SAND-MESH^{PLUS} – A PARAMETER CONTROLLED FINITE ELEMENT PRE-PROCESSOR FOR COMPOSITE SANDWICH STRUCTURES

M. Streubig, K. Wolf
Institute of Aerospace Engineering
Technische Universität Dresden
01062 Dresden, Germany

1 ABSTRACT

The parameter controlled finite element pre-processor *SAND-MESH^{plus}* is presented which has been developed at the Institute of Aerospace Engineering of TU Dresden. This code enables users to generate finite element models of composite sandwich structures with various types of core configurations. Particularly, models for the simulation of mechanical test methods (impact, compression-after-impact, tension, compression, bending etc.) are provided. The pre-processor supports several types of simulation problems such as static strength, non-linear buckling or impact analyses by generating not only appropriate finite element meshes but also the matching boundary and loading conditions.

2 INTRODUCTION

Driven by stringent weight saving requirements the composite sandwich construction has evolved as one of the promising structural design concepts for load-carrying components of advanced aeroplanes and helicopters. Particularly, sandwich using laminated composites as face sheets and light weight core materials such as foam or honeycomb is increasingly used due to features such as high strength-to-weight and stiffness-to-weight ratios as well as excellent fatigue properties. While offering unique advantages, sandwich is also prone to a range of defects and damages. Due to the thin brittle skins and the weak core material CFRP sandwich structures are particularly susceptible to impact loading which may accidentally occur during assembly or operation of aircraft. Since these damages may have detrimental effects on the load carrying capability, they have to be considered in the damage tolerant design of aircraft structures. For that purpose it is necessary to determine the extent of damage in sandwich structures, resulting from impact events such as tool drop or thrown up debris. Also, knowledge of the residual strength of the damaged components is required. Up to now these data are mainly determined by experimental investigations. Since the used test procedures are rather costly and time consuming, there is a clear need to supplement them by reliable numerical simulation tools. Usually, finite element methods are employed for this task.

For a reliable simulation of the mechanical behaviour of sandwich under impact loading as well as the numerical analysis of the failure of damaged and un-damaged sandwich very detailed finite element models have to be applied. This is particularly true for the very complex topology of open core materials such as honeycombs or corrugated sheets. Also the various boundary conditions of different test set-ups have to be appropriately considered in these models. The generation of detailed finite element meshes for this kind of sandwich structures is a very time-consuming effort. In order to solve this task more efficiently the pre-processor *SAND-MESH^{plus}* has been developed at the Institute of Aerospace Engineering of TU Dresden. The application of this tool helps to reduce the testing effort as well as the development time of composite sandwich structures. In the following sections a more detailed description of the code will be presented.

3 GENERAL FEATURES

SAND-MESH^{plus} is a parameter controlled finite element pre-processor code that has been tailor-made for composite sandwich structures. It is mainly based on the *ANSYS* Parametric Design Language APDL [1].

Originally, *SAND-MESH^{plus}* had been developed to generate finite element meshes for impact and compression after impact simulations of sandwich structures. Since the code has a modular structure, new sandwich geometries as well as load conditions can be implemented easily. Therefore, models required for the simulation of basic sandwich test procedures have been added. Typical examples are compression, tension and bending tests on laminates, core materials and sandwich configurations.

One of the essential features of the programme is the ability to deal with different core materials. Currently, the five basic core types *foam*, *honeycombs, corrugated sheets, pin-reinforced foam* and *truss core* (Fig. 1 to 5) are supported. Additionally, different core materials can be stacked to form layered sandwich structures. Optional, interface layers may be added between the cores (Fig. 6). Plane as well as curved sandwich structures can be modelled with all core types available (Fig. 7). Also, imperfections such as deviations from the nominal sandwich geometry as well as varying material properties can be considered. The dimensions and positions of all structural parts can be selected separately. This permits for example to rotate core structures by arbitrary angles. Thus, sandwich structures with almost arbitrary topologies can be generated, requiring very little input data.

For the investigation of damaged structures different defects such as skin dents, skin cracks and crushed cores can be modelled. Also, boundary and loading conditions required for the simulation of impact, residual strength and material properties tests are available.

