20th International Conference on Composite Materials
Copenhagen, 19-24th July 2015

VIRTUAL TESTING OF NOMEX HONEYCOMB SANDWICH PANEL INSERTS

Ralf Seemann[1] and Dieter Krause[2]

[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, Fully potted insert, Explicit FEA, Building block approach

ABSTRACT

The present study develops a detailed progressive failure model based on the Finite Element Method (FEM) for the widely used fully potted threaded sandwich insert under pull-out loading. The investigated sandwich panel consists of phenolic resin reinforced by glass fiber fabric as face sheet and a Nomex honeycomb core. The model is developed for three configurations where the core height is the only varying parameter. The numerical results are compared with extensive experimental data for each configuration. The model shows good agreement between numerical and experimental results well beyond the first failure mechanism of cell wall buckling for all three configurations. With increasing core height, the debonding of honeycomb core and face sheet becomes dominating with regards to the global joint strength. However, this failure mechanism has not yet been included in the model. It therefore overestimates the joint strength for the configurations with increased core height.

1 INTRODUCTION

Despite the increasing application of detailed finite element (FE) models for local numerical analyses of honeycomb sandwich structures, only few examples of virtual tests on sandwich panel joints are evident in the literature. However, due to the great variety in joint geometry and possible material combinations, it is desirable to virtually test sandwich panel joints in order to reduce cost and time imposed by real testing during the design phase.

A standard fastening element in honeycomb sandwich panel joints are potted threaded inserts [1, 2]. Due to their wide distribution, these fasteners have been investigated extensively in experimental studies in the past. Song et al. [3] investigated potted inserts in various configurations of Nomex honeycomb sandwich panels with carbon epoxy face sheets. They find that the insert pull-out strength largely depends on the core height and density as well as the face sheet thickness, while the shear-out joint strength is dominated by the face thickness. Kim and Lee [4] studied the load transfer characteristics of partially potted inserts in composite sandwich panels with respect to the insert geometry. Demelio et al. [5] tested different combinations of honeycomb sandwich panel fasteners under shear and pull-out static and fatigue loading. They report, that skin reinforcement and core height dominate the fatigue strength.

Many of the published studies on sandwich panel inserts additionally complement experimental studies with simulation models for a better understanding of the failure mechanisms or for the prediction of failure. These available models can be divided in analytical and FE-models. Thomsen and Rits [6, 7] developed one of the first models for the prediction of the sandwich panel joint strength based on high-order sandwich plate theory. Due to the made simplifications, namely smeared honeycomb core properties as well as disregard of the irregular potting and honeycomb interface, this numerically solved analytical model is only suitable for early design estimations and for deriving design guidelines. In a later study by Bull and Thomsen [8] this model has been implemented in a design tool for initial dimensioning of inserts in sandwich panels and the model performance is compared with experimental data as well as finite element analysis (FEA) predictions. In a more recent study by Smith and Banerjee [9], the analytical Thomsen model has been applied to perform reliability analyses on the strength of sandwich panel inserts comparing different reliability analysis methods. However, the majority of available simulation models are based on FEA, while the level detail of these models greatly varies.

Earlier FE-models tend to be axi-symmetric, thus neglecting potting to honeycomb interfaces similar to the analytical model of Thomsen and his colleagues [8, 10-12]. With increasing computational capabilities during the last decade more detailed 3D FE-models emerged. Bunyawanichakul et al. [13, 14] experimentally investigate the strength of countersunk titanium fasteners in an epoxy potting using a testrig that allows to apply pre-stress to the fastener. They then developed a FE-model supported by constituent tests on the potting material and the honeycomb core, while the honeycomb core is modeled using 3D-continuum elements. Nguyen et al. [15] studied different configurations of foam based sandwich joints through experiments and develop a FE model, while comparing different failure modeling methods. Heimbs and Pein [16] tested different configurations of corner joints and inserts for Nomex honeycomb sandwich panels and derived simplified 3D FE-models based on spotweld elements for an implementation in a global non-linear model of aircraft interior components. In addition, they develop one of the first detailed FE-models of a honeycomb sandwich insert, where the hexagon core geometry is modeled accurately. Such detailed meso-scale models of sandwich panels with structured cores have seen an increasing number of applications mainly for impact and crushing analyses of sandwich panels [17]. Additional applications of such detailed meso-models for honeycomb sandwich inserts include the linear model of Bianchi et al. [19], who investigated hot and cold bonded procedures of honeycomb sandwich insert manufacturing. They conclude that the stiffness of the potting material has a significant impact on the insert joint strength. Roy et al. [18] applied experimental studies on honeycomb sandwich panel inserts, to derive the orthotropic material properties of the Nomex core material using a detailed meso-model of the joint. Furthermore, meso-models of honeycomb cores have also been successfully applied to analyze the thermal coupling of sandwich inserts used in satellites [20].

The reviewed literature, consistently reports a complicated progressive failure behaviour of honeycomb sandwich inserts driven by multiple failure mechanisms, while the failure is usually initiated by local buckling of the cell walls adjacent to the potting. Modelling this initial cell wall failure requires detailed FE models with accurate cell wall representation. As shown, the literature provides some numerical studies that implement such detailed models for a better understanding of the failure mechanisms. However, a virtual test capable of predicting the progressive failure behaviour of potted sandwich inserts up until total failure is not yet evident. The present contribution describes the development of such a virtual testing framework using the Building Block Approach (BBA).

