Modelling of thick UD composites for Type IV pressure vessels

Ralf Matheis[1], Helmi Murnisya[1], Thomas Johansson[2]

[1]Forschungsgesellschaft Kraftfahrwesen mbH Aachen
[2]DYNAmore Nordic AB

1 Introduction

Increasing energy costs, limitation of crude oil resources as well as constantly intensifying emission targets (especially w.r.t. CO_2) are a pivotal driver for current automotive research and development [1]. In this regard alternatively powered vehicles (APV) with drive-trains, that differ from conventional internal-combustion engines (ICE) supplied with petrol or diesel fuel, show high potential for energy economical and environmentally friendly propulsion [2]. On the other hand lightweight structures are a fundamental requirement in order to achieve a low demand of energy and to compensate possible higher structural masses of alternatively powered drive-trains [3].
An alternatively powered drive-train that is on the one hand cost efficient since it requires comparatively minor design changes to conventional vehicles (primarily ignition and valve-train system) and on the other hand has a perspective to evoke low CO_2 emissions, is an ICE operated with compressed natural gas (CNG) [4], [5]. This energy source requires high-pressure storage tanks that have to withstand the high mechanical demands of internal pressures of 200 bar to 250 bar. In order to achieve energy storage that is simultaneously light and safe, pressure vessels of the Type IV that are entirely made of fibre reinforced polymers (FRP) are currently state of the art [6].
For the proper and optimal integration of the tanks into the vehicle during the APV design and development process at industrial level, moreover the predictability of the material and component behaviour using the finite element method (FEM) is indispensable [7]. To make a step forward in the capability of the automotive industry to model, predict and optimise the crash behaviour of vehicles equipped with Type IV tanks, a virtual testing methodology (VTM) was developed within the project MATISSE funded by European Commission's 7th Framework Programme. As a demonstrator a belly mounted CNG tank consisting of a combined carbon fibre reinforced polymer (CFRP) and glass fibre reinforced polymer (GFRP) wet wound structure with a polyamide (PA) liner and aluminium bosses was used. The reference structure was provided by the MATISSE partner Xperion Energy & Environment GmbH and is presented in Fig.1.

Fig.1: Type IV CNG tank

As a key element of the VTM a modelling approach for the impact behaviour of thick unidirectional (UD) reinforced materials was developed and applied on the tank level. It bases on a combined stacked/layered laminate set-up that uses thick shell (t-shell) and cohesive elements and an orthotropic continuum damage material model.
The validation of the approach was based on a "reverse finite element method" which is necessary since no fully reliable test procedures for the characterisation of the relevant materials were identified. Therefore, material values determined by calculation or retrieved from literature were applied. The

validation was carried out on the basis of three point bending tests (3PB) on FRP tubes. The tank model is subsequently verified on pressurisation and impact tests conducted on physical tanks.
In a last step, several measures of model simplification are carried out on the tank model in order to achieve reasonable simulation costs with the perspective of application in full vehicle simulations.

2 Modelling approach

The MATISSE reference tank comprises three main components: a pair of aluminium bosses, a blow moulded PA liner and a winding structure combining CFRP and GFRP. As different the applied materials are so different the modelling approaches had to be chosen. This comprised the material model, the element type and size as well as the laminate set-up for the FRP layers. In the following the modelling choices for each component are presented. All approaches consider LS-DYNA version R7.0.

2.1 Element type and size

For each sub-component of the tank an appropriate element type and mesh size is chosen based on the geometry and material of the component to be modelled.

2.1.1 *Wound GFRP and CFRP*

The winding structure of the tank is modelled using the ***ELEMENT_TSHELL_COMPOSITE** definition. This ply-based composite model is able to define laminate properties (material and fibre angle) in each element belonging to one ***PART**. Every unidirectional (UD) composite ply is modelled with one layer of t-shell elements with selective reduced in plane integration (type 2 on ***SECTION_TSHELL**) with three integration points. The mesh of the wound structure comprises of 359,424 eight-node hexahedrons between 1.2 mm and 3.1 mm. The usage of pentahedrons was averted since it resulted in unphysical stress peaks. Additionally, the model contains an interface between the t-shell element layers that is modelled with solid cohesive elements with zero thickness of type 19 (***SECTION_SOLID**). Furthermore, a number of tetra elements was applied in order to fill the void in the layer trail zone. A section of the full mesh of the winding structure is shown in Fig.2.

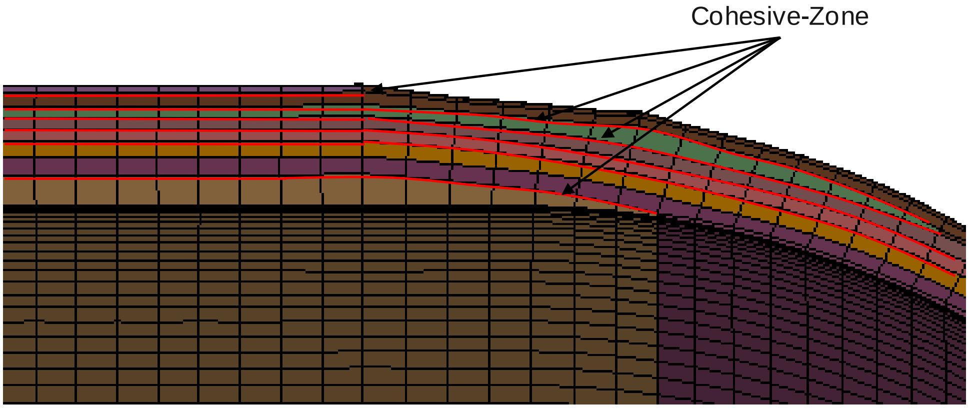

Fig.2: Sectional view of the mesh of the winding structure

2.1.2 *Liner*

The interior liner is due to the relatively thin wall thickness of 2 mm and isotropic character of PA predestined for a shell modelling. Because of the simple structure and the predicted marginal influence on the structural behaviour, an under-integrated element formulation is suitable and thus the type 2 is chosen for the ***SECTION_SHELL** property card. For linear materials the results are comparable to those achieved with fully integrated elements [8]. The element area geometry and size is congruent to the mesh of the wound structure and has common nodes with these t-shell elements. Thus, the liner mesh comprises of 9,250 trias and 42,625 quad elements of 0.2 mm to 6.2 mm and is shown in Fig.3Fig.3:.

Fig.3: Mesh of liner

2.1.3 *Bosses*

The bosses are designed as solid aluminium components which are considered to be rigid due to their thick dimensions in all directions and the high Young's modulus of the applied aluminium material. This assumption allows the usage of very simple tetrahedral solid elements. Furthermore, no specific demands are made on the element formulation and type 1 "constant stress solid" is chosen on the **`*SECTION_SOLID`** card. The boss components that are modelled for the CNG tank model consist of 100,000 elements with element sizes between 0.1 mm and 8.4 mm (Fig.4).

