A New Feature to Model Shell-Like Structures with Stacked Elements

Tobias Erhart

Dynamore GmbH, Stuttgart, Germany

1 Introduction

A computational method for the numerical simulation of stacked shell-like structures will be presented. These types of plane load-bearing components possess a thickness which is small compared to its other (in-plane) dimensions and physical properties vary in thickness direction in distinct layers. Such materials are made from two or more constituent materials with different physical or chemical properties, that when combined, produce a material with characteristics different from the individual components. The individual components remain separate and distinct within the finished structure. The new combined material is often stronger, lighter or less expensive when compared to traditional raw materials. Examples would be sandwich plate systems (SPS), composite laminates, plywood plates or laminated glass as shown in Figure 1.

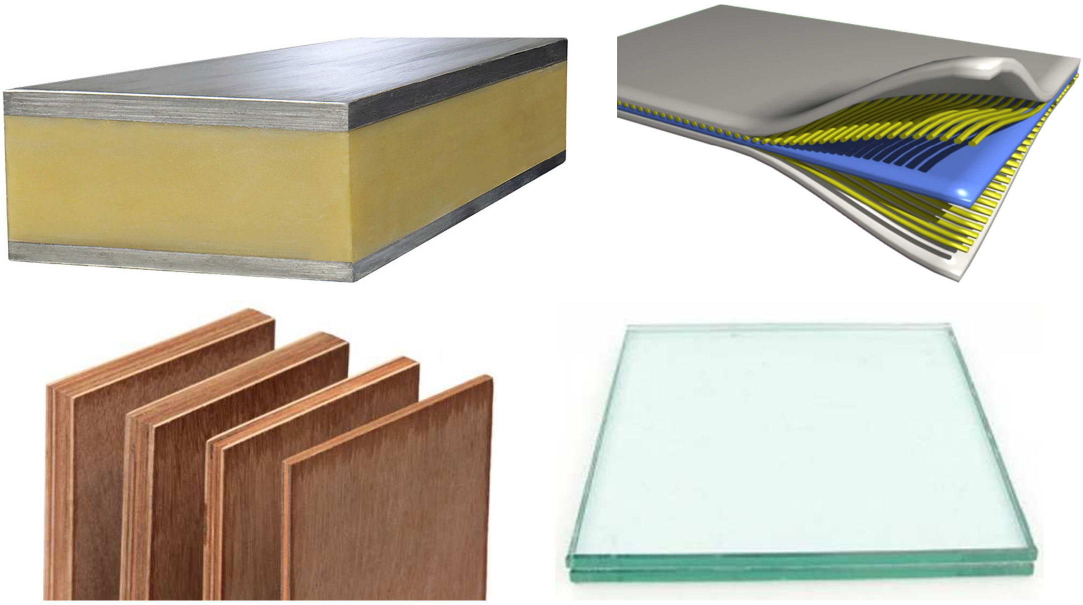

Fig.1: Typical layered shell-like structures: sandwich plate system (top left, picture credit: Wikipedia), composite laminate (top right, Wikipedia), plywood (bottom left, kalingapanels.com), laminated glass (bottom right, seekglass.com).

Such stacked two-dimensional structures could simply be modelled through shell or thick shell elements with a pre-defined number of integration points over the thickness. This is already possible in LS-DYNA with keywords like *INTEGRATION_SHELL or *PART_COMPOSITE, where each through-thickness integration point is assigned to a ply [1]. This allows for different definitions of material properties, thickness and fiber orientation for each layer. But the kinematic description is still “trapped” in the shell or thick shell element formulation, i.e. no real discontinuous deformations over the thickness are possible even with relief efforts such as the usage of lamination theory corrections. Therefore, physical effects like large transverse shear in individuals plies, delamination, or even failure of intermediate layers cannot really be captured with a pure shell element discretization.

A favourable approach for this kind of applications is to discretize them by an arbitrary layup sequence of shell and solid elements over the thickness. This gives much more flexibility to describe discontinuous deformations of distinct layers in a realistic fashion, however it must be mentioned that this comes with a higher computational effort. The decision if a physical ply is discretized by shell or solid elements depends on the individual thickness and other mechanical properties. For instance, thin layers will probably most often be modeled by shell elements, whereas solid elements better represent thicker layers. Another important property could be the transverse shear stiffness. If a ply is relatively soft in that mode, then solid elements might capture large shear deformations better than shell elements. In the end, it is up to the computational mechanics engineer to choose the best way to model his or her composite material at hand, see e.g. [2].

