Engineering Structures 132 (2017) 249–260

Contents lists available at ScienceDirect

Engineering Structures

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

Static and dynamic analyses of insulated concrete sandwich panels using a unified non-linear finite element model

Paul M. Hopkins [a], An Chen [b,*], Mostafa Yossef [b]

[a] TD&H Engineering, Media, PA 19063, USA
[b] Department of Civil, Construction and Environmental Engineering, Iowa State University, Ames, IA 50010, USA

ARTICLE INFO

Article history:
Received 3 November 2015
Revised 4 November 2016
Accepted 5 November 2016

Keywords:
Finite element analysis
Insulated concrete sandwich panel
Fiber-Reinforced Polymer shear connector
Bending behavior
Blast loading

ABSTRACT

Insulated concrete sandwich panels are comprised of two outer concrete wythes and an inner layer of foam insulation. They have been increasingly used because of their advantages of light weight and energy efficiency. Various shear connectors can be used to connect the two outer concrete wythes. More recently, Fiber-Reinforced Polymer (FRP) shear connectors have been used, which can eliminate thermal bridging and improve the thermal performance. Typical approaches to Finite Element (FE) analysis treat static and dynamic analyses separately. However, due to the flexibility of the FRP shear connectors and the cracking of the concrete in insulated concrete sandwich panels, a nonlinear static analysis model would often diverge early based on a preliminary FE study conducted by the authors. To address this issue, a nonlinear explicit dynamic FE model using ABAQUS© was developed, which can study both the panels' static behavior under typical flexural loading and dynamic behavior under blast loading. Nonlinear material properties were incorporated and damaged plasticity model was used to model concrete in both compression and tension. In order to simulate the static behavior, the time loading control was applied to the FE model to slow down the rate of loading to smoothly capture the response. For dynamic analysis under blast loading, a verification study was conducted first using the developed FE model, where good correlations can be obtained between the FE and test results on a panel tested in a previous study. The FE model was further used to study the dynamic behavior of two panels under blast loading: one is a solid concrete panel and the other is an insulated concrete sandwich panel. It can be concluded that, although the insulated concrete sandwich panel is lighter, it still performs relatively well compared to the solid panel under blast loading. Therefore, it is promising to use insulated concrete sandwich panels for both conventional and blast-resistant structures.

1. Introduction

As shown in Fig. 1, insulated concrete sandwich panels are comprised of two layers of concrete, known as wythes, separated by a layer of rigid foam plastic insulation. The two wythes are connected by some form of shear transferring mechanism, generally using solid concrete web zones, metal connectors, plastic connectors, or a combination of these elements. The panels can provide dual function of transferring load and insulating the structure among other desirable characteristics of normal concrete panels, such as durability, low-cost, fire resistance, etc. [19–25] Therefore, they have been increasingly used in building industry.

More recently, Fiber-Reinforced Polymer (FRP) shear connectors have been used, which can eliminate thermal bridging and improve the thermal performance. Various types of FRP connectors have been studied, including C-grid by Frankl et al. [1], G-grid by Soriano and Rizkalla [2], and Glass FRP bar connector by Woltman et al. [3]. More recently, the authors have developed an innovative FRP plate shear connector based on comprehensive bending tests and linear finite element analysis (FEA) [4,5]. Four types of panels were built and tested which included sandwich panels with discrete FRP shear connectors, segmental FRP shear connectors (Fig. 1), and continuous FRP shear connectors, and solid reinforced panels (Fig. 2). The dimensions of the segmental FRP shear connectors are shown in Fig. 3 and a photo of the assembly of the insulation with the segmental FRP connector is shown in Fig. 4. All the above studies showed that the sandwich panels performed well compared to solid reinforced concrete panels and can be used as structural members.

* Corresponding author.
E-mail addresses: paul.hopkins@tdhengineering.com (P.M. Hopkins), achen@iastate.edu (A. Chen).