The modular structure of *SAND-MESH^{plus}* makes it easy to implement new core structures or boundary conditions with minor effort. The basic model data are edited in standard input-files while the output-files provide interfaces to standard finite element simulation tools such as *LS-DYNA3D* or *ANSYS* or in-house software.

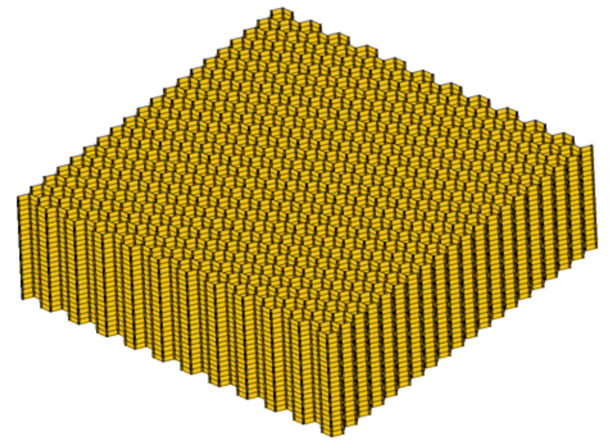

FIG. 1: Honeycomb core

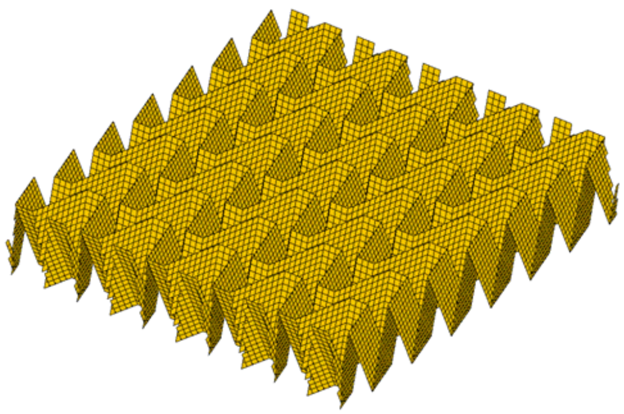

FIG. 2: Corrugated sheet core

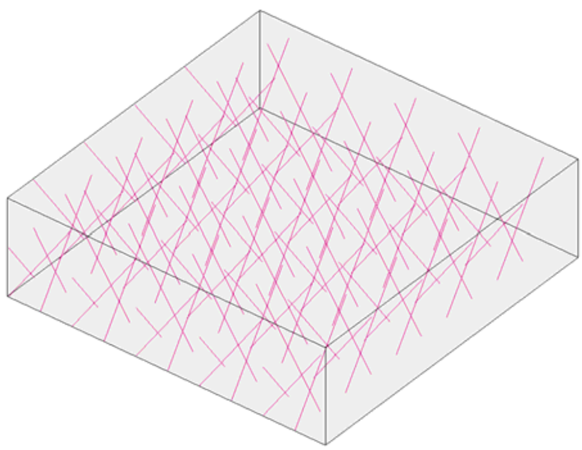

FIG. 3: Pin-reinforced foam core

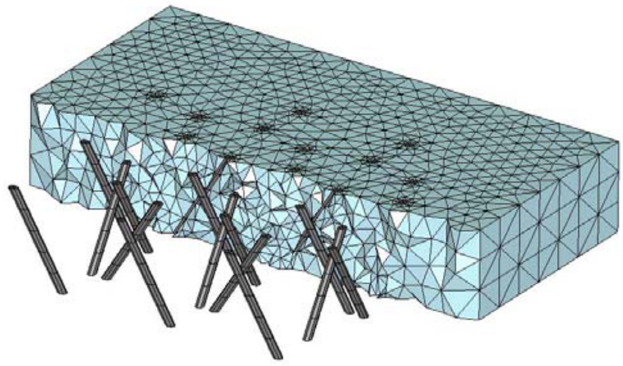

FIG. 4: Pin-reinforced foam core (detailed discretization)