It is already possible to set up such models with stacked shell and solid elements in common FE preprocessor tools, e.g. LS-PrePost offers all necessary meshing capabilities for this purpose. But sometimes it would be more convenient only to provide the essential information to the solver (LS-DYNA) and then let it do the rest. The proposed method only needs two initial items: Starting from a pure shell element mesh (i) and a user-defined layup sequence of elements in thickness direction given in new keyword *PART_STACKED_ELEMENTS (ii), new nodes and elements are automatically generated. Extrusion methods are used to determine the new node locations, i.e. the description of the out-of-plane geometry. Each layer gets its own predefined properties such as individual thickness, material characteristic and element type. With the proposed manner to describe the stacking sequence alltogether in one keyword, the solver knows better about the coherence of those layers as it would know from an upstream graphical pre-processor software that produces separate parts in the input file. This knowledge could then be used to compute resultant output values such as thickness change of the whole stack, not only individual plies.

The paper is organized as follows. In the next section, it is explained which information has to be provided by the user to apply the new feature. Then, the internal generation of the stacked Finite Element mesh based on that information will be explained in detail in chapter 3. Afterwards, some numerical examples are shown in section 4 to illustrate the new approach. The paper finally concludes with a short summary.

2 Definition of Geometry and Layup Sequence

As an input from the user, a shell element mesh and a specification of the layup sequence is needed. The shell element mesh (or “reference mesh” in the following) consists of nodes with their coordinates given in a global cartesian system (*NODE) and shell elements defined by a sequence of three or four nodes for each element as usual (*ELEMENT_SHELL). This reference mesh should be located at the virtual middle plane of the actual layered structure as shown in Figure 2.

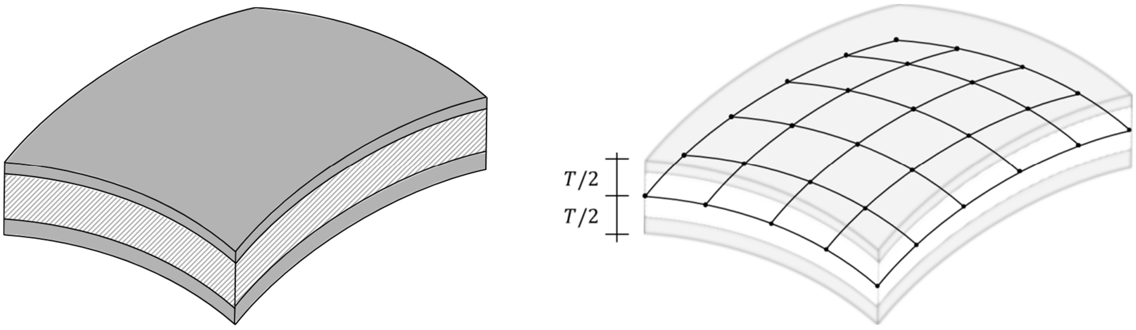

Fig.2: Physical shape of a cut through a layered shell-like structure, in this case two thin outer layers and one thicker inner layer (left) and corresponding shell element mesh (right).

The layup sequence of the composite material is given by the total number of layers n_{lay} and the following properties for each layer:

- Identifier i
- Element type: shell or solid elements (reference to *SECTION_... and *HOURGLASS_...)
- Material type (reference to *MAT_...)
- Layer thickness t_i (total thickness of the whole stack: $T = \sum_{i=1}^{n_{lay}} t_i$)
- Number of elements over thickness (only needed for solid element layers)

In LS-DYNA, this stacking sequence can be defined in new keyword *PART_STACKED_ELEMENTS. The input looks similar to the standard *PART definition, only that several parts are now combined in one place. An example for a three layer sandwich structure (shell-solid-shell) is given through the format in Figure 3.

```
*PART_STACKED_ELEMENTS
$ title
sandwich
$   pidref    numlay
        11         3
$#      pid       sid       mid      hgid      tmid       thk    numsld
       100       200         1         1         0      0.25         0
       101       201         2         0         0      0.60         3
       102       200         1         1         0      0.15         0
*SECTION_SHELL
$      sid    elform      shrf       nip     propt   qr/irid
       200         2     0.833       5.0       1.0       0.0
$       t1        t2        t3        t4      nloc
      0.25      0.25      0.25      0.25
*SECTION_SOLID
$      sid    elform
       201        -1
```

Fig.3: Keyword input format for the definition of a stacked element layup sequence (in this case three layers: shell – 3 solids – shell).

That means that a sandwich structure is supposed to be discretized by shell elements (sid=200) as outer layers with part identifiers 100 and 102 and three solid elements over thickness (sid=201, numsld=3) for the inner layer with part id 101. The reference shell mesh in this case belongs to part 11 (pidref). The thickness of each layer is defined by parameter "thk", which overwrites the thickness values (t1, t2, …) from *SECTION_SHELL. Related materials (mid, tmid) and hourglass types (hgid) are treated as usual and therefore not shown here. Solid element formulation ELFORM = -1 or -2 is recommended for solid elements with bad aspect ratio [1], e.g. flat hexahedrals, which are likely to occur in present situations.

3 Layered Mesh Generation

If all the previous information is provided, the generation of the stacked layer model can commence. It will consist of several layers of nodes as well as shell and/or solid elements and will be called "new mesh" in the following.