2 SANDWICH JOINT MATERIALS

The present study investigates the insert SL607-08-6S from Shur Lok [21] fully potted in a typical sandwich panel for an application in aircraft interior components. The symmetric face sheets are made of phenolic resin impregnated glass fiber fabric prepregs (one layer each of PHG600-44-50 and PHG600-68-50). Nomex honeycomb with a cell size of 3.2mm and a density of 48kg/m³ serves as core material, while the two component adhesive Ureol 1356 a/b is used to bond the insert in the panel. Three different configurations with varying core height are studied (Figure 1).

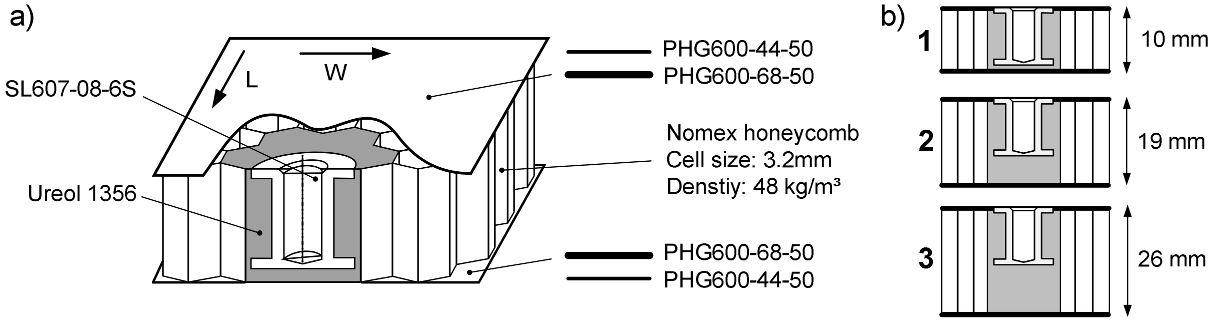

Figure 1: a) Tested sandwich joint configuration, b) Three tested core heights

3 VIRTUAL TESTING

Virtual testing intended to reduce real testing effort goes beyond the mere application of advanced simulation models. Predicting the non-linear material behavior of light weight structures requires a system of hierarchical models, engineering tests, and specialized laboratory experiments, supported by the application of information science, model-based statistical analysis, and decision theory [26]. The development of such a system for the proposed problem of honeycomb sandwich panel inserts is described in the following. More background information on virtual testing and particularly the methodology using the same example as in the present study is given in [25].

3.1 Methodology

The development of the proposed virtual testing framework is based on the Building Block Approach [22], whereby a synergetic combination of tests and analysis methods are applied on several levels of structural complexity of a given structure, in order to predict its mechanical performance. These complexity levels are also referred to as building blocks. Integrating the gained knowledge at lower levels into higher levels as well as extensive verification on each level ensures high fidelity of the developed analysis model. The implemented approach is illustrated in Figure 2. The hierarchical system of models is preceded by the problem analysis and the definition of the required level of detail. Based on this, three building blocks are defined. The first is the constituent level, which includes the Nomex honeycomb core material and the potting mass. Secondly there is the component level. Here, the bonded sandwich structure including the glass-fiber reinforced face sheets is analysed. Lastly the verified sub-models from the lower levels are integrated in the model of the final structure. All models are implemented using the commercial FE-software ABAQUS. The development of the models in each building block goes through the same general process. Firstly structural tests are performed, which eventually serve as benchmark for the model. At the same time supporting analysis methods, such as microscopy, are performed in order to characterize the structure and to be able to model it appropriately. Subsequently the FE-model of the previously performed test is generated. This model then has to be checked for its plausibility and sensitivity to certain modelling details (i.e. mesh size or scale). Lastly the model is verified using the initial test results and model parameter are adjusted respectively.

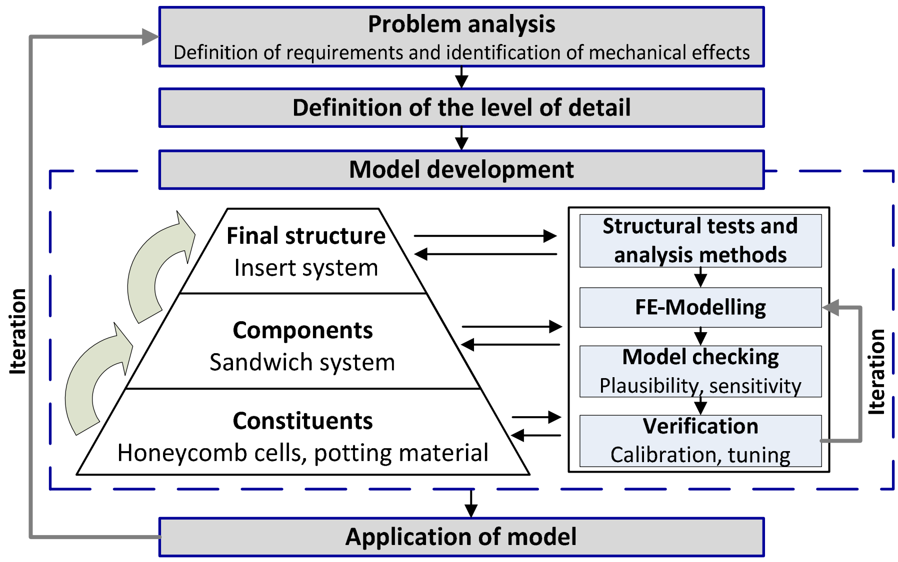

Figure 2: Applied approach for the development of a virtual testrig for sandwich panel inserts

