

An assessment of the new LS-DYNA layered solid element: basics, patch simulation and its potential for thick composite structure analysis

Madhukar Chatiri¹, Thomas Güll², Prof. Anton Matzenmiller³

¹CADFEM GmbH, Grafing b. Munich, Germany

²Adam Opel GmbH, Mainz-Kastel, Germany

³Institute of Mechanics, University of Kassel, Germany

Summary:

One major component of fuel cell vehicles is the hydrogen storage system. A promising and nowadays mostly used approach is to store hydrogen in wet wound carbon fiber reinforced plastic (CFRP) vessels manufactured by filament winding process with an operating pressure of up to 70 MPa (hereafter referred as H2 vessel). Due to the inherent complexity and 3-dimensional nature, accurate behavior of such thick composite structures in impact simulations needs an adequate representation of the composite plies.

Modeling thick composite structures with 2-dimensional elements will produce inaccurate results in transverse normal direction. Thus 3D modeling should be used but to model each ply with one solid element leads to undesirably big models and is impractical for large structures. Thus representation of several plies in one solid element and more such elements across thickness is desired. Also, solid elements are needed to represent the 3-dimensional state of stress and impact direction normal to the outer vessel surface. A new layered solid element formulation is implemented in LS-DYNA® Version 971 R4 allowing the definition of multiple integration points through the thickness in combination with arbitrary material orientation.

The above new element formulation is presented in this paper describing different patch simulation results and simulation results for thick composite structures such as hydrogen storage H2 vessels.

Keywords:

Layered solid, filament winding, thick composites, H2 vessel analysis

1 Basics and modeling of thick composites

1.1 Introduction

Finite element modeling of thick composite structures such as high pressure hydrogen tank manufactured by filament winding is relatively complicated. The filament wound pressure tanks have remarkable properties in the specific modulus, thermal resistance and endurance compared with the metallic fuel tanks. In spite of their good performance, pressure tanks made by filament winding are complex in analyzing and modeling. In these high pressure fuel tanks, the fiber orientation varies in the longitudinal and thickness directions. Also the thickness of the tank is in the range of 40 to 60 mm. To reduce the modeling effort of thick composite structures, they have been modeled with two-dimensional elements. However, this will produce inaccurate results because structures are so thick that plane stress condition is not applicable. Three-dimensional elements should be used for this case. But in three-dimensional modeling, it is difficult to model all the layers with different material properties with each solid element. Thus a new layered solid element wherein several plies with varying fiber orientations, thickness inside one solid element is needed.

In the present paper, a new layered solid element in LS-DYNA, its formulation, numerical patch tests and potential for analysis of thick composite structures is presented.

1.2 Different ways of modeling thick composite laminates

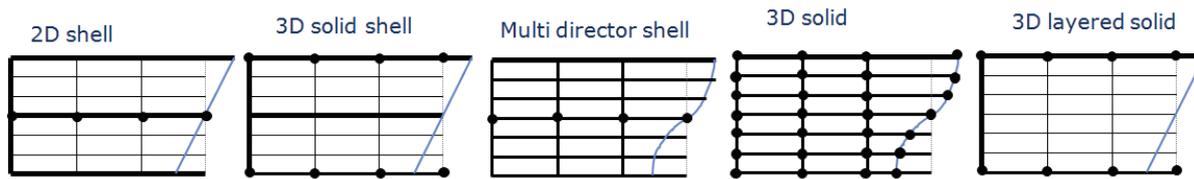


Fig. 1.1: Kinematic modeling of composite structures with different elements [8]

Modeling of thick composite structures should not only involve the global structural behavior but should also include 3D stress state in thickness direction. Traditionally different approaches have been used to model thick composite structures as seen in Fig.1.1. They can be modeled with shell elements wherein the normal stresses are neglected (which have 6 degrees of freedom, 3 translations and 3 rotations). Also, solid shell elements which have only displacements degrees of freedom have been used. In addition to convenient formulation for constitutive law, these elements provide a straightforward extension to geometrically non-linear problems, particularly in the presence of large rotations, since only translational degrees of freedom are involved. In the third method, transverse shear stresses are improved by introducing a director for each layer (multi director formulation) [2]. The next approach is to use one solid element per layer but this approach is not practical as it leads to large amount of CPU time. Apart from the above element modeling, different homogenization techniques like classical laminate theory which give smeared properties for the laminate have been used. These methods help in saving CPU time and can be used to represent global structural behavior. But they can't be used for detailed 3D stress-strain analysis specifically in thickness direction.

The last approach is the focus of the present paper wherein the new layered solid element which can represent different material layers with varying fiber angles, variable thickness inside one element with multiple integration points through the thickness. By defining several layers with different materials and ply orientation inside one layered solid, number of elements through the thickness is remarkably reduced and still the result is close to the one obtained from the detailed finite element model of one brick element per layer.

2 Element formulation

2.1 Explicit eight node layered solid element

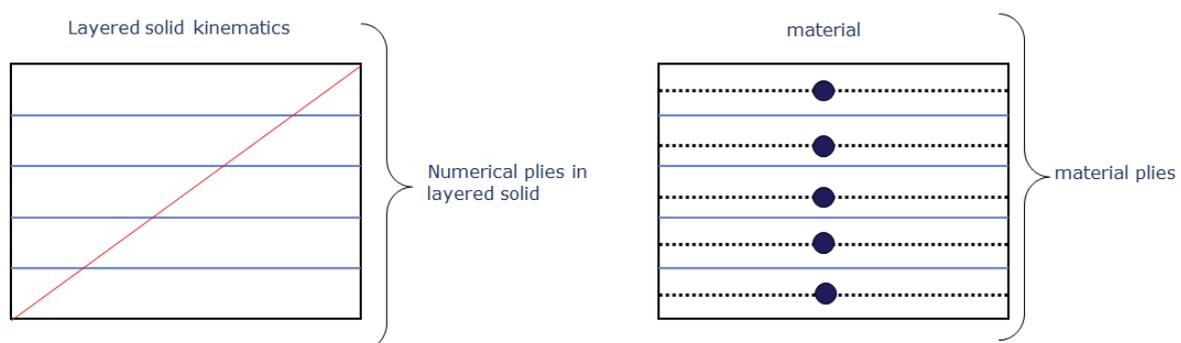


Fig. 2.1: Linear layered solid element in LS-DYNA representing a laminate configuration with 5 different plies

The 8-node layered solid element in LS-DYNA (which can be accessed under thick shell element formulation 5) uses one integration point per layer and any number of integration points through the thickness. Like any brick element, it uses the 3D stress updates. Different material models may be assigned to each layer using *PART_COMPOSITE or *INTEGRATION_SHELL cards. The trilinear solid element is the starting point for the strain field, but assumptions about the strain remove shear locking and volumetric locking. Also, a modification to the thickness strain is made for anisotropic materials and mixed materials to assure that the thickness stress is constant through the thickness. Zero energy modes that result from using only 1 integration point per layer are eliminated by hourglass control derived from the type 6 hourglass control in LS-DYNA.