First of all, the total number of new nodes and elements to be created has to be determined to assign the amount of storage that is needed for the new mesh. Therefore, an important difference between shell elements and solid elements has to be emphasized first. A shell element is described by its mid-plane nodal coordinates and a thickness value. The actual physical top and bottom surfaces are derived from that, but no real nodes are present there. On the other hand, solid element nodes genuinely describe the real geometry of the discretized body, i.e. corresponding nodes actually lie on the surface. This difference leads to the fact that the combination of shell and solid elements in thickness direction of a lamina system needs a specific strategy. In our approach, two consecutive layers always share the same nodes in between them, i.e. all layers are firmly connected. This condition and the previously described difference between shell and solid elements leads to the necessity that shell and solid elements partly overlap if they follow each other in the stacking sequence. This well-known deficiency can be corrected afterwards by already established strategies like subsequent relocation of the shell mid-surfaces via NLOC on *SECTION_SHELL (only works for first or last layer in this stacked element approach) or appropriate adjustment of the material stiffness for the solid elements. These countermeasures are already available and not further discussed here. Another consequence of the outlined procedure is that shell element layers cannot directly follow each other in through-thickness direction, because they cannot share the same nodes. This situation could be handled by a tied or tiebreak contact in the future, but this is not yet incorporated in the current approach. Up to now, one has to put at least one solid element layer between two shell element layers.

We will continue with the stepwise or “layer-wise” generation of nodes and elements of the new mesh. The nodes of each layer are just a copy of the “reference mesh” nodes, only shifted in the thickness direction. Therefore, the distance of each layer from the “reference mesh” mid-plane and the local direction vectors orthogonal to that plane are needed (see Fig. 4).

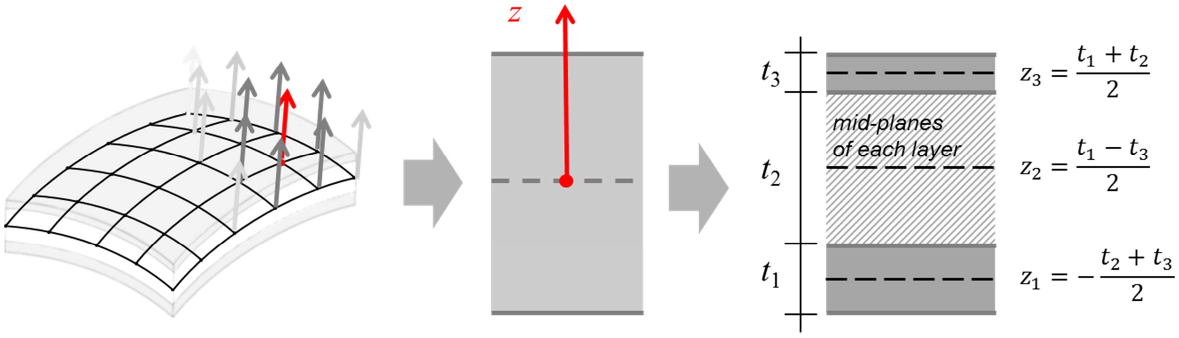
z
t_3
t_2
t_1
mid-planes of each layer
$z_3 = \frac{t_1 + t_2}{2}$
$z_2 = \frac{t_1 - t_3}{2}$
$z_1 = -\frac{t_2 + t_3}{2}$

Fig.4: Nodal orthogonal vectors of the shell mesh are used to obtain a one-dimensional problem: Determination of z-coordinates of mid-planes for each layer (in this case three layers).

For new shell element layers, there is only one distance value per layer, because there is only one mid-plane. For new solid elements, two distances per layer have to be taken into account, namely the lower and upper side distance. There it is always important to know if the solid elements layer is adjacent to a shell elements layer, because in that case the thickness of the solid elements layer has to be increased by half the thickness of that neighboring shell elements layer. All that information can be taken from the user input of layer sequence properties as shown above. The presented way of combining shell and solid elements is one key idea of this approach, therefore the determination of lower and upper side location in thickness direction for a solid elements layer is exemplary shown in Figure 5.