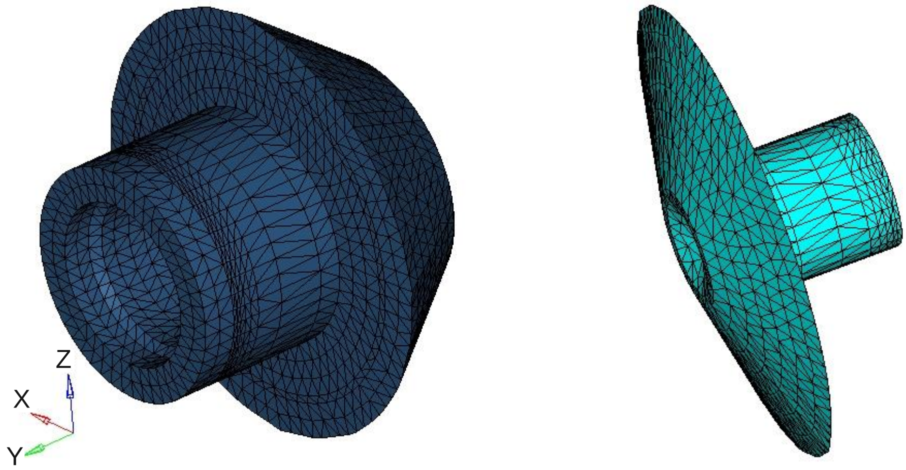

Fig.4: Mesh of bosses

2.2 Laminate set-up

For the wound structure (GFRP as well as CFRP) a combination of the layered thick shell approach and stacked thick shell approach, was applied. This means that a t-shell element is provided with a number of integration points via the **`*ELEMENT_TSHELL_COMPOSITE`** that defines the thickness and orientation of a layer (layered t-shell approach). The laminate is split into different element layers of one to eight winding layers and thicknesses from 0.835 mm to 2.7 mm (stacked t-shell approach) The combined approach is shown in Fig.5.

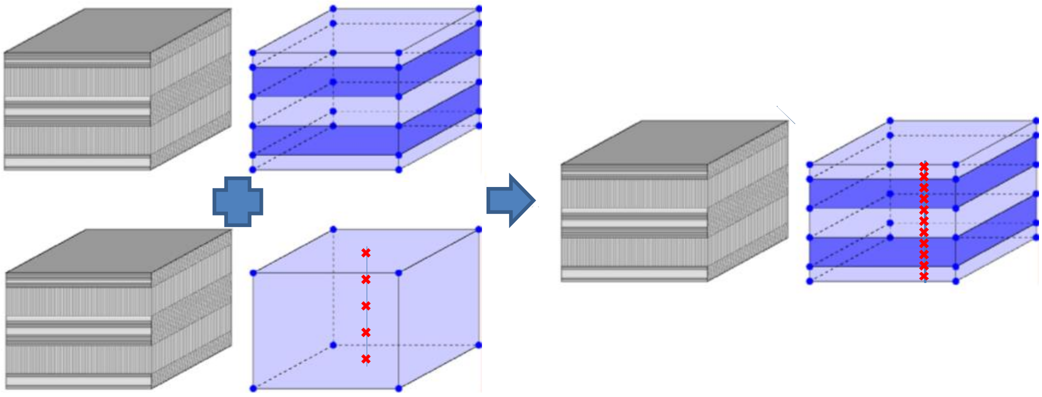

Fig.5: Combined stacked and layered t-shell approach on the basis of [9]

However, in this case the approach does not provide bonding between the element layers since no common nodes were defined. Instead, cohesive elements were placed between the element layers for the consideration of delamination (see 2.4). Since a cohesive layer between each pair of winding layers would lead to an overly complex model, a compromise between accuracy and simulation cost had to be made. However, no testing results that prove the onset of delamination in this area and hence the legitimacy of the simplification were available. Furthermore, for the interface between the liner and the FRP as well as for the interface between the two FRP materials no decohesian model is applied since no corresponding data was available, leading to further uncertainties of the delamination modelling.

2.3 Material modelling

For each component an appropriate material model was chosen depending on the applied material and assumed loading, damage and failure behaviour.

2.3.1 *Wound GFRP and CFRP*

For the wound materials the LS-DYNA material model **`*MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO`** (**`*MAT_262`**) was chosen. The main reasons for the choice are the good correlation that was achieved in a preliminary simulative assessment and the sophisticated physical based approach that makes the model on the one hand comprehensible and on the other hand convenient for data acquisition. **`*MAT_262`** is an orthotropic continuum damage model for laminated fibre reinforced composites and is based on the work of Maimí, Camanho, Mayugo and Dávila [10]. It is a constitutive model whose failure criteria are derived from physical findings and it considers simplified non-linear in-plane shear behaviour [11]. It uses maximum stress criteria in a plane stress assumption and a constant fibre misalignment instead of fibre kinking. The failure surfaces are not controlled by the model but assumed to be constant. It is implemented for solid, shell and thick shell elements [12].
As already mentioned in section 2.1.1 the mesh of the wound materials comprises also a number of tetra elements that are applied to fill small structural voids. However, **`*MAT_262`** is not implemented for this element form, so that a **`*MAT_ELASTIC`** definition is chosen for them instead.

2.3.2 *Liner*

The material model for the PA liner is defined comparatively simple. This is on the one hand based on the isotropic elastic/plastic behaviour and on the other hand based on the in comparison to the wound structure much lower thickness and stiffness. The material behaviour is assumed to be generally elastic. For this reason **`*MAT_ELASTIC`** is applied.

2.3.3 *Bosses*

As already mentioned the boss parts are considered to be undeformable. For this reason **`*MAT_RIGID`** is chosen.

2.4 Delamination modelling

The before described material model for composites is able to predict the onset and propagation of intra-laminar failure mechanisms such as fibre tension, fibre compression and transversal matrix failure in two fracture angles in respect to the thickness direction in the laminate. inter-laminar failure or crack caused by the loss of adhesion between two consecutive laminates, known under the term “delamination”, cannot explicitly be described within the composite material model. Although, delamination is one of the most common damage types in laminated fibre-reinforced plastic. In order to model this failure type appropriately two delamination modes are considered to be relevant within the tank model: mode I (opening delamination) and mode II (shear forward delamination). For this reason a decohesian or damage zone model is adopted. This damage model is based on the concept of the cohesive crack development model near the crack front. This model does not represent any physical material, but relates tractions to displacement jumps at an interface where a crack may occur. Damage initiation is related to the interfacial strength, i.e., the maximum traction. When the area under the traction-displacement jump relation is equal to the fracture toughness G, the traction is reduced to zero and new crack surfaces are formed (Fig.6).