Formulation Type 6 hourglass control in LS-DYNA is based on ASQBI (assumed strain quintessential bending incompressible) hourglass control in Belytschko and Bindeman [1]. In formulae 3.22 through 3.27 in the above mentioned reference paper, it is demonstrated that the terms used to evaluate hourglass control can also be used to evaluate the strain at any point in the element domain. These formulae allow the strain tensor to be a measure at very low cost since the **B** matrix does not need to be evaluated for each integration point. It is possible to use the ASQBI strain fields with multiple integration points without encountering locking since this strain field maintains zero dilatation in the non constant part of the strain field. Also, the ASQBI assumed strain field eliminates shear locking and volumetric locking which make the element more accurate. For isotropic material with the same properties in each layer, this strain field is used directly with a 3D stress update routine. For anisotropic materials with principal directions that vary by layer, an additional correction is needed for the thickness strain term due to Poisson's affects that are directionally dependent and therefore vary by layer. Similarly, mixing materials in the layers requires a correction due to both Poisson's affects and a variation in stiffness.

Also, the layered solid requires hourglass control because it is not rank sufficient. ASQBI hourglass control is designed for brick elements that have 12 zero energy deformation modes. With more than one integration point through its thickness, 4 of these modes, two that correspond to bending and two that correspond to torsion, result in non-zero strains so that only 8 zero energy modes remain. Each of these modes can be identified as terms in ASQBI hourglass control and only the 8 that are needed are used. In this way, the hourglass control is orthogonal to the strain fields in the element.

2.2 Steps used in the element calculation

The following steps can be summarized for the layered solid element calculation

- Calculate a global to local co-ordinate transformation system
- Transform nodal co-ordinates and velocities to the local system
- Calculate the shape function derivatives at the mid-plane
- Calculate the rate of deformation at the mid-plane
- Calculate miscellaneous terms which are used in rate of deformation calculations at integration points, hourglass control calculations, time step calculations
- Loop over integration points and calculate weighted thickness average z stresses from the previous cycle
- Loop over integration points again
 - calculate modified z strain using average z stress to estimate error in the previous cycle and in this cycle
 - calculate the weighted thickness average modified z strain
- Loop over integration points again
 - calculate the rate of deformation at the integration applying yet another correction to the z strain to make sure that the element thickness is consistent with the assumed z strain values
 - calculate the new stress using the modified rate of deformation, history data, material constitutive data
 - Calculate nodal forces from the new stress
- Calculate and add hourglass control to the nodal forces
- Transform nodal forces from the local to global co-ordinate system
- Gather nodal forces into the nodal forces array

2.3 Element description in LS-DYNA

Presently the new layered solid in LS-DYNA can be accessed under thick shell formulation 5 (tshell=5). The layered section can be defined with the *SECTION_TSHELL in combination with the *INTEGRATION_SHELL cards. Total number of layers can be defined under NIP flag (number of through thickness integration points). Also, every single fiber orientation can be defined under *SECTION_TSHELL card. Therefore the option ICOMP (flag for layered composite material mode) must be set to 1. Presently, older version of defining the modeling layup is supported i.e. an integration rule describing the relative thickness is needed. The newer and easier version of defining modeling layup with *PART_COMPOSITE card will be supported in the future versions. Exemplarily, modeling a laminate with [+45-45]_s configuration can be seen in appendix 1.

3 Numerical examples

A pragmatic approach has been used to test the new layered solid element in LS-DYNA. Numerical testing was done in three parts consisting of one element tests, composite benchmark tests as given in the literature, and component tests as performed for OPEL.

3.1 One element examples

Initially the new layered solid element was tested with different one element tests (with and without layered configurations). A test matrix was prepared which contained new layered solid, shell and solid elements with layer orientations for e.g. [0-0-0], [0-90]_s, [+45-45]_s as seen in Fig.3.2 and also without layer orientations. The new layered solid element was tested in tension, compression, shear and bending load cases both in longitudinal and transverse fiber directions. The results were compared with analytical solutions from structural mechanics and also with standard shell, one point hexahedron solid elements in LS-DYNA.

3.1.1 Non layered configuration tests

Firstly, the layered solid was verified with standard solid and shell elements without layered configurations. A hexahedron in the form of regular 1mmX1mmX1mm (Length*width*thickness) cube for solid and layered solid elements, square plate of 1mmX1mm with thickness 1mm for shell element is used for one element tests. For all the simulation models 1, 2, 3 refer to principal material directions. The magnitude of the applied loads (uniformly prescribed displacement loading) is small enough and thus linear material response can be approximated. *MAT_COMPOSITE_DAMAGE (*MAT_022) from LS-DYNA has been used for all the simulations.

Orthotropic material parameters for the one element test are $E_{11} = 1.32 \times 10^5 \text{ N/mm}^2$, $E_{22} = E_{33} = 10755 \text{ N/mm}^2$, $\nu_{21} = \nu_{31} = 0.019$, $\nu_{23} = 0.490$, $G_{12} = G_{13} = 5653 \text{ N/mm}^2$, $G_{23} = 3378 \text{ N/mm}^2$.

