Final Report of

Finite Element Modeling for Composite Structural Insulated Panels (CSIPs) under Simulated Wind Pressure

Li Dong

Department of Civil Engineering

12/15/2011

Final Report of

Finite Element Modeling for Composite Structural Insulated Panels (CSIPs) under Simulated Wind Pressure

Abstract

This paper presents a finite element (FE) modeling for an innovative Composite Structural Insulated Panels (CSIP) under simulated dynamic wind pressure. Load history was obtained from the High Airflow Pressure Loading Actuator (HAPLA) which generated dynamic wind pressure. CSIP is a sandwich panel with E-Glass/polypropylene facesheets and expanded polystyrene (EPS) foam core. In the FE modeling, the deflection of a panel with density of 1 pcf was validated. Then parametric study was conducted regarding to core thickness.

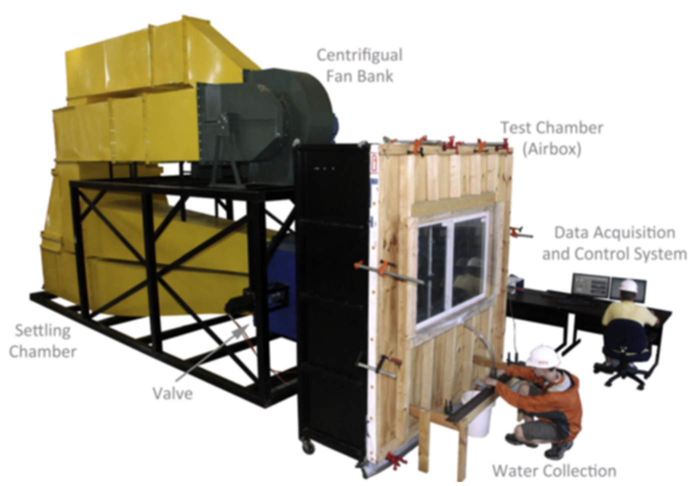
Introduction

Atlantic hurricanes have caused more than $113 billion dollars in insured losses during last two decades [1]. Through post-storm investigations, FEMA [2, 3], NIST [4], and others [5-7] have found that building envelope failures are the primary reason of collapses. Every wind disaster proves that traditional houses are too fragile to withstand the split-second high air pressure as well as the aerial debris.

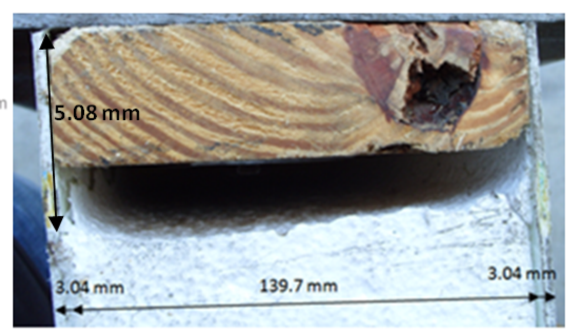
The proposed Composite Structural Insulated Panels (CSIPs) are made of low-cost thermoplastic orthotropic glass/poly-propylene (glass-PP) laminate as a facesheet and Expanded Polystyrene (EPS) foam as a core with very high facesheet/core moduli ratio. The innovative CSIPs system shows excellent wind-resistance performance for structural wall applications [8]. This report first introduces the in-field experiment briefly, and then presents the finite element modeling methodology with LS-DYNA in details.