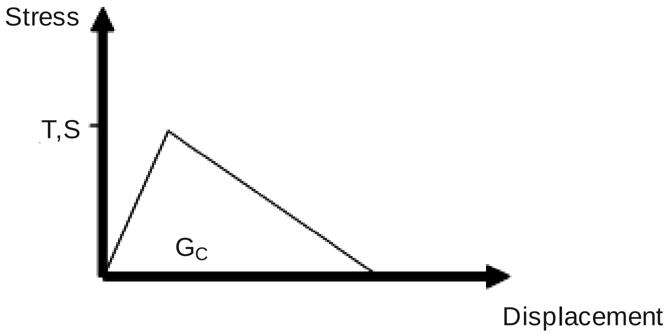

Fig.6: Bi-linear traction separation law for pure mode [13]

For the decohesive elements between the FRP layers **`*MAT_138`** (**`*MAT_COHESIVE_MIXED_MODE`**) is applied. Two different models are defined, one for the delamination of CFRP and one for GFRP.

2.5 Full tank model

The presented meshes, material models, laminate and delamination approaches for the constituent tank components are combined to a full tank model presented in Fig.7. All sub-components are combined via common nodes.

Fig.7: Full tank model (sectional view)

3 Validation approach

3.1 Material model validation

Conventionally, in order to obtain an adequate material model for unidirectional reinforced FRP laminates, first of all a number of quasi-static tests are conducted to obtain the basic material parameters. The test types depend on the values that are required by the material model. Typically, tests that investigate the tensile, compression and shear behaviour are conducted. Furthermore, tests that are aimed at the delamination behaviour should be foreseen if this failure type is relevant. Other tests that could be of interest are for example punch or bearing stress test. A good overview over tests

that are relevant for FRP materials is given in [14] and [15]. Here, the relevant regulations foresee flat specimens. However, it might be necessary to test ring shape structures in order to obtain results that are closer to the application to be analysed (e.g. pultruded tubes).
Besides the quasi-static tests also a test for a validation of the simulation approach is required in many cases. With the results derived from the program of quasi-static tests and a validation load case the process of material calibration and eventually validation can be carried out.
For the structures investigated in the MATISSE project a different approach is followed. This is mainly based on the applied production process used to manufacture the investigated Type IV CNG tanks, wet filament winding. This does not allow the production of flat specimens for tensile, compression or shear tests that are comparable to the material structure which is actually present in the tank's mantle. The production of flat test pieces by employing the filament winding process and the resulting material values are controversial issues being discussed among experts in this field. Another possibility is the production of ring shape specimens. Here, no unidirectional materials can be produced since the lay-down in the filament winding can only take place under longitudinal movement of the fibre. This would only allow unidirectional structures that are radial reinforced. This very weak laminate structure would not provide much information about the material properties in tensile or compression test.
For these reasons a completely different approach was executed for both materials, CFRP and GFRP. The material properties were acquired in a reverse FEM approach that is based on a quasi-static three point bending test (3PB) of a tubular specimen (see Fig.8). In order to have a broader test basis different winding structures were investigated (see Table 1:).

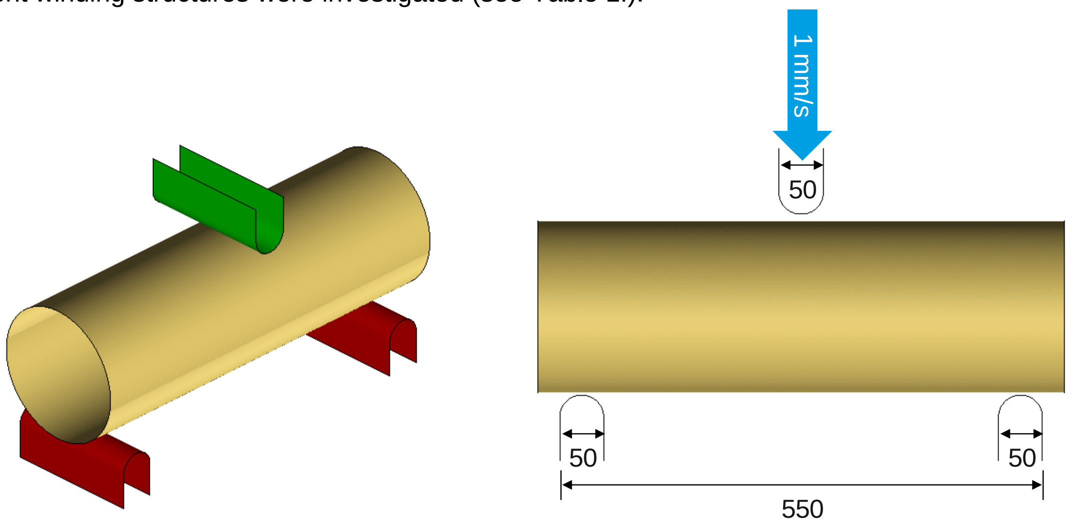

Fig.8: 3PB test

Set-up Type [-]	Layer [-]	Angle [°]	Layer Thickness [mm]	Laminate Thickness [mm]
1	1	+/-10	1.04	2.1
	2	+/-10	1.04	
2	1	+/-10	1.04	2.1
	2	90	0.53	
	3	90	0.53	
3	1	+/-30	1.05	1.6
	2	90	0.53	

Table 1: Specimen for 3PB

Another step of the approach was the assumption of the basic material values for a first loop. This was done on the basis of the fibre volume content measured by pyrolysis and the basic material properties of the constituents. These input values allow an estimation of material values from calculation as well as from literature values. Special focus was set on the elastic moduli, the failure stresses as well as strains and the fracture toughnesses for tension and compression in longitudinal and transversal direction as well as for shear loading.

With the preliminary material value set and the test results from the 3PB a validation process could be carried out. The procedure is presented in Fig.9. With the estimated material values the quasi-static 3PB was simulated for all three layer set-ups. The tube was modelled using the material model and element definition described in section 2. However, no delamination approach was applied since on the one hand no validated delamination model was available in this stage of the project and on the other hand the influence of delamination in the bending was considered to be secondary.
Additionally to the 3PB, the quasi-static coupon tests that could not be conducted in reality were simulated in order to evaluate the models plausibility in a first step. Here, three test procedures according to German standard regulations were modelled and calculated in accordance to the 3PB:

- DIN EN ISO 527: Tensile test [16]
- DIN EN ISO 14126: Compression test [17]
- DIN EN ISO 14129: Tensile test for shear properties [18]

The simulation results were then evaluated concerning force, displacement, stress, strain and failure behaviour.