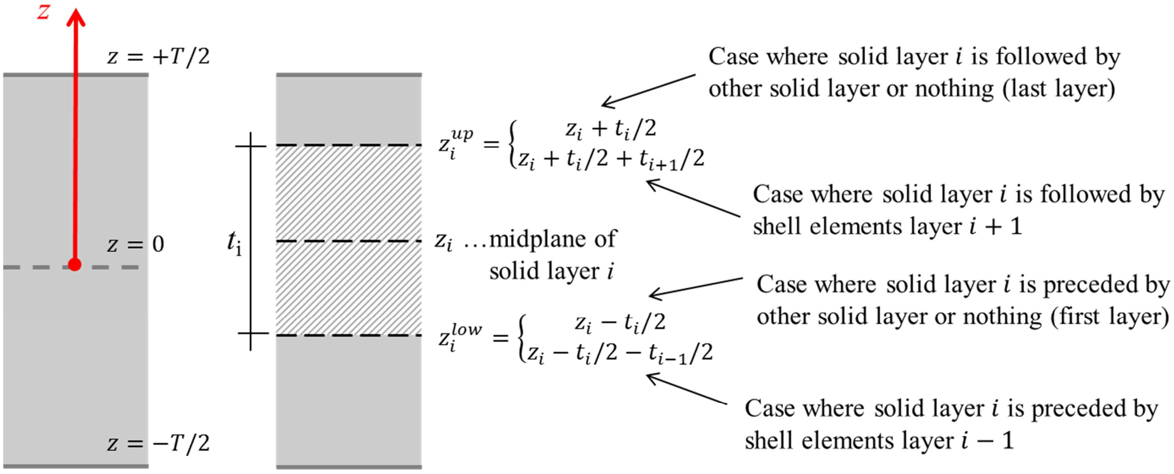
z
$z = +T/2$
$z = 0$
$z = -T/2$
t_i
$z_i^{up} = \begin{cases} z_i + t_i/2 \\ z_i + t_i/2 + t_{i+1}/2 \end{cases}$
z_i ...midplane of solid layer i
$z_i^{low} = \begin{cases} z_i - t_i/2 \\ z_i - t_i/2 - t_{i-1}/2 \end{cases}$
Case where solid layer i is followed by other solid layer or nothing (last layer)
Case where solid layer i is followed by shell elements layer $i + 1$
Case where solid layer i is preceded by other solid layer or nothing (first layer)
Case where solid layer i is preceded by shell elements layer $i - 1$

Fig.5: Determination of lower and upper side distance (z-coordinate) for a solid elements layer.

The required direction vector from Figure 4 needs to be determined for each node of the “reference mesh”. It is calculated by averaging the normal vectors of surrounding elements, see Figure 6. With that approach, even curved geometries of the reference mesh can be treated up to a certain level. If that curvature would be too high, the extrusion procedure could lead to distorted (unusable) elements, but this is a well-known issue that experienced users are already aware of and have possibilities to avoid.

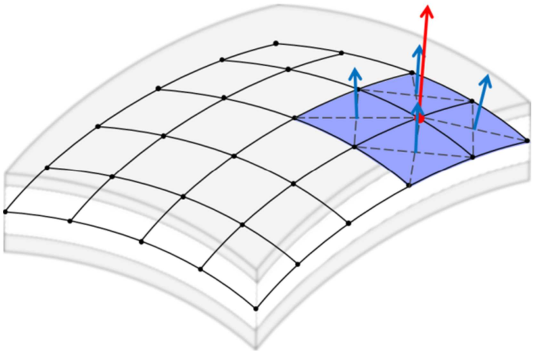

Fig.6: Normal (orthogonal) vector for a node of the reference mesh: normalized average of normal vectors of surrounding elements, where each element normal is determined from the cross product of the two element diagonals.

Now, after collecting all of the following data:

- nodal coordinates of all reference mesh nodes,
- normal direction vectors for all reference mesh nodes,
- element connectivity for reference mesh elements,
- distance values of all nodes for every layer,
- thickness of each layer,
- element type of each layer,
- number of element over thickness of each layer (only needed for solids),

the generation of new nodes and elements can be addressed. Therefore, we start from the bottom layer and then go through the total number of layers up to the top. The new nodes of each shell element layer get their coordinates by multiplying the respective distance with the normalized direction vector. If several solid elements over thickness have to be created for one layer (parameter “numsld” in Figure 3), the corresponding nodes are equally distributed over the height. New elements are generated adopting the in-plane connectivity logic in the reference mesh, see Figure 7. Finally, each layer has to be assigned with its own identifier (ID), material properties, and particular element type.

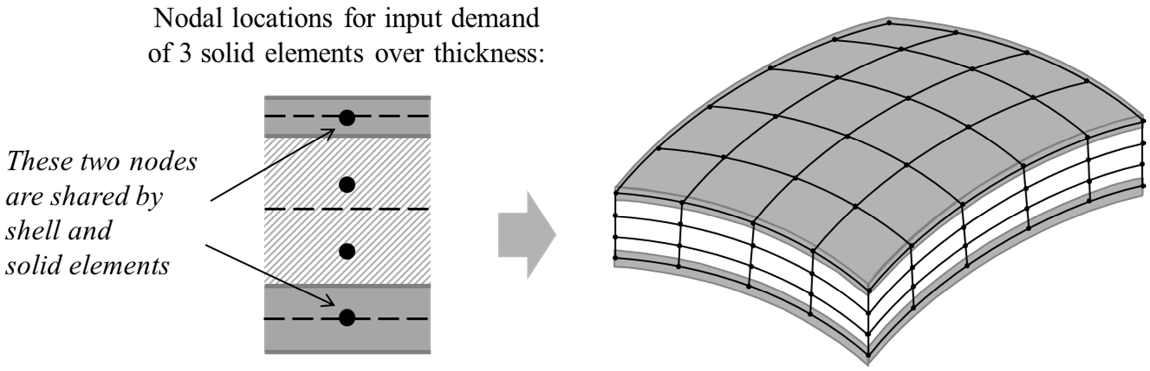
Nodal locations for input demand
of 3 solid elements over thickness:
These two nodes
are shared by
shell and
solid elements

Fig.7: New Finite Element mesh with shell and solid elements (in this case shell elements as outer layers - thickness indicated as grey shading - and three solid elements over thickness for the inner layer).