Experiment

Dynamic pressure loading was generated by the High Airflow Pressure Loading Actuator (HAPLA) system shown in Fig. 1 a, which is developed by University of Florida. The HAPLA consists of two 75 HP Centrifugal Backward Inclined Class IV SWSI that may be configured to operate in series or parallel. The ducting connects to an air valve with five ports: supply (from fan), return (to fan), exhaust (open atmosphere), intake (open atmosphere) and service (connects to the test chamber). The valve simultaneously modulates the amount of air traveling from (to) the test chamber to the exhaust (intake) port. The difference between the exhaust rate and intake rate is equivalent to the air leakage through the specimen. The valve disk is connected to a rotary actuation system comprised of a Metronix ARS2340 servo positioning controller, a Sumimoto Drive Technologies CHF 6135 Y-11 11:1 drive reduction and a Metronix SBL-T6-2900 brushless servo motor. Positioning feedback is provided by a two-pole resolver in the motor. Custom Labview 8.5 software running on a National Instruments PXI system controls the pressure in the chamber through a 50 Hz PID controller that receives feedback from a pressure transducer attached to the test chamber or “airbox,” which is a stiff 2.4 m X 2.4 m X 0.3 m (8 ft X 8 ft X 1 ft) hollow steel frame box lined with 14 gauge galvanized sheet steel on five sides. The sixth side accommodates the test specimen (Fig. 1: c, d) [9]. A customized steel frame, two wood frames, two straps, and several C-clamps were used to hold the CSIP in place during experiments. A laser deflection device was used to measure the mid-span deflection of the panel. Six strain gauges were installed at quarter span points of exterior and interior side.

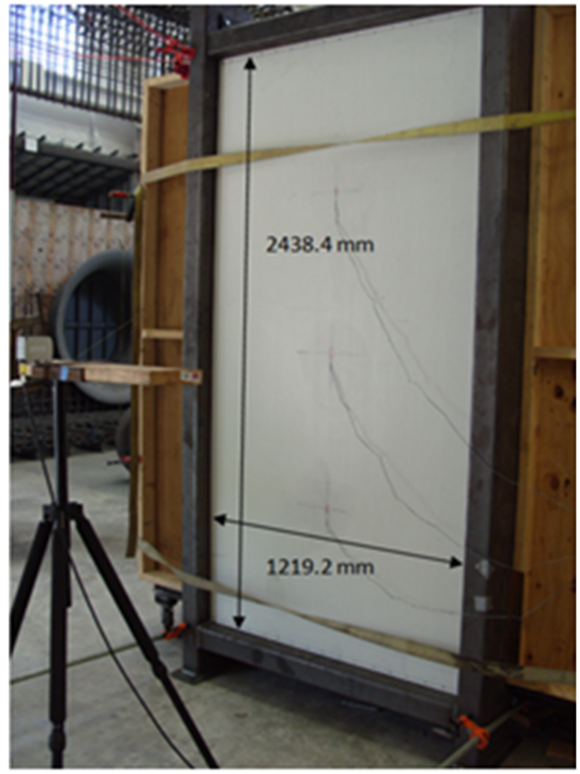


a High Airflow Pressure Loading Actuator (HAPLA) [9]



b Thickness and connection details

c Interior of Airbox



d Geometry of CSIPs

Figure 1 Configuration of the experiment

In order to evaluate the wind-resistance capacity of the CSIPs, panels with dimensions of 2.4 m X 1.2 m X 0.14 m (8 ft X 4 ft X 5.5 in) with densities of 1 pcf (16 kg/m3) were tested. The recorded dynamic simulated wind pressure was as high as 7.25 kPa, which was about 280 miles per hour of wind speed. The connection of the panel used a 2 in X 6 in (actually size 1.5 in X 5.5 in, 38 mm X 152 mm) southern pine lumber. CSIPs wall system failed at very high air pressures. All face sheets and core were found to be structurally intact. Connection failure was observed, and minor cracks were found in the connections in dynamic tests.

Simulation Details

Through the finite element modeling, material models, explicit and implicit methods, and element numbers were investigated in order to find the model which generates the best matched deflection – time curve. LOAD_SEGMENT_SET and SET_SEGMENT were used to produce the time-changing pressure, which varies every 0.01 second. The load history can be referred to Figure 2. For simplicity, the wood connection was not considered in the modeling, and the edge nodes were constrained for all six degree of freedom, which can be seen in Figure 3. The material properties used in the model can be referred to Table 1.Due to the long lasting time of the experiment and the computation characteristics of LS-DYNA, only the data of first second was used for the validation. After the modeling validation, parametric study regarding to the core thickness was conducted.