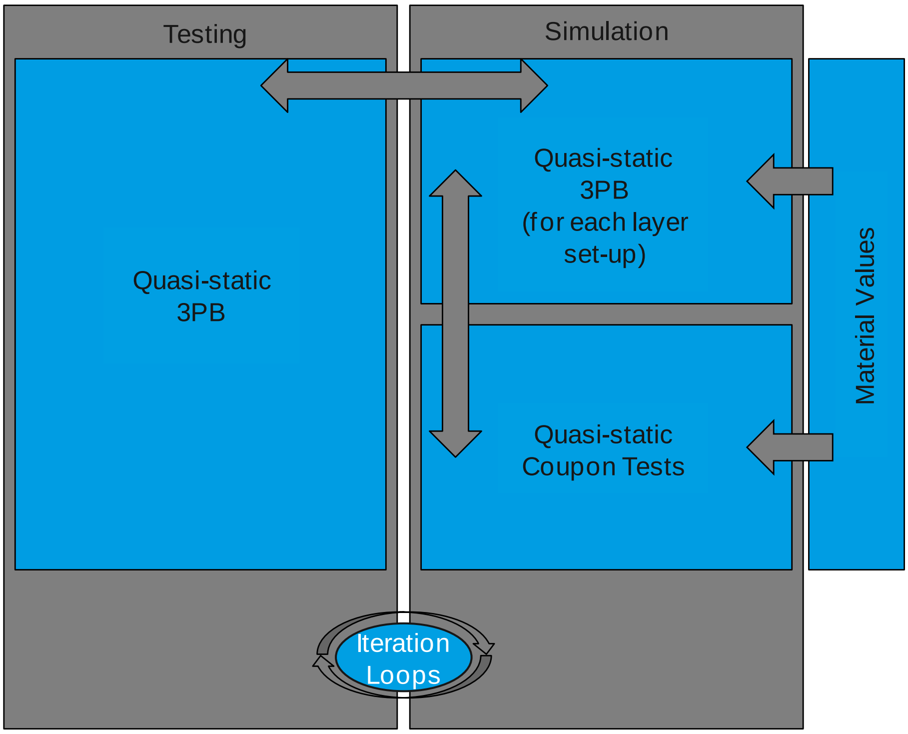

Fig.9: Material validation process for thick UD structures of MATISSE

After several loops of evaluation, reconsideration and adaption of values, for both FRP materials a **`*MAT_262`** card was acquired that is on the one hand suitable to achieve good accordance between simulation and tests for the 3PB and is on the other hand plausible in terms of the simulation of the considered quasi-static coupon tests. Fig.10 shows an exemplary result for the simulation of 3PB on a GFRP tube with set-up type 2, Fig.11 shows the results for the same set-up for CFRP. Furthermore, the model for the liner was validated similar on a 3PB test of a PA tube.

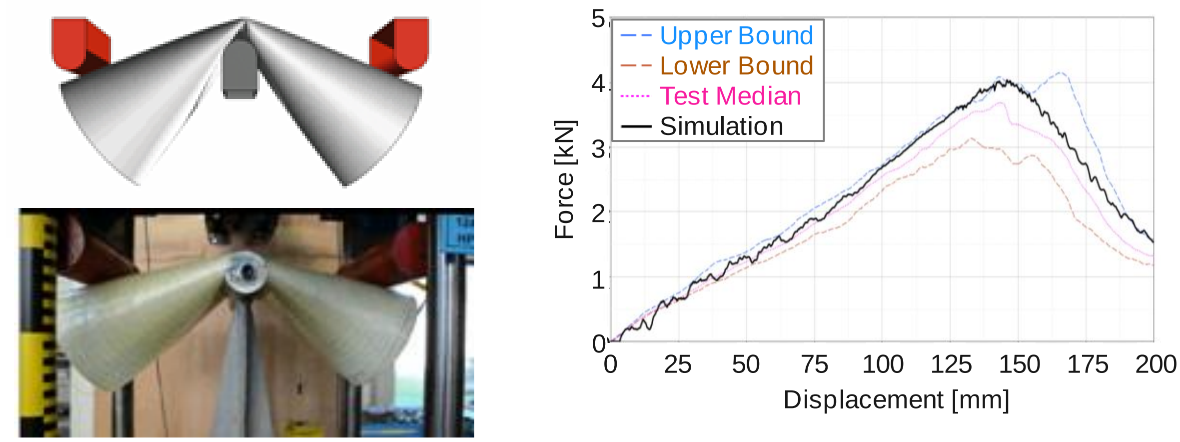

Fig.10: Comparison of test and simulation of 3PB on tubular GFRP specimen with set-up type 2

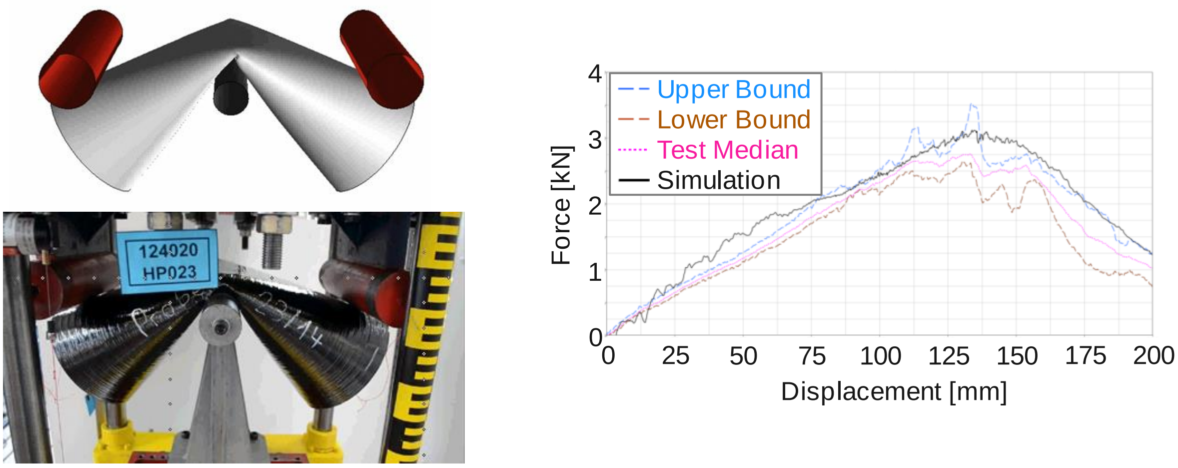

Fig.11: Comparison of test and simulation of 3PB on tubular CFRP specimen with set-up type 2

3.2 Decohesive model validation

Two tests configurations have been chosen for calibration and validation process in order to find the suitable parameter for the decohesian modelling. These tests are the Double Cantilever Beam (DCB) test to represent a pure mode I delamination (opening mode) and the End Notched Flexure (ENF) test to represent a pure mode II delamination (forward shear mode). Here again, no test result is available as reference and therefore an analytical approach based on work of [19] is used for the calibration and validation. This analytical approach is only applicable for delamination of the material with the same mechanical properties; therefore decohesian modelling between the material pair CFRP-GFRP is neglected. In the calibration process, variations of the penalty stiffness values (EN/ET) and peak traction values (T/S) in the material model were examined until proper simulation accuracy with analytical approach was reached.

