MODELLING OF WOUND, THICK UD COMPOSITES FOR HIGH-SPEED IMPACTS AND SUBSEQUENT DAMAGE EVALUATION

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

[1]Forschungsgesellschaft Kraftfahrwesen mbH Aachen
Steinbachstraße 7, D-52074 Aachen, Germany
Email: matheis@fka.de, web page: http://www.fka.de

[2]DYNAmore Nordic AB
Brigadgatan 14, SE-58758 Linköping, Sweden
Email: thomas.johansson@dynamore.se, web page: http://www.dynamore.se

Keywords: Damage evaluation, FEM simulation, Filament winding, Thick UD composites, Type IV CNG tank

ABSTRACT

The project MATISSE funded by European Commission's 7th Framework Programme aims to make a significant step forward in the capability of the automotive industry to model, predict and optimise the crash behaviour of mass produced fibre reinforced polymers (FRP) with the focus set on components for alternatively powered vehicles (APV). One of the project's main research goals is the development of a general virtual testing methodology (VTM) for the development of APV driven by compressed natural gas (CNG) stored in type IV pressure vessels made of hybrid composite material.

In this context a modelling approach for a reference tank system produced via filament winding and making use of carbon as well as glass fibre reinforced polymers (CFRP, GFRP) with aluminium bosses and a polymeric liner, is developed. The modelling bases on a "reverse finite element method (FEM)" approach 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 are applied. The values as well as the complete modelling approach are validated on the macroscopic scale on the basis of three point bending tests on CFRP and respectively GFRP tubes.

The modelling approach is based on a combination of stacked and layered thick-shell (t-shell) laminated set up, combining stacks of same orientated layers within t-shell element layers that are themselves bounded with releasable cohesive elements. The material model itself is an orthotropic continuum damage model that uses physically based failure criteria based on the work of Camanho et. al..

The validated model for the wound FRP structure is applied in a complex full tank model that also comprises the bosses, which are assumed to be rigid, and the liner that is elastically modelled. The composite layer modelling is carried out in a ply based laminate modelling which allows defining the layer definition on element basis.

The tank model is subsequently verified on pressurisation and impact tests conducted on physical tanks. Concerning the pressurisation general plausibility of the model as well as stresses and strains are investigated. For the impacts the force and deformation as well as the damage area predictability are evaluated concerning the model's accuracy.

In a last step several measures of model simplifications are carried out on the tank model in order to achieve reasonable simulation costs with the perspective of application in full vehicle simulations. Each simplification step is evaluated concerning the benefit in calculation time and potential losses of predictability and accuracy.

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 drivetrains, 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 drivetrains [3].

An alternatively powered drivetrain 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 an 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].

One of the major development goals within the automotive industry is the safety of vehicle passengers and possible opponents in a crash case. In this regard the finite element method (FEM) with explicit time integration is the core tool for an adequate and efficient structural vehicle development. Moreover, it can be stated that a validated simulation approach which is on the one hand reliable and on the other hand efficient in terms of simulation costs may be the pivotal criterion for the vehicle application of a material [7]. Beside other topics, the project MATISSE funded by European Comission's 7th Framework Programme aims at the improvement of the capability of the prediction and optimisation of the crash behaviour of Type IV CNG tanks leading to a model that is suitable for full vehicle analyses. For this reason, for such components a simulation approach was developed as part of a general virtual testing methodology (VTM). The VTM is demonstrated on reference 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 an aluminium boss at both tank poles was used. The VTM comprises a detailed model for an analysis of simplified tests on the tank level as well as a simplified model that is suitable for the application in a full vehicle analysis. The reference structure was provided by the MATISSE partner Xperion Energy & Environment GmbH and is presented in Figure 1. All presented work is carried out for the solver LS-DYNA V971 R7.0.

Figure 1: Type IV CNG tank.

2 MODELLING APPROACH

In order to evaluate the impact behaviour of the tank a suitable model should on the one hand consider all relevant damage or failure criteria but should on the other hand macroscopically consider the full tank for analyses of general impact incidents. For this reason a modelling approach is proposed that assumes several simplifications but is rather detailed from a crash simulation point of view.

