EXPERIMENTAL AND NUMERICAL ANALYSIS OF NOMEX HONEYCOMB SANDWICH PANEL INSERTS PARALLEL TO THE FACE SHEETS

R. Seemann[1] and D. Krause[1]

[1]Institute of Product Development and Mechanical Engineering Design
Technische Universität Hamburg-Harburg (TUHH)
Denickestr. 17, 21073 Hamburg, Germany
Email: ralf.seemann@tuhh.de, web page: http://www.tuhh.de/pkt

Keywords: Nomex honeycomb, sandwich, potted insert, explicit FEA, virtual testing

Abstract
The present work investigates custom made sandwich inserts, which are placed parallel to the face sheets, under tensile (pull-out) loading in both experimental and numerical studies. The objective is to develop a simulation model, which is capable of predicting the strength of the joint. This is achieved using a 3D-continuum model, where the core is modelled using 3D-brick elements. Over all, the simulations results are in good agreement with the test results.

1. Introduction

Sandwich panels find numerous applications in lightweight structures that require large planar segments. A prominent example is the secondary structure of aircraft, such as cabin components, landing gear flaps or engine nacelles [7]. These components are typically assembled from multiple sandwich panels. For the internal and external interfaces of these sandwich panel structures, adequate joining elements are required. Typically it is desired to enable detachable joints via bolted connections. For this purpose a great variety of threaded inserts, which are bonded into or onto the sandwich panel, exists [13]. These inserts are generally placed either vertical to the face sheets to fasten components on the panel surface, or they are placed parallel to the face sheets at the edge of the panel to fasten or join the panel with other panels or the surrounding structure.
Since sandwich panels are especially prone to local load introduction, the sandwich panel joints receive particular attention in the design and substantiation process of sandwich structures [13]. Therefore, extensive testing of sandwich panel joints is performed during the design phase, which is also evident in the numerous literature on this topic [4, 8, 9, 12]. In order to reduce the high cost for component testing, the industry is increasingly seeking to implement numerical simulations to predict the strength of their products. As a result, several numerical studies on the failure prediction of sandwich panel joints are available. Bunyawanichakul et al. [2, 3] developed a FE-model of a countersunk titanium fasteners in a Nomex honeycomb sandwich panel using 3D-continuum elements for the core. Nguyen et al. [9] developed a FE-model of foam based sandwich joints and compare different failure modeling methods. Heimbs and Pein [6] derived simplified FE-models where joints are modeled using spotweld elements for an implementation in a global non-linear model of aircraft interior components. In addition, they develop a detailed FE-model of a honeycomb sandwich insert, where the hexagon core geometry is modeled accurately using 4-node shell elements. Roy et al. [10] derived the orthotropic material properties of Nomex honeycomb cell walls using a detailed meso-model of a threaded insert under pull-out. The vast majority of the available studies, focuses on inserts perpendicular to the face sheets. In contrast, the present work investigates threaded inserts parallel to the face sheets in experimental and numerical studies.

Excerpt from ISBN 978-3-00-053387-7

The decisive load cases for inserts are generally pull-out and shear. In case of edgewise sandwich panel inserts, there are two distinct shear directions (in-plane and out of plane). Therefore, a total of three load cases is of interest (Figure 1).

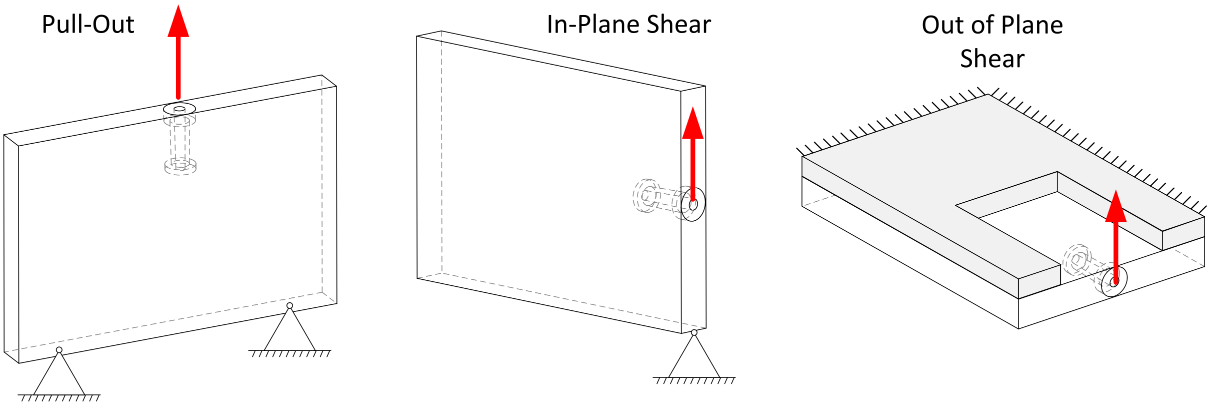

Figure 1 Critical load cases for sandwich panel inserts parallel to the face sheet