The novelty of this approach is not only the fact that the Finite Element mesh of the stacked shell-like structure is automatically generated based on a given layup sequence, but also that different element types, namely shells and solids, can be put on top of each other in an arbitrary fashion, see Figure 8.

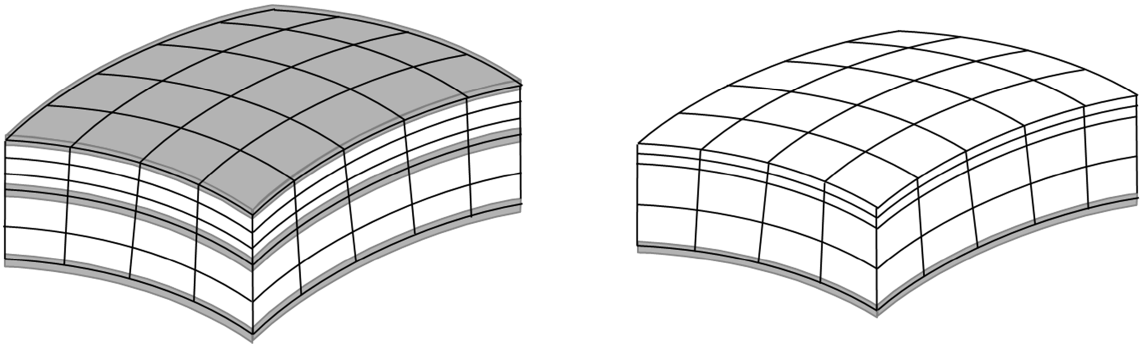

Fig.8: Other examples for possible stacking sequences, e.g. shell – 2 thick solids – shell – 3 thin solids – shell (left), shell – 2 thick solids – 2 thin solids (right).

It should be mentioned that the keyword *PART_STACKED_ELEMENTS can also be used in multi-stage process simulations. That means if *INTERFACE_SPRINGBACK_LSDYNA is applied to get all relevant informations (deformed geometry, stresses, strains, etc.) written into a dynain file at the end of a calculation and *INCLUDE with filename=dynain is used in the following operation step, no new mesh will be created, but just the layup sequence is used. LS-DYNA automatically detects if a reference shell element mesh is present or not.

4 Numerical Examples

In this section, a few possible applications for the presented feature will be shown. After some verification tests in the first subsection, more practical examples will follow to demonstrate the ability of this approach, e.g. to model sandwich structures or phenomena like delamination in composite materials.

4.1 Small Mesh Generation Tests

A few small test cases will be used to demonstrate the capabilities and limits of the presented stacked element meshing approach. For instance, two small subcomponents with mixed element types (quadrilateral and triangular) and curved surfaces are shown in Figure 9. It can be seen that the presented meshing procedure automatically leads to the desired discretization with stacked shell and solid elements. In a second example three different stacking sequences are compared, see Figure 10. These different meshes are obtained by very few modifications in the parameter settings of the new keyword *PART_STACKED_ELEMENTS.