3.2.1 *Mode I*

Fig.12 shows the general test set-up of the DCB test. The test specimen is 200 mm long (l), 20 mm wide (b) and composed of two thick beams (h = 1.55 mm) made of thick shell elements. Between these beams a thin cohesive layer (~ 0.01 mm) is inserted with an initial crack length (a_0) of 30 mm. It is already verified in diverse publications [20] that cohesive zone modelling has a strong mesh size dependency, so that in terms of the calibration process, the test specimen is built up in three different mesh sizes (1.0 mm, 2.0 mm and 3.0 mm), which is based on the mesh size range included in the stage two CNG tank FE model. The beam is modelled as isotropic with the Young's modulus in fibre direction (EA) of the composite materials as reference for **`*MAT_ELASTIC`**. The separation progress is triggered by applying a constant displacement rate of 10 mm/s in opposite direction at the beam edge nodes. **`*CONTACT_ERODING_SINGLE_SURFACE`** is used to maintain the contact between the beams after failure of cohesive elements.

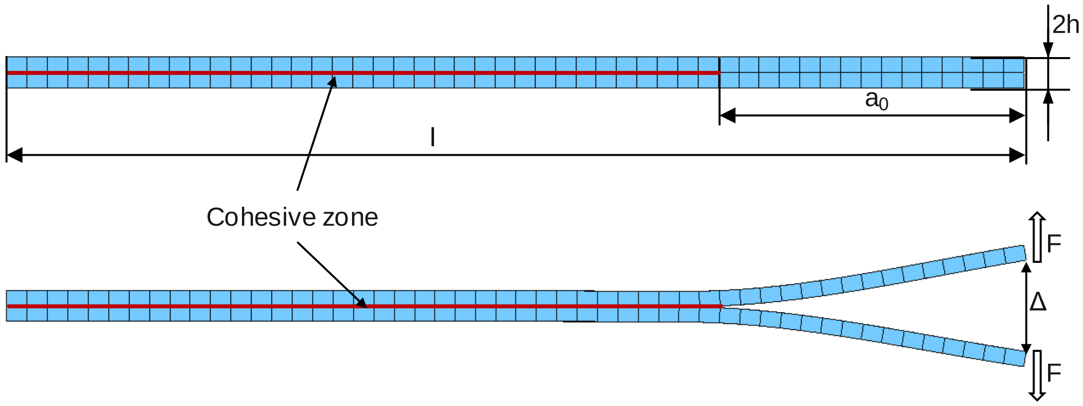

Fig.12: FE model of DCB test specimen in undeformed (top) and deformed (bottom) shape

As already mentioned before, a theoretical approach is used as reference for the calibration and validation process. Based on work of [19], the analytical approach for DCB test uses the beam theory for the analytical linear part and the constant energy release rate approach as analytical delamination propagation. The simulation results show that the finer the model is meshed, the better the simulation results are. The predominate mesh size of the target tank model is 3.0 mm. For this reason the corresponding results for the two FRP materials are presented in Fig.13 and Fig.14.

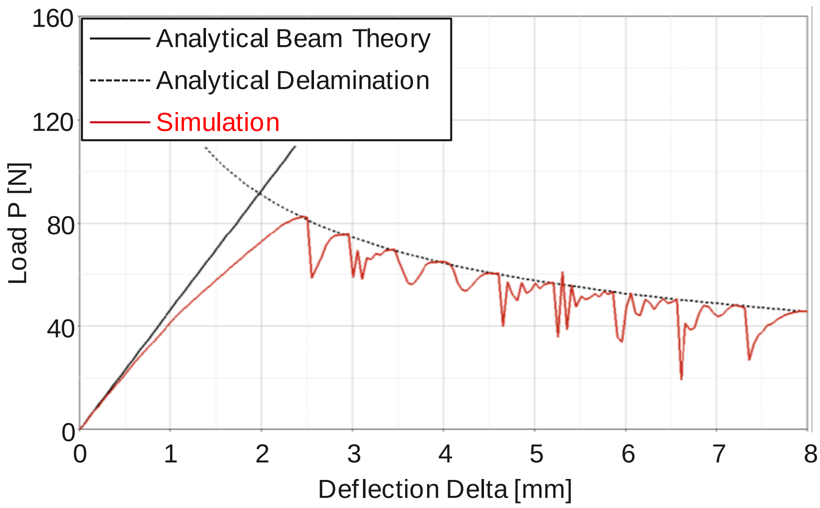

Fig.13: Simulation result of DCB test of CFRP with an element size of 3.0 mm

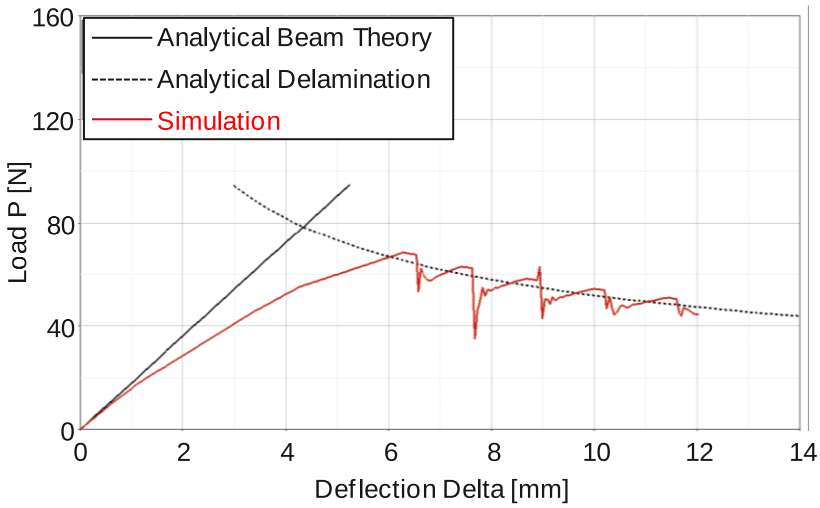

Fig.14: Simulation result of DCB test of GFRP with an element size of 3.0 mm

3.2.2 *Mode II*

Fig.15 shows the general test set-up of the ENF test. The test specimen set-up including its initial crack length is identical with the test specimen for the DCB test. In this test a constant displacement rate of 10 mm/s is applied in the middle of the upper beam. A nodal constraint is applied on the edge nodes of the lower beam which is fixed in all degree of freedom on the left side. On the initial crack side only translation in x-direction is possible. The analytical approach for the ENF test is also based on the beam-theory solution with three reference curves [19]. The general simulation procedure is the same as for mode I. The corresponding results for the two FRP materials are presented in Fig.16 and Fig.17.

