Abaqus/Explicit Simulation of the Low Velocity Impact of an Aluminum Honeycomb Sandwich Panel

Abaqus Technology Brief

Summary

Honeycomb core sandwich plates and shells possess significant advantages over monocoque construction and are widely used in the aerospace, naval, railway, and building industries. The complexity of a honeycomb cell structure presents modelling challenges. A plug-in extension for Abaqus/CAE permits one to create honeycomb structure geometry quickly and accurately. The geometry can be meshed with conventional shell elements and used directly, or it can be utilized to determine the mechanical properties of a specific honeycomb structure through virtual testing, replacing laborious and expensive physical testing. The properties obtained can be subsequently used to discretize the sandwich core with continuum solid elements.

In this Technology Brief, we describe an approach for modelling impact on honeycomb sandwich structures using the new plug-in and Abaqus/Explicit. Very good agreement between the numerical results and experimental data is demonstrated.

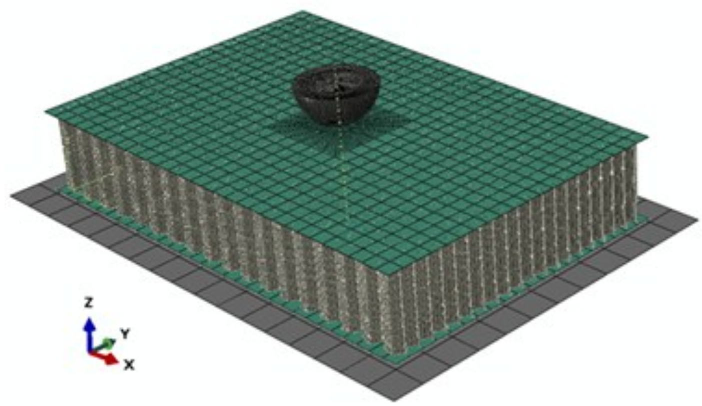

Figure 1: Abaqus/CAE model

Key Abaqus Features and Benefits

- Abaqus/CAE plug-in to automatically build honeycomb structure models
- Abaqus/Explicit for robust solution of complex, highly nonlinear models including contact, collapse, buckling, and plasticity

Background

The significant advantages of sandwich construction over monocoque construction are widely exploited in the aerospace, naval, railway, and building industries [1]. Twin-skinned sandwich plates and shells with a honeycomb core are especially attractive for aerospace applications: for the price of small additional weight compared to monocoque construction, very high specific strength and stiffness are obtained. Moreover, the topology and materials of a sandwich structure, including the core geometry, can be optimized for a given application such as an aircraft fuselage, engine cowling, or impact protection shield.

A successful numerical simulation of honeycomb sandwich structures depends heavily on the proper modelling of the core. One approach is the use of continuum solid elements to represent the core material. This, however, requires physical or virtual [2] testing to determine the mechanical properties for a specific core type. Another approach is to explicitly model the cellular structure of the core using conventional shell elements. To create complicated honeycomb geometry quickly and accurately, a dedicated Abaqus/CAE plug-in has been developed. The cellular geometry can be subjected to virtual testing (such as compression, tension, and shear tests) in order to determine the mechanical properties of a given core type. The properties obtained can be subsequently used to discretize the sandwich core with continuum solid elements. Alternatively, the honeycomb geometry can be meshed and used directly in the analysis.

The present Abaqus/Explicit simulation examines the low velocity impact response of a sandwich specimen with aluminum face sheets and an aluminum honeycomb core. The geometry and material properties of the specimen were taken from [3] and [4]. The in-plane dimensions of the specimen are 150 by 100 mm. The face sheets are made of 2024-T8 aluminum alloy. The thickness of the top and bottom face sheets is 0.508 mm and 0.813 mm, respectively. The hexagonal aluminum honeycomb core is made of untreated 5056 aluminum alloy. The thickness of the core is 25.4 mm, its cell size is 4.7625 mm (3/16"), and the foil gauge is 0.0508 mm (0.002"). The aluminum alloys used in the face sheets and core are assumed to behave as nonlinear, isotropic, elasto-plastic materials with the properties given in Table 1. A 25 mm diameter hemispherical impactor is used with an energy of 15 J.

Finite Element Analysis

The model was built in Abaqus/CAE (Fig. 1) using the honeycomb geometry plug-in. The core geometry (Fig. 2) was meshed and used directly. Tie constraints were used to attach the face sheets to the core.