The out-of-plane shear load case is similar to the pull-out loading of standard inserts perpendicular to the face sheets, which has been studied in a previous study of the authors [11]. The two in-plane load cases are characterized by direct load transmission from the insert via the potting into the face sheet. Both load cases exhibit similar damage mechanisms, while the pull-out load case appears to cause a more complicated stress state in the face sheets, due to high shear loads and to some extend even bending. Therefore, the present study focuses on the pull-out load case.

2. Materials

The investigated sandwich panels consist of glass fiber fabric reinforced phenolic resin prepregs as face sheets and a Nomex honeycomb core. Two prepregs following the Airbus material performance specifications ABS5047-02 and ABS5047-08 are used in different lay ups. The honeycomb core has a cell size of 3.2 mm and a density of 48 kg/m³ [5]. The inserts are cold bonded after panel manufacturing using the two component adhesive Ureol 1356 A/B as potting. The inserts are custom made from aluminum alloy (Figure 2). The manufacturing process is similar to standard perpendicular inserts. The core is locally removed, the insert is placed in the hole with the help of a jig and the potting mass is injected.

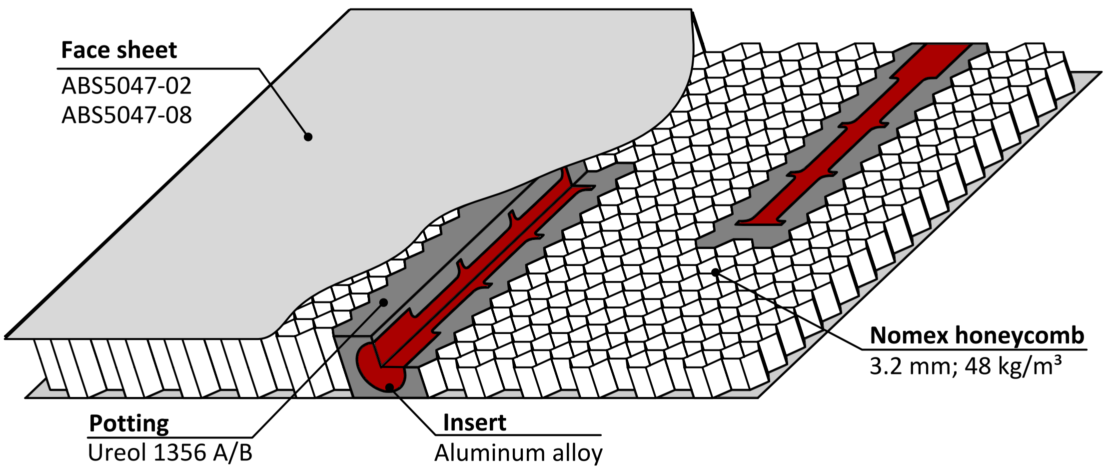

Figure 2 Investigated materials

Excerpt from ISBN 978-3-00-053387-7

In total five different configurations are studied, whereby four configurations are derived from a standard configuration. Based on this standard configuration, the length of the insert as well as the number of face sheet prepregs is varied in two increments (Figure 3). All panels have a nominal core height of 18.5 mm, while the total panel thickness depends on the face sheet layup. The panels are cut into dimensions of 230 and 180 mm.

