BEHAVIOUR OF COMPOSITE FLOOR SLABS UNDER FIRE CONDITIONS

M. M. Florides & K. A. Cashell

Department of Mechanical, Aerospace and Civil Engineering, Brunel University, London, UK
E-mails: Marios.Florides@brunel.ac.uk, Katherine.Cashell@brunel.ac.uk

Keywords:

Composite slabs; fire; numerical modelling; tensile membrane action; BRE method.

A B S T R A C T

This paper is concerned with the ultimate behaviour of composite floor slabs during fire scenarios. 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 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 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 be the key to preventing disproportionate collapse of the overall 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 and is 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. Bailey (2004), Cashell et al. (2011a, b). This mainly followed observations during real building fires such as the Broadgate and Basingstoke fires when buildings with composite floors performed much better than expected due to the ability of the slabs to survive and re-distribute loads around the structure. Large-scale experiments were conducted at Cardington (Martin & Moore 1999) 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 loads 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 slab during a fire is particularly complex owing to the inter-related material and geometric nonlinearities which develop with increasing deflections 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 assessment of the behaviour of floor slab must be attained.

Towards this end, a limited number of experimental programmes have studied the performance of isolated slab elements both at ambient and elevated temperature (e.g. Bailey & Toh 2007). However, large-scale experiments are prohibitively expensive and so a full examination of the various parameters affecting the behaviour is not 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 (e.g. Izzuddin 1991, Huang et al. 2003) in particular under fire loading scenarios. 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 steel framed buildings with composite floor slabs in fire has been proposed (e.g. Bailey & Moore 2000, Newman et al. 2000) to help engineers achieve safe and efficient designs. However, several shortcomings of the BRE method have been presented by various researchers (e.g. Cashell et al. 2011, Burgess 2014) including a lack 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 ratio of the slab.

The main focus in this paper is the response of isolated simply-supported slab panels in both ambient and elevated temperature conditions. 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. Accordingly, the paper proceeds with an overview of recent studies carried out to examine the ultimate performance of floor slabs. A finite model is developed using the ABAQUS software. The model is first described for ambient conditions and validated against a series of experiments on isolated slabs. 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. The elevated temperature model is then described and validated against data available in the literature.

2. AMBIENT TEMPERATURE RESPONSE

2.1 *Development of the numerical model*

A finite element model (FEM) has been developed using the commercially-available ABAQUS software, (ABAQUS 2013) which is capable of achieving numerical convergence despite the geometric and material nonlinearities. Many different FE packages can be used for this purpose and have been by other researchers. However, ABAQUS was selected for this project 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.

The model represents the key components of a composite floor such as the concrete slab and steel reinforcement. The steel decking is not included in the analysis because it was observed at Cardington that it de-bonded from the concrete slab at an early stage during the fire tests and ceased contributing to the load-carrying capacity of the slab (Bailey & Moore 2000).

The concrete slab panels are modelled using 3D solid elements with reduced integration (C3D8R) from the ABAQUS library. The model employs a mesh comprising of 20mm cubic elements, based on a mesh sensitivity assessment. Solid elements are employed because although they are computationally expensive, they better represent damage in the material and the relationship between the concrete and the embedded reinforcement at elevated temperature. The reinforcement is modelled using linear 3D truss elements (T3D2) which are embedded in the solid slab elements. Due to symmetry, a quarter of the slab is modelled (Fig. 1). The slab is simply supported and free to move both rotationally and laterally at the supports. The loading arrangement can be varied (e.g. uniformly distributed, point loads, etc.) and the slabs are generally 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 is employed.