http://dx.doi.org/10.1016/j.engstruct.2016.11.017
0141-0296/

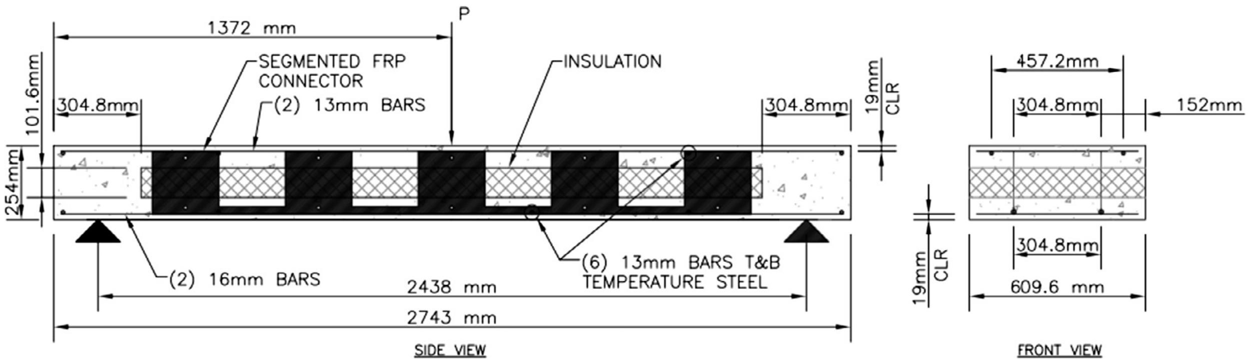

Fig. 1. Insulated concrete sandwich panel.

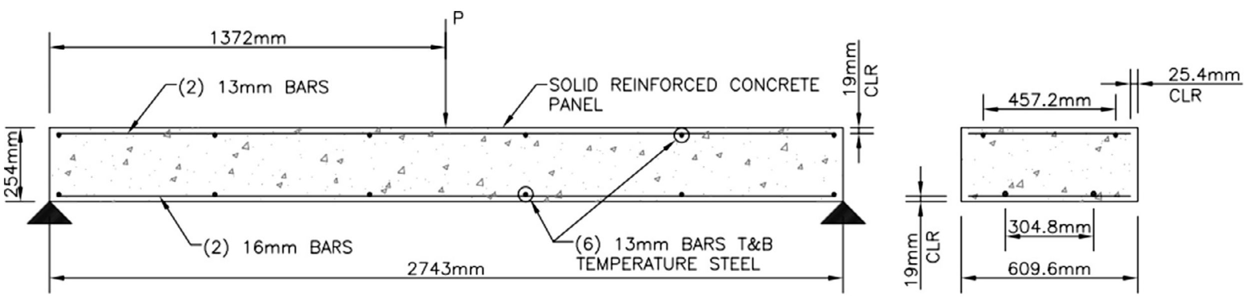

Fig. 2. Solid reinforced concrete panel.

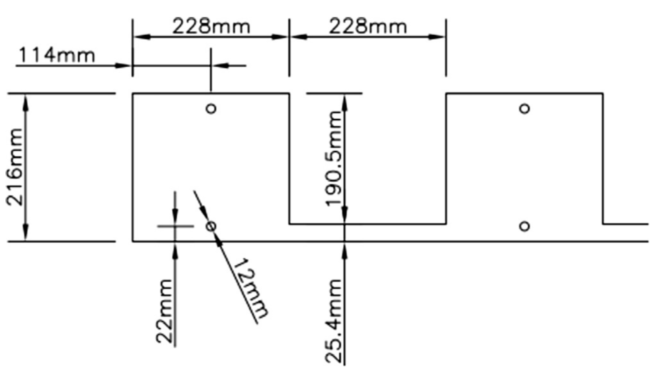

Fig. 3. Segmental FRP shear connector.

Fig. 4. Assembly with segmental FRP connector.

Constructing and testing panels, although a necessity to prove and verify strength and durability, can be expensive. Therefore, a reliable FEA model is needed so that parametric studies and advanced analyses can be performed, such as the ones initiated in the research [18], which is the objective of this study. ABAQUS© [6] is used for this purpose. However, there are many types of solvers and elements to choose from. This paper provides a summary on the nonlinear analysis approaches which can be taken to obtain a solution to the insulated concrete sandwich panel modeling, based on which an explicit dynamic model is proposed.

The United States Army Corps of Engineers have years of published research on the topic of blast loading and structural hardening [14,15]. Many typical structural building materials, such as masonry, reinforced concrete and steel were labelled as preferred and tested materials to be used in buildings designed for blast resistance. Precast concrete materials, specifically sandwich panels, were mentioned in these publications. But they were mainly used as envelope or cladding type materials, which are secondary structures. Limited studies are available on the dynamic behavior of insulated concrete sandwich panels under blast loading, which will also be studied in this paper.

2. Finite element analysis

The goal of this paper is to develop a reliable FEA solver that can predict the strength and deflection of the insulated concrete sandwich panels. Simplifying the model into shell elements or utilizing other simplistic forms was eliminated so that we can capture cracking patterns and full failure mechanisms. ABAQUS offers a