Analysis of an Automobile Roof Panel under Strongly Coupled Fluid Structure Interaction using LS-DYNA

Yong-Bae Cho, Duane Detwiler[1]

Kevin A. Gardner, Jeremy D. Seidt, Amos Gilat[2]

[1]Honda R&D Americas, Inc., Raymond, OH, USA

[2] The Ohio State University, Department of Mechanical and Aerospace Engineering, OH, USA

1 Extended Abstract

Accurately and efficiently simulating strongly coupled fluid structure interactions has been a challenge. In this presented study, the interaction of a lab-scale experiment using an automotive roof panel with an air flow applied through a pipe from a compressed air tank was analyzed using LS-DYNA multi-physics solver. The purpose of the analysis was the evaluation of the accuracy of deformation, strain, and the pressure distribution on the panel due to the fluid structure interaction. The oil-canning phenomenon is a complex response that is governed by many different factors including the geometry of the sample, the sample material, the forming history of the sample, the boundary conditions, and the ratio of the jet area to the sample area. In order to replicate real word situations it was decided to design a test rig capable of testing full scale automotive roof panels. The steel roof panel was modeled with material strain rate properties and the post stamping simulation results applied to take into account the thinning and work hardening effects.

The simulated pressure distribution on the panel is shown in Fig. 1. The maximum pressure on the roof surface occurred at the flow center on the panel and was predicted to be 15.84 kPa. There is a region of negative pressure on the roof panel. After the air flow comes out of the pipe, it impacts the roof panel and causes a bowl shaped (oil canning) deformation in the roof panel, and in turn the air flows along the deformed surface. When the flow comes out of the bowl, it draws up the air nearby as shown in Fig. 3, velocity fringe pattern, and negative pressure occurs, which an uncoupled CFD/structural analysis does not predict.

The center cross-section pressure distributions from the pipe outlet to roof panel are shown in Fig. 2, and Fig. 3 describes the velocity distribution at the same location. Along the panel surface the velocity was very low. The directions of air flow changes from perpendicular direction, to the tangential direction of the panel, then the maximum velocity is reached. The maximum speed of the flow in the tangential direction to the panel was about 176.3 m/sec, which was very similar to the 173.9 m/sec measured in the test at the end of the pipe. From Fig. 2 & 3, it is apparent that the velocity decreases and the pressure increases simultaneously as the flow approaches to the panel like Bernoulli's principle states.