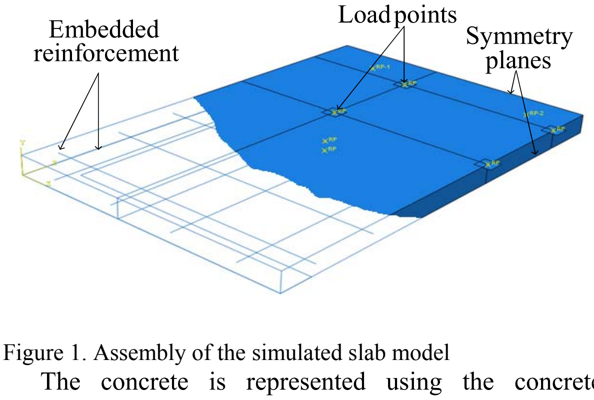


Figure 1. Assembly of the simulated slab model

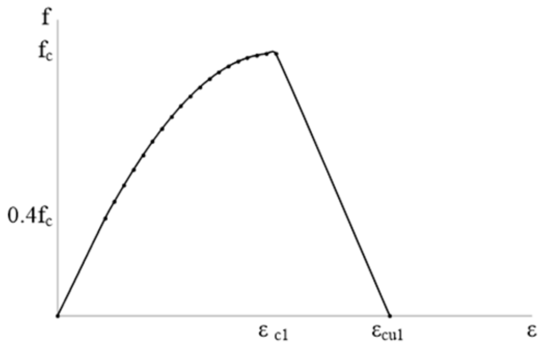
The concrete is represented using the concrete damaged plasticity model which uses the concept of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behaviour of concrete, and a combination of multi-hardening plasticity and isotropic damaged elasticity to describe the irreversible damage that occurs during the fracturing process. The tension stiffening property in this model is employed to simulate the bond between the steel reinforcement and the surrounding concrete in an indirect way. The stress-strain curves for concrete at ambient temperatures are shown in Figure 2.

Under uniaxial compression, the response of concrete is based on the equations given for non-linear analysis by BS EN 1992-1-1 (2004) while in tension the relationship specified in the CEB-FIP Model Code 90 (1990) is used. More specifically, under compression the response is linear until the proportional stress is reached (approximately equal to $0.4f_c$, where f_c is the concrete compressive strength) and then irreversible damage is included in the calculations.

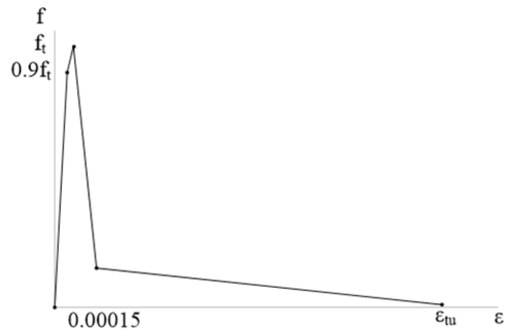
Under tension, the stress-strain curve follows a linear elastic relationship until the value of the maximum tensile capacity (f_t) after which there is a softening response to compensate for the existence of the embedded reinforcement and thus the bond between the two materials. The CEB-FIP code recommends that a stress-strain diagram should be used for uncracked concrete and a stress-crack opening diagram for the cracked section whereas ABAQUS requests the input of a single curve. The stress-crack opening was converted to the softening stress-strain part of the curve using the fracture energy G_f (energy required to propagate a tensile crack of unit area) divided by the crack band width Bažant (1983).

2.2 *Validation of the numerical model at ambient temperature*

In this section, the FEM described above is calibrated and validated using a series of ambient temperature tests on isolated slab members with various geometric and material properties. The experiments focused on lightly reinforced concrete slab specimens with the main aim of providing a greater insight into the large displacement behaviour of floor slab systems. For brevity, only a brief account of the tests will be given herein with a more comprehensive description available elsewhere (Cashell et al. 2011). Figure 3 shows the general geometry of the two different types of specimens tested (aspect ratio of 1 and 1.5), while Table 1 provides the relevant geometric and material properties for all slab specimens. The reference system adopted in labelling each specimen is kept the same as in the original work and is as follows: the first parameter denotes a rectangular (R) or square (S) slab; F40 and F60 represent the depth of the slab in mm; the third parameter describes the reinforcement used (P6 for plain bars of 6mm diameter, D6 for deformed bars of 6mm diameter, D8 for deformed bars of 8mm diameter and M6 for A142 welded mesh consisting of 6mm deformed bars spaced at 200mm centres) while A, B, C and D signify various reinforcement arrangements used. The table also includes information relating to the depth of the slab H, the long and short spans L1 and L2, respectively, and also ρ1 and ρ2, which are the reinforcement ratios in the long and short spans. All of these experiments have been modelled using the FEM but owing to space limitations, one experiment (S-F60-M6-A) is selected for detailed analysis of the load-deflection and temperature-deflection performance (see Section 3).