3.2 Problem Analysis

The present study focuses on the sandwich panel insert pull-out test as it is suggested in the Insert Design Handbook (IDH) [23]. Here, an insert is bonded in the center of a quadratic sandwich panel (100x100mm) and placed against a fixture with a circular hole (80mm). Through quasi-static cross head movement (2mm/min) of an universal material testing machine the insert is pulled out of the panel until total failure (Figure 3a). Understanding the mechanical effects that govern the failure of a structure is crucial for developing a prediction model. Therefore, the force displacement curve of the pull-out test is analysed and subdivided in four phases based on observations during the test. The identified failure mechanisms confirm the findings of previous studies in the literature. A representative force-displacement curve of one of the performed insert pull-out tests is depicted in Figure 3b. Phase ① represents the linear elastic displacement of the undamaged structure. In phase ②, a quadratic flattening of the curve occurs. This results from shear buckling of the honeycomb cell walls adjacent to the potting. Phase ③ represents another linear elastic regime, which is dominated by elastic deformation of the face sheets, while shear buckling of the core progresses. However, depending on the tested configuration, continuous debonding of the face and core might occur in phase ③ simultaneously, leading to reduced stiffness with increasing core height. Phase ④ marks total failure of the structure. This is initiated by failure of the upper face sheet followed by debonding of the potting material from the bottom face sheet. In addition, the core cell walls tear due to tensile loading in this final phase. Figure 3c illustrates the observed failure mechanisms in a section view of a failed specimen. Failure of potting has not occurred in the present study. However this is reported in some of the reviewed previous studies.

a)

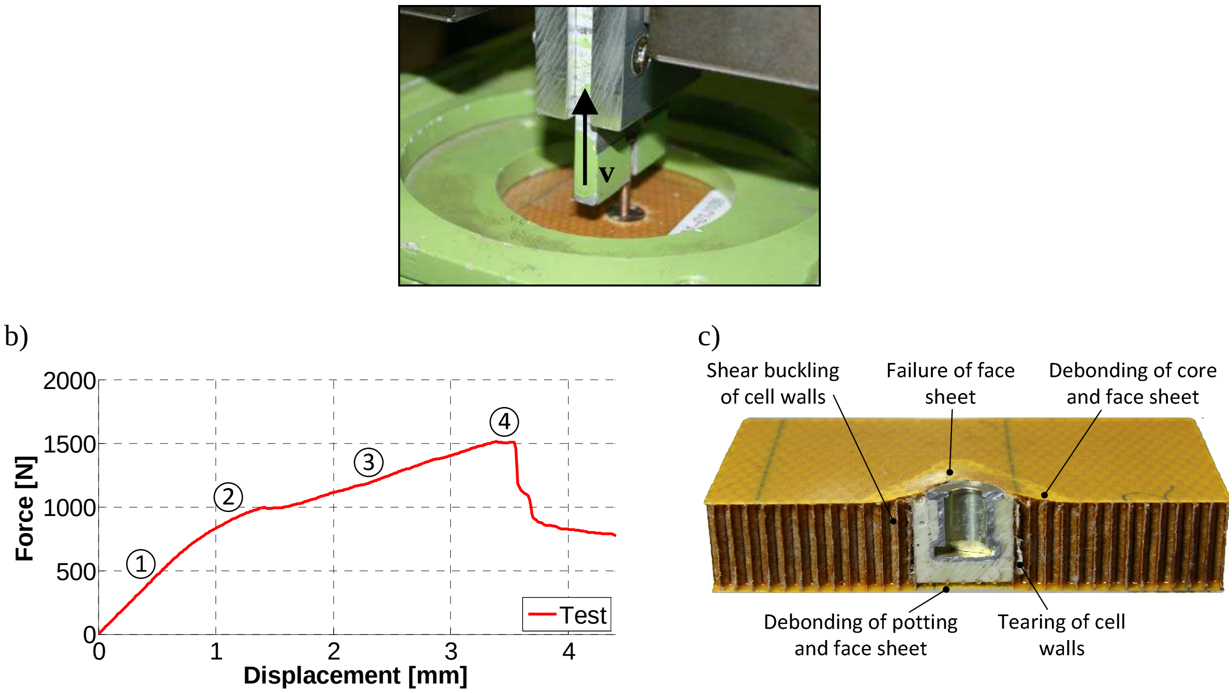

Figure 3: a) Sandwich panel insert pull-out test as suggested by the IDH [23]; b) Typical force displacement curve of insert pull out test; c) identified failure mechanisms

3.2 Constituents – Potting and Honeycomb Core

From the problem analysis it can be concluded that the honeycomb core material and the face sheets are the primary constituents that govern the failure of the insert system. The applied honeycomb core material has been studied extensively in a previous study by the authors of this contribution [24]. In this study, compressive (ASTM C363) and shear tests (ASTM C273) have been performed on the sandwich core material. These tests are then implemented in a virtual testing environment using a detailed meso-model of the honeycomb core and the material parameters of the cell walls are determined through calibration using the macroscopic test results. For the present study, the previously determined material parameters are newly calibrated using ABAQUS/Explicit as solver (Figure 4), while an orthotropic elastic material model including Hashin failure model criteria is employed. This material model is standard for fibre reinforced composite materials in ABAQUS and enables the consideration of the tensile failure of the Nomex.