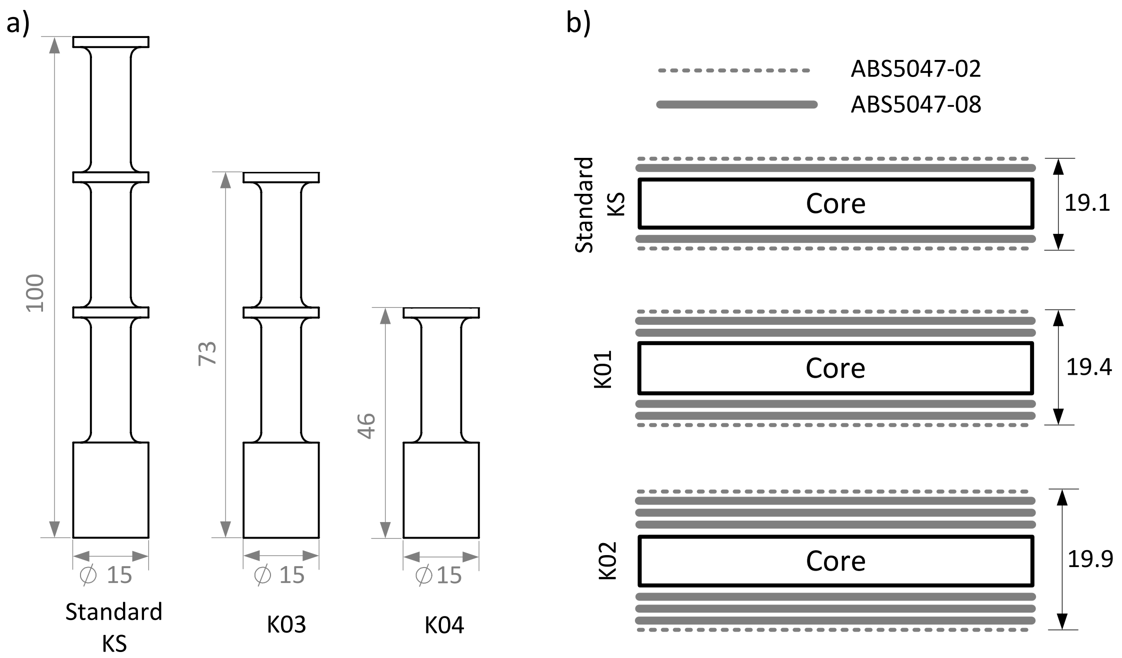

Figure 3 Five investigated configurations a) insert lengths; b) prepreg lay-ups

3. Experimental study

The test setup is designed in a way that requires each specimen to have a two-point support at the bottom (Figure 4). This is provided by two additional edgewise inserts 140 mm apart at the bottom of each panel. Therefore, each specimen contains three inserts, while it is the middle insert that is being pulled out. Hence, the setup resembles a three point bending test. The tests are performed on a Zwick/Roell Z050 TH universal testing machine using a 50kN load cell (class 0.5). Four specimens for each configuration are prepared and tested with a constant cross head velocity of 10 mm/min. The deflection is taken directly from the internal displacement measurement of the machine. In order to compensate the deflection of the setup, the stiffness of the test setup is measured prior to testing.

During the tests two distinct global damage mechanisms are evident – face sheet damage and debonding of potting and face sheet. The failure of the configurations with standard face sheet lay-up is solely driven by face sheet damage, while the configurations with added face sheet layers additionally show skin debonding. The debonding becomes the dominant damage mechanism in K02 with maximum number of face sheet layers. Due to the complicated stress state in the face sheet, the skin damage pattern shows up to three laminate damage mechanisms (Figure 5 right). The skin damage pattern varies considerably and it is difficult to assign a specific damage pattern to certain configurations. There are however trends. The tensile failure in x-direction at the top may occur in all configurations except in K03 and K04 with shorter inserts. The failure in these two configurations is initiated by y-tensile damage and propagates into shear damage shortly after. Shear damage on the other hand is not evident in the configurations K01 and K02 with added skin layers. The x-tensile damage at the top tends to initiate the failure in configuration K01 with intermediate face sheet lay-up. It assumed that this scatter in damage patterns is due to the manual manufacturing process of the joints.

