Fire Safety Journal 68 (2014) 1–15

Contents lists available at ScienceDirect

Fire Safety Journal

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

A software independent tool for mapping thermal results to structural model

Dilip K. Banerjee *

Materials Measurement Laboratory, National Institute of Standards and Technology, Gaithersburg, MD 20899, USA

ARTICLE INFO

Article history:
Received 21 February 2014
Received in revised form
4 June 2014
Accepted 6 June 2014
Available online 5 July 2014

Keywords:
Data transfer
Fire
Finite element analysis
Heat transfer model
Structural analysis model
Solid/beam/shell element

ABSTRACT

In order to model the structural behavior under fire, three separate analyses need to be conducted: (a) fire propagation and growth (fire modeling), (b) heat transfer in structural members due to fire, and (c) structural analysis to account for both thermal and mechanical load. Fire modeling is conducted with computational fluid dynamics (CFD) approach. Typical heat transfer analysis is conducted using finite element analysis (FEA) approach employing solid 3D or 2D shell elements. Structural analysis is often conducted using FEA approach but employing beam and shell elements, especially for large structures.

Common outputs of a CFD fire model relevant for a structural fire analysis are heat flux and temperature field, which are input to the heat transfer model as thermal boundary conditions. Subsequently, the transient temperatures computed by the heat transfer analysis in the structural members are inputs to the structural analysis model. However, this transfer of data is complicated because of the difference in the type of finite elements and level of discretization used in each of these two analyses. A software independent mapping tool is therefore required to transfer the thermal data from heat transfer model to structural analysis model.

This paper discusses a novel methodology that was developed to map thermal data from a heat transfer model comprising solid finite elements to a structural analysis model comprising beam and shell elements. The methodology relies on the fundamentals of finite elements pertaining to the use of element shape functions and local or natural coordinates.

Published by Elsevier Ltd.

1. Introduction

In the performance based approach for the design of structures for fire, the actual temperatures of structural components during exposure to a real fire need to be precisely known as a function of time and space. These temperatures are used in subsequent structural analysis model to determine the structural response to both thermal and mechanical load. Transient temperatures in structural members can be determined with a heat transfer model that uses the fire conditions as thermal boundary conditions. Most common heat transfer models are based on FEA (finite element analysis) methods using hexahedral solid elements or 2D shell elements. Transfer of thermal results to structural models can be easily accomplished in traditional FEA software when compatible elements are used for both heat transfer and structural analysis calculations (e.g. solid to solid or shell to shell etc.). Such an approach is practical when the structural model is simple and when the model can be described with relatively small number of elements. However, for larger structures the use of beam and/or shell elements is often necessary to ensure that a structural model of a reasonable size is developed. Currently, there is no general purpose tool available in commercially available software to map thermal data computed in a solid thermal finite element (FE) model to a beam and/or shell FEA structural model. Therefore, a general purpose and an efficient approach is needed to map thermal results from a solid thermal FE model to an equivalent structural FE model comprising line and/or shell elements. Accurate mapping of thermal results in structural analysis model is needed to account for the effects of thermal expansion/contraction and temperature dependent changes in material properties associated with fire exposure. In this approach, it is inherently assumed that thermal analysis influences the mechanical behavior of structures, while mechanical behavior has minimal influence in thermal behavior.

The text is organized as follows. First, a brief literature review is provided. Next, theoretical background of the proposed methodology is discussed. Then the verification of the proposed temperature mapping approach is discussed with two examples: (a) a single steel beam model and (b) a composite floor model used for Cardington fire test 1 [1]. Finally, the paper concludes with a brief summary.

* Tel.: +1 301 975 3538; fax: +1 301 975 4553.
E-mail addresses: Dilip.Banerjee@nist.gov, banerjee_dilip@yahoo.com

http://dx.doi.org/10.1016/j.firesaf.2014.06.002
0379-7112/Published by Elsevier Ltd.

2. Literature review

In recent times, data mapping challenges between thermal and structural models have been attempted by few researchers/groups. Each of these approaches has its advantages and disadvantages. The following sections provide a brief summary of efforts from several groups in developing data mapping algorithms between thermal and structural models.

2.1. SAFIR[1] [2]

SAFIR is a well-known software used to model structural behavior under fire. Thermal analysis in SAFIR results in temperatures across each member cross section and these temperatures are stored for subsequent structural analysis. The type of elements needed for thermal analysis depends on the type of elements that are selected for subsequent structural analysis. For the thermal analysis associated with 2D beam elements, the beam's cross-section is divided into a number of fibers, which are linear and may have three or four nodes (triangular or quadrangular elements). The beam element has a constant cross-section along the longitudinal axis. This cross-section is the same as that defined during the previous thermal analysis, which is divided into several fibers. During the mechanical analysis, the material behavior of each fiber is evaluated in the fiber's center and assumed constant for the whole fiber. Each fiber may have its own material properties. However, it is not clear how mechanical behavior of fibers corresponding to a given node is mapped to the nodal results. In addition, details on data mapping using other types of elements are not available.