In the present study, no coupon tests on the face sheet prepregs are performed, since coupon tests have limited validity in sandwich structure design. This is largely due to the telegraphing effect that leads to different material properties when comparing stand alone and honeycomb core bonded prepregs. Therefore, the material parameters given by the prepreg manufacturer are used as reference. These parameters are then adjusted using bending tests of the sandwich panel as described in the following paragraph.

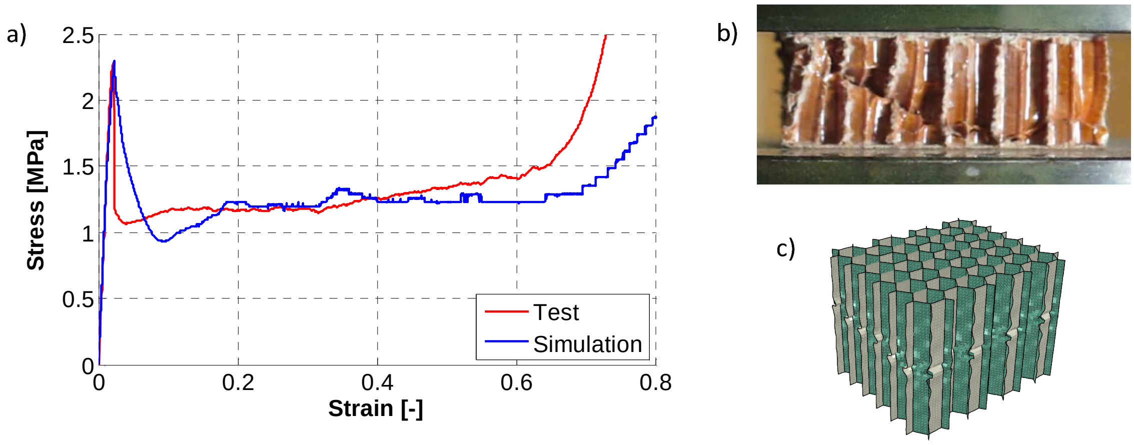

Figure 4: a) Comparison of simulation and test results for compression test of the core after parameter calibration b) Compressive core test acc. to ASTM C363; c) Simulation of compressive core test

Lastly, there is the adhesive material of the potting and the threaded inserts itself to be addressed on constituent level. Since the steel insert does not fail and the young's modulus is generally known, no additional tests or models are implemented for the insert. On the contrary, the mechanical properties of the potting material are generally not given by manufacturers. Hence, tensile (ASTM D638) and compressive (ASTM D695) constituent tests are performed on the potting material (Figure 6). The material exhibits a very ductile behaviour with a comparably low young's modulus of E_{pot} = 800MPa, while the compressive and tensile plastic behaviour differ significantly. This material is modelled using an isotropic elastic material model, complemented by a plastic behaviour that allows for separate input of stress strain curve data in compressive and tensile direction (Figure 6a).

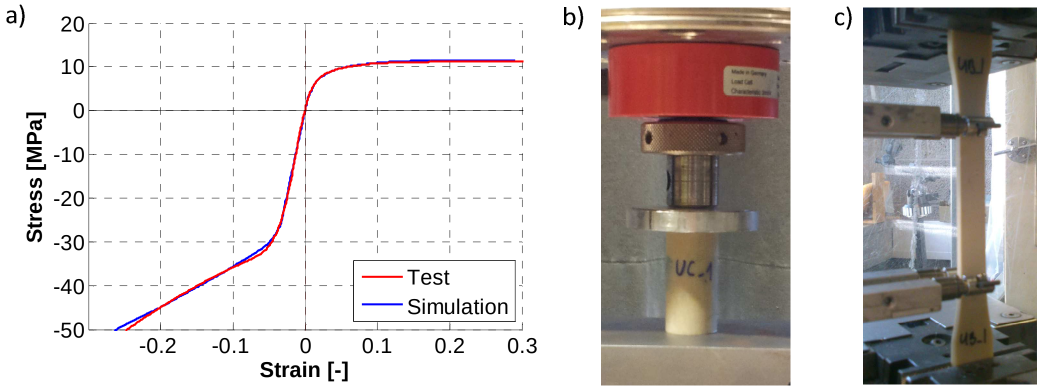

Figure 5: a) Comparison of measured potting material behaviour and implemented material model; b) compressive potting material test acc. to ASTM D695; c) tensile potting material test acc. to ASTM D638

3.4 Components – Bonded Sandwich Panel

The second building block is the bonded sandwich structure. As indicated in the previous section, 4-point bending tests according to ASTM C393 on the same sandwich panels as used for the insert pull-out tests are performed. Therefore, three different panels with varying core height but the same face sheets are tested in both material directions of the panel (L/W as in Figure 1a). These tests are then implemented in ABAQUS/Explicit using a detailed meso-scale model of the sandwich beam.