Excerpt from ISBN 978-3-00-053387-7

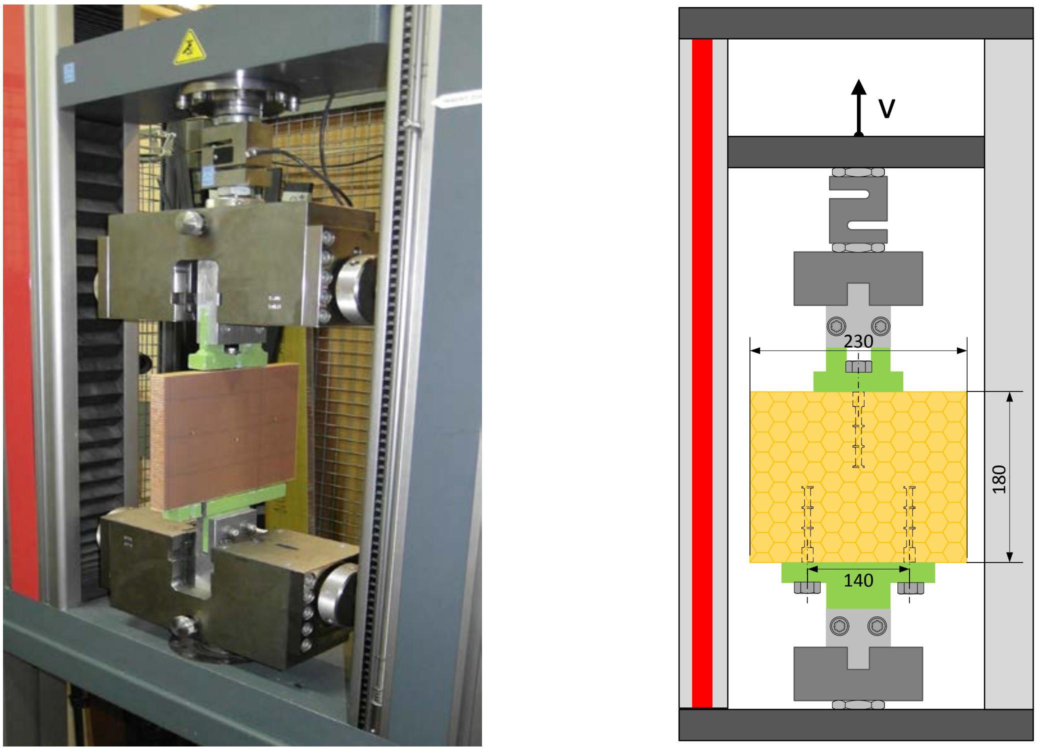

Figure 4 Experimental setup

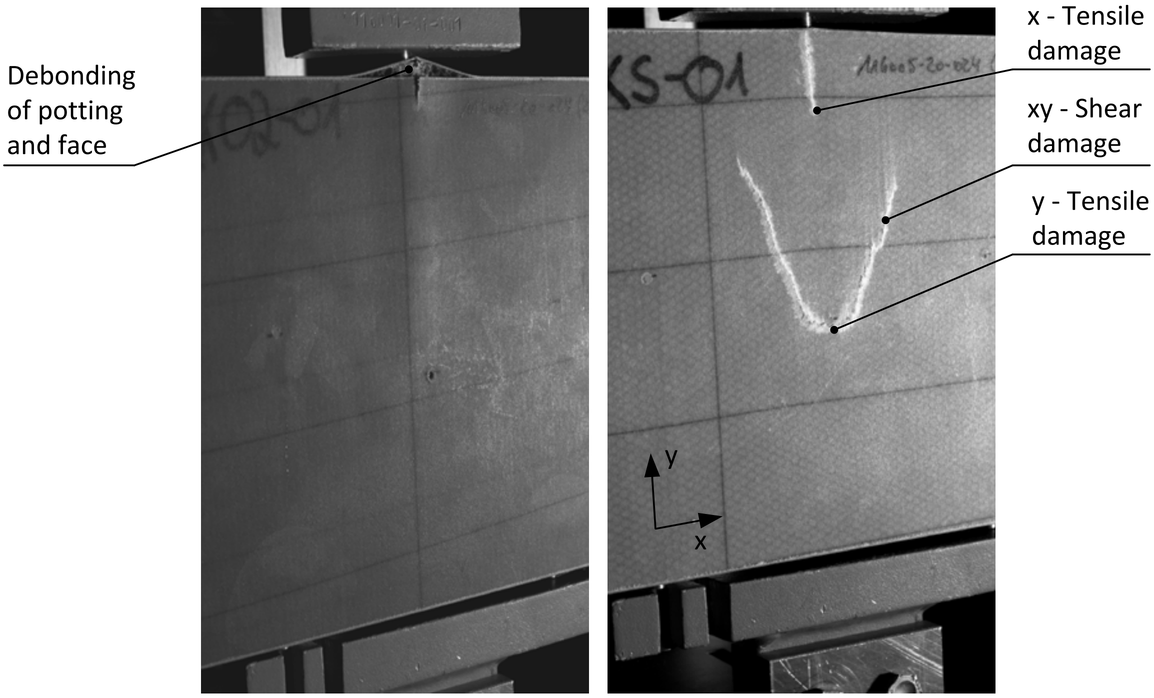

Figure 5 Damage mechanisms during tests; K02 debonding (left) and KS face sheet damage (right)