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.

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 thick shell elements (t-shell). This eight node hexahedral or respectively six node pentahedral element type is based on Liu [8]. It allows on the one hand an efficiency that is comparable to thin shell elements and on the other hand meshes that allow a higher geometry accuracy than this shell type. Furthermore, contrary to solid elements an integration point-wise layer definition is possibly. In LS-DYNA these layers can be defined in a ply-based composite model which is able to define laminate properties (material and fibre angle) in each element using the formulation *ELEMENT_TSHELL_COMPOSITE. For the tank model approach the t-shell type 2 is defined via the property definition *SECTION_TSHELL. This element type assumes selective reduced in-plane integration. Here, for each number of same-oriented unidirectional (UD) layers one element layer is defined. Additionally, the model contains an interface between the t-shell element layers that is modelled with eight node solid cohesive elements with zero thickness of the special cohesive type 19 in the property definition *SECTION_SOLID. Furthermore, a number of constant stress tetrahedral solid elements were 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 Figure 2.

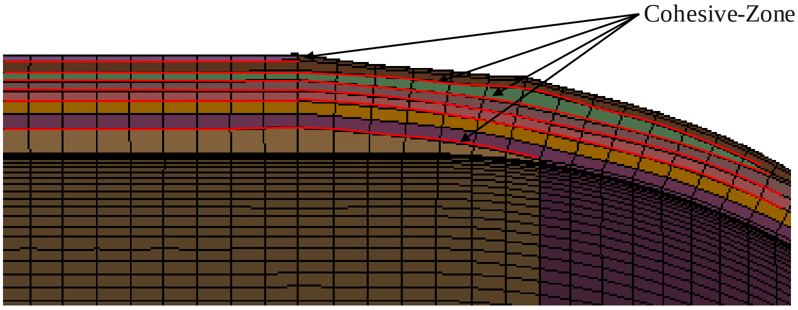

Figure 2: Sectional view of the mesh of the winding structure.

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 was defined in the property card. The element area geometry and size is congruent to the mesh of the wound structure and has common nodes with these t-shell elements. The liner mesh is shown in Figure 3.

Figure 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 constant stress tetrahedral solid elements. The boss components for both poles of the tank are shown in Figure 4.

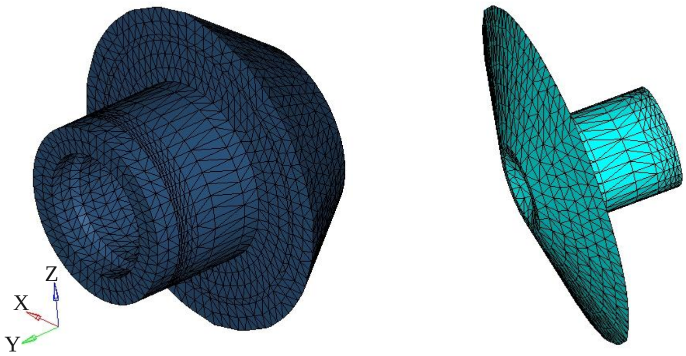

Figure 4: Mesh of bosses.

2.2 Laminate set-up

For the wound structure (GFRP as well as CFRP) a combination of a layered thick shell and a 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 global laminate is further split into different element layers of one to eight winding layers of same orientation (stacked t-shell approach). The combined approach is shown in Figure 5.

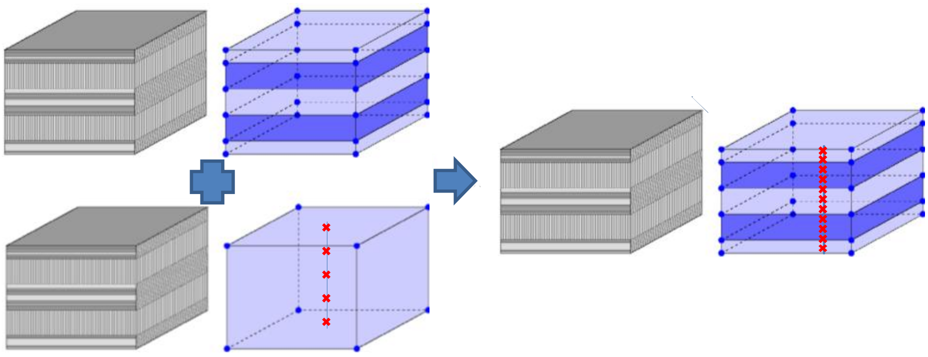

Figure 5: Combined stacked and layered t-shell approach on the basis of [9].