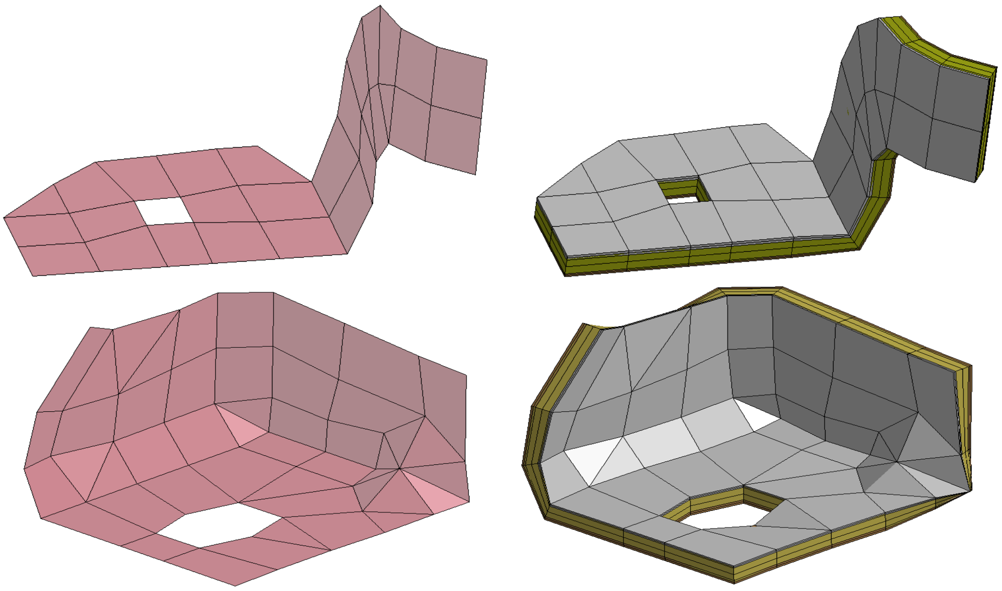

Fig.9: Reference shell element meshes with curved surfaces, mixed element types, and holes (left). Corresponding new meshes with stacked solid and shell elements (right).

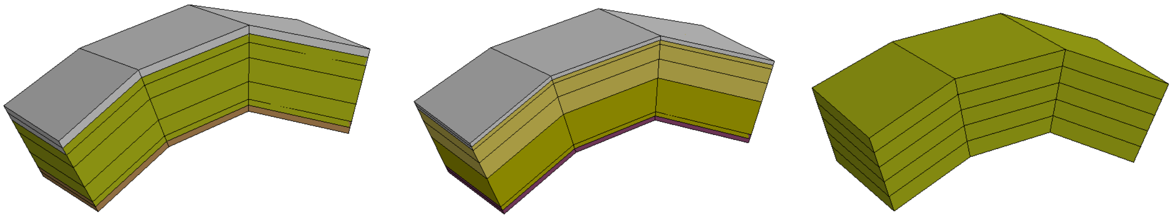

Fig.10: Examples for different stacking sequences (shell elements represented with their thickness): shell–3 solids–shell (left), shell–1 solid–2 solids–shell (middle), just 5 solids (right)

4.2 Forming of a Sandwich Plate System

In this simple cup draw example, a sandwich plate system (SPS) will be used for the workpiece. Such sandwich plate systems are often used in different engineering projects to reduce the weight of applied components while maintaining good stiffness and strength properties [3]. The SPS in this test case is composed of two thin outer steel layers (0.25 mm) and a thicker inner core made of a polyurethane elastomer (0.6 mm). The material models that were used for the steel and the polymer are *MAT_024 (von Mises) and *MAT_PLASTICITY_POLYMER. The corresponding stacked element discretization uses shell elements for the outer layers and solid elements for the inner layer as shown in Figure 11.

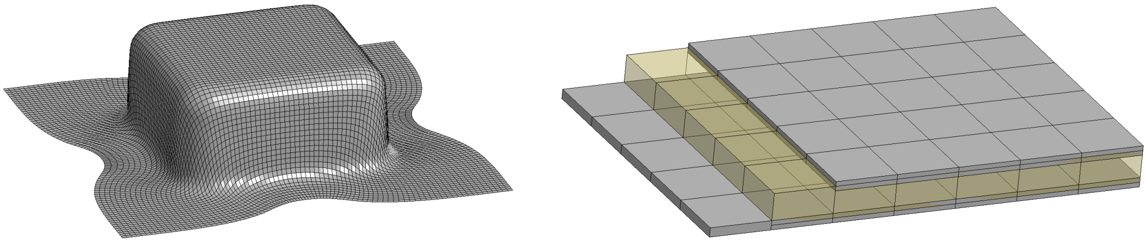

Fig.11: Formed cup (left) and sandwich plate system (right)

With the new feature *PART_STACKED_ELEMENTS, it is quite easy to define the layup sequence for that kind of sandwich structure, e.g. the number of solids in through-thickness direction can be varied just by changing one parameter (numsld). During the drawing process, different loading conditions occur in the workpiece, e.g. compression in thickness direction between binder and die or large transverse shear deformations in the region of the wall. This and the fact that the two combined materials significantly differ in their stiffness properties lead to discontinuous deformations as can be seen in Figure 12. These realistic results with three solid elements over the thickness would not be possible with a pure shell element mesh, even if one would assign different material properties to different integration points through the thickness.

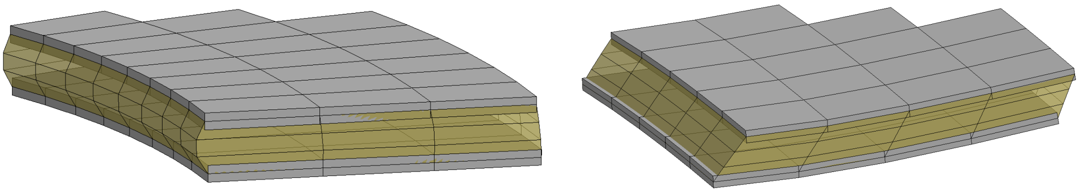

Fig.12: Sections from deformed sandwich plate system: squeezed polymer layer under binder compression (left), large shear deformations in wall region (right).