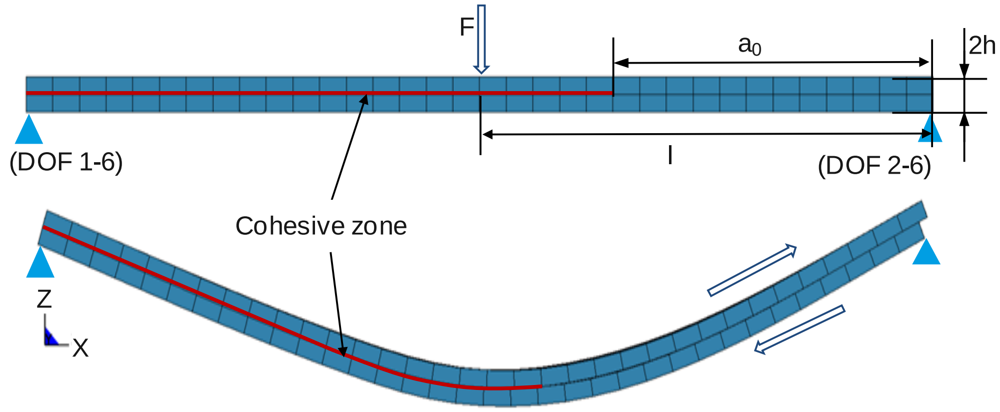

Fig.15: FE model of ENF test specimen in undeformed (top) and deformed (bottom) shape

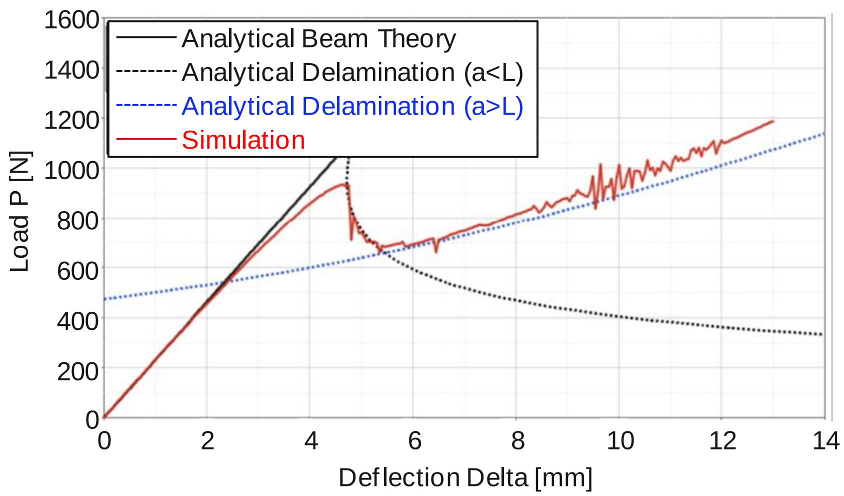

Fig.16: Simulation result of ENF test of CFRP with an element of 3.0 mm

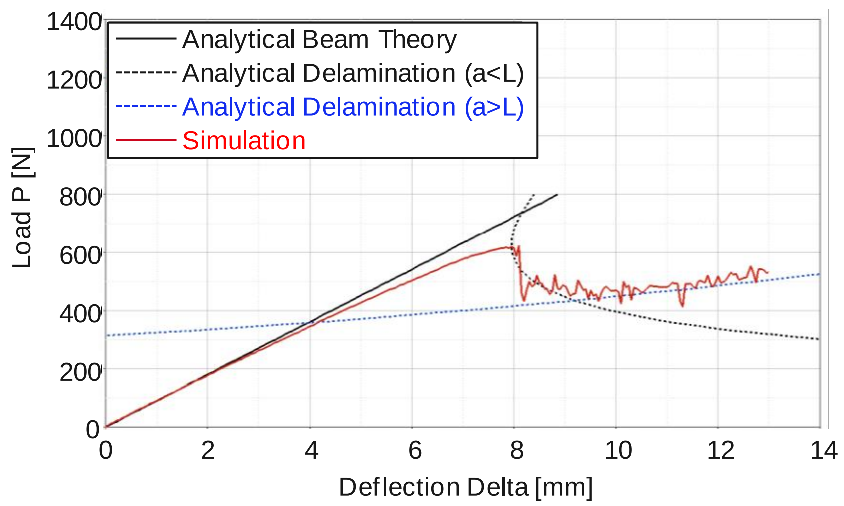

Fig.17: Simulation result of ENF test of GFRP with an element size of 3.0 mm

4 Validation on tank level

In order to verify the validity of the combination of the validated simulation models of the sub-components a further validation of the model of the full tank has to be carried out in a following step. For this reason a number of tests have been conducted on the tank level and are currently evaluated and compared to the simulation results.

As a simple indicator for the validity of the tank model simple pressurisation tests were carried out. At first tanks were filled with air to the working pressure of 200 bar and subsequently pressurised until burst. For the validation of the simulation model, besides general plausibility, the tank's extension in radial and longitudinal as well as the pressure at burst is evaluated.

Furthermore, starting from full vehicle crash simulations a suitable test rig for a crash test facility was developed within MATISSE's VTM. It was designed as a flexible set-up that allows the application of different tank sizes and mounting systems (e.g. belly and neck mounting) as well as different impact orientations, positions, masses, surfaces and velocities. Based on a set of different vehicle to vehicle collisions a number of impact configurations was defined. Among these test-rig configurations, four were identified as the most relevant ones and subsequently, for complexity reasons, only this subset of configurations was tested. Within the test rig the specimen is mounted as it is applied in the vehicle but, however, upside down, since this allows flexibility and practicability of the test rig set-up on a crash test facility. The general test rig set-up is presented in Fig.18Fig.18:, the test configurations in Fig.19.