4. Numerical study

The numerical models are implemented using the commercial FE-Software ABAQUS\Explicit. The explicit solver is chosen, in order to avoid convergence problems due to multiple non-linearities in the model. In order to reduce the model size, the symmetry of specimen and loading condition is utilized

Excerpt from ISBN 978-3-00-053387-7

by only modelling one quarter of the specimen. The required level of model detail is defined based on a problem analysis and the intended use of the simulation model. In this case, the model shall be able to predict the absolute strength of the joint, while the post failure behavior is not of interest. The identified damage mechanisms during the experimental study indicate, that it is not required to model the honeycomb core in detail. Instead a 3D-contiuum model, where the core is modeled using 8-node brick elements, is implemented. This modelling approach enables a detailed representation of the load transmission from insert via potting into the face sheet. In addition, it allows to define cohesive behavior between potting and face sheet, which his required to account for debonding of these two constituents. Since an explicit solver is applied for a quasi-static loading problem both mass scaling and increased loading rate are implemented as measures to reduce computational time. The appropriate levels of mass scaling and increased loading rate have been determined in sensitivity studies. The loading is defined as prescribed velocity on the nodes inside the bore hole of the middle insert. The model is constrained in the bore hole of the outer insert in all translatory degrees of freedom. Depending on the configuration the model has up to 76000 nodes. An illustration of the implemented model is given in Figure 6. The representation of the different constituents along with the contact definitions is described in the following.

4.1. Constituent representation

Face sheets
The face sheets are modelled with 4-node shell elements of the S4R type. A global mesh size of 1.6 mm is defined, which is refined to 0.8 mm in the vicinity of the middle insert. The standard ABAQUS material model for fiber reinforced composites is applied including damage modeling based on Hashins failure criteria. The material properties for the face sheets have been derived from 4-point bending and shear tests on bonded sandwich panels made of the same materials as studied in the present work.

Core
As initially indicated, the core is modeled using 8-node brick elements of the C3D8R type and an element size of 2 mm. An elastic orthotropic material model (*Engineering Constants*) is defined using the macroscopic material data given by the core manufacturer [5]. The unknown material properties, such as in-plane stiffness are set to be about 1% of the out-of-plane stiffness as suggested by Bitzer [1]. This ensures that the core only adds neglectable resistance to the pull-out loading, which is also confirmed by the experimental results. The low in-plane stiffness also ensures, that no damage modeling is required. The defined material parameters for the core are summarized in Table 1.

Table 1 Defined core material properties

E_1	E_2	E_3	G_{12}	G_{13}	G_{23}	v_{12}	v_{13}	v_{23}
1 MPa	1 MPa	137 MPa	0.1 MPa	30 MPa	48 MPa	1	0.01	0.01

Potting and Insert
Both potting and insert are discretized using simple 4-node tetraeder elements (C3D4). This allows efficient meshing and good geometry representation using a small element size of 1 mm at tolerable computational effort. Since the experimental results do not indicate any damage at the insert, an elastic material model is implemented based on the established stiffness of aluminum alloy (E = 69000 MPa). The potting material parameters have been determined in a previous study [11]. A bi-plastic material model is implemented based on tension and compression test results of the potting material. In a sensitivity study it has been shown that 4-node tetraeder elements are sufficient to capture the plastic behavior of the potting.

4.2. Contact definition

According to the observations during the tests, the debonding of face and potting is the only debonding mechanism that contributes to the failure of the investigated joints. Therefore, the contact

Excerpt from ISBN 978-3-00-053387-7

between potting and face is defined as a surface based cohesive contact using a linear traction-separation law. All remaining contact surfaces in the model are defined as tied contacts.