Table 1 Material Properties

Face sheet (glass/PP)										Core (EPS foam)		
RHO	EX	EY	EZ	PRXY	PRYZ	PRXZ	GXY	GYZ	GXZ	RHO	EX	PRXY
16 kg/m^3	11035 MPa	11035 MPa	1030 MPa	0.11	0.22	0.22	1800 MPa	750 MPa	750 MPa	1783 kg/m^3	1.2 - 1.5 MPa	0.25

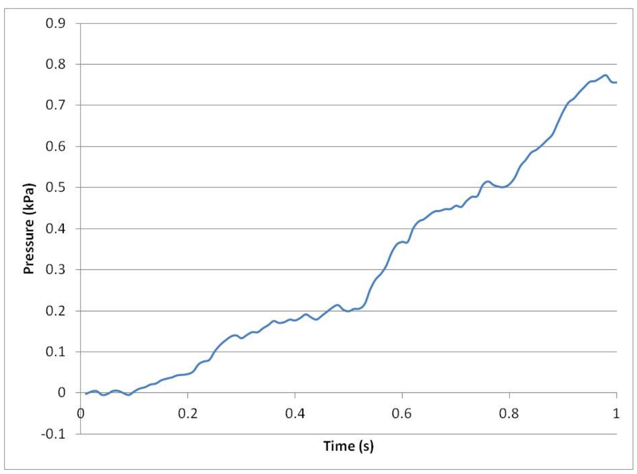


Figure 2 Load history: Pressure - Time

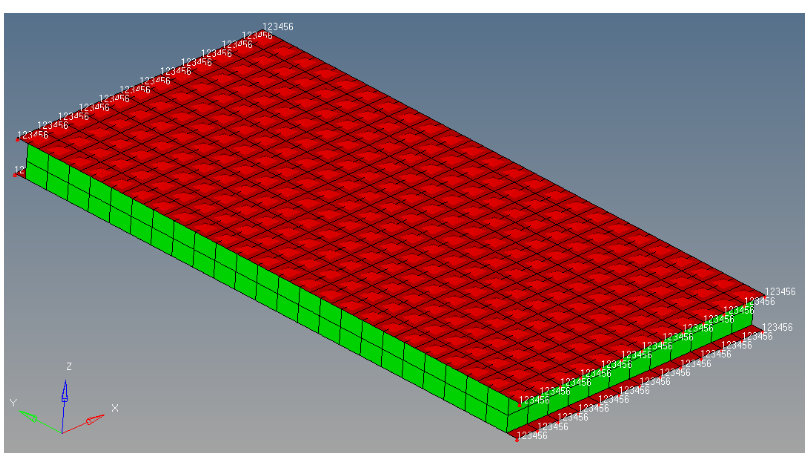


Figure 3 Geometry

Material Models

MAT161 and MAT54 were tried for the 3.04 mm glass fiber / PP face sheet, while MAT003, MAT012, and MAT063 were tried for the EPS foam core. It is noted that Figure 4 was used in the Stress – Strain curve for MAT012. The default material models for core and face sheet are MAT012 and MAT054, which are determined to be the optimum types after several trials. The results can be referred to Figure 5.

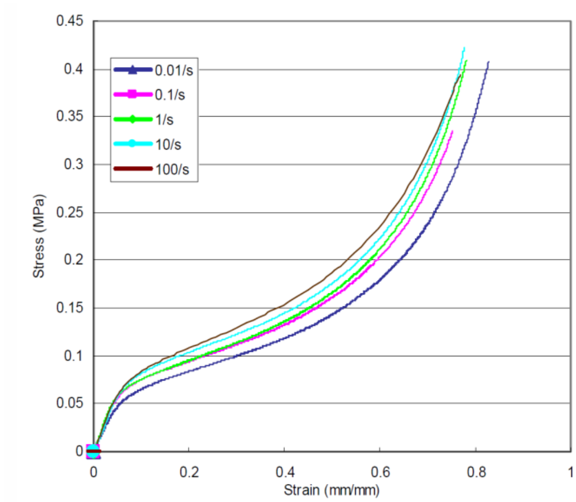


Figure 4 Rate dependent data on crushable EPS foam [11]