Next, a comparison between a pure metal sheet with a thickness of 1.1 mm and the sandwich plate with the same total thickness as described above was performed. The weight reduces by a factor of 2 with the second option, which is one main argument for the application of SPS. Nonetheless, the stiffness and strength of the produced component should not suffer. And the good correlation of plastic strain and effective stress distribution indicates the truth content of that statement, see Figures 13 and 14. But it is not the purpose of this study to verify the benefits of SPS furthermore.

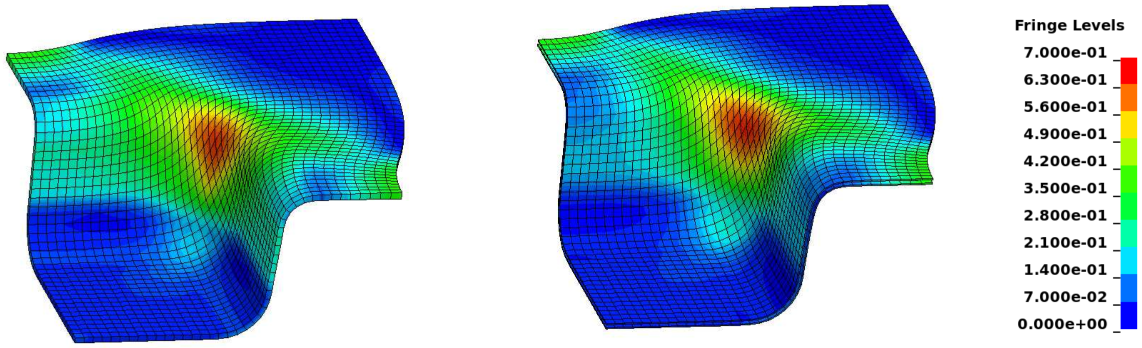

Fig.13: Contours of effective plastic strain for pure metal sheet (left) and sandwich plate (right).

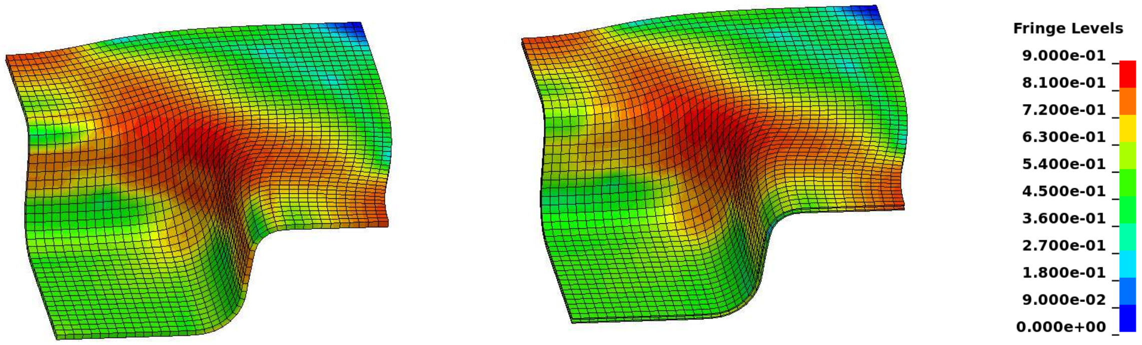

Fig.14: Contours of effective stress (GPa) for pure metal sheet (left) and sandwich plate (right).

For forming operations it would be desirable to check the thickness reduction (possible cracking) or thickness increase (wrinkling) at the end of the simulation. This is still an open task for these stacked element models and not yet implemented in the current environment. An averaging scheme will be necessary to transform the thickness strains of elements lying on top of each other to one value for the thickness change of the whole stack.

4.3 Delamination Under Impact

In this example, a hemispherical impactor hits an arch-shaped composite plate, see Figure 15. The plate is first discretized with shell elements only and all boundary nodes are completely constrained using *BOUNDARY_SPC_SET. With the new feature, a layup sequence with 4 shell elements over thickness with 3 solid elements in between is employed to reflect possible delamination. Therefore, special purpose solid elements, namely cohesive elements (*SECTION_SOLID, ELFORM=20), were used in combination with a material model including damage and failure (*MAT_186). The stacked element model is generated by LS-DYNA and the boundary conditions are automatically taken care of by internally extending the corresponding node set by the newly generated marginal nodes.