2.2. Jeffers and Sotelino [3]

The authors propose the development of a novel type of “heat transfer finite element” that can be used to model the three-dimensional thermal response of structural beams and columns subjected to elevated temperatures associated with fire. The proposed element is a three-node heat transfer element that uses a fiber discretization to account for both transverse and longitudinal temperature variations in a structural member. The authors implemented this element in ABAQUS [10] using a user-defined element subroutine. The methodology involves conducting both finite difference and finite element approximations of the governing heat conduction equations using this fiber element. The fiber heat transfer element is designed to be compatible with any fiber-based finite element software.

The proposed fiber heat transfer element is a 1-D fiber element with a fiber discretization over the member's cross section. The temperatures of each fiber are assumed to vary along its length only and temperatures are uniform across each fiber's cross-sectional area. A 3-node element is considered along the fiber length. Fiber temperatures are approximated by quadratic shape functions. Temperature variations across the cross section of a structural member are considered by writing finite difference equations describing heat flow over fiber boundaries. Each fiber is treated as a separate control volume. Fiber (i, j) is treated as a 1D finite element with n nodes along its length. The temperature $T_{i,j}$ in fiber (i, j) is interpolated from the fiber's nodal temperatures $\{T_{i,j}\}$ using the fiber's shape function matrix. Using Galerkin's finite element method the resulting element equations are formulated. By assembling equations for individual fibers, a global system of equations is obtained. For a 3-node fiber element with n_{fib} fibers, the solution vector is a vector of dimension $3n_{fib}$. Temperatures can be calculated directly at the Gauss point of the structural element using shape functions of the fiber heat transfer element. However, the implementation of this methodology requires one to develop the element formulation specifically for a commercial software, which is quite challenging. Also, it is inherently assumed that both thermal and structural models have similar discretization, which may not be attractive from a practical standpoint.

2.3. Duthinh et al. [4]

A computer program was written in ANSYS [5] APDL language for providing a 2-way interface between ANSYS thermal and ANSYS structural results. This utility allows for transfer of thermal and structural data (temperature, deflection, strain etc.) between incompatible elements. For example, one can transfer temperatures from an ANSYS heat transfer model comprising solid elements to an ANSYS structural model comprising beam elements. The elements supported for this utility are structural elements (beam and shell) and thermal elements (solid). The program can handle several types of singly symmetric and doubly symmetric cross sections. The program calculates temperatures at specific transfer points on the cross-section of structural elements by interpolating temperatures between nodes in the corresponding thermal model. In the structural model, the temperatures are included as element thermal loads as defined in the description of the ANSYS element [5].

For example, the temperatures for the ANSYS BEAM 188 element can be input as element body or thermal loads at three locations along the cross section at each end node of the beam element. At each end node of the beam, temperatures are computed at $(x, 0, 0)$, $(x, 1, 0)$, and $(x, 0, 1)$ locations as defined in ANSYS element descriptions for the BEAM 188 element. Since this approach was implemented in the ANSYS environment only, it would require additional efforts to implement this approach in other software using different element formulation methods.

2.4. Thermal Desktop [6]

Thermal Desktop is a CAD-based and FE-compatible software, which is used for thermal data mapping. Thermal model can use any combination of finite difference or finite element models. It has been implemented in MSC/NASTRAN software [7]. Thermal models and structural models can be developed independently in this approach.

Postprocessed data (temperatures, heat fluxes, etc.) calculated from a thermal model may be mapped to any set of arbitrary point locations in structural model. For each point for which data are to be mapped (e.g., a NASTRAN grid point), the software checks each thermal object for proximity to the point. If the point lies on or in the object, data for the object are interpolated at the point location. Finite difference surfaces interpolate linearly in their parametric space. Finite element primitives use their corresponding shape functions. To provide a generalized capability, locations for which thermal data is to be mapped may be input as an ASCII text file specifying a name (for example, a structural FE node ID) and a $[x, y, z]$ location. Output is generated as an ASCII file consisting of the name and the mapped thermal data at the location corresponding to the input name. Locations to be mapped may also be specified using a NASTRAN input deck. Grid points are read directly from the input deck and temperature data are output in NASTRAN load set format. Temperatures can be solved using SINDA/FLUINT [8] and results mapped onto the NASTRAN structural model.

The correct approach is to take the collection of x, y, z locations for each structural node and interpolate using the geometric

[1] Certain commercial software or materials are identified to describe a procedure or concept adequately. Such identification is not intended to imply recommendation, endorsement, or implication by NIST that the software or materials are necessarily the best available for the purpose.

Download English Version:

https://daneshyari.com/en/article/269818

Download Persian Version:

https://daneshyari.com/article/269818

Daneshyari.com