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 section as well as for the interface between CFRP and GFRP no decohesian model is applied since no corresponding data was available.

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. *MAT_262 has been implemented in release R7.0 of LS-DYNA V971. It 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 consider a simplified non-linear in-plane shear behavior. The constitutive law is established on the basis of scalar function that is corresponding to the complementary free energy density in the material. It considers damage variables that are calculated using the material's fracture toughness. The failure criteria are coupled and based on a plane stress assumption and a constant fibre misalignment instead of fibre kinking. The failure surface of *MAT_262 considers four different sub-surfaces for the failure mechanisms fibre tension, fibre compression and two transversal matrix failure effects, under 0° as well as under 53°. As long as the stress state is located within the failure surface the model behaves orthotropic elastic [11]. The model is implemented for solid, shell and thick shell elements [12].

The material model showed a good correlation in a preliminary simulative assessment and the sophisticated physical based approach makes the model on the one hand comprehensible and on the other hand convenient for data acquisition. Thus, it was considered to be suitable for the approach followed within MATISSE.

As already mentioned in section 2.1.1, the mesh of the wound materials comprises also a number of tetrahedral solid elements that are applied to fill small structural voids. However, *MAT_262 is not implemented for this element form, so that an isotropic hypoelastic definition (*MAT_ELASTIC) was 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 assumed low influence on the overall behaviour due to its low 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 the corresponding LS-DYNA model *MAT_RIGID that defines a rigid body is chosen.

2.4 Delamination modelling

The before describe material model *MAT_262 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 with 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 (*MAT_138). This damage model is based on the concept of the cohesive crack development model near the crack front. It does not represent any physical material, but relates tractions to displacement 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 relation is equal to the energy release rat G, the traction is reduced to zero and new crack surfaces are formed [11]. The traction separation law is illustrated in Figure 6. Two different models are defined, one for the delamination of CFRP and one for GFRP.

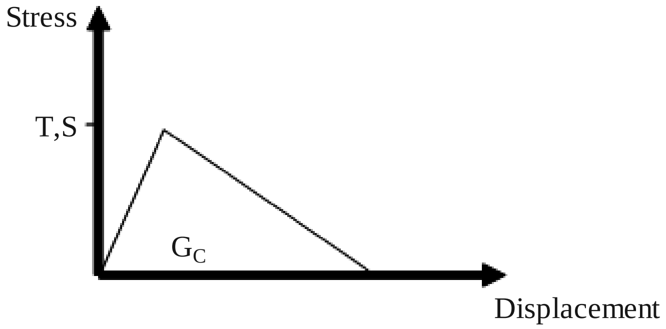

Figure 6: Bi-linear traction separation law for pure mode [13].

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 Figure 7. All sub-components are combined via common nodes.

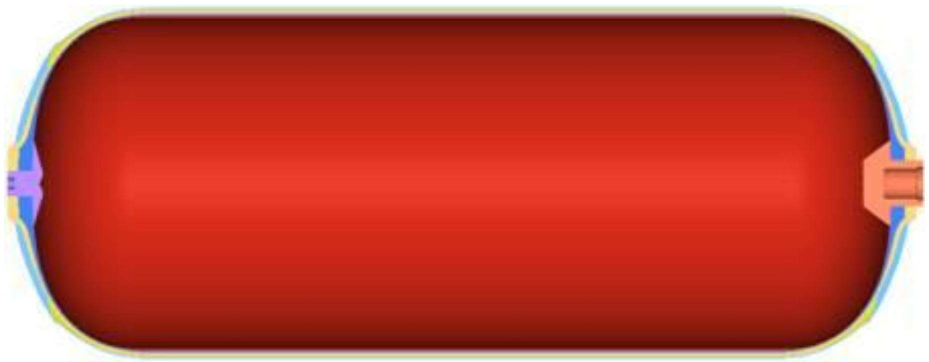

Figure 7: Full tank model.