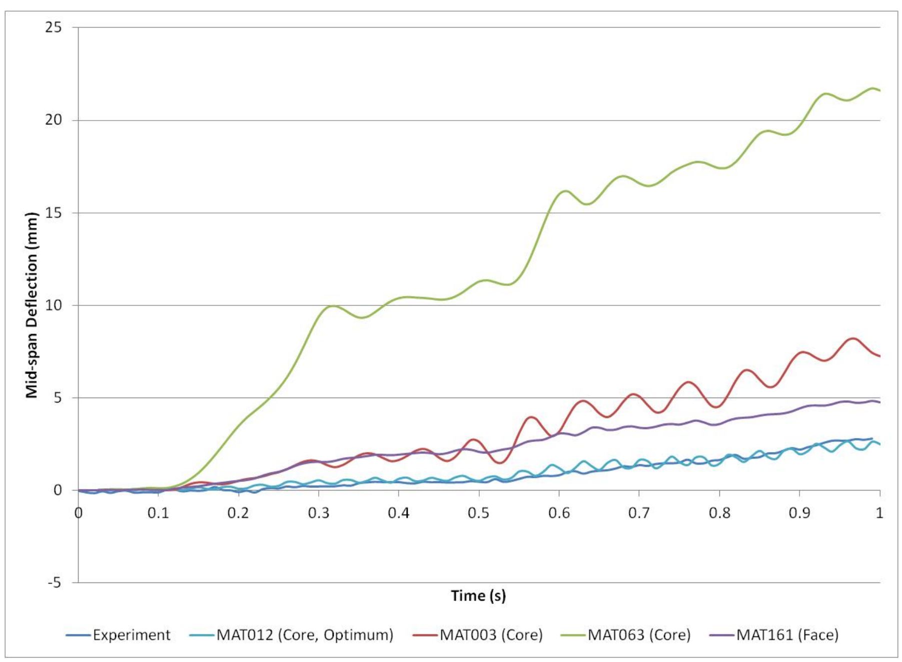


Figure 5 Deflection - Time curve of different material trials

Shell element was used instead of solid element considering the fact that the element with aspect ratio closer to unity yields better results [10] as well as the CPU cost. Although MAT161 was always considered as a perfect fit for fiber polymer materials, it did not show enough realistic behavior in this modeling. The reason may be MAT161 is better at modeling fiber failure but the relatively low pressure did not give any damage to this model. Also it is noted that MAT161 is not applicable for 4-node shell element. Due to the 3.04 mm thin face sheet and the large dimension of the model, the CPU cost would be significant if the element aspect ratio suffice the unity requirement.

Explicit and Implicit Solver

The terms implicit and explicit refer to time integration algorithms. For explicit analysis, the maximum time step size is limited by the Courant condition, and it is well suited to dynamic simulations while it can become prohibitively expensive to conduct long duration or static analyses. For implicit analysis, the time step size may be selected by the user while large numerical effort are required to form, store, and factorize the stiffness matrix [12]. CONTROL_IMPLICIT_GENERAL was used to activate the implicit solver. The comparison among experiment, explicit, and implicit can be seen in Figure 6. It can be seen the result does not vary quite much for the two methods. However, the CPU cost for implicit calculation was much larger than that for explicit method. Thus explicit was chosen for this modeling.