For post processing → internal strain energy density per unit volume (equ.3.1), energy ratio (equ.3.2), reaction forces, stresses were checked with the analytical solutions. Also, the layered solid solution was compared with the standard hexahedron, shell element solutions.

$$\text{Internal strain energy: } E = 0.5 \int (C_{ij} \varepsilon_j) \varepsilon_i dv \quad [3.1]$$

$$\text{Energy ratio: } e = \frac{E_{total}}{E_{total}^0 + W_{external}} \quad [3.2]$$

$$\text{where } \underbrace{E_{kin} + E_{int} + E_{rigidwall} + E_{hourglass} + E_{damping} + E_{sliding}}_{E_{total}} = E_{kin}^0 + E_{int}^0 + W_{external}$$

Exemplarily shown are the results from uni-axial tensile load case (Fig.3.1) results in x direction. Table 3.1 summarizes results from internal energy, reaction forces, and stress components. As seen from the table, various results with the layered solid element are in good agreement with the reference analytical solutions, shell and solid elements. Similar tests and comparisons were carried out for compression, bending and shear load cases.

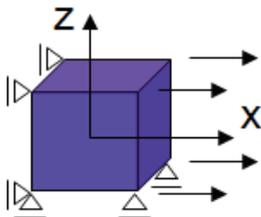


Fig.3.1: Uniaxial tensile test in the (x) direction

Component	Analytical solution	Solid type 1	Shell type 2	Layered solid (tshell type 5)
internal energy (E)	26 N-mm	26 N-mm	26 N-mm	26 N-mm
reaction force (F)	2.64kN	2.64kN	2.64kN	2.64kN
inplane stress (σ_x)	2640 MPa	2640 MPa	2640 MPa	2640 MPa

Table 3.1: Results for Uniaxial tensile test in the (x) direction

3.1.2 Layered configuration tests

The above mentioned tests were again performed with different fiber orientations (Fig.3.2) and the results of the layered solid were compared with the layered shell element (in-plane components were verified). The layered definition (ply thickness, ply orientation) as described previously is defined by *SECTION_TSHELL (layered solid), *SECTION_SHELL (shell) and *INTEGRATION_SHELL cards in LS-DYNA.

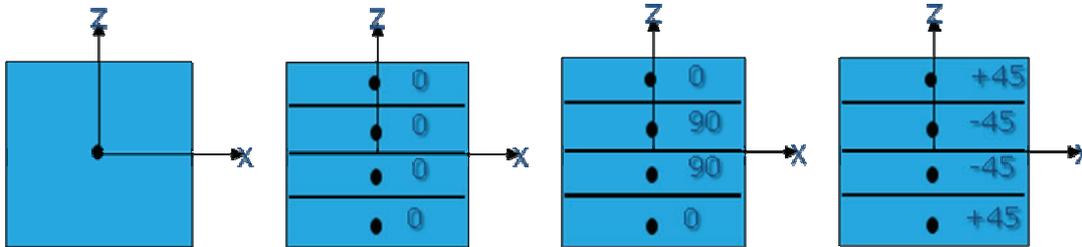


Fig.3.2: Laminate configurations for one element test

Exemplarily shown are the results from uni-axial tensile load case on a [0-0]_s, [0-90]_s and [+45-45]_s (Table. 3.2) laminate configurations in x direction. The inplane stress component σ_x in the layer co-ordinate system for the new layered solid element are verified with analytical and layered shell element solutions. The layered solid element results are in excellent agreement with the layered shell element.

Laminate configuration	Analytical solution	Shell type 2	Layered solid (tshell type 5)
0-0-0-0	2640 MPa	2640 MPa	2640 MPa
0-90-90-0	2640 MPa	2640 MPa	2640 MPa
+45-45-45+45	355 MPa	355 MPa	355 MPa

Table 3.2: Comparison of inplane stress σ_x in layer co-ordinate system

3.1.3 One element tests in thickness direction for layered configuration

In order to verify the thickness components, a solid hexahedron was stacked on top of each other to compare with the layered solid element. Uniaxial compression in thickness (z) direction is done on layered solid and stacked hexahedron solid elements (Fig.3.3 and 3.4). The layered configuration is 0-90-90-0. One point hexahedron solid are stacked on top of each other to make the 0-90-90-0 configurations and is achieved by *ELEMENT_SOLID_ORTHO definition in LS-DYNA.

One element test
Time = 0

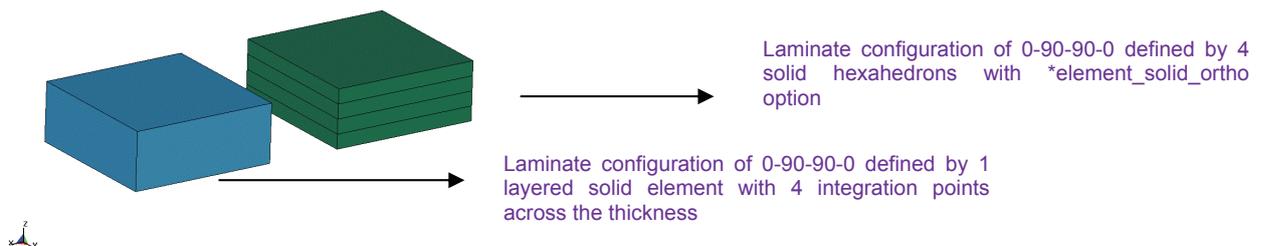


Fig.3.3: Layered solid and stacked one point solid

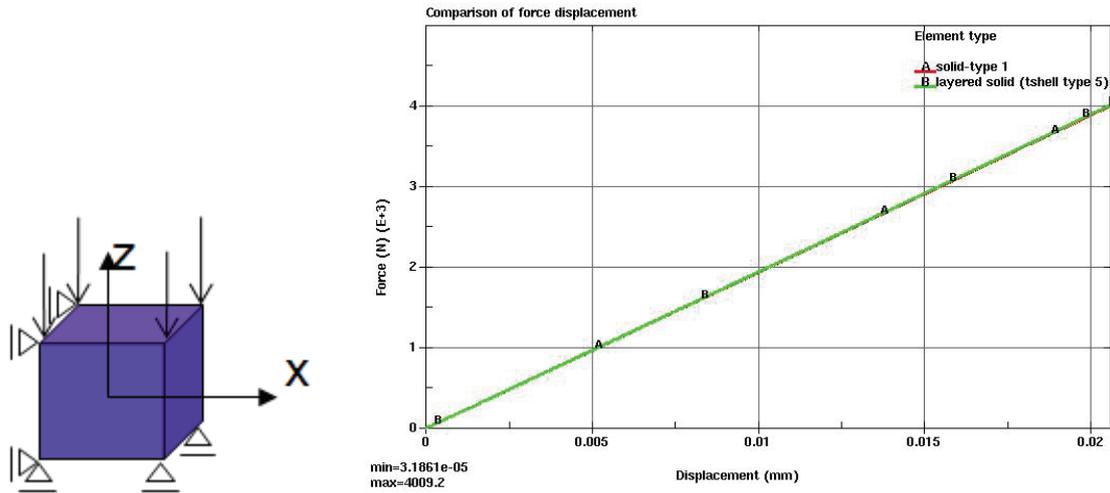


Fig.3.4: Uniaxial compression test

in the thickness (z) direction Fig.3.5: Comparison of force-displacement curves for solid and layered solid elements