3 VALIDATION APPROACH

3.3 Material model validation

Conventionally, in the field of crash modelling 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, test 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. Typically, 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 specimen for tensile, compression or shear tests, 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 winding experts. The production of ring shape specimens for wound structures can also be evaluated as unsuitable. 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. Such a laminate structure is very weak in the other material directions and would not provide much information about the material properties in tensile or compression test.

For these reasons, in MATISSE a completely different approach was executed for the CFRP and GFRP material. 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 Figure 8). In order to have a broader test basis, different winding structures were investigated (see Table 1).

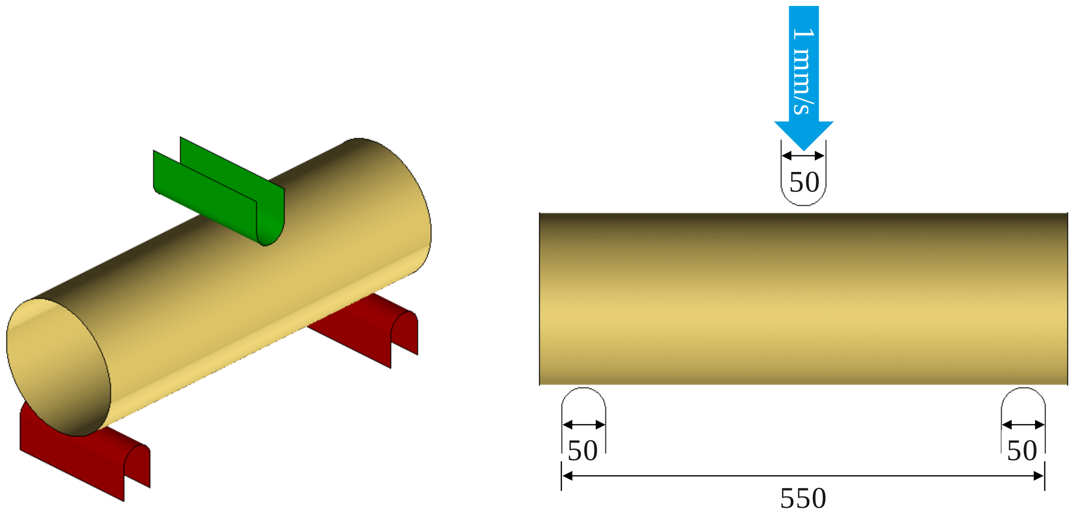

Figure 8: 3PB test.

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

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. 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, quasi-static coupon tests that could not be conducted in reality were simulated for tensile compression and shear in order to evaluate the models plausibility in a first step. The simulation results were then evaluated concerning force, displacement, stress, strain and failure behaviour.

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. Figure 9 shows an exemplary result for the simulation of 3PB on a GFRP tube with set-up type 2, Figure 10 shows the results for the same set-up for CFRP. Furthermore, the model for the liner was validated in similar way on a 3PB test of a PA tube.