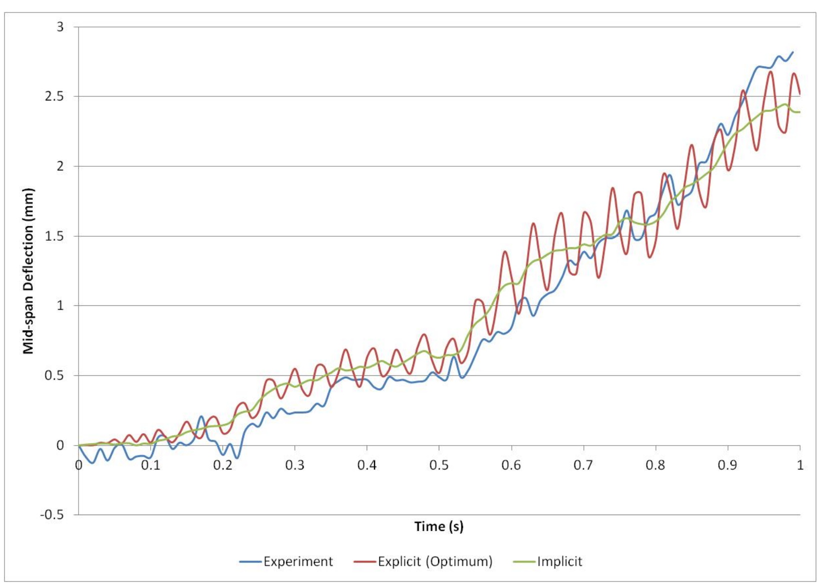


Figure 6 Deflection – Time curve of Explicit and Implicit methods

For the implicit solver, the default iterative nonlinear solver used is the BFGS method that employs a 'Quasi-Newton' method in which the global stiffness matrix is reformed only every ILIMIT steps. This default method often fails to converge when the non-linearity of the problem grows or when significant amount of contact is involved [13]. It is clear that this specific modeling has significant amount of contact between face sheet and core. In this case, as Suri Bala [13] recommends, a more expensive Full-Newton method is used by setting ILIMIT = 1 to force LS-DYNA to reform the stiffness matrix at every iterative step to yield a more accurate stiffness estimation. Also the MAXREF is set to 50 to allow sufficient number of stiffness reformations before terminating or reduction of solution time step.

Element Numbers

In order to balance the relationship between CPU cost and result consistency, different amount of element numbers were investigated for this model. With different mesh sizes, 32 elements, 96 elements, 1128 elements, and 64843 elements were tried. It can be seen from Figure 7 that the model with 64843 elements differed not much from the model with 1128 elements, and both of them are in good consistent with the experiment curve. From the curve, it is safe to use the model with 1128 elements.