(a)



(b)

Figure 2. Representation of the concrete characteristics in (a) compression and (b) tension

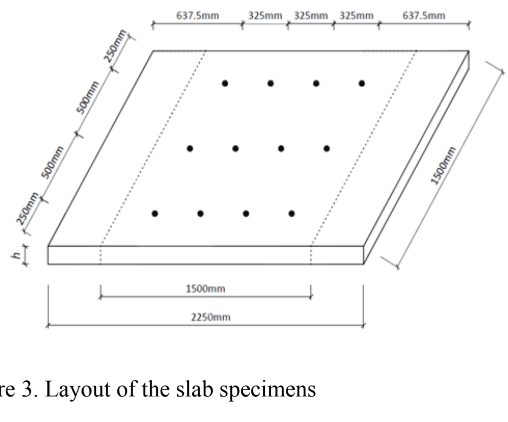


Figure 3. Layout of the slab specimens

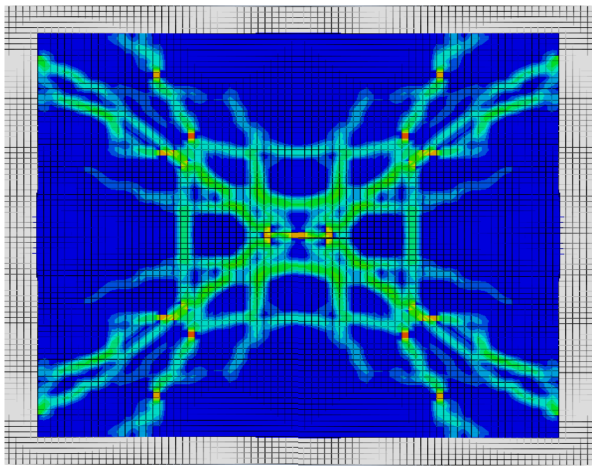
Table 1 Properties of the slab tests (Cashell et al. 2011)

Model	L1 (mm)	L2 (mm)	H (mm)	Bar type	ρ1 (%)	ρ2 (%)
R-F60-M6-A	2250	1500	60	M6	0.24	0.24
R-F60-P6-A	2250	1500	60	P6	0.24	0.24
S-F60-M6-A	1500	1500	60	M6	0.24	0.24
R-F40-D6-B	2250	1500	40	D6	0.35	0.35
R-F60-D6-C	2250	1500	60	D6	0.24	0.48
R-F60-D6-A	2250	1500	60	D6	0.24	0.24
S-F60-D6-A	1500	1500	60	D6	0.24	0.24
S-F60-D6-D	1500	1500	60	D6	0.52	0.52
S-F60-D8-D	1500	1500	60	D8	0.52	0.52
S-F60-P6-A	1500	1500	60	P6	0.24	0.24
R-F60-M6-A	2250	1500	60	M6	0.24	0.24
R-F40-M6-B	2250	1500	40	M6	0.35	0.35
R-F60-D8-A	2250	1500	60	D8	0.28	0.28
R-F60-D8-C	2250	1500	60	D8	0.28	0.56

Figure 4(a) shows the crack pattern for specimen S-F60-M6-A whilst the FEM prediction for the same specimen is given in Figure 4(b). Clearly, the model depicts the development of conventional yield line cracks as well as the additional cracks in the centre of the slab. The experimental and numerical load-deflection response for slab specimen S-F60-M6-A is given in Figure 5. Also presented in the graph is the simulation from the BRE analytical method as well as the prediction from another commonly-used finite element model, VULCAN. The BRE simplified design method estimates the load-carrying capacity of a lightly reinforced concrete slab as a function of the vertical displacement based on the in-plane stresses (membrane action) in the slab. VULCAN, on the other hand, is a finite element package which was first developed at the University of Sheffield to model the 3-dimensional nonlinear behaviour of composite steel-framed buildings under fire conditions. It has been extensively validated since its development and is used herein to ratify the results obtained from the ABAQUS FEM.