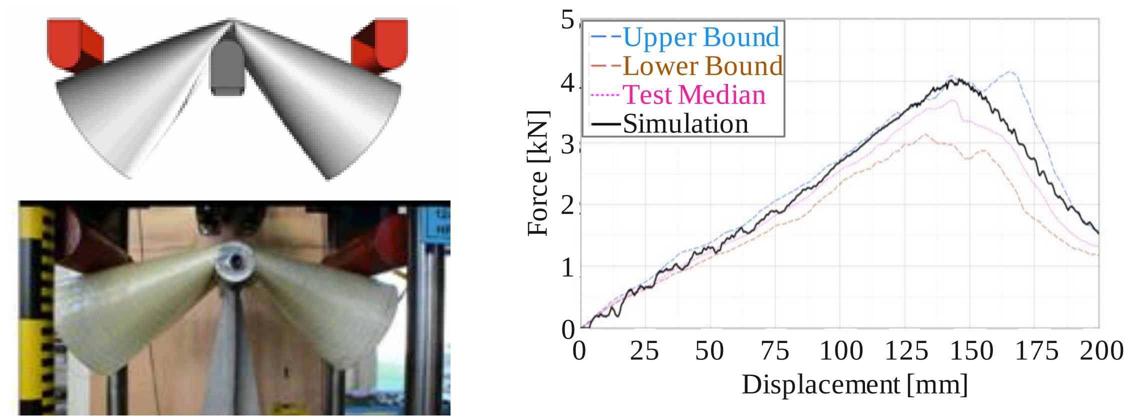

Figure 9: Comparison of test and simulation of 3PB on tubular GFRP specimen with set-up type 2.

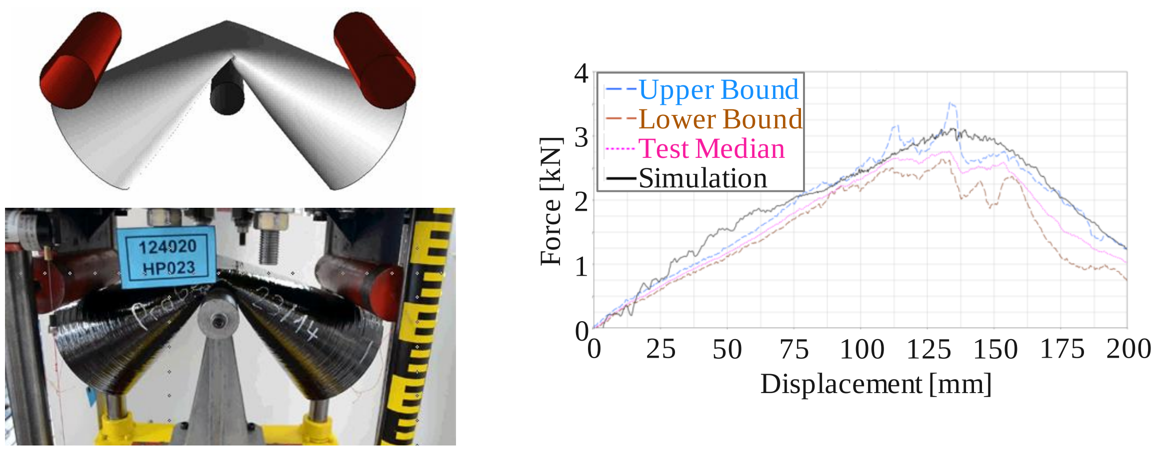

Figure 10: Comparison of test and simulation of 3PB on tubular CFRP specimen with set-up type 2.

3.4 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 the work of [14] is used for the calibration and validation. This analytical approach is only applicable for delamination of laminate layers 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.

3.4.1 Mode I

Figure 11 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 [15] 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 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 the *MAT_ELASTIC model. The separation progress is triggered by applying a constant displacement rate of 10 mm/s in opposite direction at the beam edge nodes.

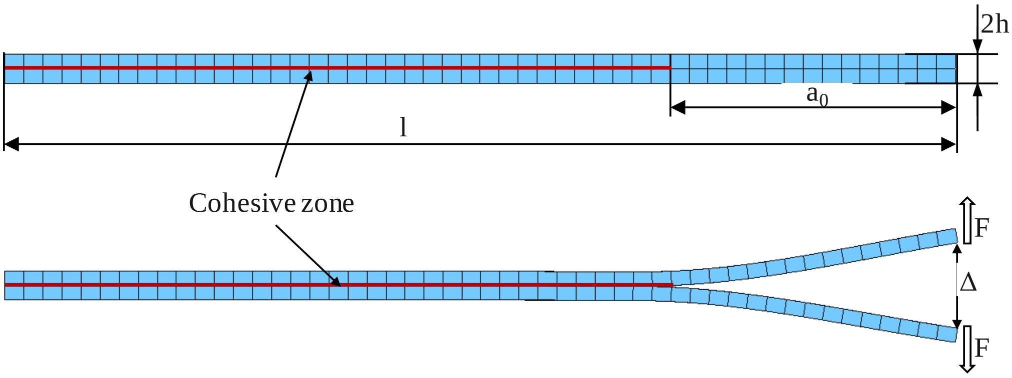

Figure 11: FE model of DCB test specimen in undeformed (top) and deformed (bottom) shape.

