Structures xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Structures

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

Numerical Modelling of Composite Floor Slabs Subject to Large Deflections

M.M. Florides, K.A. Cashell *

Department of Mechanical, Aerospace and Civil Engineering, Brunel University, London, UK

ARTICLE INFO

Article history:
Received 14 June 2016
Received in revised form 21 September 2016
Accepted 17 October 2016
Available online xxxx

Keywords:
Composite slabs
Fire
Numerical modelling
Tensile membrane action
BRE method

ABSTRACT

This paper is concerned with the ultimate behaviour of composite floor slabs. Steel/concrete composite structures are increasingly common in the UK and worldwide, particularly for multi-storey construction. The popularity of this construction form is mainly due to the excellent efficiency offered in terms of structural behaviour, construction time and material usage all of which are particularly attractive given the ever-increasing demands for improved sustainability in construction. In this context, the engineering research community has focused considerable effort in recent years towards understanding the response of composite structures during extreme events, such as fires. In particular, the contribution made by the floor slab system is of crucial importance as its ability to undergo secondary load-carrying mechanisms (e.g. membrane action) once conventional strength limits have been reached may prevent overall collapse of the structure. Researchers have focused on developing the fundamental understanding of the complex behaviour of floor slabs and also improving the methods of analysis. Building on this work, the current paper describes the development and validation of a finite element model which can simulate the response of floor slab systems until failure, both at ambient and elevated temperature. The model can represent the complexities of the behaviour including the temperature-dependent material and geometric nonlinearities. It is first developed at ambient temperature and validated using a series of experiments on isolated slab elements. The most salient parameters are identified and studied. Thereafter, the model is extended to include the effects of elevated temperature so it can be employed to investigate the behaviour under these conditions. Comparisons with current design procedures are assessed and discussed.

1. Introduction

Over the last number of years, the performance of buildings with steel/concrete composite floors during fire conditions has received increasing attention from the structural engineering community (e.g. [1–3]). This mainly followed observations during real building fires such as the Broadgate and Basingstoke fires where buildings with composite floors performed much better than expected due to the ability of the slabs to survive and redistribute loads around the structure. Large-scale experiments were conducted at Cardington [4] to investigate the behaviour under more controlled conditions and it was observed that traditional prescriptive design methods are overly conservative and steel-framed buildings with composite floors inherently possess sufficient ductility and resistance during extreme events to delay or even prevent failure. The Cardington experiments led to a surge in interest from the engineering research community with work focused on developing a greater understanding of the behaviour through further experimental and numerical analysis.

The response of a two-way spanning floor slab during a fire is particularly complex owing to the interrelated material and geometric nonlinearities which develop with increasing levels of deflection and temperature. Although the slab exhibits significantly lower bending capacity in a fire due to the degradation of material strength and stiffness, the development of tensile membrane action can lead to a greater overall capacity than predicted by the design codes (e.g. Eurocodes). However, before tensile membrane action can be incorporated into design standards, a detailed and fundamental understanding of the behaviour of floor slab must be attained.

Towards this end, a limited number of experimental programmes have studied the large-deflection performance of isolated slab elements both at ambient and elevated temperature (e.g. [2,5]). However, large-scale experiments are prohibitively expensive and time-consuming and so a full examination of the various parameters affecting the behaviour is not realistically feasible. Therefore, a number of purpose built numerical models have been developed by the research community to study the effects influencing the response of structures and floor slabs in particular under fire loading scenarios (e.g. [6,7]). Although these models have led to considerable advancement in the understanding of structures in fire, they are often not suitable for design as they can be computationally expensive and the scale of the structures may be difficult to realistically represent. Practical design guidance and software for

---

* Corresponding author.
E-mail addresses: Marios.Florides@brunel.ac.uk (M.M. Florides), Katherine.Cashell@brunel.ac.uk (K.A. Cashell).

http://dx.doi.org/10.1016/j.istruc.2016.10.003

Please cite this article as: Florides MM, Cashell KA, Numerical Modelling of Composite Floor Slabs Subject to Large Deflections, Structures (2016), http://dx.doi.org/10.1016/j.istruc.2016.10.003

steel framed buildings with composite floor slabs in fire have been proposed, the most well-known of which is the BRE method [8,9], to help engineers achieve safe and efficient designs. However, several shortcomings of the BRE method have been presented by various researchers (e.g. 3, [10]) including an absence of appropriate failure criteria and also a lack of consistency in terms of the method providing a conservative or unconservative assessment of the load carrying capacity, depending on the aspect and reinforcement ratios of the slab.

The main focus in this paper is the development of a numerical model that can accurately predict the response of isolated simply-supported strip and slab elements in both ambient and elevated temperature conditions until failure. This work is part of a larger research programme which will investigate the effects of various boundary and geometric properties and also propose analytical solutions. Importantly, the model considers the highly influential effect of bond strength between the steel reinforcement and surrounding concrete on the ultimate performance; this has already been shown to be critical for assessing the level of load and deflection which can be sustained before failure [2,3]. Accordingly, this paper proceeds with an overview of recent studies carried out to examine the ultimate performance of one- and two-way spanning lightly reinforced elements, such as composite floor slabs. A finite element model is developed using the ABAQUS software; the model is first described for ambient conditions and validated against a series of experiments on isolated members. Subsequently, the results are compared with those obtained utilising the finite element software VULCAN and the BRE method. This is an essential precursor to further expansion of the model to incorporate the effects of elevated temperature, such as those which occur during a fire.