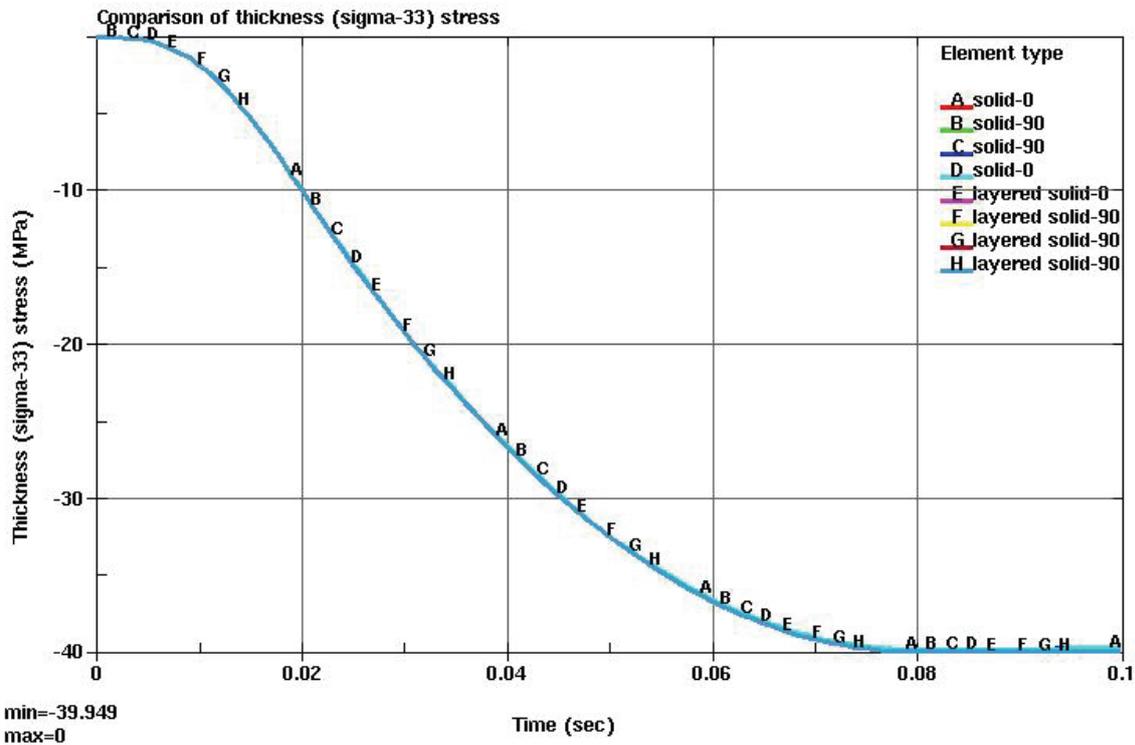


Fig.3.6: Comparison of thickness (σ_{33}) stresses in individual stacked solid elements and different layers of layered solid element.

As previously discussed about the constant thickness stresses for the layered solid element, Fig. 3.6 depict that the thickness stresses (σ_{33}) are constant across thickness for all the layers (0-90-90-0) inside the layered solid element. Also, the thickness stresses are in excellent agreement with the stacked hexahedron solid elements.

3.2 Cantilever beam

After the one element tests, layered solid element was tested with various composite benchmark examples from literature. Fig 3.7 shows a linear elastic cantilever beam with a point load at its end.

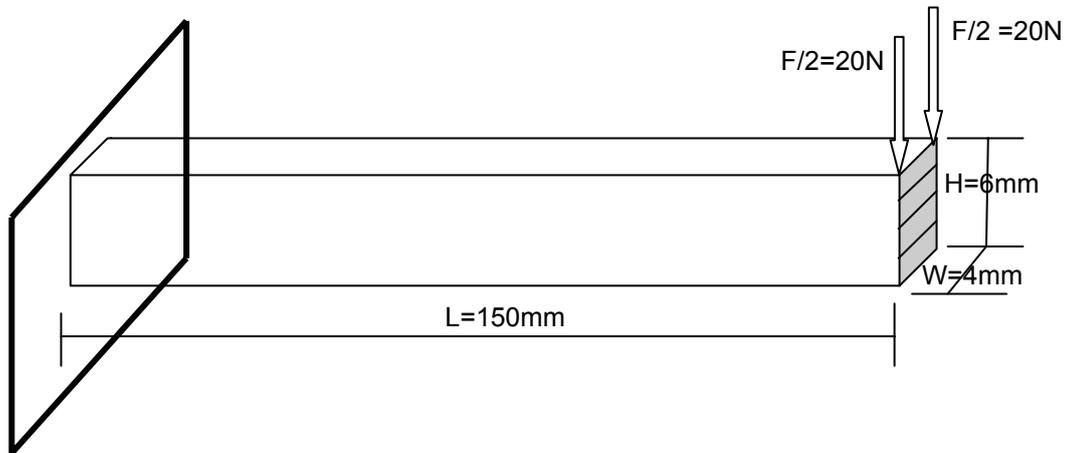


Fig.3.7: Problem description for Cantilever beam subjected to point load

Orthotropic material parameters for the cantilever test are $E_{11} = 146.9$ GPa, $E_{22} = E_{33} = 10.9$ GPa, $V_{21} = V_{31} = 0.028$, $V_{23} = 0.776$, $G_{12} = G_{13} = 10.89$ GPa, $G_{23} = 6.4$ GPa.