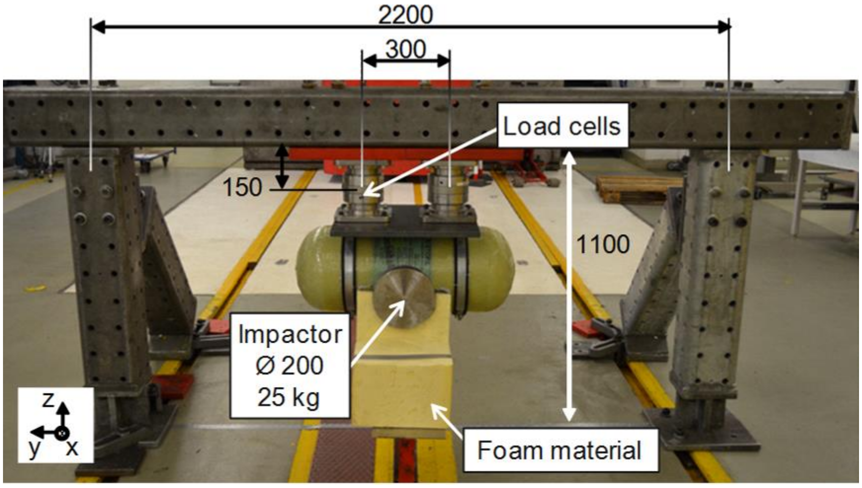

Fig.18: Test rig for impact tests on tank level

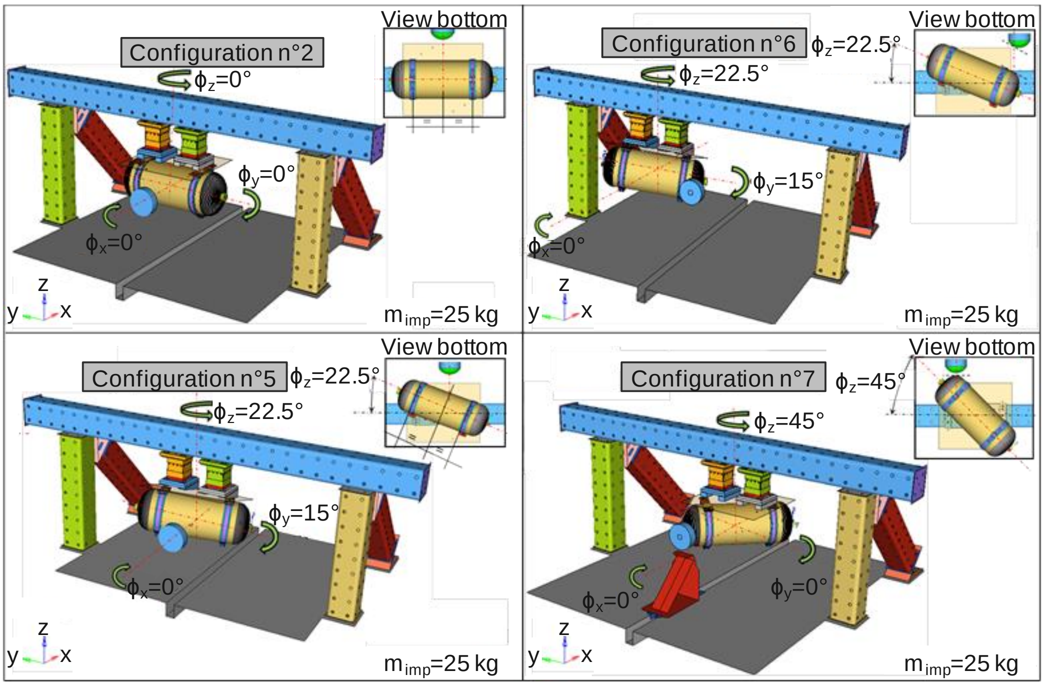

Fig.19: Impact configurations for test programme

After testing several evaluations were carried out. These were on the macroscopic level the analysis of force and acceleration curves measured on the test rig as well as optical checks of tank outer and inner surface and of the mounting straps. On the microscopic level light microscopy and computed tomography (CT) of cut-out specimens were carried out. In order to verify the simulation of the failure mechanisms, currently an evaluation and comparison of simulation and test is carried out. On the one hand the macroscopic test behaviour is analysed while on the other hand failure and damage mechanisms are investigated. This procedure is presented qualitatively in Fig.20.

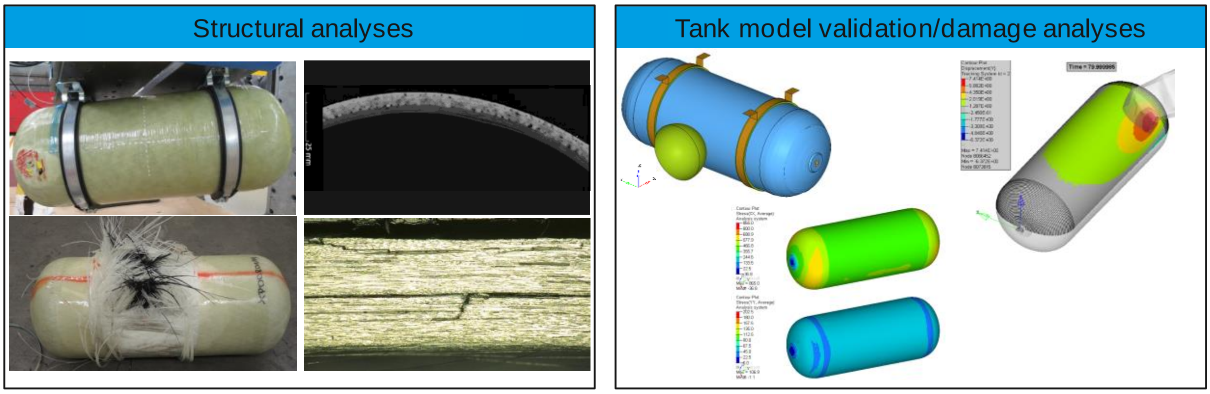

Fig.20: Damage evaluation

5 Simplification of tank model

The model presented before is a very detailed and complex model, taking in account numerous properties of the CNG tanks. It can be considered as a useable tool for a component simulation in a reduced analysis system but as too detailed for a reasonable simulation of a full vehicle impact in terms of simulation time and amount of data. For this reason currently a simplified but still significant model is in development.

In order to achieve this goal an iterative simplification process is applied as presented in Fig.21. Here, five general steps of model simplification are foreseen that are sequentially carried out. After each step, simulation results for pressurisation and impact are evaluated concerning simulation quality in comparison to the basis model. However, losses in the simulation accuracy are expected and in many cases inevitable. If all model quality criteria that are previously defined are met, the next step of simplification is carried out. If the results are not sufficient, the step is revised or dropped. In this case the following step is carried out on the model of the previous step.