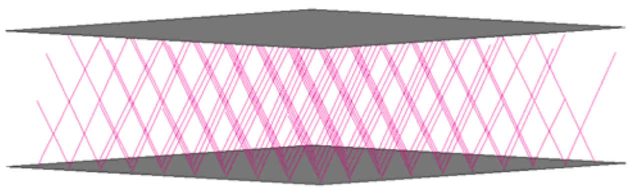

FIG. 5: Truss core

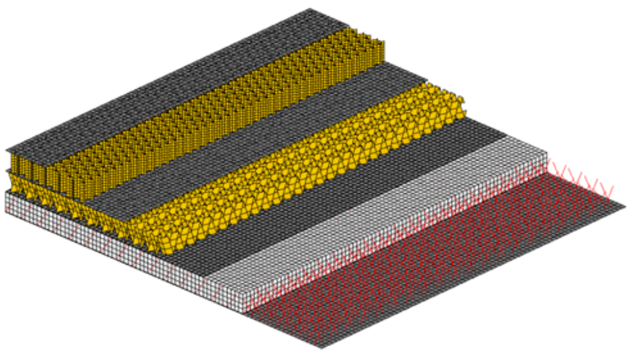

FIG. 6: Layered sandwich with interface sheets

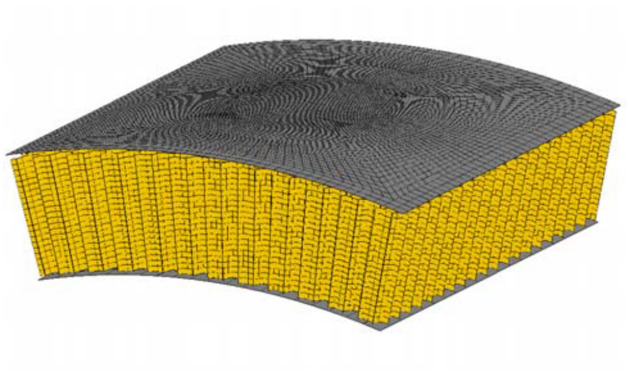

FIG. 7: Curved sandwich

4 FACE SHEETS

The finite element discretization of a structure depends mainly on the structural phenomena that have to be dealt with by the simulation model (buckling, delamination etc.) as well as on the type of loading. Thus, different models have to be used for the face sheets. This is particularly true, if the laminates contain damages. In order to deal with damaged structures *SAND-MESH^{plus}* has the capability to consider defect types, which may have detrimental effects on the load carrying capability. Details on this are given in the following section.

4.1 Delamination

Bending as well as impact loads cause transverse shear stresses in the laminated face sheets. These stresses may result in delaminated plies and a reduced flexural rigidity. This affects the deformation of the whole structure and, in case of impact loading, the core damage and finally the residual strength. Therefore it is important to include delaminations in the simulation model.

In *SAND-MESH^{plus}* a laminated face sheet is modelled by using separate layers of shell or solid elements (Fig. 8) for each ply. These layers are linked by appropriate contact conditions to form the laminate. This permits to consider delaminations between the plies.

A simpler approach can be chosen, if delamination is not a basic failure mode (very thin face sheets etc.). In this case a laminated face sheet is modelled by only one layer of elements. These layered shell elements take into account the laminate lay-up schemes of the faces through special integration rules.

4.2 Imperfections

For the residual strength of impact damaged sandwich structures failure due to instability plays a key role. In order to simulate the buckling behaviour of the face sheets the finite element model should be able to consider imperfections. This is particularly true for explicit finite element codes. *SAND-MESH^{plus}* generates imperfections through stochastic node displacements. The maximum amplitude can be chosen depending on the problem.

4.3 Impact damage

Impact loads on monolithic composite plates or sandwich with composite face sheets often cause complex damage states. Depending on the size and severity this damage may have a significant effect on the mechanical behaviour of the structure. This is particularly true in case of in-plane compression or shear loading. Therefore, the damage topology as well as the resulting material degradation has to be included in the simulation models.

In order to get a detailed damage map through impact simulation the element meshes have to be refined in the vicinity of the impact center. This feature is also supported by *SAND-MESH^{plus}* which permits to generate locally different types of element grids. The mesh density is parameter controlled.