The above cantilever bending problem was simulated with two models. The first one was to check bending behavior of a normal orthotropic material without layer configuration and the second one with 0-90-90-0 layer configuration. According to the structural mechanics, deflection at the end of the beam would be $w = FL^3/3EI = 4.25$ mm. But this equation can't be used for comparison here because of orthotropic material. With the convergence studies it was possible to get a converged solution of 4.30 mm.

The second model was simulated with one layered solid element across thickness with 0-90-90-0 configuration and the displacement achieved was 5.01 mm. The converged displacement solution in reference [7] is 4.83 mm.

3.3 Plate under uniform load

In this example, a simply-supported square cross-ply laminated plate (Fig. 3.8) is subjected to a uniform pressure p_0 and the center deflection δ (Z-direction) of the plate due to the pressure load is determined. The stacking sequence of the plies is symmetric about the middle plane. This problem has been verified from the reference [4, 10]. Also, the same problem has been simulated with layered elements from ANSYS [10]. The LS-DYNA layered solid element is in good agreement with the expected results (Table. 3.3).

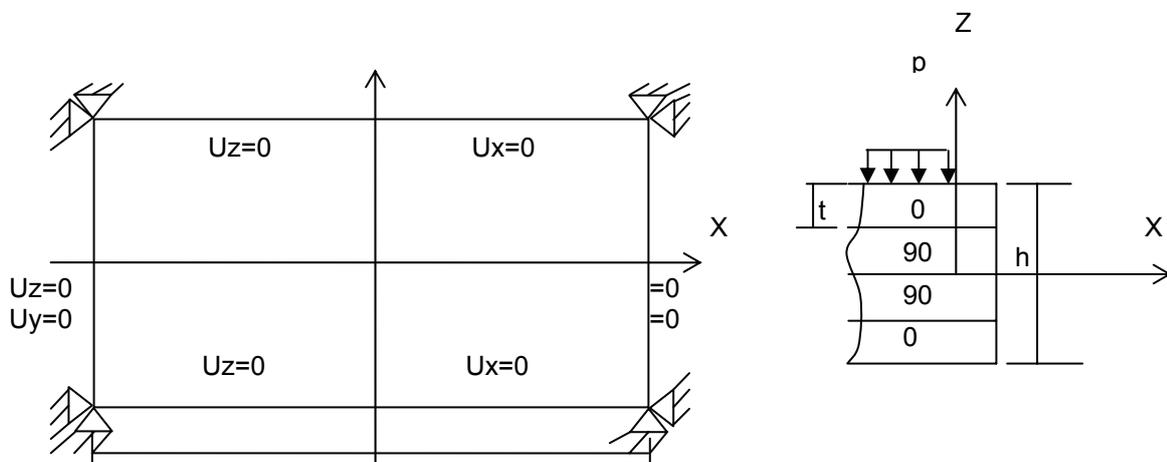


Fig.3.8: Problem description for plate under uniform pressure loading [10]

Orthotropic material parameters for the above test are $E_{11} = 25 \times 10^6 \text{ N/m}^2$, $E_{22} = E_{33} = 1 \times 10^6 \text{ N/m}^2$, $V_{21} = V_{31} = 0.028$, $V_{23} = 0.776$, $G_{12} = G_{13} = 0.5 \times 10^6 \text{ N/m}^2$, $G_{23} = 0.2 \times 10^6 \text{ N/m}^2$.

Element type	Target value for centre deflection 'w' according to [4]	Obtained value
ANSYS layered solid 46	0.0683	0.0676
ANSYS layered solid 185	0.0683	0.0676
ANSYS layered shell 181	0.0683	0.0681
LS-DYNA layered solid (tshell type5)	0.0683	0.0659

Table 3.3: Result comparison with various layered elements

3.4 Pinched cylinder

This problem is a severe benchmark test for checking shear locking in thin shell structures. Fig. 3.9 shows a pinched cylinder subjected to a pair of concentrated loads, both ends of the cylinder are assumed to be free [5]. The parameters for the above test are $E = 1.05 \times 10^6 \text{ psi}$, $\nu = 0.3125$, $L = 10.35 \text{ in}$, $R = 1.0 \text{ in}$, $t = 0.094 \text{ in}$, $P = 100 \text{ lb}$.

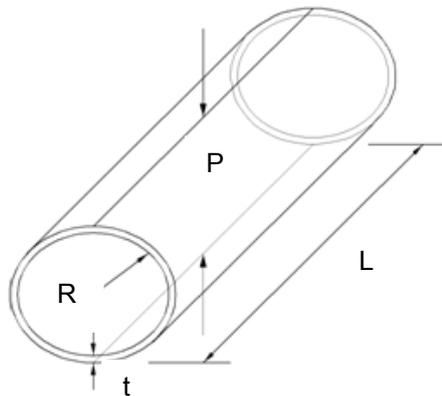


Fig.3.9: Pinched cylinder with dimensions [5]

The analytical solution for displacement at the loading point of pinched cylinder was calculated as $w_{\max} = 0.1137$. The problem was simulated with the two different mesh densities as shown below.

Mesh	16x16x1	20x20x1
Layered solid solution	0.051	0.0651

Table 3.4: Displacement at loading point of pinched cylinder

We can see from the above table (Table 3.4) that shear locking can be avoided by the layered solid element to certain extent but layered solid element based on ASQBI hourglass control [1] tend to be stiffer for thin shell applications and thus should be avoided

3.5 Material property tests

GM Fuel Cell Activities is presently developing a systematic approach for the substantiation of CFRP H_2 high pressure vessel design and its integration in the vehicle [6]. Different tests (ELIP, combined T/C-T etc.) have been performed on unidirectional cylindrical specimens in order to obtain the material properties for the H_2 vessel. In the ELIP test (elastomer inner pressure test), a ring is subjected to inner pressure. It is used to gain the material properties E_{11} , ν_{12} . In the T/C-T (Combined tension/compression and torsion test), a cylindrical tube is subjected to the torsion as shown in the Fig. 3.10. It is used to gain shear modulus G_{12} . Initially, these tests were simulated with solid elements. But since each layer was represented with one solid element, it was leading to high CPU times. Next, homogenized (smeared) properties of different layers were taken and simulated with solid

elements. This saved CPU time but detailed layer configuration can't be used. The same simulations were carried out with the new layered solid element with detailed layer configuration and the results were compared with the standard type 1 solid hexahedron available in LS-DYNA. As we can see in Fig. 3.11, we get similar results with solid as well as new layered solid element but the CPU time gained is significant (Table 3.5).