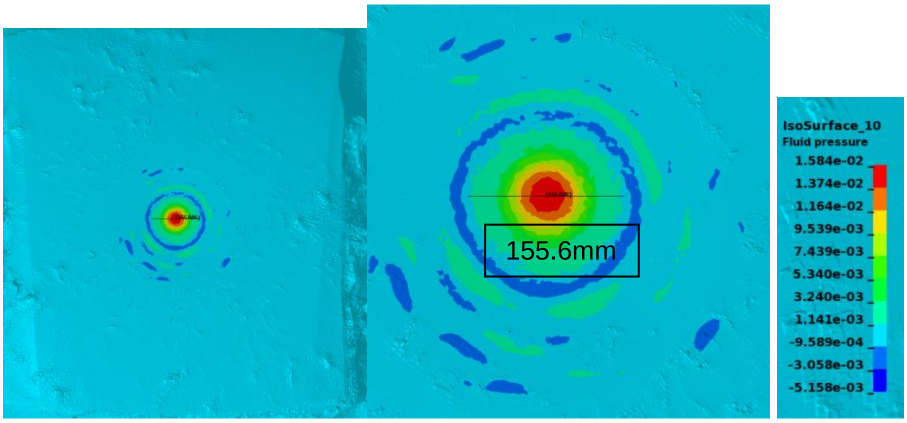

Fig.1: The pressure distribution on the surface of the roof

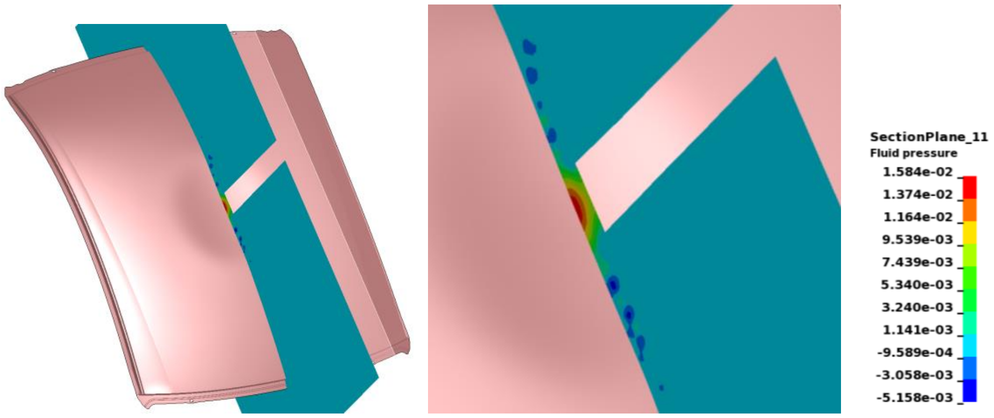

Fig.2: The pressure distributions of the cross section at the center

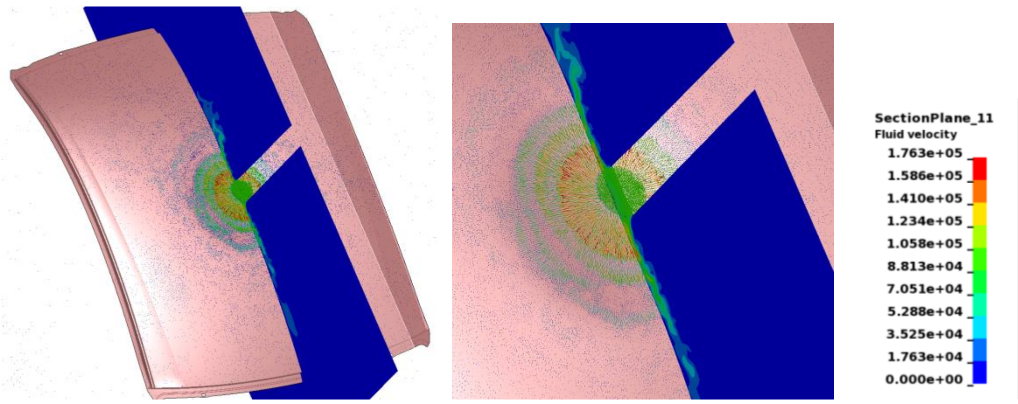

Fig.3: The velocity distributions on the roof surface and velocity vector plot of the cross section at the center

The predicted panel force versus displacement is shown In Fig. 4. The maximum force was 43 N, and the force fluctuates even with the applied damping properties.

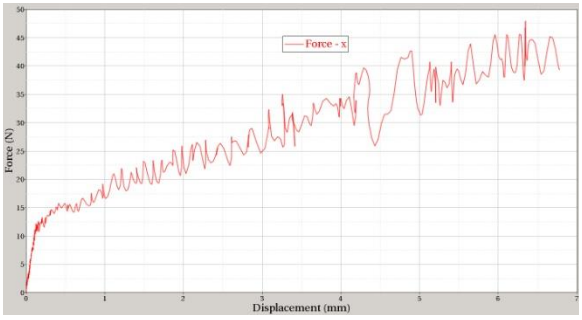

Fig.4: Force vs. Displacement

Further investigation is needed regarding the best method to apply a measured fluid velocity profile to the model's fluid inlet, and the proper damping effects for the loaded steel panel. Strongly coupled fluid structure analysis enables us to predict the deformation of a structure due to fluid flow, which in turn alters the flow pattern and pressure due to the structure's deformation. This predictive capability allows us to design structures more optimally for real world loading conditions.

2 Literature

[1] Pin, F.D. "Advances on the Incompressible CFD Solver in LS-DYNA", *11th International LS-DYNA Users Conference, 2011*

[2] Facundo Del Pin, Inaki Caldichoury, Rodrigo R. Paz, ICFD Theory Manual, Incompressible Fluid solver in LS-DYNA, 2014