Tunnelling and Underground Space Technology 77 (2018) 177–187

Contents lists available at ScienceDirect

Tunnelling and Underground Space Technology

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

Performance of reinforced concrete slabs under hydrocarbon fire exposure

R.A. Hawileh[a,*], V.K.R. Kodur[b]

[a] Department of Civil Engineering, American University of Sharjah, P.O. Box 26666, Sharjah, United Arab Emirates
[b] Department of Civil and Environmental Engineering, Michigan State University, East Lansing, United States

ARTICLE INFO

Keywords:
Concrete slabs
Hydrocarbon fire
Tunnels
Modeling

ABSTRACT

This paper presents numerical results on the fire resistance of reinforced concrete (RC) slabs subjected to the severe hydrocarbon fire exposure. The hydrocarbon fire curve represents a possible fire scenario in parking structures, tunnels, and in petro-chemical facilities. A three-dimensional (3D) nonlinear finite element (FE) model is developed to simulate the thermo-mechanical response of a RC slab that incorporates temperature dependent thermal and mechanical properties of concrete and steel reinforcement, respectively. Transient thermal-stress analysis is performed to study the cross-sectional temperature distribution and resulting deflections in the slab during simultaneous application of heat exposure and sustained loading. The predicted temperature distribution within the slab and mid-span deflection are compared with published experimental data. The predicted FE results are in good agreement with the experimental data throughout entire fire exposure duration. The validated FE model is applied in a parametric study to study the fire resistance of RC slabs under different slab configurations and hydrocarbon fire exposure. The varied parameters included fire exposure scenario, service load level, aggregate type, and concrete cover thickness. Results from the numerical analysis showed that hydrocarbon fire exposure has a significant influence on the fire resistance of RC slabs. In particular, the fire resistance of a slab under hydrocarbon fire exposure is 22.4% lower than that of a slab under the standard ISO834 fire exposure. Further, load level has a major effect on the fire resistance of RC slabs under hydrocarbon fire.

1. Introduction

Experimental and numerical research studies over the last few decades focused predominately on the fire resistance of normal-weight reinforced concrete (RC) slabs exposed to standard fires in a furnace (Lie and Leir, 1979; Gamal and Hurst, 1995; Wang et al., 2013; Huang et al., 1999; Cooke, 2001; Gillie et al., 2001; Huang et al., 2003; Huang et al., 2003; Bailey and Toh, 2007; Bailey and Toh, 2007; Huang, 2010; Guo and Bailey, 2011; Allam et al., 2013; Shakya and Kodur, 2015). The conventional temperature-time fire exposures ISO834 (ISO834-1975, 1975) and ASTM E119 (ASTM Test Method E119, 2002), which are quite similar, are simulated to study the fire performance of structural members (slabs, beams, columns, walls), in buildings. In addition, the guidelines on the temperature distribution and fire resistance of RC slabs in international building codes of standards, such as Eurocode 2 (2004) and ACI Committee 216.1-14 (2014) are based on such standard fire exposures.

In a standard fire test of a loaded RC slab, the ISO834 or ASTM E119 temperature versus time fire curve is applied to the bottom surface of the slab in a furnace to evaluate fire resistance. The temperature increases with time until failure of the slab, or until a target (fire resistance rating) is attained (ISO834-1975, 1975; ASTM Test Method E119, 2002).

The behavior of a RC slab during fire depends on number of factors including fire scenario, load level, boundary conditions, and thermal and mechanical properties of the concrete and steel rebars at elevated temperatures. The mechanical properties such as the elastic modulus and compressive strength of concrete and the elastic modulus, yield strength and tensile strength of the steel reinforcement degrade dramatically at elevated temperatures especially beyond 400 °C (Eurocode 2, 2004; ACI Committee 216.1-14, 2014). Thus, during the fire event, the temperature of concrete and steel reinforcement will increase leading to a loss of stiffness and strength in materials which in turn increases deformation in the structural member and possible damage (Buchanan, 2001; Kodur and Dwaikat, 2007; Kodur and Dwaikat, 2011; Kodur et al., 2013).

In previous experimental and numerical research investigations, a static sustained loading was applied on RC slab specimens and subjected solely to a standard fire exposure to develop fire resistance ratings under different configurations. The temperature limit in the steel

* Corresponding author.
E-mail address: rhaweeleh@aus.edu (R.A. Hawileh).

https://doi.org/10.1016/j.tust.2018.03.024
Received 12 July 2017; Received in revised form 11 March 2018; Accepted 24 March 2018

reinforcement, temperature of the top unexposed slab surface, and deflection limits are used in most studies as the limiting criteria to evaluate failure of RC slabs. The varied parameters in the slab included concrete cover thickness to the tensile steel reinforcement, load level, slab thickness, concrete strength, and type of aggregate. However, there is lacking the literature is lack of information on the performance and fire resistance of RC slabs when subjected to severe hydrocarbon fires, which represent a possible scenario in parking structures and tunnels as well as in the petroleum and chemical facilities. Thus, the performance and fire resistance of RC slabs under severe hydrocarbon exposure is not fully quantified and established.