As already mentioned before, a theoretical approach is used as reference for the calibration and validation process. Based on work of [14], 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 Figure 12.

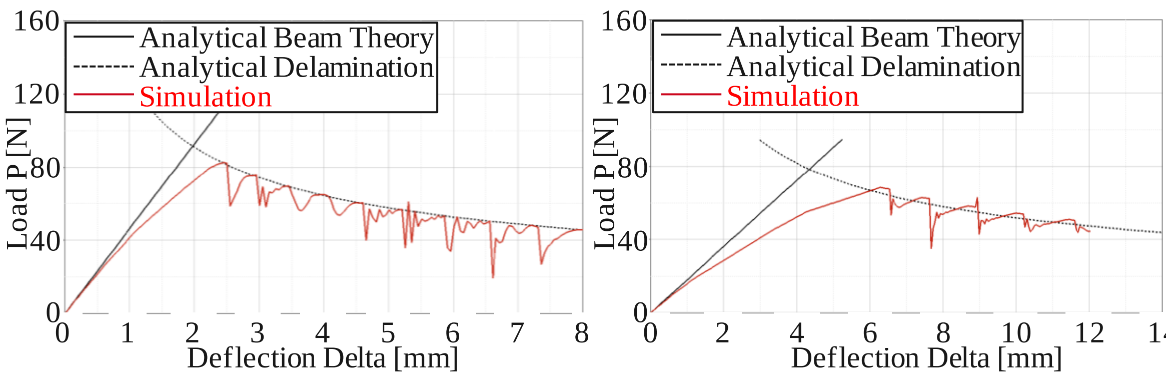

Figure 12: Simulation result of DCB test of CFRP (left) and GFRP with an element size of 3.0 mm.

3.4.2 Mode II

Figure 13 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 degrees of freedom on the left side. On the initial crack side only translation in x-direction is possible.

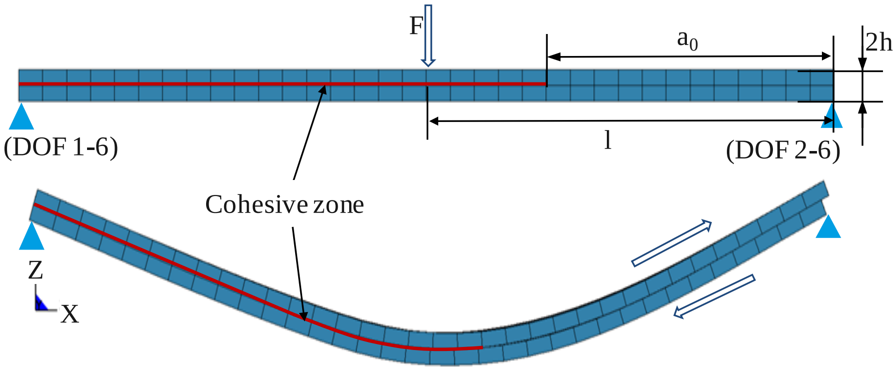

Figure 13: FE model of ENF test specimen in undeformed (top) and deformed (bottom) shape.

The analytical approach for the ENF test is also based on the beam-theory solution with three reference curves [14]. The general simulation procedure is the same as for mode I. The corresponding results for the two FRP materials are presented in Figure 14.