4.3.1 Indentation

The damaged area in the vicinity of an impact is modelled by an indentation with an elliptic plan view. The cross-section shape of the dent is approximated by a cosine function. Size, position and direction of the ellipse can be adjusted by parameters (Fig. 9). A separate node set is defined in this area which can be used to simulate debonding between face sheet laminate and core. This failure mode is particularly observed with foam core sandwich. Also the properties of the face sheet material can be reduced in the impact area to consider local damages such as broken fibres und cracked matrix material.

At high energy levels the impactor may penetrate the face sheet completely, leaving a hole. This kind of damage also can be modelled by *SAND-MESH^{plus}*.

4.3.2 Hidden damage

Damage may also occur outside of the indentation, e. g. delaminations, micro cracks, fiber and matrix cracks. Although these damages are barely or non visual, they contribute to a reduction of stiffness and strength and may decrease the buckling stability. In order to include these effects a second elliptic damage area can be defined outside of the indentation. This area can be rotated to fit the real damage size and shape. Delaminations are modelled according to chapter 4.1, whereas other defects are considered through reduced material properties.

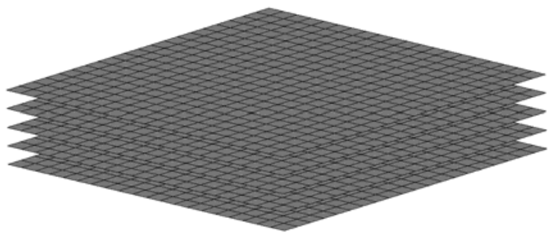

FIG. 8: Composite face sheet with discretized plies

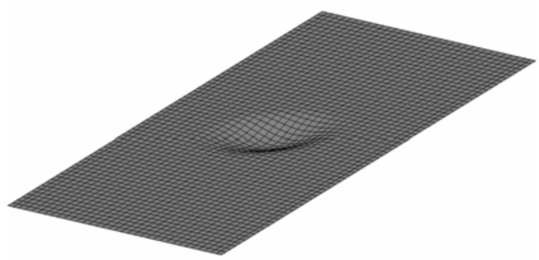

FIG. 9: Composite face sheet with impact dent

5 FOAM CORE

Foam cores made of materials such as ROHACELL, AIREX etc. are treated as macroscopic homogeneous continua. They are discretized by applying solid elements.

5.1 Impact damage

Impact loads on cellular foams cause severe damages on the cellular level due to collapsed cell walls. This may lead to the complete destruction of the microstructure. The affected core region is no longer able to carry any loads. Therefore, there is no need to model that volume and the impact damage is considered only as a dent in the core. Additionally, a separate node set is used on the contact surface between face sheet and core which permits to model debonding effects, since this kind of failure is often observed with foam core sandwich. Below the dent a separate core volume with reduced material properties can be defined in order to take less severe damage into account.

6 TRUSS CORE

The implemented truss core configuration (Fig. 5) is similar to that used in pin-reinforced foams (Fig. 3). The trusses are made of composite pins that are inclined and arranged in rows. In each row the pins have the same orientation angle and distance. The angle, the diameter and the material of the pins as well as the distance between two pins are variable. The pins are modelled by using finite beam elements.

6.1 Impact damage

Impact loads on core structures consisting of composite pins cause primarily damage due to stability failure. Similar to cellular foams the material in the affected region is severely crushed. Since the pins are not able to carry any further loads, this region of the core is approximated by a dent. A separate node set is used to consider face sheet debonding.

7 PIN-REINFORCED FOAM CORE

Pin-reinforced foam cores are a combination of foam and truss cores. The finite element models of both constituents have independent meshes. That permits the use of solid-elements and a coarser mesh for the foam. The interaction between pins and foam is provided by special coupling algorithms which depend on the simulation software applied.

A more complex way to discretize pin-reinforced foam cores is to use a detailed finite element model, i. e. solid elements for the pins and corresponding holes in the foam model (see Fig. 4). Due to the highly complex geometry very small tetrahedron elements are required for the foam model which increases the computational effort considerably. Appropriate contact conditions have to be defined to link pins and foam.

The damage modes which can be considered are similar to those described in section 5 and 6.

8 HONEYCOMB CORE

The cell walls of honeycomb cores are modelled using shell elements. The mesh density can be adjusted to the investigated problem by parameters.