Element type	Number of elements	CPU time, Intel Xeon-4 CPU
one solid per ply	38000	660 sec
Layered solid (tshell elform 5)	9600	105 sec

Table 3.5: Comparison of simulation models

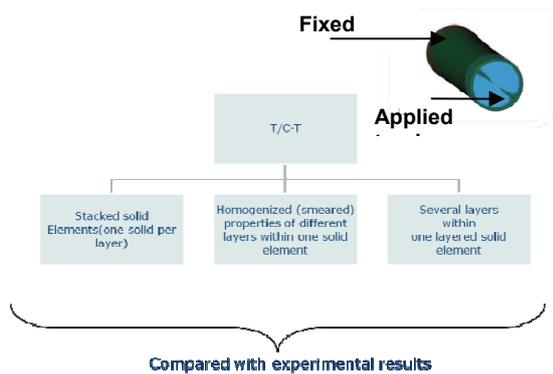


Fig.3.10: Different modeling ways for Combined tension/compression-torsion test

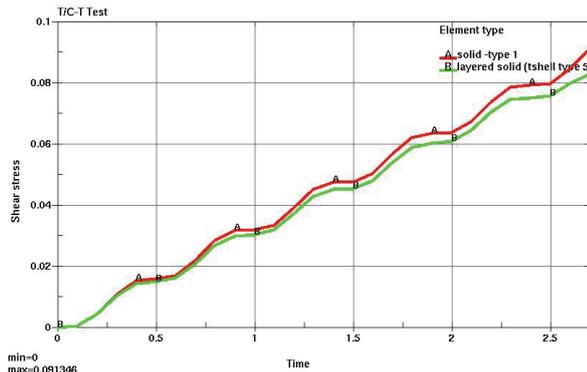


Fig. 3.11: Comparison of shear stress between solid and layered solid models

4 Component test with H2 vessel

4.1 Test setup and component

To verify stability, cpu-cost as well as pre- and postprocessor handlings of the new layered solid element, simulations with H2 vessel were performed. Cylindrical segments which were cut out of H2 vessels were impacted, intrusion and forces are measured. Fig.4.1: Test setup H2 vessel segment impact shows the principal setup of the test. The test conditions are chosen to perform 'test to failure' and don't represent loads which are measured in vehicle crash simulations.

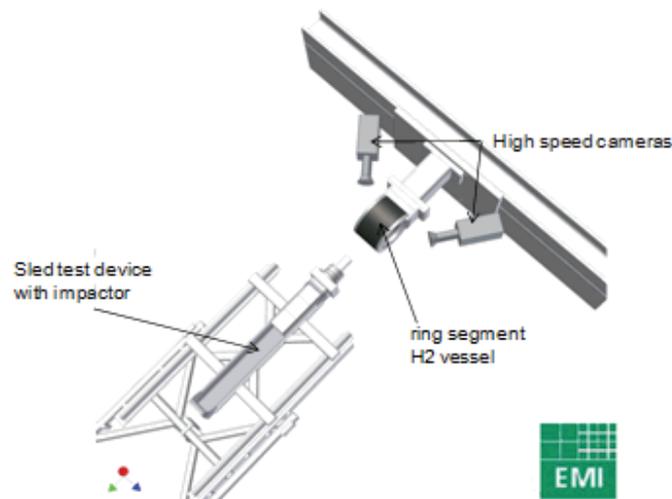


Fig.4.1: Test setup H2 vessel segment impact-picture by courtesy of Ernst-Mach-Institut

4.2 LS-Dyna model

The test result (Fig. 4.1) were compared with 3 different models, which are explained below.

4.2.1 TSHELL model with layered solid element

The LS-DYNA version 971_d_R4.2 used for the following simulation results allows the use of the new layered solid element with material 22, *MAT_COMPOSITE_DAMAGE. Delamination is represented with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK contacts. More than 100 plies of carbon fiber are represented with 5 TSHELL elements. Every fiber orientation is stated in the *SECTION_TSHELL card. Therefore the option ICOMP (flag for layered composite material mode) must be set to 1 and an integration rule describing the relative thickness is needed.

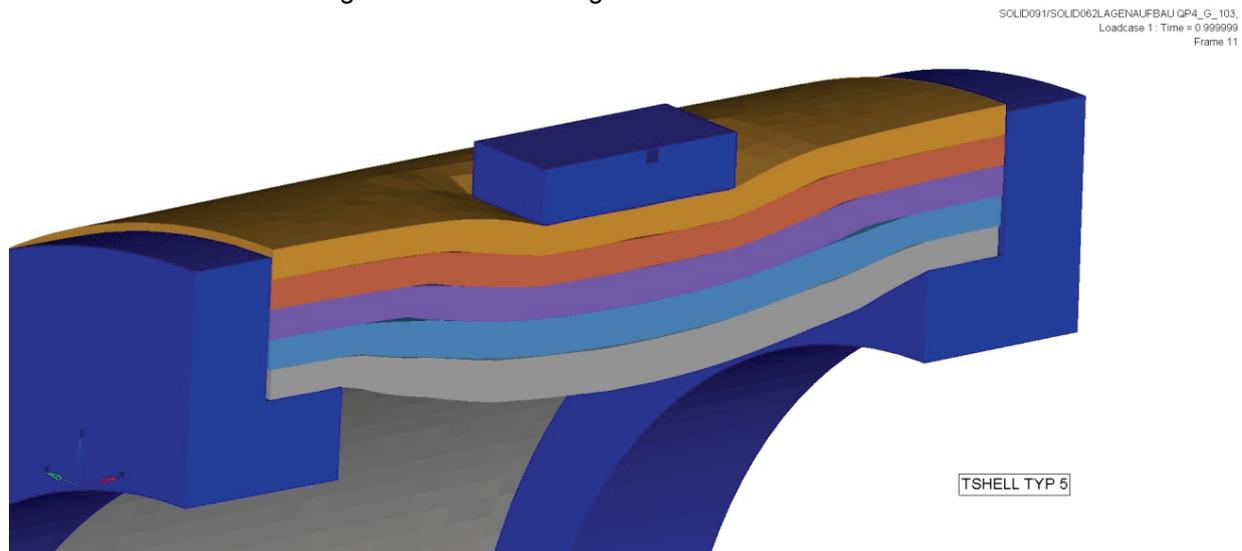


Fig.4.2: Test setup H2 vessel segment impact modeled with layered solid elements (tshell elform5)

4.2.2 One solid per ply model

This model uses default solid elements (type 1) with one integration point per ply. Ply direction for more than 100 plies is represented with the use of the AOPT-option. For every fiber direction an own material card with an applicable orientation vector is needed.