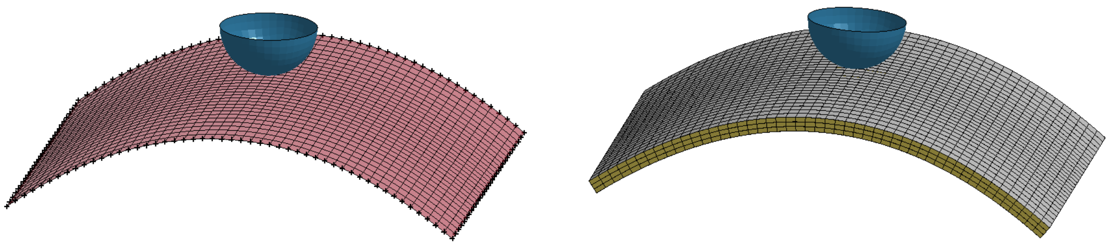

Fig.15: Curved composite plate with impactor, reference mesh (left) and stacked element model (right)

The delamination progress can be observed in Figures 16 and 17. With increasing deflection through the impactor, cohesive elements are eroded due to failure. Contact between emerging free shell surfaces is taken care of in the input definitions. In Figure 16 it can be seen that the elements initially fail around the impactor but not directly below it. This indicates that failure happens due to tension and shear, but not under pure compression, which makes sense from a physical point of view. Figure 17 shows a cross-sectional view of one element row.

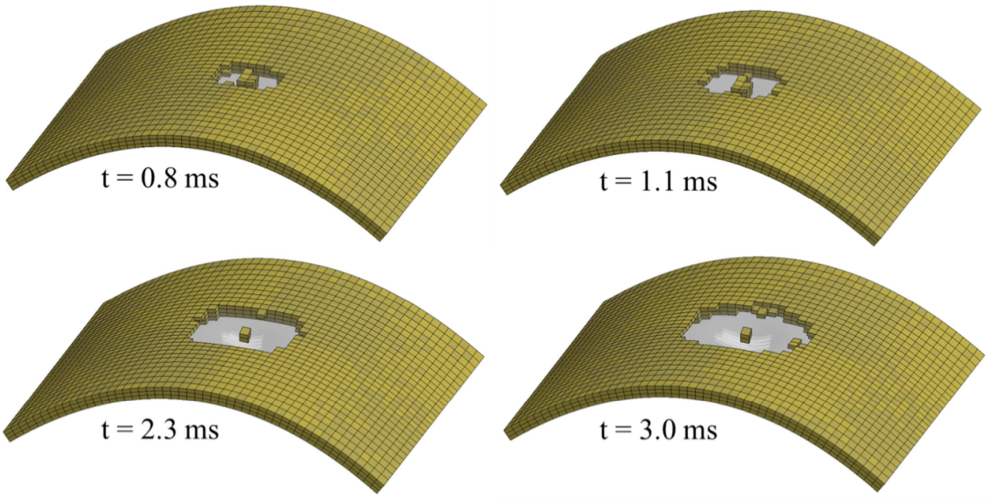

Fig.16: Successive failure of cohesive elements replicates delamination of intermediate layers (impactor not shown and shell elements plotted transparently)

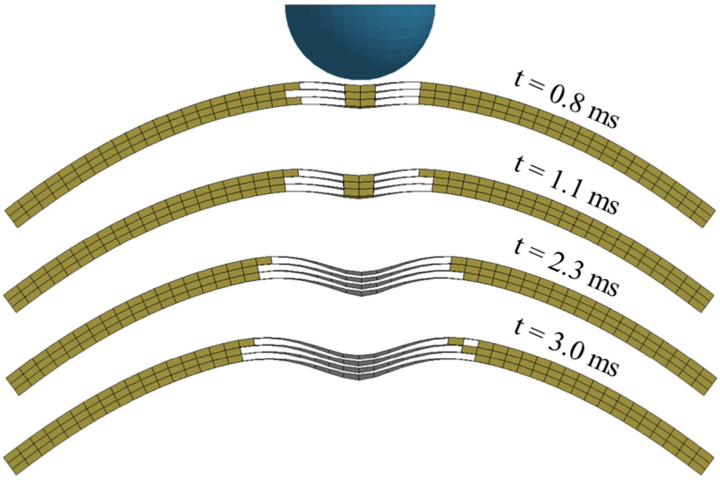

Fig.17: Successive failure of cohesive elements in cross-sectional view

5 Summary

A new feature for the numerical simulation of plane composite materials with distinct layers was presented. It simplifies the setup of numerical models for such stacked shell-like structures by only asking for a minimal amount of user input, such as geometry description (pure shell element mesh) and layer design (number of layers and properties of each layer). The mesh generation procedure was presented in detail and possible areas of application were shown.

6 Literature

[1] LS-DYNA Keyword User's Manual, Volume I & II, LS-DYNA R8.0, Lawrence Livermore Software Technology Corporation (LSTC), 2015.

[2] Muflahi, S.A., Mohamed, G., Hallett, S.R., *Investigation of Delamination Modeling Capabilities for Thin Composite Structures in LS-DYNA*, 13th International LS-DYNA Users Conference, Detroit, USA, 2014.

[3] Braun, D., Kennedy, S.J., Kennedy, D.J.L., Allen, D.E., *Sandwich Plate System Risers For Stadia*, SSRC 2002 Annual Stability Conference, Seattle, Washington, USA.