Numerical modelling and structural behaviour." *Fire Saf. J.*, 35(3), 165–188.

Scott, M. H., Fenves, G. L., McKenna, F., and Filippou, F. C. (2008). "Software patterns for nonlinear beam-column models." *J. Struct. Eng.*, 10.1061/(ASCE)0733-9445(2008)134:4(562), 562–571.

Shi, J. Y., Li, Y. Q., and Chen, H. C. (2008). "Application of computer integration technology for fire safety analysis." *Tsinghua Science and Technology*, 13(S1), 387–392.

Spacone, E., and Filippou, F. C. (1992). *A beam element for seismic damage analysis*, Univ. of California, Berkeley, CA.

Usmani, A. S., Rotter, J. M., Lamont, S., Sanad, A. M., and Gillie, M. (2001). "Fundamental principles of structural behavior under thermal effects." *Fire Saf. J.*, 36(8), 721–744.

Usmani, A. S., Zhang, J., Jiang, J., Jiang, Y. Q., and May, I. (2012). "Using OpenSees for structures in fire." *J. Struct. Fire Eng.*, 3(1), 57–70.

Vembe, B. E., Lilleheie, N. I., Holen, J. K., and Magnussen, B. F. (1998). "Kameleon FireEX, A simulator for gas dispersion and fires." *Int. Gas Research Conf.*, Gas Research Institute, Chicago, IL.

Vila Real, P. M. M., Lopes, N., Silva, L. S., Piloto, P., and Franssen, F. M. (2004). "Numerical modelling of steel beam-column in case of fire-comparisons with Eurocode 3." *Fire Saf. J.*, 39(1), 23–39.

Wang, Y. C., Lennon, T., and Moore, D. B. (1995). "The behaviour of steel frames subject to fire." *J. Constr. Steel Res.*, 35(3), 291–322.

Wickstrom, U., Duthinh, D., and McGrattan, K. (2007). "Adiabatic surface temperature for calculating heat transfer to fire exposed structures." *Proc., 11th Int. Conf. on Fire Science and Engineering Interflam*, Royal Holloway College, Univ. of London, U.K.

Yang, Y. B., and Kuo, S. R. (1994). *Theory and analysis of nonlinear framed structures*, Prentice Hall, Singapore.

Downloaded from ascelibrary.org by Edinburgh University on 03/12/15. Copyright ASCE. For personal use only; all rights reserved.