The sandwich specimen was meshed using S4R elements. According to [3] the actual contact diameter between the impactor and the top face sheet was 3 mm. Therefore, to accurately model the deformations at the impact region, the 3 by 3 mm area at the center of the top face sheet was discretized using a small element size of 0.5 mm. Beyond this area, the element size was gradually increased up to a value of 5 mm. Similarly, the area of roughly 30 by 30 mm in the center of the honeycomb core was meshed using an element size of approximately

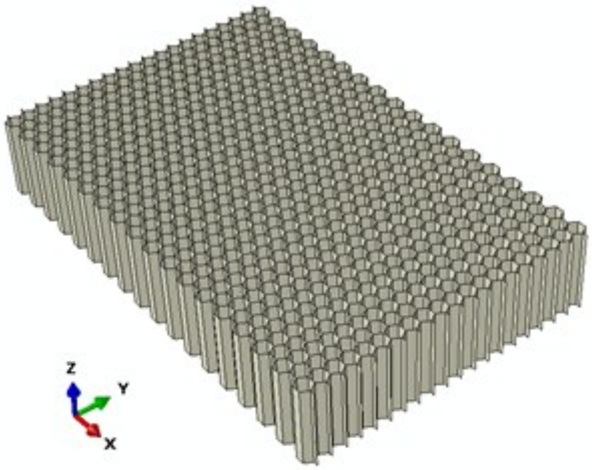

Figure 2: Honeycomb geometry

0.5 mm, and the rest of the core was discretized with rectangular elements with a size of approximately 1.4 by 0.5 mm. In total, 242,964 conventional shell elements were used to model the core. Such a fine mesh was necessary to properly capture the crushing response of the honeycomb.

The impactor was discretized with C3D8R elements. The mesh was gradually refined toward the impact area, where the element size was approximately 0.25 mm, so that several impactor nodes fell within the perimeter of each element located in the center of the top face sheet. This was done to accurately simulate the pressure stresses in the impact region. The resulting finite element model had approximately 1.5 million degrees of freedom.

Constitutive model

The constitutive response of the aluminium alloys was described with an isotropic elasto-plastic material model (Table 1). The values of the plastic stress, σ_{pl}, and the plastic strain, ε_{pl}, were calculated from the following expression [5] based on the data in Table 1:

$$\sigma_{pl} = \sigma_y + \frac{E_X E_T}{E_X - E_T} \varepsilon_{pl} \quad (1)$$

	Face sheets	Core
Young's modulus, E_x [GPa]	72.4	69.6
Tangent modulus, E_T [GPa]	50.7	2.5
Yield stress, σ_y [MPa]	393	65.5
Poisson's ratio, ν	0.33	0.33
Density, ρ [kg/m^3]	2768	2685

Table 1: Material properties of the sandwich constituents [3]

Namely, the stress at zero plastic strain equals σ_y. The values of the plastic strain defining isotropic hardening for the face sheets and the core were calculated from Equation (1) by prescribing the corresponding values of the ultimate tensile strength for the face sheets and the core. These values were taken as 478 MPa [6] and 82.4 MPa, respectively.

Boundary conditions

The sandwich specimen was positioned on a rigid base plate. Friction between the bottom face sheet and the base plate was neglected because it did not influence the response. Dry sliding friction was assumed between the impactor and the top face sheet, with a friction coefficient equal to 0.45, corresponding to the friction between steel and aluminum.

Initial conditions

Reference [3] provided the value of the impact energy but did not specify the impactor velocity, drop height, or mass. Instead, the maximum and minimum values of the impactor mass (between 0.5 kg and 2.5 kg) and the maximum drop height (4 m) that were allowed by the testing equipment were noted. Because the impactor mass and its velocity at the moment of impact can be immediately determined given the drop height and the impact energy (15 J), several impact analyses corresponding to different drop height choices, starting with the maximum value of 4 m, were performed to match the predicted impulse with the measured one. The results reported here are given for an impactor mass of 805 g and an impact velocity of 6 m/s.

Results

Close agreement between the experimental [3] and simulated force-time histories is obtained, as shown in Figure 3. The difference between the measured and calculated peak impact forces is 1.4%. It can be seen from this figure that both curves exhibit repeated reductions and increases in their slopes that reflect the crushing process in the honeycomb core. It is important to note that the simulation closely follows not only the ascending part of the experimental force-time history but also its fairly steep unloading portion. The nature of the crushing process of the honeycomb structure is revealed in Figure 4. Figures 3 and 4 suggest that proper mesh refinement, especially in the core, is vital for accurate simulation.

Conclusions

An Abaqus/Explicit simulation of the low-velocity impact of a honeycomb sandwich specimen was conducted. A new Abaqus/CAE plug-in for the automatic definition of honeycomb cellular structures was used to create the honeycomb geometry that was directly meshed and used in the analysis. The analysis results showed excellent agreement with experimental data.

The new plug-in provides a means for efficient and accurate modeling of honeycomb sandwich structures. The modeling approach described here is suitable for the analysis of relatively small sandwich structural parts. If large sandwich structures should be modeled, the plug-in can be used to determine the mechanical properties of a specific honeycomb structure by virtual testing rather than conducting expensive and time-consuming physical experiments. The honeycomb mechanical properties obtained from such virtual testing can be then used to model the sandwich core with continuum solid elements.