Assuming, that the core material properties are known from the first building block, the bending tests enable the calibration of the face sheet material parameters, by matching the experimental results with the virtual test results of the simulation. This process is illustrated in Figure 6.

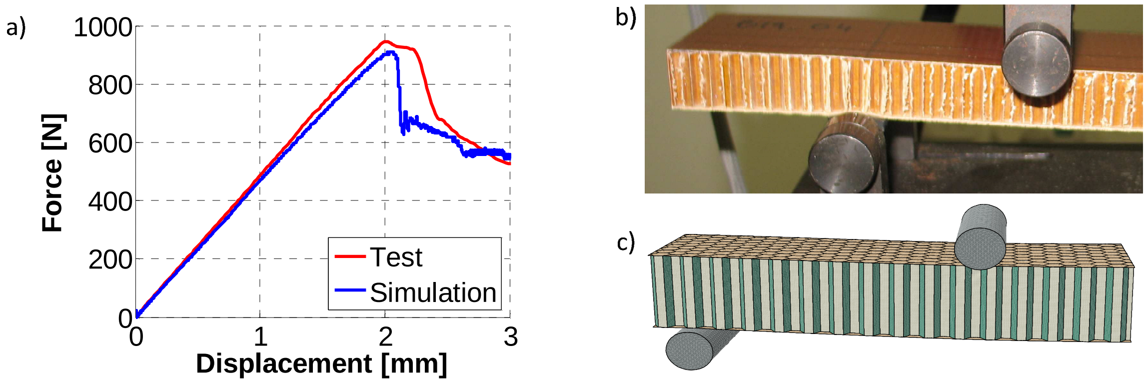

Figure 6: a) Comparison of force displacement curves of experiment and simulation for 4-point bending test of 19mm panel; b) 4-point bending test; c) virtual 4-point bending test on meso-scale

The identified debonding effects can be considered as part of the component building block as well. However, in order to reduce the complexity of the final model, the consideration of these effects has been simplified in the present study. The debonding of face sheet and core is neglected completely. This failure mechanism does not occur in the 10.5mm panel. It is therefore assumed, that this configuration is predicted best by the developed model. In a later stage, the debonding of core and face can be implemented based on available drum peel test results. The debonding of potting and face sheet is more difficult to characterize, as there are no standardized test procedures. However, since this failure mechanism occurs in all configurations it is implemented in the final model without constituent tests. It is assumed that the calibration of the corresponding cohesive contact can be done using the final model, considering that all remaining model parameters are known.

3.5 Final Structure – Fully Potted Sandwich Insert

The previously verified sub-models are lastly integrated in the full model of the insert-pull out test. In order to find a good trade-off between numerical accuracy and computational effort, numerous preliminary studies on the final model are performed. In this regard, implicit and explicit integration schemes have been compared as well. Due to the multitude of non-linear effects of the final model, ABAQUS/Explicit is generally favoured. However, in the reviewed literature both, implicit and explicit numerical models are evident for sandwich fastener simulation applications. Despite the presence of many non-linearities, implicit integration can be also justified, since quasi-static loadings are considered exclusively, making the choice of the favourable integration scheme not trivial for the proposed problem. Therefore, the final model is implemented in ABAQUS/Standard as well as in ABAQUS/Explicit and the processes are compared. It can be concluded that both integration schemes lead to comparable results at comparable computational effort. However, with ABAQUS/Standard multiple convergence issues appear throughout the model setup, requiring constant adjustment of the respective numerical stabilization parameters. Since simulation with ABAQUS/Explicit is considerably more robust, it is continued with the explicit integration scheme.

In addition, several mesh convergence studies are performed. In the first building block it has been established that a mesh size of 0.4mm for the honeycomb cell walls offers a good trade-off between convergence and computational effort to capture the cell wall buckling [24]. However, since buckling only occurs in the cell walls adjacent to the potting it is established through further convergence studies that about five cells surrounding the potting require a fine mesh, while the outer cells can be coarse. Similarly, the required mesh size for the face sheets in the area of damage is established.

When performing quasi-static simulations using an explicit solver, mass scaling and increased displacement rates are common practice to reduce the computational effort. In order to ensure that these model adjustments do not significantly affect the numerical results, several convergence studies are performed in each building block in order to establish suitable and continuous mass scaling and displacement rates throughout the different hierarchical models.

Regarding the contact formulations, the bond between potting and cell walls is modelled as tied contact, since the problem analysis does not indicate significant failure of this bond. In contrast to that, the bond between face sheet and core does fail in two of the tested configurations. However, in order to reduce the complexity of the model, this contact is simplified to a tied contact in the current stage. Only the bond between potting and face sheet is implemented as cohesive contact, since it occurs in all tested configurations. The contact between fixture and panel is modelled using a standard penalty contact. The threaded insert is simplified to a rigid body that comprises directly the nodes on the inner surface of the potting. The deformation of the insert is therefore neglected, which is supported by a preliminary comparative study where the insert has been included. The final model setup is summarized in Figure 7.