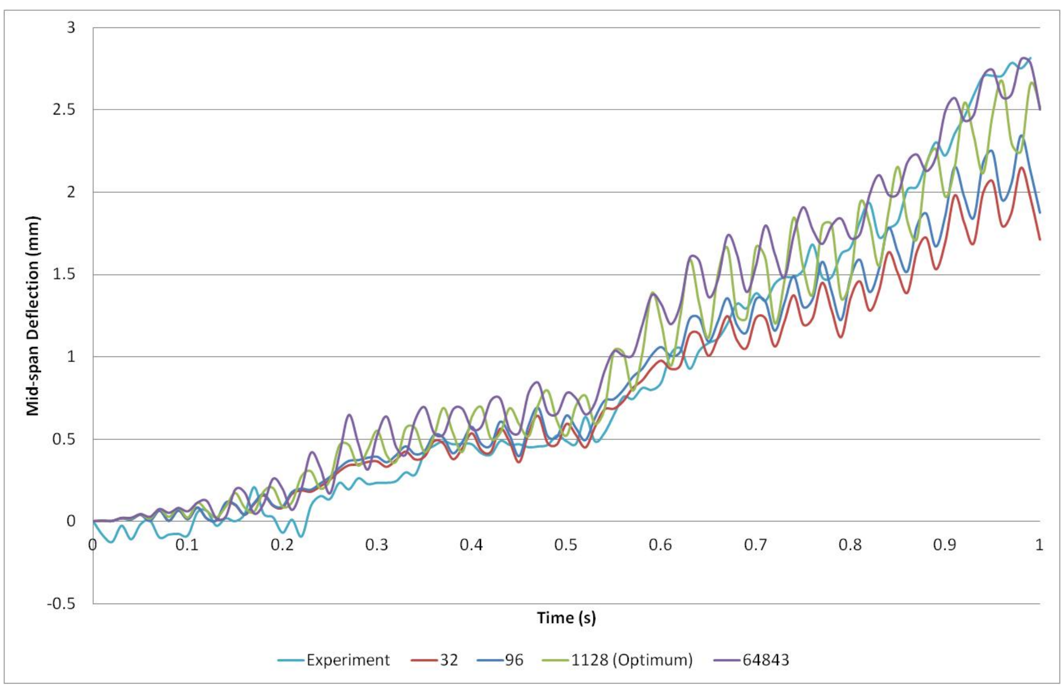


Figure 7 Deflection – Time curve of different element numbers

Although the model has quite a large dimension of 2.4 m X 1.2 m X 0.14 m (8 ft X 4 ft X 5.5 in), the result is consistent enough with mesh size of 100 mm and element number of 1128. The reason why this model is not very sensitive to the element size may be the simple geometry and load pattern.

Results