8.1 Imperfection

Honeycomb structures often show a number of manufacturing defects. Examples are deviations from the exact hexagonal shape or s-shaped walls. These defects affect the structural stiffness of honeycomb cores, particularly under compression loads. Therefore, *SAND-MESH*^{plus} offers options to include these defect modes in the core model. For this task algorithms are employed which are similar to those used for the face sheets (see chapter 4.2).

8.2 Location of impact center

The damage size caused by impacts depends among others on the location of the impact center. An impactor hitting directly a honeycomb wall causes other damage modes than an impactor that hits the empty space between the cell walls. Furthermore, as a result of the production process cell walls have two different thicknesses and hence show a different mechanical behaviour. In order to take these effects into account *SAND-MESH*^{plus} permits to locate the impact center at an arbitrary point relative to the honeycomb geometry.

8.3 Impact damage

Compression loads on honeycomb structures resulting from high impact energy levels lead to stability failures of the honeycomb walls and leave a dent. This dent is modelled by *SAND-MESH*^{plus} in combination with the face sheet damage (see section 4). Also a separate node set is defined in order to implement face sheet debonding in the affected area if required. Some of the failed honeycomb cells may have a residual strength and stiffness. Thus, similar to the other core structures, below the dent a separate volume with reduced material properties can be defined (see Fig. 10).

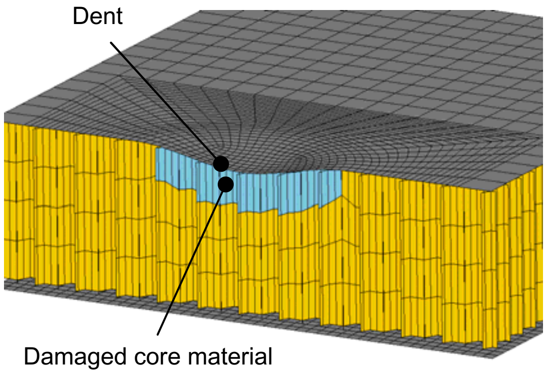

FIG. 10: Impact damage

9 CORRUGATED SHEET CORE

SAND-MESH^{plus} offers two different ways to prepare finite element models of corrugated sheet cores. Either the core geometry is generated by the code based on basic parameters. Or in the second approach geometric input data are used which are directly provided by the manufacturers of corrugated sheet materials. This ensures that *SAND-MESH*^{plus} provides finite element models of core configurations which can be actually produced.

9.1 Location of impact center

As with honeycomb cores the location of the impact center affects the damage state in the face sheets as well as in the cores. Therefore, *SAND-MESH*^{plus} offers to locate the impact center at an arbitrary point relative to the wall geometry.

9.2 Impact damage

Impact damages are modelled similar to honeycomb cores as described in chapter 8.2.

10 BOUNDARY CONDITIONS

SAND-MESH^{plus} provides not only the finite element meshes but also the boundary conditions which are required for a range of experimental procedures such as tension, compression, bending, impact and residual strength tests.

10.1 Tension / compression tests

Loading and supports are defined by using separate node sets and corresponding constraints. Displacement as well as load controlled test procedures with arbitrary crosshead rates are supported by the pre-processor.

10.2 3-point-bending tests

Additional to the test specimens *SAND-MESH*^{plus} provides finite element models of the supports and the loading nose as well as the required contact conditions (Fig. 11).

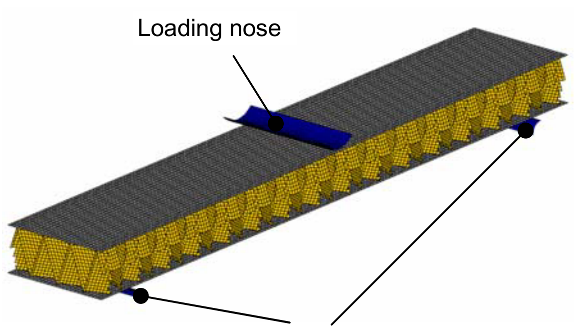

FIG. 11: 3-point-bending test of a sandwich beam

10.3 Impact

Additional to the finite element model of the sandwich structure *SAND-MESH*^{plus} provides a model of the object which hits this structure (Fig. 12). The contact conditions that are required by the simulation code are also supplied. Furthermore the coordinates of the impact location as well as the impact energy can be specified.