In the VULCAN model, a 9-noded quadrilateral plate element (as shown in Fig. 6) is used to model the slab specimens which can capture both the bending and membrane effects. Each element is divided into 16 layers of which 14 represent the concrete with the remainder used for steel reinforcement. The configuration of the model is an exact replicate of the experimental conditions at ambient temperature. More advanced elements are available in VULCAN which is capable of representing the interaction between the steel reinforcement and concrete but these are beyond the scope of the current work and will be incorporated into future models.

(a)



(b)

Figure 4. Cracking pattern for S-F60-M6-A (a) from the experimental programme (Cashell et al. 2010) and (b) from ABAQUS

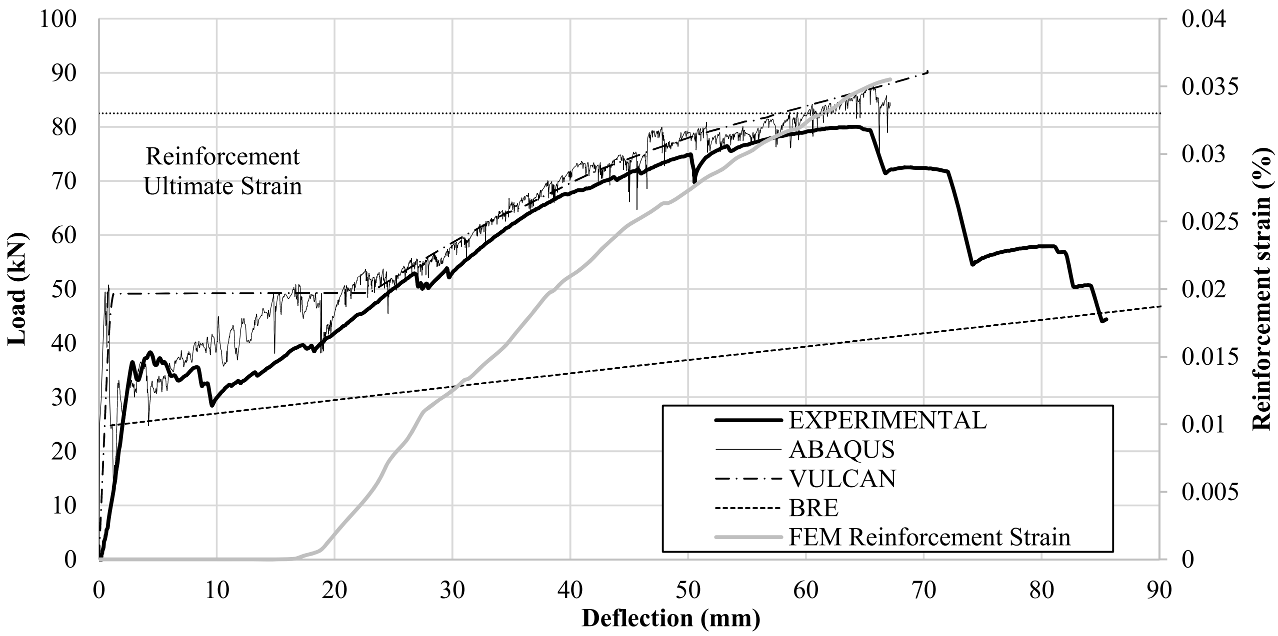


Figure 5. Load-displacement response of S-F60-M6-A (primary axis) and strain induced in the reinforcement along the formed yield line for the corresponding vertical displacement (secondary axis)

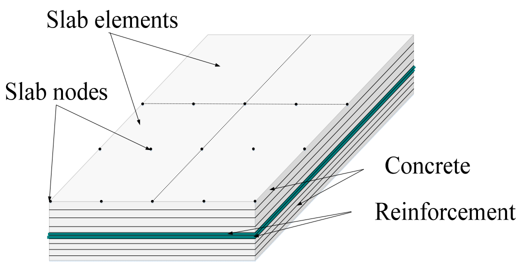


Figure 6. 9-noded quadrilateral plate element employed for analysis in VULCAN

The load-deflection response shown in Figure 5 for the specimen S-F60-M6-A illustrates that the overall behaviour of the slabs is well described by the FEM. First cracking of the concrete, evidenced by the drop in load at small levels of deflection, is well predicted. Furthermore, yielding of the reinforcing steel is also well simulated with reasonable accuracy until failure.