As all the above validation modeling shows, a finite model with MAT054 for face sheet, MAT012 for core, CONTACT_TIED_SURFACE_TO_SURFACE between face sheet and core, mesh size of 100 mm with element number of 1128 were proven to be very consistent with the experiment using explicit solver. It can be seen from Figure 8 that the variance is within acceptable range. The deflection and stress contours can be referred to Figure 9.

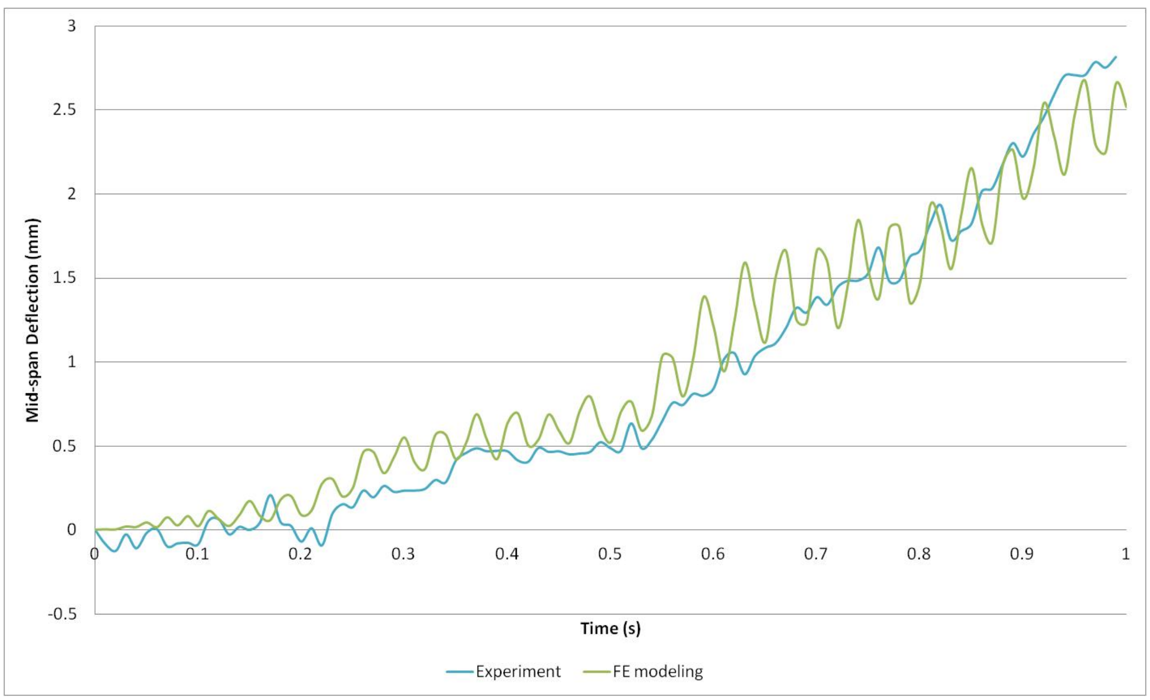
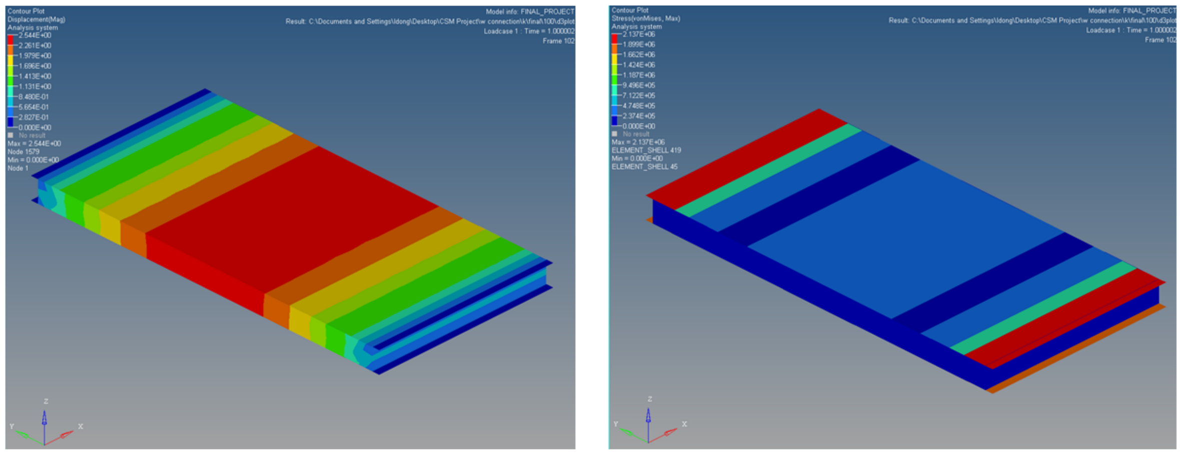