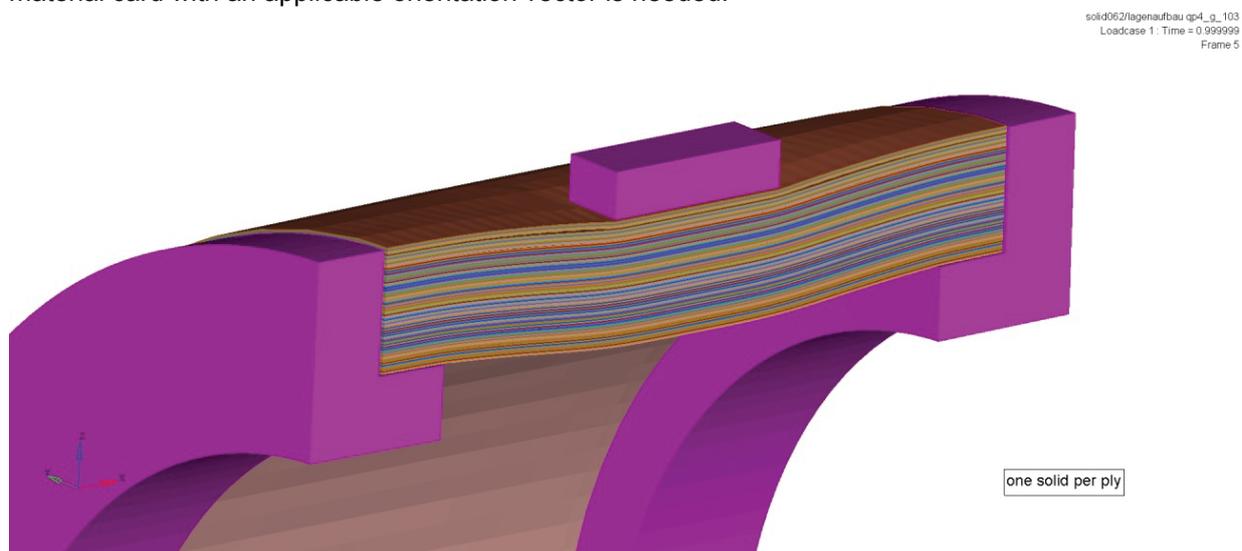


Fig.4.3: Test setup H2 vessel segment impact each ply modeled with one solid element

4.2.3 Model with engineering constants

This model uses 'smeared' (homogenized) stiffness properties calculated with the classical laminate theory. More than 100 plies of carbon fiber are represented with 5 default solid elements (type 1).

4.3 Simulation results

All three models can detect the onset of failure for a H2 vessel in a crash event. Failure due to delamination is modeled with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK. The author didn't get aware of higher numerical instabilities.

Model type	# elements	CPU time [h]/ Win XP with 3 CPU
one solid per ply	1.9 Mio	81.5
Layered solid (tshell elform 5)	35.000	13.5
Engineering constants	35.000	5

Table 4.1: Comparison of simulation models

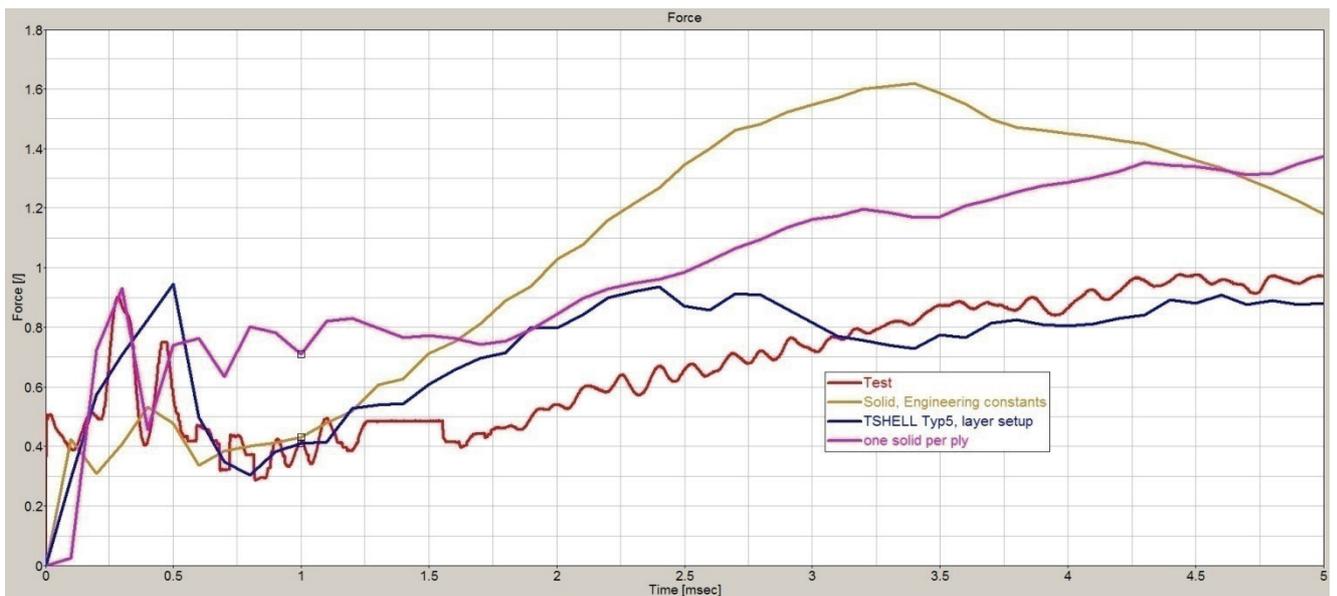


Fig.4.4: Result H2 vessel segment impact → measured force over time with different element types