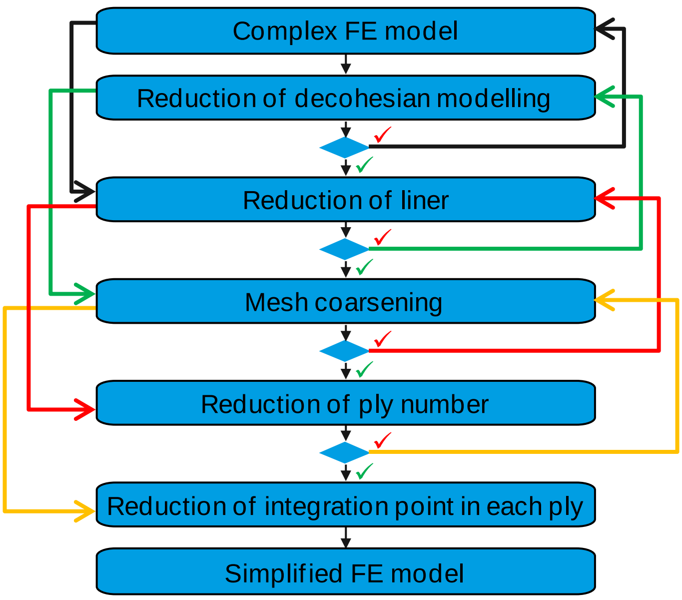

Fig.21: Iterative simplification process

6 Summary

For the means of a virtual testing methodology an LS-DYNA model of a Type IV CNG tank with wound CFRP and GFRP layers was developed. Each sub-component of the tank was modelled appropriately. For the aluminium bosses a rigid model in combination with solid elements was chosen; the polymeric inner liner was modelled elastically with a shell mesh. The modelling of wound FRP layers was considered to be the essential part of the tank model. Thus, a very complex model was applied, making use of an orthotropic continuum damage model that considers a number of physically based failure and damage criteria. In order to accurately model the winding structure and to consider delamination as well, a so-called combined stacked and layered t-shell approach was applied. This comprises t-shell elements as well as decohesive elements with a special decohesive material model.
Because of a lack of the availability of significant test specimens or respectively test procedures to estimate the material values for the FRP structure a so-called reverse FEM approach was applied in order to validate the material models for the wound CFRP and GFRP material. This comprised the testing and simulation of tubes with representative winding structures to validate a number of calculated, researched and assumed material values. Furthermore, a set of test procedures was carried out on the virtual level in order to monitor the plausibility of the material behaviour. Additionally the material model for the liner was validated on the basis of a 3PB. For the validation of the decohesive model no test results were available and consequently the validation of the model was done theoretically. Therefore, force deflection curves of the simulation of two test procedures considering relevant delamination modes and curves calculated using fundamental mechanic assumptions were compared. All sub-component models were combined in a full tank model.
Furthermore, a number of tests on the tank level were conducted that are based on the tank's operating conditions and potential impact conditions based on full vehicle scenarios.
In order to validate the full tank model, evaluations and comparisons of the conducted tests and their virtual counterparts are currently ongoing. Special focus is set on the accurate evaluation of material damage of the wound layers.
In a final step a simplification of the complex model is carried out, allowing an application in a full vehicle simulation with reasonable simulation costs. For this reason an iterative simplification procedure is applied.

7 Acknowledgement

All the presented work was carried out within the project MATISSE funded by European Commission's 7th Framework Programme.

The authors would like to acknowledge the MATISSE partners Xperion Energy & Environment GmbH and Centro Ricerche Fiat S.C.p.A. for the collaboration since the very beginning of the project.

8 Literature

[1] Eichhorn, U.; "Mobility for the Future"; Proceedings, 21st Aachen Colloquium Automobile and Engine Technology; 2012
[2] Wu, J., Dunn, S. C.; "Environmentally responsible logistics systems"; International Journal of Physical Distribution & Logistics Management, Vol. 25 Iss.: 2, 1995; pp.20 – 38
[3] N.N.; "Fortschrittsbericht der Nationalen Plattform Elektromobilität (Dritter Bericht)"; Bundesministerium für Umwelt, Naturschutz, Bau und Reaktorsicherheit, Germany; 2012
[4] da Silveria, B. H., Benvenuti, L. H., Werninghaus, E., Coelho, E. P. D., Silva, C. D., de Olivera, F. R., Silva, J. C. S.;"Development of a Concept Vehicle for Compressed Natural Gas"; SAE Technical Paper Series, Paper No. 2004-01-3452; 2004
[5] Gaudio, R., Volpi, E.; "Position Paper: Natural Gas and CO2 - Natural gas is a champion in road transport and also saving CO2 emissions"; Position Paper of NGVA Europe; 2009
[6] Hansen, G. P., Sato, M., Yan, Y.; "Pressure Vessels for Hydrogen Vehicles: An OEM Perspective"; International Hydrogen Fuel and Pressure Vessel Forum, Beijing; 2010
[7] Liu, G. R., Quek, S. S.; "Finite Element Method – A Practical Course"; Elsevier B.V.; 2003
[8] Stelzmann, U., Stühmeyer, A.; "Einführung in LS-DYNA und ANSYS/LS-DYNA" CADFEM GmbH; 2012

[9] Maier, M., Schweizerhof, K., Schmeer, S., Magin, M., Mattern, S.; "Statusbericht zur Berechnung von CFK-Strukturen im Automobilbau"; Forschungsvereinigung Automobiltechnik; 2007

[10] Maimi, P., Camanho, P, Mayugo, J., Davila, C.; "A continuum damage model for composite laminates: Part II – Computational implementation and validation"; Mechanics of Materials; Iss.: 39; 2007; pp. 909-919

[11] N.N.; "LS-DYNA Keyword User's Manual - Volume II Material Models"; Revision 3372 (Draft); Livermore Software Technology Corporation; 2013

[12] Hartmann, S.; "Neue Materialmodelle für Composites in LS-DYNA; Infotag: Composite Berechnung mit LS-DYNA"; 2013

[13] Marzi, S., Ramon-Villalonga, L., Poklitar, M., Kleiner, F.; "Usage of Cohesive Elements in Crash Analysis of Large, Bonded Vehicle Structures – Experimental Tests and Simulation"; LS-DYNA Anwenderforum; Bamberg; 2008

[14] Fell, S., Schütz, T., Schultheis, V., Kang, S., Gama, B.; "Crash Modeling of High-Pressure Wet Wound Composite Vessels"; SAE International, 2001

[15] N.N.; "AVK-TV Handbuch - Faserverstärkte Kunststoffe und duroplastische Formmassen"; Arbeitsgemeinschaf Verstärkte Kunststoffe – Technische Vereinigung e.V.; 2004

[16] N.N.; "DIN EN ISO 527-1 - Kunststoffe – Bestimmung der Zugeigenschaften - Teil 1: Allgemeine Grundsätze"; Beuth Verlag; 1996

[17] N.N.; "DIN EN ISO 14126 - Bestimmung der Druckeigenschaften in der Laminatebene"; Beuth Verlag; 2000

[18] N.N.; "DIN EN ISO 14129 - Zugversuch an 45°-Laminaten zur Bestimmung der Schubspannung/Schubverformungs-Kurve, des Schubmoduls in der Lagenebene"; Beuth Verlag; 1998

[19] Mi, Y., Crisfeld, M. A., Davies, G. A. O., Hellweg, H. B.; "Progressive Delamination Using Interface Elements"; Journal of Composite Materials; Iss.: 32; 1998 ; p.p. 1246-1272

[20] Feucht, M., Haufe, A.; "Modeling connections with LS-DYNA"; DYNAmore GmbH; 2008