Discrepancies in the results are attributed to the effect of the smeared cracked approach associated with the concrete damaged plasticity model which is reliant on the mesh size. Although the gross crack patterns remain the same regardless of the mesh size, the load capacity is inaccurate if the mesh density is relatively coarse. A mesh sensitivity study has been completed in order to calibrate the elements used in the current work. The local fluctuations in the numerical results are a numerical issue due to the simulation of the interaction between the rigid elements (loading plates and base) and the concrete slab. A finer mesh at the interaction interface would reduce this effect as the distortion of the contact elements would be smaller. Nevertheless, the results obtained are reliable with this level of noise.

The VULCAN model also provides a very good representation of the nonlinear behaviour of the slabs although only one slab is presented herein owing to space limitations, the other slabs in this test programme have also been studied and show similarly good results. The BRE method provides a reasonable prediction of the response at small deflections in ambient temperatures but this prediction becomes less representative and overly conservative at large deflections.

In terms of failure, the slab failed by fracturing of the steel reinforcement across a localized through-depth crack across the shorter span. The ultimate strain of the steel reinforcement in the tests was found through tensile testing to be 0.033%, as indicated in the Figure 5. Also presented in the graph using the secondary y-axis is the FEM prediction for the strain induced in the reinforcement along the failure crack for the corresponding vertical displacement of the slab. Clearly, failure of the reinforcement is well predicted although it is important to note that this prediction is very much dependent on the depiction of bond strength in the model, which is currently represented through the tension stiffening property in the concrete damaged plasticity model. In this respect, the experiments calibrate the numerical model rather than validate the behaviour.

3. ELEVATED TEMPERATURE RESPONSE

In this section, the ABAQUS FEM developed previously for ambient temperature conditions is extended to consider the effects of elevated temperature.

3.1 *Elevated temperature numerical model*

The sequentially coupled thermal-stress analysis procedure in ABAQUS/Explicit is used to represent the slabs behaviour at elevated temperature. This approach has been used by other researchers (e.g. Nguyen, T-T. et al. 2015) and is employed because the stress/displacement solution is dependent on a temperature field without any inverse dependency. The sequentially coupled thermal-stress analysis is performed by first performing a heat transfer analysis to obtain the temperature distribution through the cross-section of the slab and then reading these values into the stress analysis.

ABAQUS allows different meshes to be used between the two stages of the analysis and so a finer mesh was employed in the heat transfer stage, particularly through the depth. This was to ensure that the nodes at the reinforcement level in the stress analysis are matched by the nodes at the same locations in the heat transfer analysis and the temperature of the embedded reinforcement is depicted accurately.

As in the ambient temperature model, the concrete is depicted using the damaged plasticity model whilst the reinforcement is modelled as an elastic/plastic material. The material mechanical properties for both concrete and the reinforcement at elevated temperatures are represented using the material reduction factors and properties specified in EN 1992-1-2.

3.2 *Results from the elevated temperature numerical model*

The same slab previously modelled at ambient temperature (i.e. S-F60-M6-A) is again presented in this section to illustrate the results at elevated temperature. A uniformly distributed load of 15kN/m^2 was applied to the concrete surface; this load level results in an elastic response at ambient temperature (Fig. 5). The slab is then subjected to the standard ISO 834 fire curve, as included in BS EN 1991-1-2 (1991), for 90 minutes. The temperature values for the top, bottom and reinforcement layers are presented in Figure 7.

Figure 8 presents the crack pattern simulated by the ABAQUS model. It is observed to be very similar to that presented in Figure 4 for the same slab at ambient temperature and also to the pattern predicted by yield line theory. This indicates that a similar failure mechanism is likely to occur both at ambient and elevated temperature.