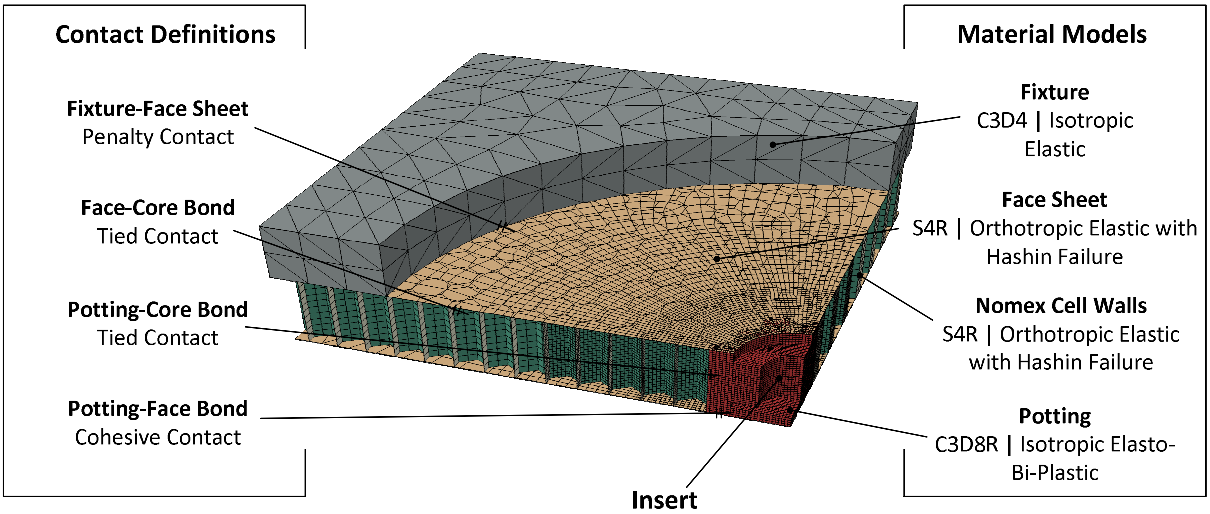

Figure 7: Summary of model setup of the sandwich insert pull-out test

4 RESULTS

For each of the here considered configurations 21 specimen have been prepared and tested, thus allowing conclusion regarding the statistical scatter and uncertainty of the results. The test results are plotted in Figure 8 as force-displacement curves in the background of the simulation results. Each plot contains the scatter of the test results, while one selected test curve is highlighted as reference curve, indicating the average of the test results. The test results have not been truly averaged, in order to ensure that the reference curve has the same characteristic curve progression as the actual test results. It can be seen that all three configurations have a similar absolute strength, with the 10.5mm panel being slightly weaker. Regarding the maximum strength, the scatter is about 15% for the 10 and 26mm panel, while the middle panel has an increased scatter of about 20%. When also considering the failure displacement, the 19mm panel exhibits the highest scatter over all.

Regarding the numerical results, it can be noted that the simulation is capable of representing all implemented failure mechanisms, thus leading to the characteristic failure curve as in Figure 3b.

5 DISCUSSION

Before the simulation results in comparison to the experimental results are discussed, a brief review of the experimental is given.

The test results indicate, that the initial stiffness (phase ①) increases with the core height, while the stiffness in phase ③ seems to decrease with increasing core height. In addition, phase ③ is gradually shorter with increasing core height, leading to decreasing failure displacement. As indicated in the problem analysis, it is assumed that this behavior can be attributed to the debonding of face sheet and honeycomb core. This debonding gradually increases from *no debonding* in the 10mm panel, to *partial debonding* in the 19mm panel and lastly to *full debonding* in the area surrounding the insert in the 26mm panel. All tested configurations exhibit significant scatter of the test results. The 19mm panel might have the highest scatter due to the fact that, this panel is in the transition zone where the face-core debond becomes dominant for the failure of the joint. Some of the tested 19mm specimens do not show significant debonding while others exhibit considerable face-core debonding, thus leading to high scatter. The generally high scatter of sandwich insert pull-out tests is reported by previous studies as well. Raghu et al. [10] attribute this largely to the discrete nature of the core, leading to varying potting shapes, radii and local cell wall properties.

The simulation predicts a gradually increasing absolute strength with increasing core height, while the failure displacement decreases with the core height. When comparing simulation and experiment it can be observed that the virtual testrig predicts the linear behavior as well as the first failure of local shear buckling of the cell walls for all configurations accurately. As failure progresses, the simulation becomes increasingly inaccurate for the configurations with increased core height (19mm and 26mm), while the 10mm panel is predicted well by the simulation model for the entire progressive failure curve. This is expected due to the neglect of the core-face debond, which becomes increasingly dominant with increasing core height. The prediction of the second configuration with 19mm panel can be considered reasonable, as the face-core debond is not as pronounced yet. Here the joint strength is overestimated by about 25%. This changes in case of the last configuration with 26mm panel, where the joint strength is overestimated by 60%, due to the fully pronounced face-core debonding, which is neglected in the model.