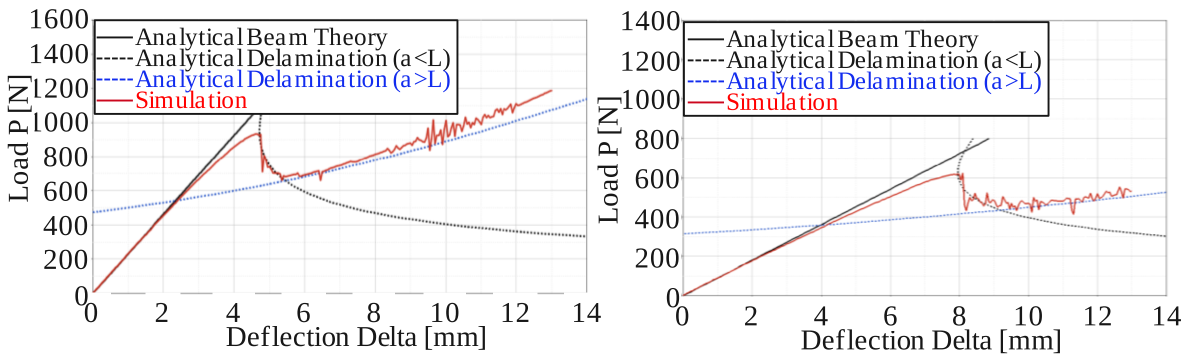

Figure 14: Simulation result of ENF test of CFRP (left) and GFRP with an element of 3.0 mm.

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 expansion 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 have been 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 Figure 15.

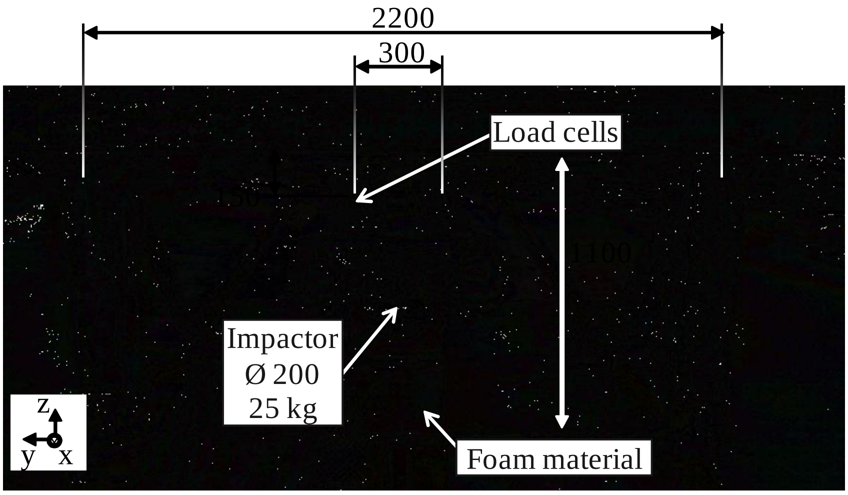

Figure 15: Test rig for impact tests on tank level.

Several evaluations of the test results 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 Figure 16.

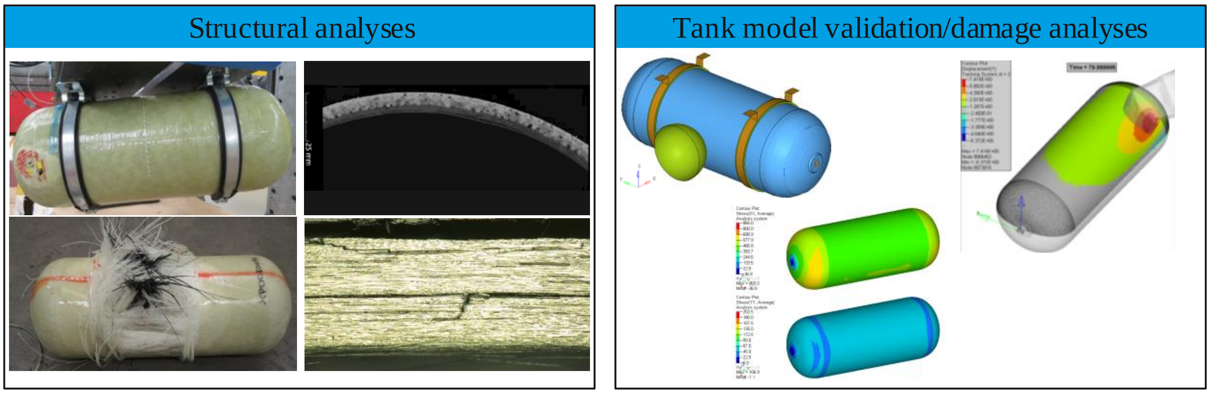

Figure 16: Damage evaluation.