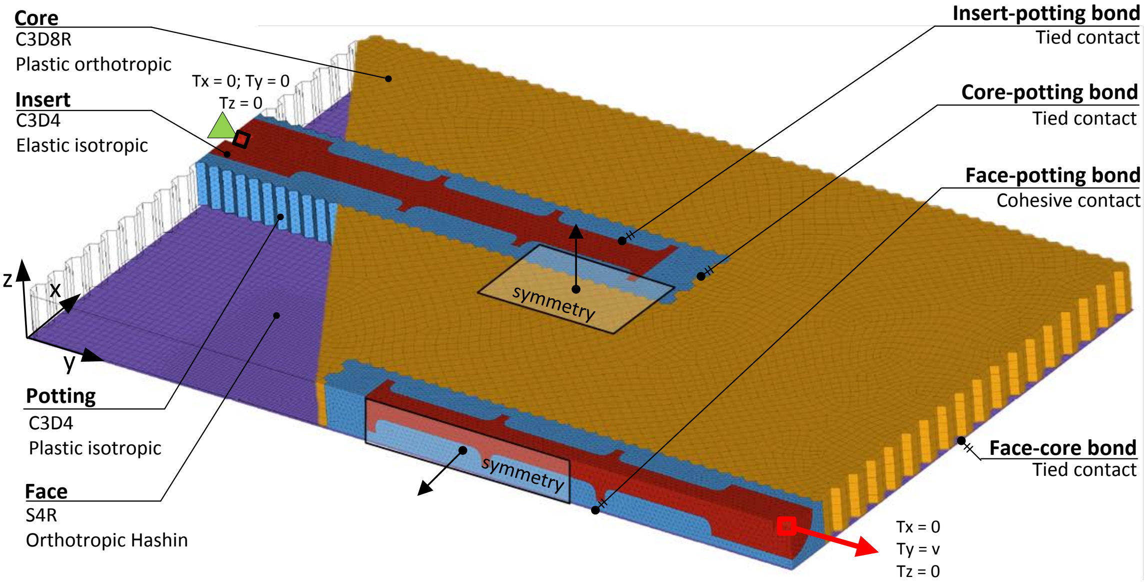

Figure 6 Model definition

5. Results

The test results along with the simulations results in terms of force displacement relationship are given in Figure 7. The experimental results largely reflect the expectations. Increasing the insert length leads to increased load bearing capacity, as the load is distributed to a larger face sheet area. However, the strength of configuration K03 is only by a small margin lower than the standard configuration KS. The reason for this has been found in the depth of the potting mass in the panel. The specimens of both KS and K03 appear to have almost the same volume of potting mass, despite K03 having a considerably shorter insert. This is not intended and can be attributed to incorrected manufacturing. As a consequence both configurations show similar strengths, while KS still slightly outperforms K03 due to the stiffening effect of the longer insert. K04 does not have this manufacturing error and therefore achieves strengths that are considerably lower than K03 and KS.

The simulation results agree well with all three configurations with standard lay-up in terms of both stiffness and strength (deviation is below 10%). The exception is the strength of K03, which is notably underestimated. This is due to aforementioned manufacturing error resulting in added potting mass, which is not considered in the model.

For the configurations with added face sheet layers, it can be said that the simulation overestimates the measured test results with regards to stiffness, while the strength is predicted well. These two configurations show considerable debonding of face sheet and potting mass. This effect is captured well by the cohesive contact in the model, leading to a good match of test and simulation in terms of strength. Regarding the stiffness, it can be noted that in all configurations the simulation lies above test results, suggesting that the young's modulus of the face sheet prepregs is overestimated. However, in case of the configurations K01 and K02, this mismatch is more evident. Therefore, it is assumed that another effect is the cause of this deviation of simulation and test results. One possible explanation is, that K01 and K02 were prepared with prepregs from a different manufacturer if compared to the other configurations.

Regarding the damage mechanisms, it is notable that the model predicts the failure mode for all configurations correctly with the damage pattern resembling the specimens after failure (Figure 8).