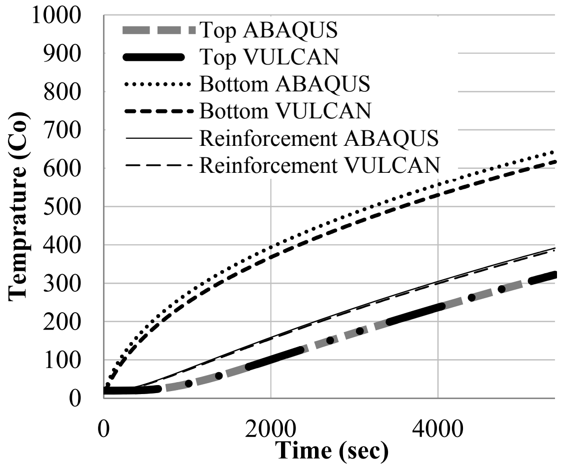


Figure 7. Temperatures at key layers for ABAQUS and VULCAN stress analysis

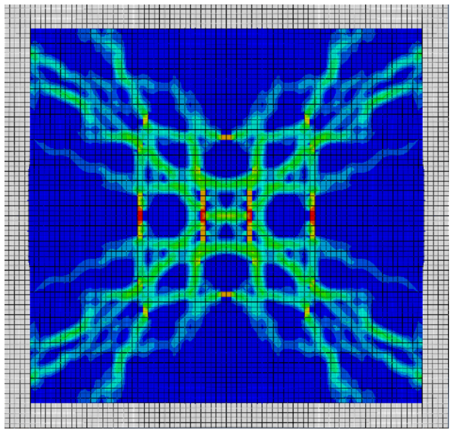


Figure 8. Crack pattern for S-F60-M6-A after the elevated temperature analysis

3.3 *Validation of the elevated temperature numerical model*

In order to verify the results obtained from the ABAQUS numerical model, a comparison of the behaviour is made with the FE code VULCAN which has been extensively validated for similar elevated-temperature applications (e.g. Huang et al. 2004). VULCAN considers the nonlinear elevated-temperature material behaviour.

The model created in VULCAN has similar material and loading properties to those in the ABAQUS model. The slab shown in Figure 9 is simulated and meshed so that the layers are compatible with the VULCAN layered quadrilateral plate element. The heat transfer component of the analysis is performed in ABAQUS because VULCAN does not readily have this capability and then the temperature values in the various layers are imported into the VULCAN model.

As before, during the heat transfer analysis, the slab is subjected to the standard ISO 834 fire curve for 90 minutes. Assuming a linear distribution of temperature along each layer of the VULCAN model, the nodal temperatures from the ABAQUS heat transfer analysis are used to calculate the average temperature in each layer. The difference between the ABAQUS nodal temperatures and the VULCAN average layer temperatures reduces towards the unheated site of the slab. Of particular interest is the temperature on the reinforcement layer where the difference is insignificant.

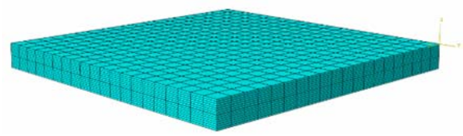


Figure 9. Heat transfer analysis model used to extract the VULCAN temperatures

The graph in Figure 10 presents the results of the ABAQUS and VULCAN models in solid black and grey lines, respectively. It is observed the two approaches are in reasonable agreement. Discrepancies between the two models are most likely due to differences in the development of cracks.

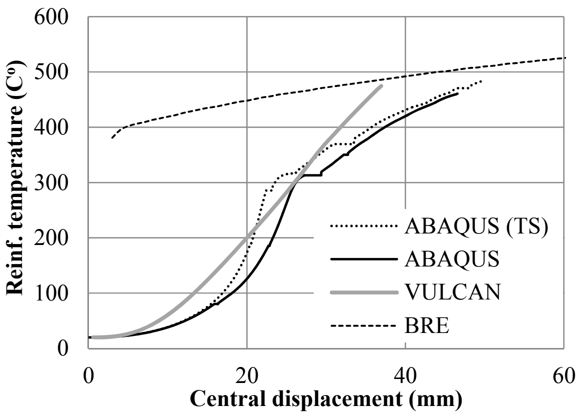


Figure 10. Temperature-displacement response of S-F60-M6-A

The BRE simplified design method has also been used to analyse this slab at elevated temperatures and the results compared to those from the ABAQUS and VULCAN FE models, and these predictions are also presented in Figure 10. This analytical method estimates the load-carrying capacity of a lightly reinforced concrete slab as a function of the vertical displacement based on the in-plane stresses (membrane action) in the slab. By incorporating the thermal effects on the material properties in the equations the temperature-displacement response of the slab can be extracted. As shown in Figure 10, the results demonstrate that the BRE simplified design method is in good agreement at elevated temperature and large deflection levels with the response predicted by the two models, contrary to the estimated response at ambient temperatures.

4. EFFECT OF TENSION STIFFENING/BOND

The model in ABAQUS is used to test a range of salient factors that alter the response of the slab at elevated temperatures, including the effect of the tension stiffening property used in the concrete damage plasticity model to simulate the bond, which will be discussed herein. As evidenced by Cashell et al. (2010), bond is of great importance in the evaluation of lightly reinforced concrete members in large displacements, particularly failure. The line labelled "ABAQUS (TS)" in Figure 10 illustrates the response of the same slab specimen with altered tension stiffening concrete properties (tensile strength of concrete remains the same). In this case the crack band width used to transform the stress-crack opening to stress-strain is smaller than that suggested by Bažant (1983) (w_c=2d_a instead of 5d_a, where w_c is the width of the crack and d_a is the maximum aggregate size equal to 10 mm for the tests presented in this paper).

A simple three-point bending experimental setup as shown in Figure 11 has been created in ABAQUS to investigate the effect of tension stiffening and highlight its effect on bond simulation. Two sets of three simply supported concrete elements with a single embedded reinforcing bar are created. The reinforcement bar has a different tension stiffening value for each set, related to the crack band width, and each element within the set has a different cover distance (either 20, 35 or 50 mm).

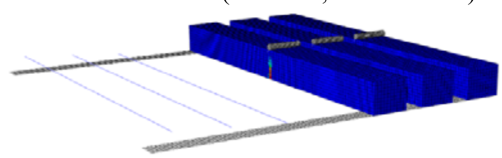


Figure 11. Model created for the investigation of tension stiffening in the concrete damage plasticity model

The results from the model show that the tension stiffening property of concrete is affected by the position of the reinforcement relative to the cracking face of the concrete. The development of the crack is disrupted by the embedded reinforcement (Bažant 1985) and stress is concentrated in the reinforcing bar at the position of the crack. If the fracture energy is increased (i.e. smaller w_c) the strain in the reinforcement is less localised. To obtain the results presented for S-F60-M6-A (Figs. 8 and 10) a bilinear relationship is used to represent the post-cracking stress strain relationship; the values used in the model were calibrated using the test results of Cashell et al. (2010). It is possible to use a linear relationship post-cracking instead with reduced ultimate strain values, as shown in the descending branch in Figure 12. This model can be used to simulate a range of bond conditions accordingly through the slope of the descending branch. A more detailed study into the effect that bond and tensioning stiffening has on the behaviour is currently underway and will be presented in future publications.

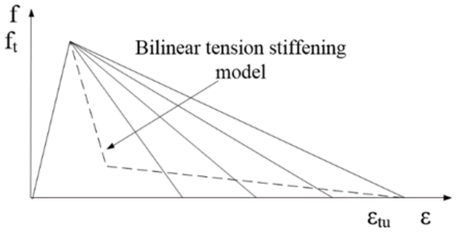


Figure 12. Stress-strain diagrams assumed to assess the tensions stiffening effect in the ABAQUS model.

5. CONCLUSIONS

This paper presents a numerical model using the commercially-available ABAQUS software which is capable of depicting the nonlinear response of steel-concrete composite slabs in fire conditions. The model has been validated using the previously-validated VULCAN software, and compared to the BRE simplified design method for both ambient and elevated temperature. Important parameters that influence the outcome of the FEM have been identified and discussed where appropriate. The work described in this paper is the first step in a larger research programme and the future targets include:

- Expanding the model to include the influence of neighbouring compartments on the overall behaviour in fire;
- Using the validated model to develop an understanding of the most salient parameters such as boundary conditions, continuity, bond strength and various other material and geometric properties under ambient and elevated temperatures on the overall response; and
- Proposing performance-based expressions which can be used in design for the ultimate response of floor slabs under fire conditions

The results of this investigation will offer detailed insights into the key factors that govern the ultimate behaviour of buildings with composite floor systems under extreme loading conditions, and provide the essential background to enable the development of more performance-based design expressions.

6. REFERENCES

ABAQUS 2013. "*ABAQUS user's manual Version 6.13-2.*" Hibbitt, Karlsson & Sorensen, Inc, Rhode Island, USA.

Bailey, C.G. & Moore, D. 2000. "*The structural behaviour of steel frames with composite floorslabs subject to fire. Pt. 1: theory.*" The Structural Engineer.

Bailey, C.G. & Moore, D. 2000. "*The structural behaviour of steel frames with composite floorslabs subject to fire. Pt. 2: design.*" The Structural Engineer.

Bailey, C.G. 2004. "*Membrane action of slab/beam composite floor systems in fire.*" Engineering Structures 26(12): 1691-1703.

Bailey, C.G. & Toh, W. 2007. "Behaviour of concrete floor slabs at ambient and elevated temperatures." Fire Safety Journal 42(6): 425-436.

Bažant, Z.P., & Oh, B.H. 1983. "*Crack band theory for fracture of concrete.*" Materials and Structures, 16:155-177.

Bažant, Z.P. 1985. "*Fracture in concrete and reinforced concrete.*" in Mechanics of Geomaterials, ed. Bažant Z.P., John Wiley & Sons Ltd.

CEB-FIP 1991. "*Model Code for Concrete Structures MC90*", Thomas Telford, London.

Burgess, I.W., Huang, S.S., & Staikova, S. 2014. "*A re-examination of the mechanics on tensile membrane action in composite floor slabs in fire.*" 8th International Conference on Structures in Fire, Shanghai, China.

Cashell, K.A., Elghazouli, A.Y., & Izzuddin, B.A. 2010 "*Experimental and analytical assessment of ductility in lightly reinforced concrete members.*" Engineering Structures 32(9): 2729-2743.

Cashell, K.A., Elghazouli, A.Y., & Izzuddin, B.A. 2011. "*Failure assessment of lightly reinforced floor slabs. I: Experimental investigation.*" Journal of Structural Engineering 137(9): 977-988.

Cashell, K.A., Elghazouli, A.Y., & Izzuddin, B.A. 2011. "*Failure assessment of lightly reinforced floor slabs. II: analytical studies.*" Journal of Structural Engineering 137(9): 989-1001.

EN 1991-1-2. 2002. "*Eurocode 1: Actions on structures—Part 1-2: General actions—Actions on structures exposed to fire.*" British Standards Institution, UK.

EN 1992-1-2. 2004. "*Eurocode 2: Design of concrete structures-Part 1-2: General rules-Structural fire design.*" British Standards Institution, UK.

Huang, Z., Burgess, I.W., & Plank, R.J. 2003. "*Modeling membrane action of concrete slabs in composite buildings in fire. I: Theoretical development.*" Journal of Structural Engineering 129(8): 1093–1102.

Huang, Z., Burgess, I.W., Plank, R.J., & Bailey, C. 2004. "*Comparison of BRE simple design method for composite floor slabs in fire with non-linear FE modelling.*" Fire and Materials 28(2-4): 127-138.

Izzuddin, B.A. 1990. "*Nonlinear dynamic analysis of framed structures*" PhD Thesis, Imperial College London.

Martin, D. & Moore, D. 1999. "*The behaviour of multi-storey steel framed buildings in fire.*" British Steel Plc, UK.

Newman, G.M., Robinson J.T. & Bailey C.G. 2000. *"A New Approach to Multi-Storey Steel-Framed Buildings P288"*, The Steel Construction Institute, Ascot, UK.

Nguyen, T. T., Tan, K.H. & Burgess, I.W. 2015."*Behaviour of composite slab-beam systems at elevated temperatures: Experimental and numerical investigation.*" Engineering Structures 82: 199-213.