3 SIMPLIFICATION OF TANK MODEL

The model presented before is in terms of crash simulation very detailed and complex. 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. For this reason several steps of simplification are carried out and subsequently the loss in accuracy is evaluated.

4 CONCLUSION

A modelling approach for the crash analysis of Type IV CNG tanks made by filament winding was proposed in this contribution. The approach allows the damage evaluation on a full tank level and considers the failure mechanisms fibre tension, fibre compression and two transversal matrix failure effects as well as a simplified delamination. Furthermore, an unconventional validation approach is proposed, allowing material modelling of wound materials for structures that cannot adequately be captured with conventional flat or ring specimens. The approach is currently validated against burst and impact tests. For the means of impact loading a special simplified crash test programme was derived from full vehicle simulations and subsequently carried out. In order to efficiently apply the tank model in a full vehicle crash simulation, the model is currently in a simplification routine.

ACKNOWLEDGEMENTS

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.

REFERENCES

[1] U. Eichhorn, Mobility for the Future, *Proceedings of the 21st Aachen Colloquium Automobile and Engine Technology 2012, Aachen, Germany, October 8-10, 2012*

[2] J. Wu, S. C. Dunn, Environmentally responsible logistics systems, *International Journal of Physical Distribution & Logistics Management*, **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] B.H. da Silveria, L. H. Benvenuti, E. Werninghaus, E. P. D Coelho, C. D. Silva, F. R. de Olivera, J. C. S. Silva, Development of a Concept Vehicle for Compressed Natural Gas, *SAE Technical Paper Series*, Paper No. 2004-01-3452, 2004

[5] R. Gaudio, E. Volpi, Position Paper: Natural Gas and CO2 - Natural gas is a champion in road transport and also saving CO2 emissions, NGVA Europe, Brussels, Belgium, 2009

[6] G. P. Hansen, M. Sato, Y. Yan, Pressure Vessels for Hydrogen Vehicles: An OEM Perspective, *Proceedings of International Hydrogen Fuel and Pressure Vessel Forum, Beijing, China, September 27-29 2010*

[7] G. R. Liu, S. S. Quek, Finite Element Method – A Practical Course, Elsevier B.V., Amsterdam, Netherlands, 2003

[8] J. O. Hallquist, LS-DYNA Theory Manual, Livermore Software Technology Corporation, Livermore, USA, 2006

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

[10] P. Maimi, P. Camanho, J. Mayugo, C. Davila, A continuum damage model for composite laminates: Part I – Constitutive model, *Mechanics of Materials*, **39**, pp. 897-908, 2007

[11] N.N., LS-DYNA Keyword User's Manual - Volume II Material Models, Revision 6307, Livermore Software Technology Corporation, Livermore, USA, 2015

[12] S. Hartmann, Neue Materialmodelle für Composites in LS-DYNA - Infotag: Composite Berechnung mit LS-DYNA, Stuttgart, DYNAmore GmbH, Stuttgart, Germany, 2013

[13] S. Marzi, L. Ramon-Villalonga, M. Poklitar, F. Kleiner, Usage of Cohesive Elements in Crash Analysis of Large, Bonded Vehicle Structures – Experimental Tests and Simulation, *Proceedings of LS-DYNA Anwenderforum, Bamberg, Germany, September 30 – October 01 2008*

[14] Y. Mi, M. A. Crisfeld, G. A. Davies, H. B. Hellweg,, Progressive Delamination Using Interface Elements, *Journal of Composite Materials*, **32,** p.p. 1246-1272, 1998

[15] M. Feucht, A. Haufe, Modeling connections with LS-DYNA, DYNAmore GmbH, Stuttgart, Germany, 2008