Excerpt from ISBN 978-3-00-053387-7

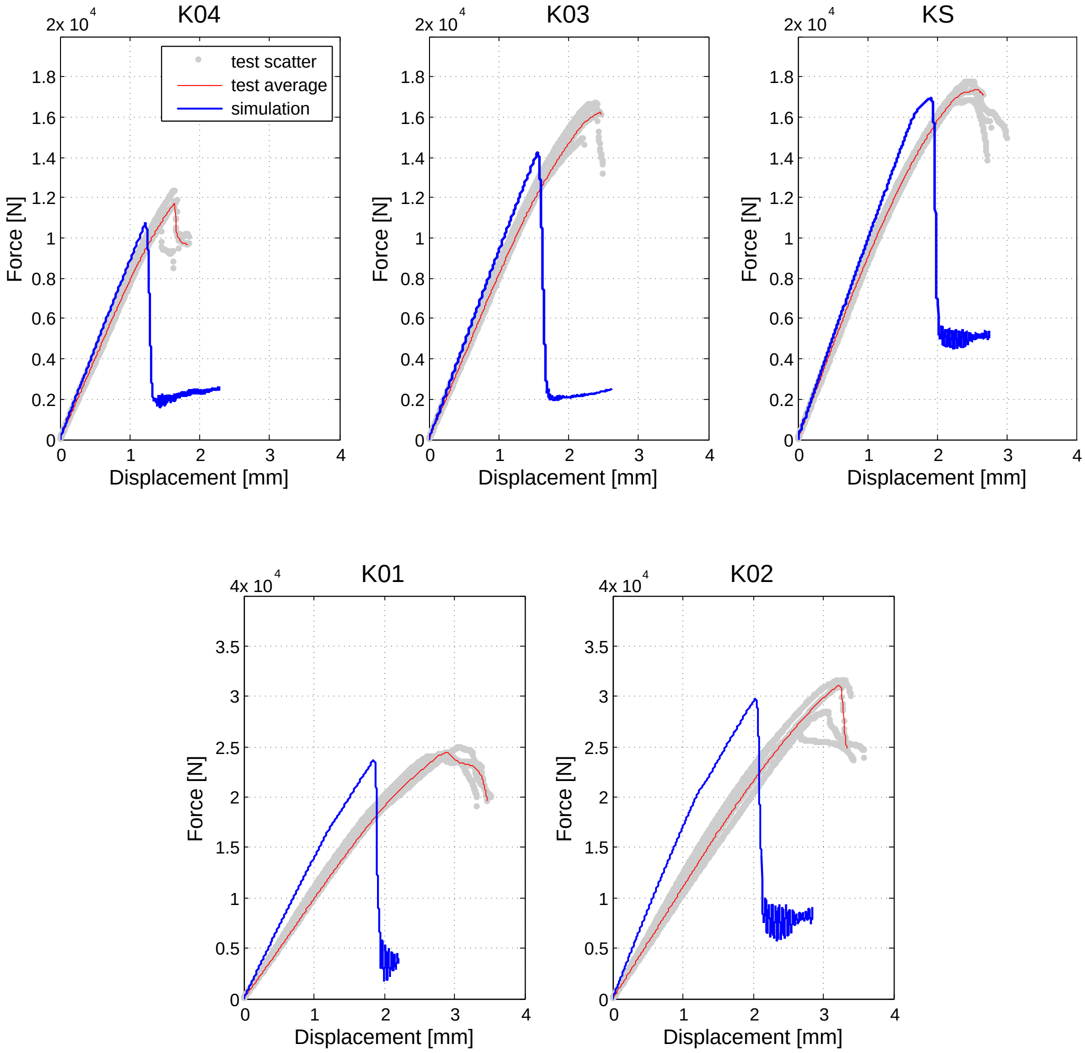

Figure 7 Comparison of force displacement curves from test and simulation

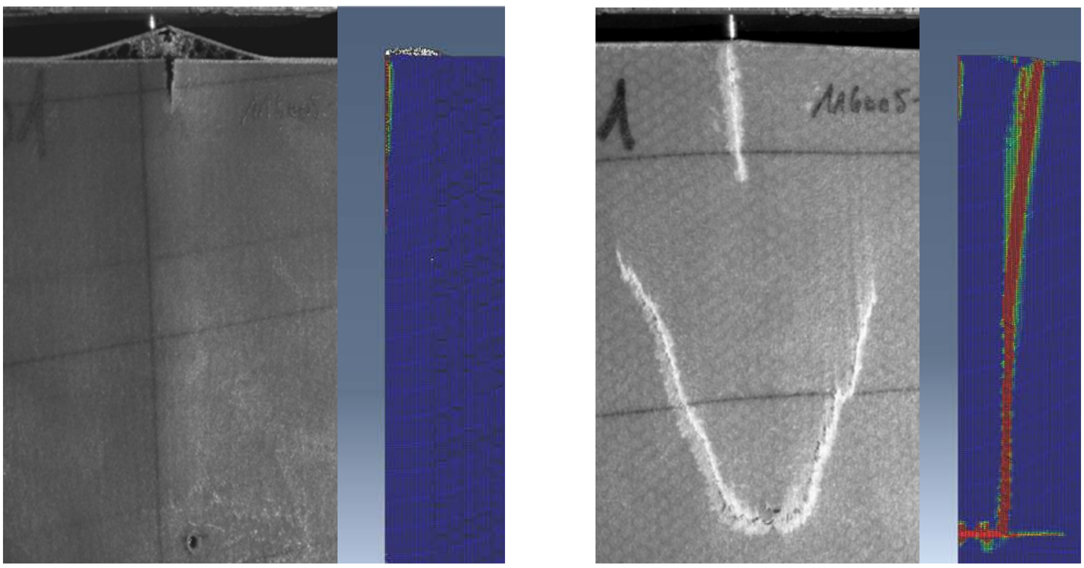

Figure 8 Comparison of damage pattern in simulation and test, left debonding, right face sheet failure

Excerpt from ISBN 978-3-00-053387-7

6. Conclusion

In this work, tensile tests on custom made sandwich inserts parallel to the face sheets were performed on five different configurations. The tests revealed two failure mechanisms – face sheet failure and debonding of potting and face sheet, while the face sheet failure was complex with multiple damage patterns. A simulation model was implemented using a virtual testing approach in order to predict the test results for future use in design studies. The model was implemented as 3D-continuum model, where the core is modeled using 8-node brick elements and the face using 4-node shell elements. The debonding of face sheet and potting was modeled as surface contact with cohesive behavior.
Simulation results agree well with the experimental results, with the exception of the stiffness in the configurations with added face sheet layers. This is to be investigated further.

Acknowledgments

The presented work was funded by the German Federal Ministry for Economic Affairs and Energy, through a subaward received by DIEHL Aerosystems in the framework of LuFo V-2. The authors want to thank the DIEHL Service Modules GmbH for the preparation of the specimens and conducting the tests.

References

[1] Bitzer, T. 1997. *Honeycomb technology*. *Materials, design, manufacturing, applications and testing*. Chapman & Hall, London, New York.
[2] Bunyawanichakul, P., Castanie, B., and Barrau, J.-J. 2005. Experimental and Numerical Analysis of Inserts in Sandwich Structures. *Appl Compos Mater* 12, 3-4, 177–191.
