5th National Congress on Civil Engineering,
May 4-6, 2010, Ferdowsi University of Mashhad, Mashhad, Iran

Parametric Studies on the FEM Analysis of Foam Core Sandwich Panels with Holes

Ali Vakilazad Sarabi, Masood Mofid

1-Post Graduate Student, Civil Engineering Department, Sharif University of Technology, Tehran, Iran

2- Associate Professor, Civil Engineering Department, Sharif University of Technology, Tehran, Iran

Alivakilazad@gmail.com
Mofid@sharif.edu

Abstract

The aim of the work presented in this research is deal with some of the aspects in the FEM analysis of sandwich panels containing holes which comprised with foam core. In this research, the FEM modeling was produced, analyzed and computed considering laboratory conditions. An extensive parametric study was investigated under different load conditions; different geometrical parameters, such as; dimensions, face thickness, core thickness, size and location of the opening.

Keywords: Sandwich structure, Finite element method analysis, Foam core, Opening.

1. INTRODUCTION

A typical sandwich panel has a three-layer structure. The rigid faces with a relatively high modulus of elasticity are kept apart by the much lighter core which has shear stiffness sufficient to carry most of shear force. The core also acts as a highly effective thermal insulation layer. Variations of this pattern also exist. In multi-layer panels, the faces consist of more than one component and the purpose of the extra layers is usually to improve the performance in fire.

Very light closed-cell polymeric (Divinycell) foam is a favored material for the core of sandwich structures. In classical works (e.g., Gibson and Ashby [1], Shipsha et al. [2], the failure of this material in tension or compression has been described as ductile, failure criteria expressed in terms of stresses have been used. Such failure can exhibit no size effect.

Such ductile response, however, does not take place when high tensile stress concentrations exist, induced for example by notches in laboratory specimens, or various structural holes or accidental damage of real structures. In such a case, the failure of the foam may be brittle. This was first revealed by the notched specimen tests of Zenkert [3] in which the fracture toughness of foam was measured, by the tests of holed panels tests by Fleck and co-workers in Cambridge, and by the finite element studies of foam based on microplane model for foam by Brocca et al. Consequently, the foam, when notched or damaged, must be expected to exhibit size effect, and this is what is confirmed by a recent study (Bazant, Zhou, Novak and Daniel), in which the size effect in Divinycell H100 foam noticed[4].

2. MODELING THE BEHAVIOR OF SANDWICH PANELS

The purpose of the finite element analysis was to predict the behavior of foam core sandwich composites by taking into account both physical and geometrical non-linearity.

The finite element model of an indented sandwich beam was based on the geometry and dimensions of the specimens as used in the indentation tests. The numerical simulations were carried out by using a 3D computational model. The length of the model was reduced to 250 mm. In this way, the number of the elements and the computational time was decreased. The modeling was performed using a finite element computer code. The test specimens were supported by a rigid substrate. Thus, all degrees of freedom were constrained at the lower boundary of the model. The nodes on the vertical axis of symmetry were restrained in horizontal direction.