Figure 8 Comparison between experiment and FE modeling



a Deflection contour

b vonMises stress contour

Figure 9 Model contours

Parametric study

A parametric study regarding to core thickness was conducted to see the influence that the EPS core will do to the sandwich structure system. Besides the original thickness of 5.5 in, 4.5 in, 7.25 in, and 9.25 in were selected because they are dimensions of standard lumbers. As the core thickness increases, the four plots in Figure 10 show decreasing mid-span deflection as expected.

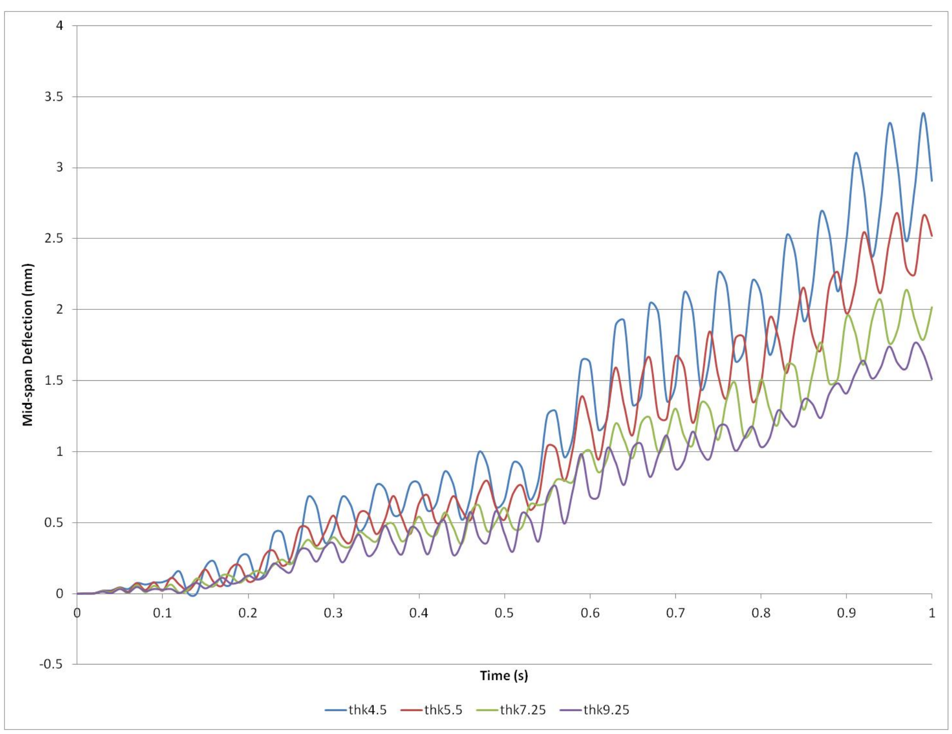


Figure 10 Deflection – Time curve of different core thickness

Summary

This report presents the finite element validation and parametric study of the hurricane–resistant sandwich structure system, the composite structural insulated panel which is made of glass/polypropylene face sheet and expanded polystyrene core. The simplified model was created without wood connections and bolts. The air pressure inside the air chamber was simulated as distributed pressure in LS-DYNA. After trials of different material types of MAT003, MAT012, MAT063 for core and MAT054 and MAT161 for face sheet, different solvers of explicit and implicit, and different amount of element numbers of 32, 96, 1128, and 64843, finally a model with MAT054 for face sheet, MAT012 for core, CONTACT_TIED_SURFACE_TO_SURFACE between face sheet and core, mesh size of 100 mm with element number of 1128 were found to be in good agreement with the experiment using explicit solver. Using the validated model, a parametric study was conducted regarding to the core thickness. A decreasing mid-span deflection was found as the core thickness increases.

References

1. Insurance Information Institute. Available at, www.iii.org/disasters/Hurricane.
2. Federal Emergency Management Agency. FEMA 490 Summary report on building performance, 2004 Hurricane season. Washington, DC: Federal Emergency Management Agency; 2005.
3. Federal Emergency Management Agency. FEMA 549, Summary report on building performance: Hurricane Katrina 2005. Washington, DC: Federal Emergency Management Agency; 2006.
4. National Institute of Standards and Technology. Performance of physical Structures in Hurricane Katrina and Hurricane Rita: a Reconnaissance Report; 2005. NIST Technical Note 1476.
5. Gurley K, Masters F. Post 2004 hurricane field survey of residential building performance. Natural Hazards Review, in press.
6. Mehta KC, Minor JE, Reinhold TA. Wind speed-damage correlation in Hurricane Frederic. Journal of Structural Engineering 1983;1:37-49.
7. Guillermo F, Green R, Khazai B, Smyth A, Deodatis G. Field damage survey of New Orleans homes in the aftermath of Hurricane Katrina. Natural Hazards Review 2010;11:7-18.
8. Mousa M. Composite Structural Insulated Panels (CSIPs) for Hazards Resistant Structures. University of Alabama at Birmingham, Ph.D. Dissertation, 2011.
9. Lopez C, Masters F.J, Bolton S. Water Penetration Resistance of Residential Window and Wall Systems Subjected to Steady and Unsteady Wind Loading. Journal of Building and Environment 2011; 46: 1329-1342.

10. Bhavikati S.S. (2005), Finite Element Analysis. New Age International (P) Ltd., Publishers, New Delhi, India.
11. Croop B, Lobo H. Selecting Material Models for the Simulation of Foams in LS-DYNA. 7th European LS-DYNA Conference Proceedings 2009.
12. LS-DYNA Version 971 Keyword User's Manual, 2006.
13. http://blog2.d3view.com/ Website visited on Dec 15, 2011.