The structural response to an impact load depends strongly on the boundary conditions applied. A fully supported sandwich absorbs the impact energy by local elastic and plastic deformation as well as local damage of the upper face sheet and the core. A sandwich which is supported only on the edges shows some elastic behaviour. In this case only a part of the impact energy results in local damages in the face sheet and the core. Therefore, *SAND-MESH*^{plus} offers procedures to create appropriate boundary conditions for both cases in the form of constraints or finite element models of the supports and corresponding contact conditions.

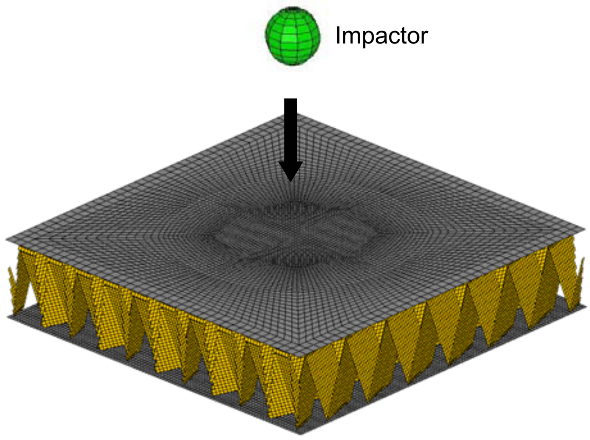

FIG. 12: Simulation of an impact test

10.4 Residual compression strength

Performing experiments to determine the compression after impact strength (CAI) of sandwich structures requires some preparation of the test specimens. Normally, the compression load is applied through special load introduction elements which replace the core material at both ends of the specimens. *SAND-MESH*^{plus} generates finite element models of the specimens without the core material in the load introduction region (Fig. 13). Then the loading conditions of the test are modelled using appropriate constraints which are applied to the element nodes in this area.

A serious problem related with compression testing is structural failure due to instability. Therefore, often anti-buckling guides are used to prevent global buckling of the specimen. These devices are modelled in *SAND-MESH*^{plus} by special contact conditions (Fig. 13).

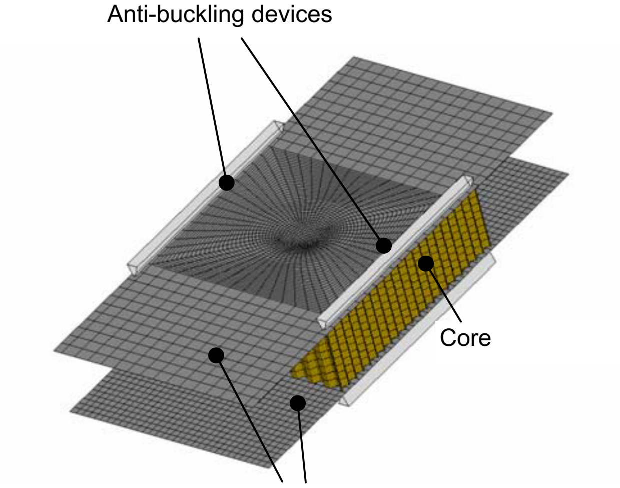

FIG. 13: CAI specimen with anti-buckling devices

11 CONCLUSION

The presented parameter controlled pre-processor software *SAND-MESH*^{plus} offers a variety of features to generate finite element models for composite sandwich structures with various core types. Particularly, models for the simulation of test procedures (tension, compression, bending, impact and residual strength) are provided. The model data such as element and node numbers, node coordinates, material laws, contact and boundary conditions are compiled into input files for commercial finite element simulation tools. Currently, interfaces for *ANSYS* and *LS-DYNA3D* as well as for in-house software are available.

SAND-MESH^{plus} is particularly well suited for numerical studies on the effect of complex core configurations on the mechanical performance of sandwich structures, since the core topology can be easily varied by a few number of control parameters.

12 REFERENCES

[1] ANSYS Release 10.0 Documentation, ANSYS, Inc.; 2005