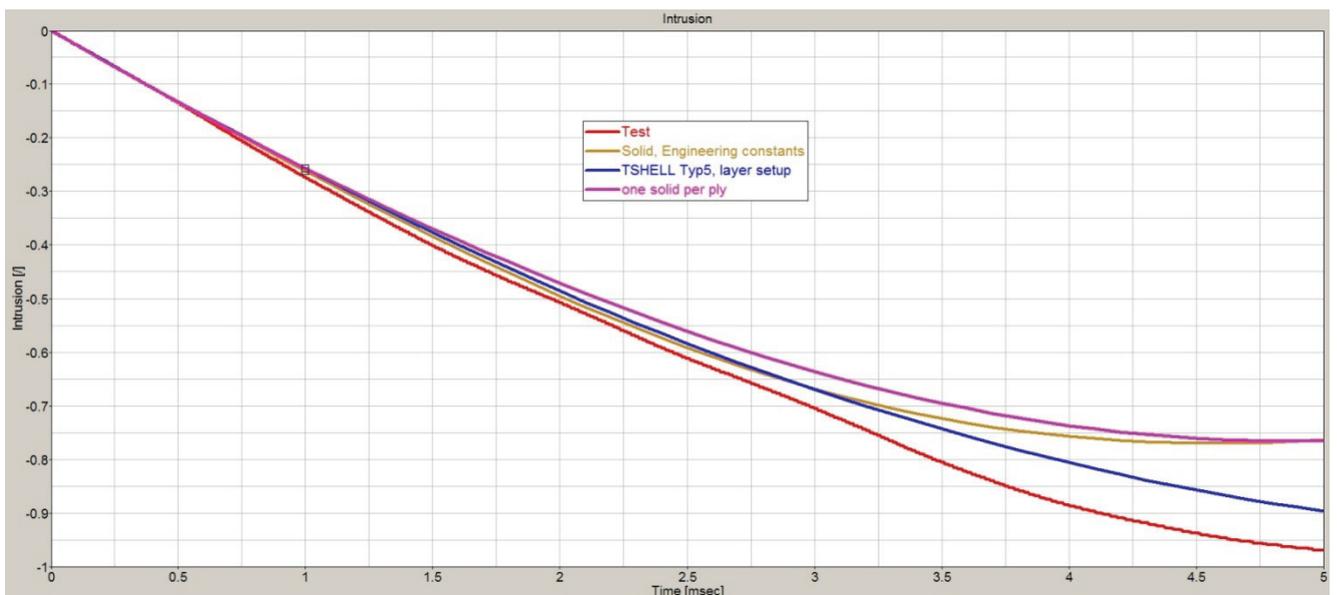


Fig.4.5: Result H2 vessel segment impact → measured intrusion over time with different element types

5 Summary and outlook

The use of layered solid (TSHELL elform=5) elements offers a possibility to model thick composite lay-ups in detail with acceptable CPU times as well as an acceptable model size. One can take individual fiber orientations and ply thickness in account without the need to live with 'smeared' material properties. Also, this element has several advantages like calculating thickness stresses, and also it is important for certain problems where through the thickness deformations and strains are significant.

One necessary improvement for the TSHELL Type 5 element is the usage of more composite material models like *MAT_59 & *MAT_162. *PART_COMPOSITE card in combination with layered solid would be a great addition because layered section definitions can be defined easily.

The new Layered solid element (tshell elform=5) should not used with single one point integration like hexahedron element. It is better to use hexahedron element instead. For thin shell applications, layered shell formulations must be preferred because elements based on ASQBI tend to be stiffer for thin shell bending applications.

6 Literature references

- [1] Belytschko, T and Bindeman, L.P.: Assumed strain stabilization of the eight node hexahedral element. Computer Methods in Applied Mechanics and Engineering 1993; 105:225-260
- [2] Robbins, D.H and Reddy, J.N.: Modeling of thick composites using a layerwise laminate theory. International Journal for Numerical Methods in Engineering 1993; 36:655-677
- [3] Fiolka, M and Matzenmiller, A.: On the resolution of transverse of transverse stresses in solid-shells with a multi-layer formulation. Communications in Numerical Methods in Engineering 2007; 23:313–326
- [4] Reddy, J.N.: Exact solutions of moderately thick laminated shells. ASCE Journal Engineering mechanics 1972; 110-5:805
- [5] Y.Guo: Eight node solid element for thick shell simulations. 6th International LS-DYNA conference, Detroit, 2000
- [6] Valentin, S. et al: The hierarchical building-block approach of testing and FE analysis for strength prediction of high pressure wet wound composite vessels. SIMVEC VDI tagung, 2008
- [7] Eriq, Q.S.: Shear locking and hourglassing in MSC Nastran, ABAQUS and ANSYS. MSC Software Users Meeting, 2006
- [8] Werner, W.: Simulation and FE-modeling of global and local effects in thin-walled shell structures of composite materials. NAFEMS Seminar on Simulating Composite Materials and Structures, Bad Kissingen, 2007
- [9] Nguyen, D.Q. and Matzenmiller, A.: A solid shell element with enhanced assumed strains for higher order shear deformations in laminates. Technische Mechanik, Band 28, Heft 3-4, S. 334-355, 2008
- [10] ANSYS verification manual, Release 11 documentation for ANSYS. ANSYS Inc, Pittsburg, 2007
- [11] Hallquist, J.O.: LS-DYNA Theoretical Manual. Livermore Software Technology Corporation, Livermore, 2006

Appendix 1

*SECTION_TSHELL

```
$# sid elform shrf nip propt irid icomp
   1      5 0.8333 4.0   0.0  -99    1
```

```
$# bi bi bi bi
   45.0 -45.0 -45.0 45.0
```

*INTEGRATION_SHELL

```
$# irid nip esop
   99    4    0
```

```
$ s wf opid
-0.7500000 0.2500000 1
-0.2500000 0.2500000 1
 0.2500000 0.2500000 2
 0.7500000 0.2500000 1
```