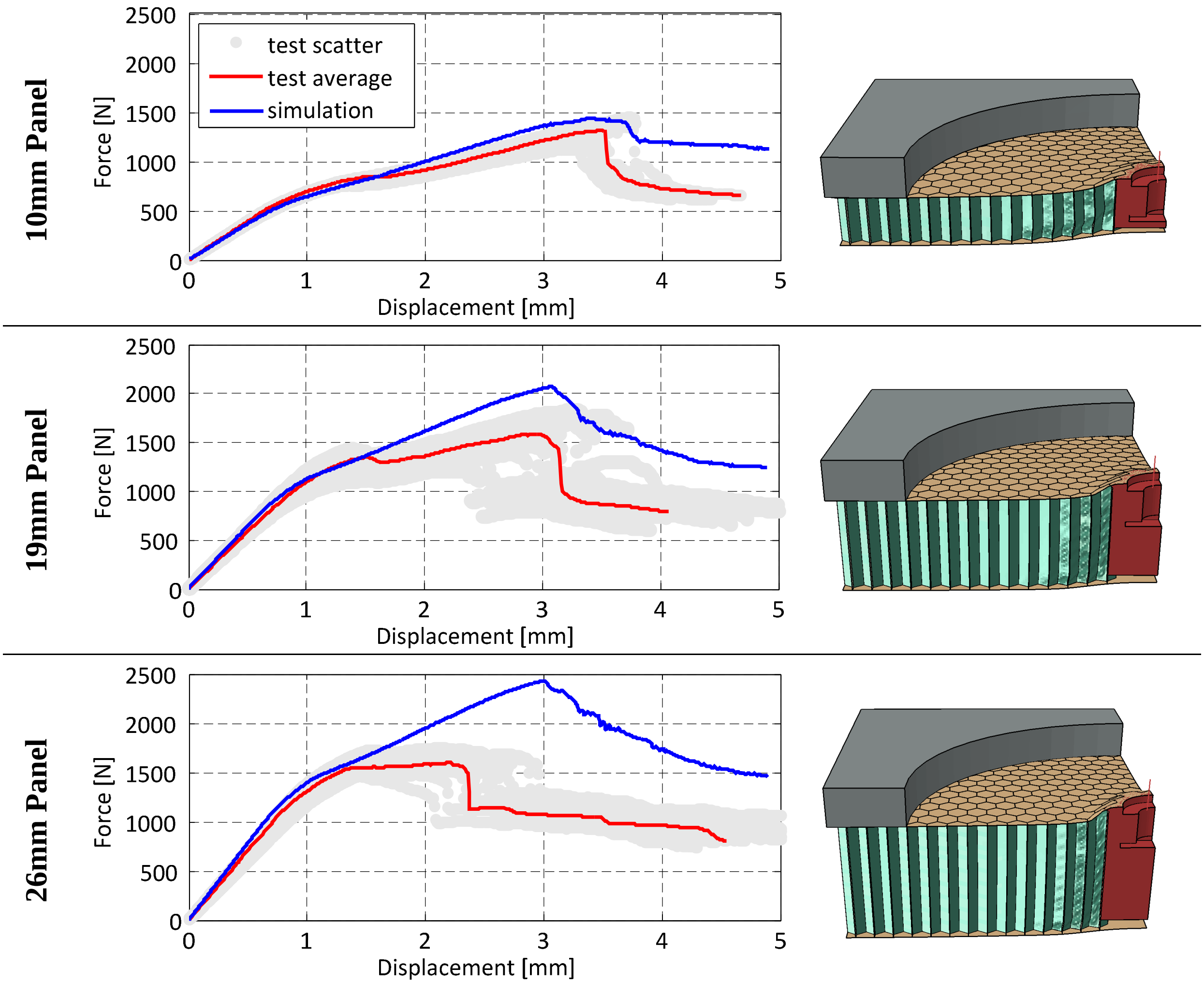

Figure 8 Comparison of simulation and experimental results for all three tested sandwich panel insert configurations

6 CONCLUSION

The present contribution describes the development of a virtual test rig for the pullout of sandwich fasteners such as inserts. The developed model is capable of considering multiple failure mechanisms, thus leading to the characteristic force-displacement curve as given by tests. However, the face-to-core bond failure is not yet implemented. Since it is this failure that dominates the global failure for the two thicker panels, the model overestimates the failure load of the 19mm and 26mm panel. The next step is the implementation of the face-to-core debonding modeled through an additional adhesive contact between core and face, followed by the consideration of the shear-out load case as well as different insert geometries.

ACKNOWLEDGEMENTS

The authors wish to thank the company DIEHL Aerosystems for providing the experimental results as well as for the fruitful discussions on the topic. The research was funded by the German Federal Ministry for Economic Affairs and Energy.

REFERENCES

[1] D. Zenkert, *The handbook of sandwich construction*, EMAS Publishing, 1997.

[2] T. Bitzer, *Honeycomb technology*, London: Chapman and Hall, 1997.

[3] K.-I. Song, J.-Y. Choi, J.-H. Kweon, J.-H. Choi and K.-S. Kim, An experimental study of the insert joint strength of composite sandwich structures, *Composite Structures,* **86**, 2008, 107-113.