[3] Bunyawanichakul, P., Castanié, B., and Barrau, J.-J. 2008. Non-linear finite element analysis of inserts in composite sandwich structures. *Composites Part B: Engineering* 39, 7-8, 1077–1092.
[4] Demelio, G., Genovese, K., and Pappalettere, C. 2001. An experimental investigation of static and fatigue behaviour of sandwich composite panels joined by fasteners. *Composites Part B: Engineering* 32, 4, 299–308.
[5] EURO-COMPOSITES. 2010. *Mechanical PropertiesECA Honeycomb (metrical, test specimen thickness 12.7 mm)*, EC536-13e.
[6] Heimbs, S. and Pein, M. 2009. Failure behaviour of honeycomb sandwich corner joints and inserts. *Composite Structures* 89, 4, 575–588.
[7] Herrmann, A. S., Zahlen, P. C., and Zuardy, I. Sandwich Structures Technology in Commercial Aviation. In *Proceedings of the 7th International Conference on Sandwich Structures*, 13–26. DOI=10.1007/1-4020-3848-8_2.
[8] Kim, B. J. and Lee, D. G. 2008. Characteristics of joining inserts for composite sandwich panels. *Composite Structures* 86, 1-3, 55–60.
[9] Nguyen, K.-H., Park, Y.-B., Kweon, J.-H., and Choi, J.-H. 2012. Failure behaviour of foam-based sandwich joints under pull-out testing. *Composite Structures* 94, 2, 617–624.
[10] Roy, R., Nguyen, K. H., Park, Y. B., Kweon, J. H., and Choi, J. H. 2014. Testing and modeling of Nomex™ honeycomb sandwich Panels with bolt insert. *Composites Part B: Engineering* 56, 762–769.
[11] Seemann, R. and Krause. D. 2015. Virtual testing of Nomex honeycomb sandwich panel inserts. *20th International Conference on Composite* Copenhagen, 19–24 July 2015.
[12] Song, K.-I., Choi, J.-Y., Kweon, J.-H., Choi, J.-H., and Kim, K.-S. 2008. An experimental study of the insert joint strength of composite sandwich structures. *Composite Structures* 86, 1-3, 107–113.
[13] Zenkert, D. 1997. *The handbook of sandwich construction*. Engineering Materials Advisory Services, Cradley Heath, West Midlands.

Excerpt from ISBN 978-3-00-053387-7