To cover this gap in the literature, this paper aims to study fire resistance of RC slabs with different design configurations under ASTM E1529 (ASTM Test Method E1529, 1993) hydrocarbon temperature exposure. A finite element (FE) numerical model is developed to evaluate the fire resistance of RC slabs under such severe (hydrocarbon) temperature loading scenarios. The developed model considers the variation of thermal and mechanical properties of concrete and steel reinforcement with temperature. Transient thermal-nonlinear stress analysis is performed using the FE software, ANSYS (ANSYS – Release Version 14.5, 2013) to study the temperature distribution and deformation response of concrete slabs during fire exposure. The developed model is validated by comparing temperature and deflection predictions with the experimental data published in the literature (Cooke, 2001) on RC slab subjected to a standard ISO834 fire and to the severe (hydrocarbon) Norwegian Petroleum Directorate (NPD) fire exposures. The developed FE model was then utilized to study the influence of fire scenario, load ratio, aggregate type, and concrete cover thickness on the fire resistance of different configurations of RC slabs under the hydrocarbon ASTM E1529 fire curve.

2. Modeling and analysis methodology

A finite element (FE) model was developed using the finite element software, ANSYS (ANSYS – Release Version 14.5, 2013). Thermo-stress analysis was performed to predict the fire resisting of RC slabs. The slabs are analyzed by subjecting them to simultaneous loading and furnace exposure at the slab's soffit. The analysis is carried out in the following steps:

1. Create two 3D models of the concrete slab using thermal and structural element, to perform thermal and stress analysis, respectively. The developed FE models comprise of the geometry, thermal and mechanical material properties of the concrete and steel reinforcement and their variation with temperature, and applicable boundary conditions.
2. Apply the transient furnace exposure scenario to the bottom surface of the thermal model in the form of a temperature versus time curve. The heat is to be transferred throughout the slab by conduction, convection, and radiation. The top sides of the slab specimen are not exposed to elevated temperatures, and thus are subjected to room temperature.
3. The developed thermal model was validated by comparing predicted and experimentally obtained temperature distribution along the slab's cross-section at specified time intervals, and progression of temperature along the steel reinforcement during fire exposure.
4. Apply a uniformly distributed load to the structural (slab) model top surface to simulate the applied gravity loading during fire exposure. This will be the first load step in the structural run. Check the deflected shape response, mid-span deflection, and support reactions to verify the behavior and accuracy of the developed model due the applied static load and boundary conditions.
5. Apply the obtained nodal temperature loads from the thermal analysis (Step 2) at several specified time steps and sub-steps for the entire furnace exposure to the structural model to obtain the slab deformations and full field stress and strain response results for the entire fire exposure duration. Thus, the analysis was performed by incrementing the time in several steps from the initiation of fire till failure of the slab specimen. The time to reach failure is defined as the fire resistance of the slab.
6. Validate the accuracy of the developed structural FE model by comparing predicted and experimentally obtained mid-span deflection response results during fire exposure.

3. Description of the developed numerical model

A numerical finite element (FE) model is developed in this study using the finite element software, ANSYS (ANSYS – Release Version 14.5, 2013) to predict the fire resisting performance of RC slabs subjected to severe ASTM E1529 temperature exposure. The developed model is based on a simply supported slab specimen tested by Cooke (2001) in a previous investigation. The total length, span length, width, and thickness of the tested slab specimen are 4700, 4500, 930, and 150 mm, respectively. This slab was made of normal weight concrete using siliceous aggregates with a density and characteristic cube strength at room temperature of 2400 kg/m^3 and 30 MPa, respectively. The slab is reinforced with 10 steel deformed longitudinal bars (BS 4449 Type 2) having a diameter and yield strength of 8 mm and 460 MPa, respectively. The concrete cover from the slab's soffit to the longitudinal steel is 25 mm. The bottom surface of the slab was subjected to the ISO834 temperature curve in a furnace. The slab's soffit was exposed to the temperature curve over a length of 4000 mm. The slab was simultaneously subjected to a uniformly distributed live load of 1.5 kN/m^2, corresponding to a load ratio (ratio of the applied load to the flexural capacity of the slab at ambient temperature) of 50%. Another non-loaded slab specimen tested by Cooke (2001) and subjected to the severe (hydrocarbon) Norwegian Petroleum Directorate (NPD) temperature-time curve in a furnace was also modeled to further validate the accuracy of the developed FE model. Based on ACI 216.1-14 prescriptive approach, the design fire resisting rating of this slab is 90 min (Cooke, 2001). The developed FE model incorporates temperature dependent thermal and mechanical properties of concrete and steel and has the same geometrical configuration and boundary conditions as that of the tested slab specimen. A full description of the development of the FE models in terms of geometry, element types, material properties, loading and boundary conditions is provided in the following subsections.

3.1. Geometry and element types

Fig. 1 shows the developed FE model for the quarter slab specimen using ANSYS version 14.5 (ANSYS – Release Version 14.5, 2013). Quarter FE models are adequate to simulate the behavior of the slab, due to the symmetry in the geometry, materials, structural and fire loading, and boundary conditions of the tested specimen. The use of a quarter models in analyzing this slab leads to a significant reduction in the computational time and effort. Symmetry is simulated by restraining the motion perpendicular to the plane of symmetry using roller supports.

The element types used to model the concrete and steel reinforcement bars in the thermal model are SOLID70 and LINK33, respectively (ANSYS – Release Version 14.5, 2013). These elements have the capability of conducting heat throughout the slab's model, due to applied transient temperature that was initiated at the bottom surface of the slab. The 3D brick SOLID70 element has a total of eight nodes and is used to model the entire slab structure. Each node of the SOLID70 element has one degree of freedom (*dof*), namely temperature (ANSYS – Release Version 14.5, 2013). The 3D spar uniaxial thermal LINK33 element is defined by two nodes, each with a temperature *dof* as well (ANSYS – Release Version 14.5, 2013). The SOLID70 and LINK333 element types can be used in both steady-state or transient thermal analysis (ANSYS – Release Version 14.5, 2013).