[4] B. J. Kim and D. G. Lee, Characteristics of joining inserts for composite sandwich panels, *Composite Structures*, **86**, 2008, pp. 55-60.

[5] G. Demelio, K. Genovese and C. Pappalettere, An experimental investigation of static and fatigue behaviour of sandwich composite panels joined by fasteners, *Composites Part B,* **32**, 2001, pp. 299–308.

[6] O. T. Thomsen and W. Rits, Analysis and design of sandwich plates with inserts - a high-order sandwich plate theory approach, *Composites Part B,* **29B**, 1998, pp. 795–807

[7] O.T. Thomsen. Sandwich plates with 'through-the-thickness' and 'fully potted' inserts, evaluation of differences in structural performance, *Composite structures*, **40**, 1998, pp.159 –174.

[8] P. H. Bull and O. T. Thomsen, Development of a Design Tool for Initial Analysis of Inserts in Sandwich Structures, *Proceedings of the 8th International Conference on Sandwich Structures (Ed. A. J. M. Ferreira), Porto, Portugal,* 2008, pp.1036-1047

[9] D. Smith and B. Banerjee, Reliability of inserts in sandwich composite panels, *Composite Structures*, **94**, 2008, pp. 820-829.

[10] N. Raghu, M. Battley and T. Southward, Strength variability of inserts in sandwich panels, *Proceedings of the 8th International Conference on Sandwich Structures (Ed. A. J. M. Ferreira), Porto, Portugal,* 2008, pp.558-569.

[11] N.G. Tsouvalis and M.J. Kollarini. Parametric study of stress concentrations caused by inserts in sandwich panels. *Proceedings to the 12th European Conference on Composite Materials*, Biarritz; 29 August – 1 September 2006.

[12] E. Bozhevolnaya, A. Lyckegaard, O. T. Thomsen and V. Skvortsov, Local effects in the vicinity of inserts in sandwich panels, *Composites: Part B 35,* **35,** 2004, 619-627.

[13] P. Bunyawanichakul, B. Castanie and J.-J. Barrau, Experimental and Numerical Analysis of Inserts in Sandwich Structures. *Appl. Compos. Mat.,* **12**, 2005 pp. 177-191

[14] P. Bunyawanichakul, B. Castanie and J. -J. Barrau, Non-linear finite element analysis of inserts in composite sandwich structures, *Composites Part B,* **39**, 2008, pp. 1077–1092.

[15] K.-H. Nguyen, Y.-B. Park, J.-H. Kweon and J.-H. Choi, Failure behaviour of foam-based sandwich joints under pull-out testing, *Composite Structures*, **94**, 2012, pp. 617-624.

[16] S. Heimbs and M. Pein, Failure behaviour of honeycomb sandwich corner joints and inserts. *Composite Structures*, **89**, 2009, pp. 575-588.

[17] S. Abrate, B. Castanié and Y.D.S. Yapa, *Dynamic Failure of Composite and Sandwich Structures*, in Solid mechanics and its applications Vol. 192, Springer, Dordrecht, 2013.

[18] R. Roy, K.H. Nguyen; Y.B. Park, J.H. Kweon and J.H. Choi, Testing and modeling of Nomex™ honeycomb sandwich Panels with bolt insert. *Composites Part B*, **56**, 2014, pp. 762–769.

[19] G. Bianchi; G. S. Aglietti and G. Richardson, Static Performance of Hot Bonded and Cold Bonded Inserts in Honeycomb Panels, *J. of Sandwich Struct. and Mat.*, **12**, 2010, pp. 59-82.

[20] A. Boudjemai, A. Mankour, H. Salem, R. Amri, R. Hocine and B. Chouchaoui, Inserts thermal coupling analysis in hexagonal honeycomb plates used for satellite structural design, *Applied Thermal Engineering*, **67**, 2014, pp. 352-361.

[21] Shur Lok Company, Fasteners for Sandwich Structure Catalog, Irvine, CA (http://www.shur-lok.com/product_dls/Sandwich_Structure.pdf)

[22] Composite Materials Handbook. Volume 3: Polymer Matrix Composites. Materials Usage, Design and Analysis (MIL-HDBK-17-3F), Washington DC, Department of Defense, 2002.

[23] Insert Design Handbook, ESA PSS-03-1202, European Space Agency, Paris; 1987.

[24] R. Seemann and D. Krause, Numerical Modelling of Nomex Honeyomb Cores for Detailed Analyses of Sandwich Panel Joints. *Proceed. of the 11th World Congress on Computational Mechanics (WCCM XI) (Eds. E. Oñate, J. Oliver and A. Huerta)*, Barcelona, ES, 2014, Paper 627

[25] R. Seemann and D. Krause, Entwicklung von virtuellen Testverfahren für Sandwichstrukturen, *Beiträge zum 25. DfX-Symposium*, Bamberg, Germany, 2014, pp. 65-76 (in German).

[26] B.N. Cox, M. Spearing and D.R. Mumm: Practical Challenges in Formulating Virtual Tests for Structural Composites, *Computational Methods in Applied Sciences*, **10**, 2008, pp 57-75.