2. Development of the numerical model

A finite element model (FEM) has been developed using the commercially-available ABAQUS software [11] which is capable of achieving numerical convergence despite the geometric and material nonlinearities of the behaviour. Many different finite element packages can be used for this purpose and have been by other researchers (e.g. [3, 7,12]). However, ABAQUS was selected for this work because great importance is given to being able to readily compare results to those from other researchers and, more importantly, developing models and procedures which are not reliant on proprietary software.

In order for the FEM to accurately predict the response of slabs with various combinations of material and geometrical properties and to ensure that the model is readily usable by designers, a number of steps were followed in the modelling procedure. During this process, importance was given to ensuring that the material and geometric properties such as concrete/reinforcing steel strength, dimensions and boundary conditions were realistically represented and yet ensuring that the model requires knowledge only of the typical properties which are known by designers. Owing to the complexity of the behaviour of two-way spanning slabs, the model was developed in three major steps: (i) one-way spanning slab strips at ambient temperature (ii) two-way spanning slabs at ambient temperature and (iii) incorporation of the effects of elevated temperature. The first two stages are very important and an essential pre-requisite for understanding the mechanical behaviour of these elements at large deflections, such as those that occur in a fire. In the following sections, the development of the one- and two-way spanning models is first described, including the material representations. The inclusion of elevated temperature effects is discussed later in Section 4.

2.1. One-way spanning slab strips

The task of creating a FEM that is user-friendly on the one hand and also able to reliably simulate the behaviour of reinforced concrete elements on the other is extremely challenging. This is mainly due to the nature of the material behaviour, particularly the interaction between the reinforcing steel and the surrounding concrete, a subject that has been the focus of many researchers (i.e. [13,14]). In the current research programme, a great deal of importance was given to the representation of bond between the steel and the concrete in the model and, in particular, ensuring that it was included in a manner which is appropriate for designers. This means using methods which only require the typical material and geometric properties which are known by engineers (e.g. material strength, stiffness, dimensions, etc.). For this reason, and since the analysis of slab elements can be based on strip element analysis (particularly of one-way spanning slab members), the FEM described hereafter was calibrated and validated using a recent series of ambient temperature tests on isolated slab strips with various geometric and material properties, including bond strength. For brevity, only a brief account of the tests is given herein with a more comprehensive description available elsewhere [14].

Fig. 1 shows the general geometry of the specimens and the testing rig, whilst more specific details of each specimen are described in Table 1 which gives the half-length (L), width (b) and depth (h) of the strips, as well as the reinforcement ratio (ρ), concrete compressive strength (f_c) and concrete tensile strength (f_t). The effective depth of the reinforcement from the compressive face is half the overall depth (i.e. $h/2$) in all cases. Four types of reinforcement were considered in the test programme, namely: (i) plain bars with a diameter of 6 mm (P6); (ii) deformed bars with a diameter of 6 mm (D6); (iii) deformed bars of 8 mm diameter (D8); and (iv) A142 welded mesh consisting of 6 mm deformed bars spaced at 200 mm centres (M6).

In the ABAQUS FEM, the concrete elements were modelled using 3D solid elements with reduced integration from the ABAQUS library (C3D8R). The model employed a square mesh comprising of 20 mm cubic elements, based on a mesh sensitivity assessment. Solid elements were employed because although computationally expensive, they better represent damage in the material and the relationship between the concrete and the embedded reinforcement at elevated temperature, which is particularly important in this study. The reinforcement was modelled using linear 3D truss elements (T3D2) which were embedded in the solid concrete elements. Due to symmetry, only a part of the specimens was modelled such that one reinforcement bar was included in the concrete at any time (Fig. 2). The strips rest on a fixed rigid frame and are free to move both rotationally and laterally. The loading arrangement was identical to that in the testing procedure which is shown in Fig. 1 and the elements were loaded in displacement-control. Although ABAQUS includes several static analysis methods, in order to facilitate both the ambient and elevated-temperature loading, a quasi-static dynamic, implicit analysis was employed in this study.

2.2. Two-way spanning slabs

The one-way spanning strip model described in the previous section was extended for two-way spanning slabs. The steel decking was not included in the analysis because it was observed during real fire tests that it de-bonded from the concrete slab at an early stage and ceased to contribute to the load-carrying capacity of the slab [8]. Similarly to the one-way spanning elements, the concrete slab panels were modelled using

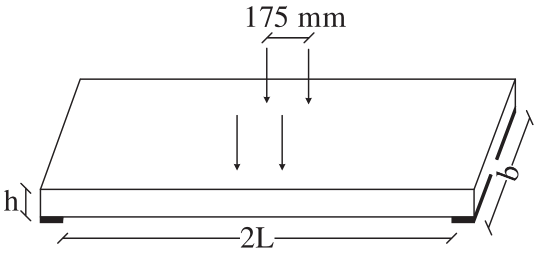

Fig. 1. General schematic of strip tests and test rig [14].

Please cite this article as: Florides MM, Cashell KA, Numerical Modelling of Composite Floor Slabs Subject to Large Deflections, Structures (2016), http://dx.doi.org/10.1016/j.istruc.2016.10.003