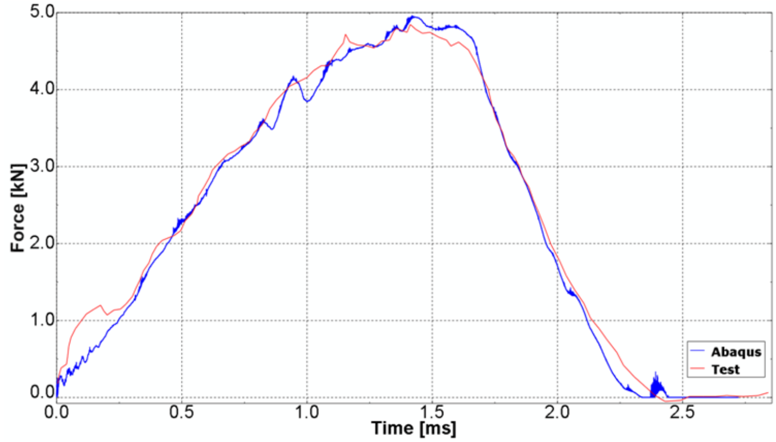

Figure 3: Experimental and predicted force-time histories

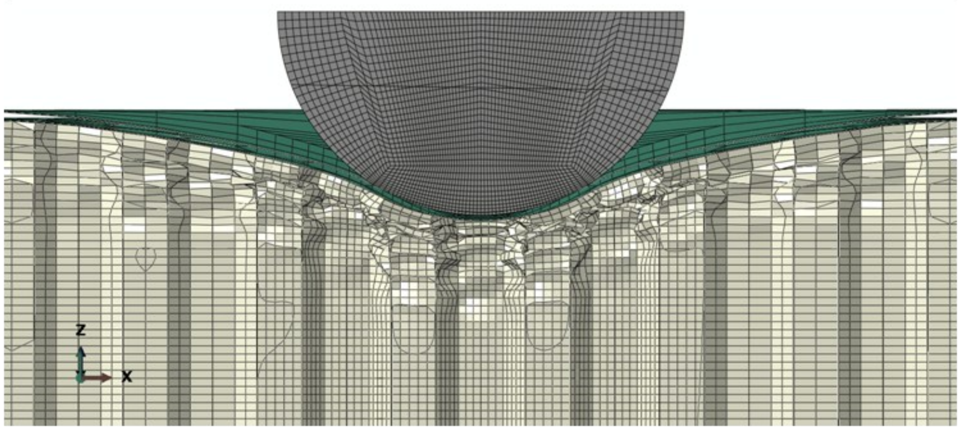

Figure 4: Deformed shape of the sandwich specimen at the maximum displacement of 6.4 mm

References

1. Vinson, J.R., *Plate and Panel Structures of Isotropic, Composite and Piezoelectric Materials, Including Sandwich Construction*, Springer, 2005.
2. Heimbs, S., "Virtual Testing of Sandwich Core Structures with LS-DYNA," ANSYS Conference & 26th CADFEM Users' Meeting, 2008.
3. Nguyen, M.Q., Jacombs, S.S., Thomson, R.S., Hachenberg, D., Scott, M.L., "Simulation of Impact on Sandwich Structures," *Composite Structures*, Vol. 67, Issue 2, 2005.
4. Rider, N.A., "Degradation of Peel and Tensile Strength of Bonded Panels Exposed to High Humidity," Technical Report DSTO-TR-1272, DSTO Aeronautical and Maritime Research Laboratory, 2001.
5. Simo, J.C., and Hughes, T.J.R., *Computational Inelasticity*, Springer-Verlag New York, Inc., 1998.
6. Srivatsan, T.S., Vasudevan, S., and Lederich, R.J., "Influence of Friction Stir Welding on Microstructural Development, Mechanical Response and Fracture Behavior of Aluminum Alloy 2024," 15th International Symposium on the Processing and Fabrication of Advanced Materials, 2006.

SIMULIA References

For additional information on the Abaqus capabilities referred to in this brief please see:

- "Abaqus/CAE plug-in to automatically create honeycomb structure models," DS Knowledge Base Article QA00000023102

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors.

DASSAULT SYSTEMES

Delivering Best-in-Class Products

CATIA
Virtual Product Design

EXALEAD
Information Intelligence

SOLIDWORKS
3D for Professionals

NETVIBES
Dashboard Intelligence

SIMULIA
Realistic Simulation

3DSWYM
Social Innovation

DELMIA
Virtual Production

3DVIA
Online 3D Lifelike Experiences

ENOVIA
Global Collaborative Lifecycle Management

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes in all industries in more than 80 countries. For more information, visit www.3ds.com.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA