Project Number	Client			Report Number
P102024	Energimyndigheten, Gestamp Hardtech AB			TR21-001
Date	Reference			Revision
2021-11-26	D1.2			-
Registered by	Issued by	Checked by	Approved	Classification
EJ	RO	DB	CC	OPEN

IMKO final project report

Robin Olsson

RISE SICOMP AB, Box 104, SE-431 22 Mölndal

Abstract

This report describes the development of a unified model for prediction of damage initiation and growth in composites with unidirectional fibres under triaxial loads. The model, aimed for industrial use, is obtained by merging a previous model for fibre kinking with a model for transverse loading and shear, using a unified finite deformation formulation. The new model has been implemented as a user material subroutine in Abaqus and LS-Dyna and has been verified for accuracy and stability for various load cases on single finite elements, without any significant differences between the two software packages. The model has subsequently been validated by comparison with existing experiments involving a three-point bend test and axial crushing of a crash tube. These tests revealed a slightly larger mesh sensitivity of LS-Dyna and the importance of inclusion of the correct number of delaminations developing during crushing. Issues remaining for further research include apparent differences in the logarithmic strain defined in Abaqus and LS-Dyna, and incorrect prediction of the fibre rotation during finite deformations.

Keywords: CFRP, Crash, Crushing, Damage-mechanics, Energimyndigheten, Modelling

Distribution list (only for confidential reports)

Organisation	Name	Copies

Digitally signed by Christopher J. Cameron
DN: cn=Christopher J. Cameron, o=RISE Research Institutes of Sweden AB, ou=Department of Polymers, Fibres and Composites, email=christopher.cameron@ri.se, c=SE
Date: 2021.12.22 09:57:35 +01'00'

Contents

1. Introduction

1.1. Background

The structural weight is a key factor for the energy consumption during the operation of vehicles. The concern for global climate effects has prompted consumer awareness and legislation to reduce the weight of passenger cars and other vehicles. Fibre reinforced polymer (FRP) composites, particularly carbon fibre FRP (CFRP), have high stiffness and strength per weight, and are therefore of increasing interest for the automotive industry. CFRP is significantly more expensive than sheet metal. Furthermore, forming of composite components typically requires longer process times. On the other hand, tools can be made from aluminium or composite materials rather than from expensive tooling steels. For these reasons CFRP has primarily been introduced in high end cars produced in shorter series.

Recently, faster manufacturing has been offered by introduction of resin transfer moulding (RTM) or hot-pressing of preforms with pre-impregnated fibres. Such methods are facilitated by use of thermoplastics and fast curing thermosets. The expansion of the market for carbon fibres from aeronautics to wind turbines and automotive applications has also resulted in cheaper carbon fibres. Furthermore, leading automotive manufacturers, e.g. BMW and Audi, have introduced methods for press forming of sheet metal with reinforcing CFRP patches, which allows simultaneous forming and curing of a thermoset resin. These methods are also actively studied by Gestamp Hardtech.

1.2. Structure and failure of composites

FRP composites consist of stiff fibres embedded in a softer and weaker polymer (matrix). To simplify handling and manufacture it is common to provide dry fibres in various types of preforms. Textile preforms with fibres are suitable for manufacturing involving impregnation of dry fibres, but the weaving procedure causes some crimp of the fibres which reduces the stiffness and compressive strength of the fibres. Non-crimp fabrics (NCF:s) involve nominally straight reinforcement fibre bundles stitched or knitted together to a preform. In most cases NCF:s are multidirectional, but they may also be unidirectional. “Uniweaves” consist of unidirectional reinforcement fibre bundles woven with sparsely distributed transverse “warp” yarns and are often considered as a special type of NCF.

Virtually all failure mechanisms of fibre composites are controlled by the matrix. These include shear or loading transverse to the fibres. In most common composites the compressive failure in the fibre direction is also controlled by the matrix and occurs due to local microbuckling of the fibres (“fibre kinking”). Tensile failure in the fibre direction is the only failure mechanism mainly controlled by the fibres.

1.3. Previous work

RISE SICOMP has participated in sequence of research projects on modelling of composites under crash situations and other complex load cases. These include the projects “Compcrash 1” fully funded by the Swedish Energy Agency (Energimyndigheten) and “Compcrash 2” with limited co-funding from Gestamp Hardtech. RISE SICOMP has also collaborated with Chalmers University, consulting companies, software providers, and other Swedish automotive companies in the projects “FFI-Crash 1” and “FFI-Crash 2” jointly funded by FFI-VINNOVA and the industry. Gestamp Hardtech has participated in “Compcrash 2” and “FFI-Crash 2”.

A key task in the above projects has been to develop reliable material models of composites for implementation in finite element (FE) analysis of crash behaviour. This necessitates reliable criteria for failure initiation under complex triaxial loads and the behaviour during subsequent deformation. The subsequent deformation involves large strains and gradual softening of the material, which is highly challenging but very important for the energy absorption during a crash. The previous model development in the above projects is briefly summarised below. Note that the references are limited to publications from RISE SICOMP. References on experimental work are limited to experimental studies of immediate use for model development or validation.

- Compcrash 1 (2012-2016): Identification of material parameters relevant for crash and thorough experimental characterisation of a uniweave material as a basis for modelling [1, 2, 6, 7]
- FFI-Crash 1 (2013-2016): Building of a simplified material model for triaxial FE analysis [3]. Development of a prototype computational model for fibre kinking [4, 5]
- Compcrash 2 (2016-2019): Experimental characterisation under biaxial loading [9] and of friction [10]. Improvement and validation of a model for shear of uniweaves [11]. Development of fibre kinking model with new methodology more suitable for FE analyses [12].
- FFI-Crash 2 (2017-2020): Improvements of the model from FFI-Crash 1 for implementation in FE analysis with shell elements [8]. Validation of the model for crushing and crushing [11] and for industrial test cases.

2. Project aims and structure

The aim of the project was to proceed the work in FFI-Crash 2 and Compcrash 2 to a coherent material model that could be used by the industry within standard commercial FE codes. The aim is reflected by the project acronym IMKO (“Industrianpassad materialmodell för kompositer” in Swedish or “Composite material model adapted for industrial use” in English). To achieve this goal the following tasks had to be addressed:

- Merge the model for damage growth under transverse loading and shear with the separate model for fibre kinking, using a unified formulation.

- Implement the developed material model in the FE software LS-Dyna preferred by the automotive industry and verify its agreement with the original implementation in Abaqus.
- Test the software for stability and accuracy.
- Validate the software by simulating and comparing with previously completed experiments on simple components.

The project was performed by the research provider RISE SICOMP and the industrial partner Gestamp Hardtech, with a total project budget of 1.5 MSEK, equally funded by the Swedish Energy Agency and Gestamp Hardtech. Formally the project period was 2020-05-01 to 2021-12-31 (extended 8 months after delayed start due to Covid-19). In practice the work has been performed from October 2020 to November 2021.

The project was coordinated by Robin Olsson at RISE SICOMP. The overall division of work was that Sergio Costa at RISE SICOMP was responsible for model development, while Hana Zrida at Gestamp Hardtech was responsible for simulation and validation. Extensive supporting simulations were, however, also performed by Sergio Costa. Implementation and testing of the model were performed jointly by Sergio Costa and Hana Zrida, where Costa was responsible for implementation and testing in Abaqus and Zrida was responsible for the corresponding tasks in LS-Dyna. Support and advice in model development and simulations was provided jointly by Robin Olsson at RISE SICOMP and Rickard Östlund at Gestamp Hardtech.

Figure 1 gives an overview of the Work Packages (WPs) and division of work in the project.

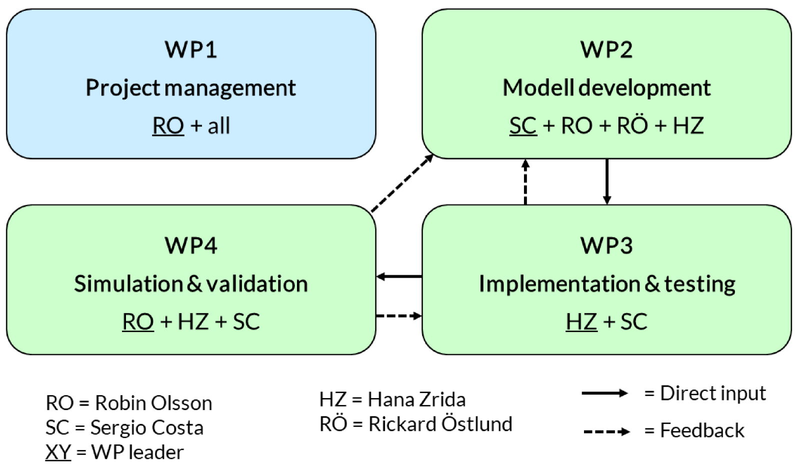

Fig. 1. Overview of work packages and division of work.

3. Model concepts and development

In the previous project Compcrash 2 separate models were developed for damage initiation and growth under transverse loading and shear on one hand and for fibre kinking on the other. The model for fibre kinking was formulated using finite deformation theory while the model for transverse loading and shear was not. A fundamental task in IMKO was therefore to formulate the model for transverse loading and shear using finite deformation theory and to merge it with the fibre kinking model into a unified material model that can be used directly in industrial simulations of general triaxial stress states. As a direct consequence transverse loading, shear and fibre kinking are all described by a damage mechanics approach using a common single damage variable, representing damage in the matrix material. The only additional damage variable is required for fibre tension, as illustrated in Fig. 2.

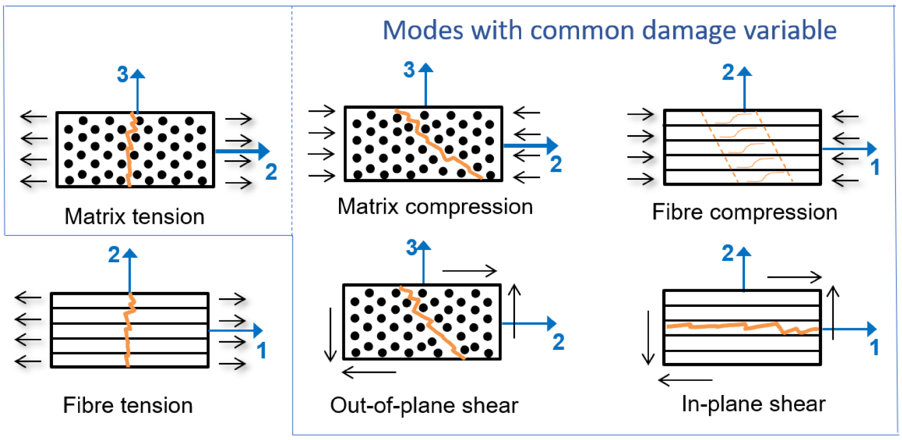

Fig. 2. Damage modes and common damage variable..

Shear deformation is involved in all four failure modes in the right of Fig. 2, and obviously requires a correct representation of the shear stress-strain relation. This is done by calibrating a nonlinear shear stress-strain curve to a corresponding experimental curve for the material, which also results in a corresponding shear strain-damage curve, Fig. 3. In this figure γ_0 and τ_0 represent onset of damage (nonlinearity) and γ_f the experimentally determined failure strain. The quantity $\delta\gamma$ is a strain increment to reach full damage with maintained numerical stability, and was set to $0.02\gamma_f$. The generation of the curves shown in Fig. 3 from experiments has been described in [6]. An inherent part of the model is also to associate damage with generation of microcrack surfaces and related friction, rather than with plasticity, which explains the remaining constant stress once full damage has been reached. This approach is more efficient than other typical approaches (based on plasticity), but also allows a fairly good representation of the hysteresis loops observed during cyclic loading, as shown in Fig. 3.

It is evident that shear failure occurs at relatively large strains, which indicates the importance of considering finite deformations in the analysis. The importance of finite deformation analysis is particularly relevant for fibre kinking, where the shearing is directly generated by the misalignment between the fibre direction and the loading direction. The generation of shear deformation and resulting shear stresses is illustrated by the right subfigure in Fig. 4. Fibre kinking initiated by a small initial fibre misalignment, where the increasing moment associated with an increasing fibre angle results in further fibre rotation, and a corresponding decrease in shear stiffness. The finite deformation analysis with correct updating of the fibre angle is particularly important for modelling fibre kinking.

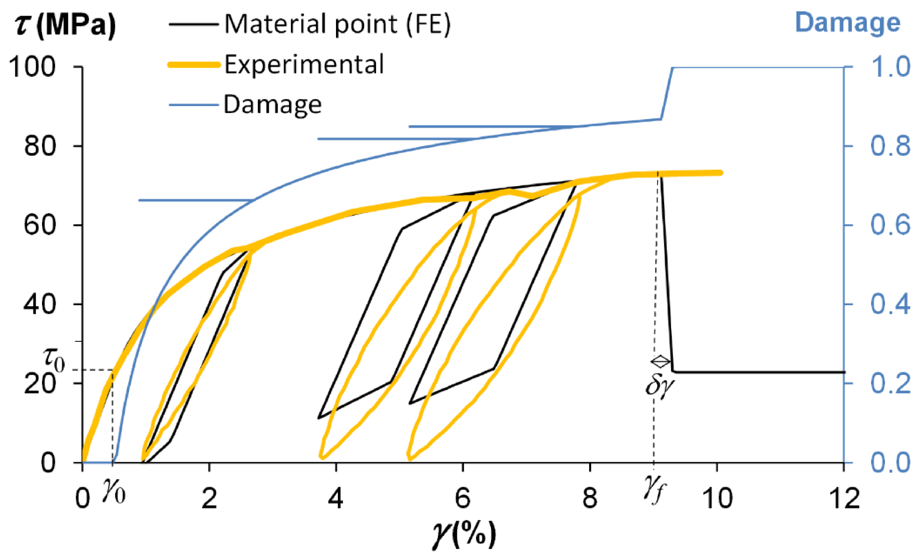

Fig. 3. Calibration of model shear stress-strain curve to experiments.

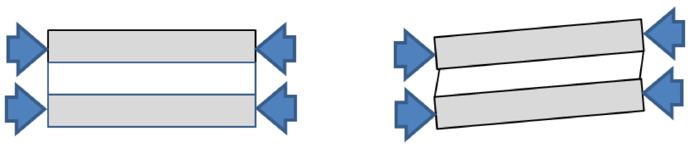

Fig. 4. Shearing due to fibre rotation.

The use of a three dimensional (3D) finite deformation formulation for all damage modes in the current unified material model is particularly beneficial for modelling the post peak behaviour during fibre kinking, but also allows an improved modelling of other deformation modes and their interaction.

A detailed description of the theory and concepts of the model is given in [13, 14, 16].

4. Tests and evaluation performed

When the unified model had been formulated in Fortran code it was implemented as a user defined Fortran subroutine (VUMAT) material model in Abaqus and tested against results from previous results from the earlier separate parts of the model. Initial "sanity" tests were also done for the output from the model.

After completed initial tests in Abaqus the Fortran code was also implemented as a user material (UMAT) in LS-Dyna. Once the unified model had been implemented in both Abaqus and LS-Dyna it was tested for a single FE element, and the results from the two FE codes were compared. Test cases included several elementary load cases, Fig. 5.

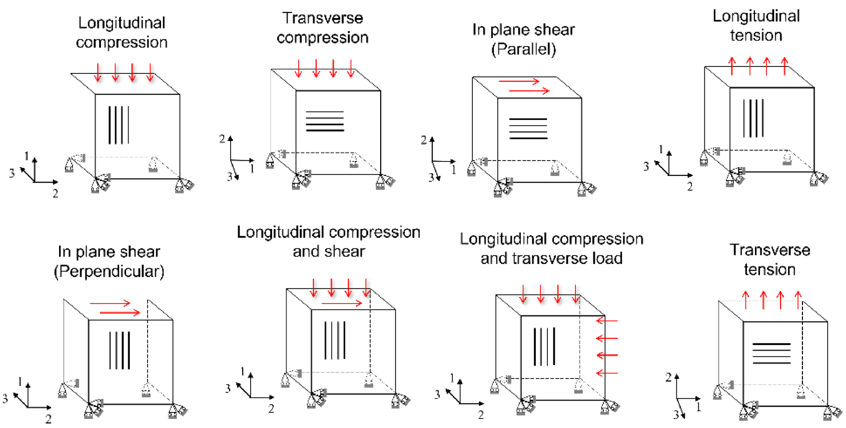

Fig. 5. Single element test cases.

In general, there was a good agreement between the predictions by Abaqus and LS-Dyna, as shown by the examples in Fig. 6.

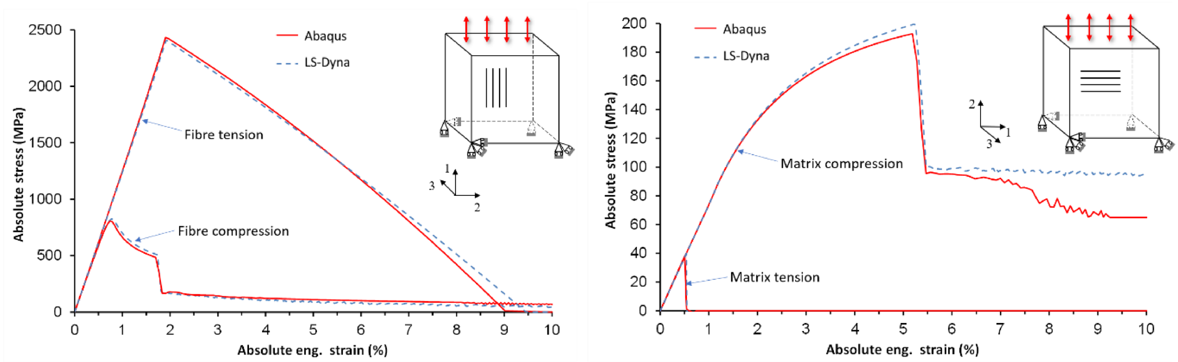

Fig. 6. Comparison of response for longitudinal and transverse loading.

Comparisons were also made for the response for compressive loading at various angles to the fibre direction, where a similar good agreement between Abaqus and LS-Dyna was observed, Fig. 7.

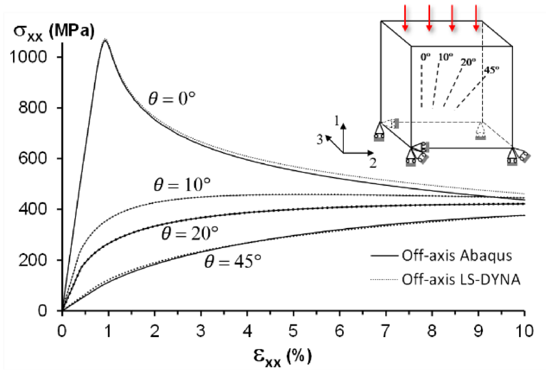

Fig. 7. Comparison of compressive response for various off-axis angles..

All the above results are based on engineering (nominal) strain, i.e. change in element length divided by original element length. For fibre compression ("fibre kinking") the results based on logarithmic ("true") strain, as provided by the FE software, deviated significantly even at strains below 3%, Fig. 8. Furthermore, for the logarithmic strain Abaqus and LS-Dyna demonstrated significantly different response. In the case of Abaqus there was also a large difference between the output from the material model (material point) and from the element integration point.

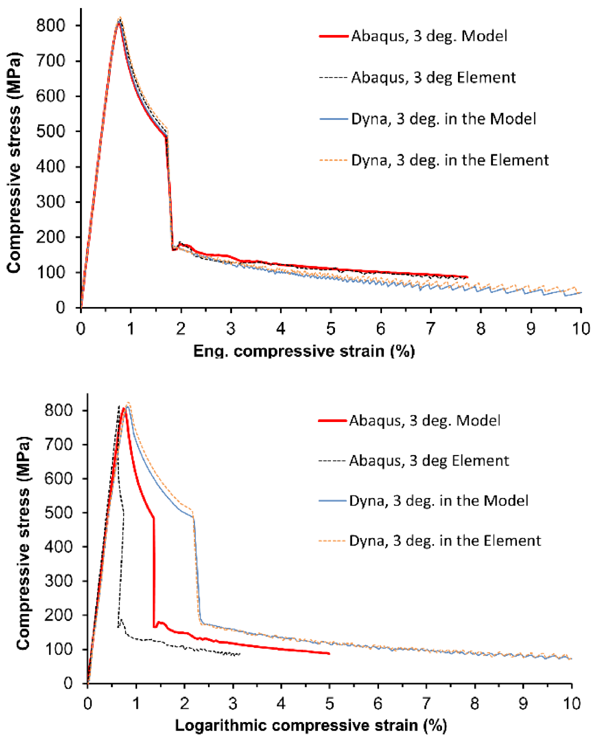

Fig. 8. Comparison of compressive response for various off-axis angles.

The reasons for these differences were discussed with the FE software providers Simulia and Dynamore without finding any clear answers. The issue was dropped as it was considered sufficient for multi-element FE models that the two software packages provide an equal response in nominal strain, i.e. in element deformation. Differences in the definition of logarithmic strains may, however, affect various strain-based calculations in the model and should be investigated further in the future.

It was also suspected that the differences for fibre compression may be due to incorrect representation of the fibre orientation during shear deformation. The dilemma is illustrated by Fig. 9, where the "element coordinates" (1_e-2_e) and "material coordinates" (1_m-2_m) coincide prior to deformation but disagree after the deformation. For a correct modelling of the composite the material axes in the deformed configuration should agree with the 1_m'-2_m'-system. The issue was studied for a single reduced order linear Abaqus element for different fibre orientations, load cases (u_1, u_2) and boundary conditions. It is evident that the material axes given by the element in the deformed configuration always disagree with the natural material coordinates. This may obviously affect all model calculations involving fibre rotations. A particular problem appears in model validation, where predicted fibre orientations will disagree with experimental results. The issue was again discussed with the software providers Simulia and Dynamore but could not be solved as the predicted rotation of the material coordinates is an inherent part of the element formulation. Thus, it appears that a user-defined material coordinate system would be required.

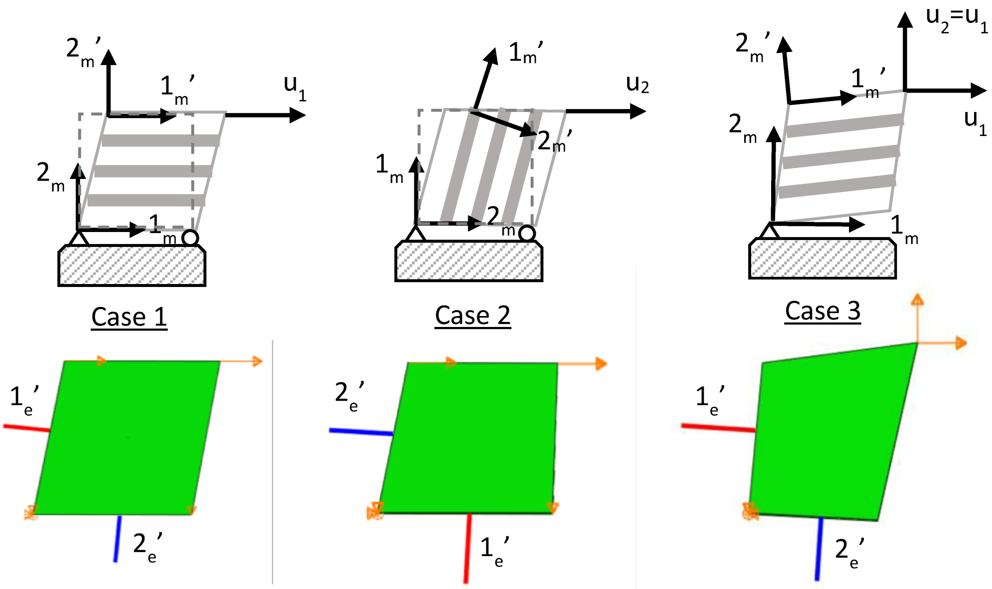

Fig. 9. Material coordinates related to fibre orientation (1_m-2_m) *and FE-element* (1_e-2_e) *before* (1-2) *and after* (1'-2') *deformation.*

Finally, it should be mentioned that comparisons between Abaqus and LS-Dyna were also made between a single element and a 2x2x2 cube (8 elements). Further details on these studies are given in [13].

5. Simulations performed

The model was validated by comparing FE simulations with solid elements with previously performed tests, including an in-house three-point bend test at Gestamp Hardtech and a static crush test on a circular crash tube, performed at Volvo Cars within the project FFI-Crash 2.

The three-point bend test, Fig. 10, involved quasi-static bending of a CFRP laminate with rectangular cross-section and a quasi-isotropic layup with fibres equally distributed in the directions 0°, 90° and ±45°. Figure 11 shows the predicted and measured load-deflection relation. The different failure mechanisms in different plies were identified by using Abaqus to plot damage parameters for fibre tension and matrix damage for each ply.

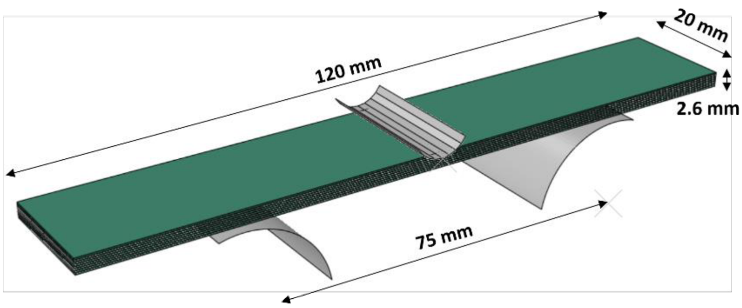

Fig. 10. Geometry of the three-point bend test.

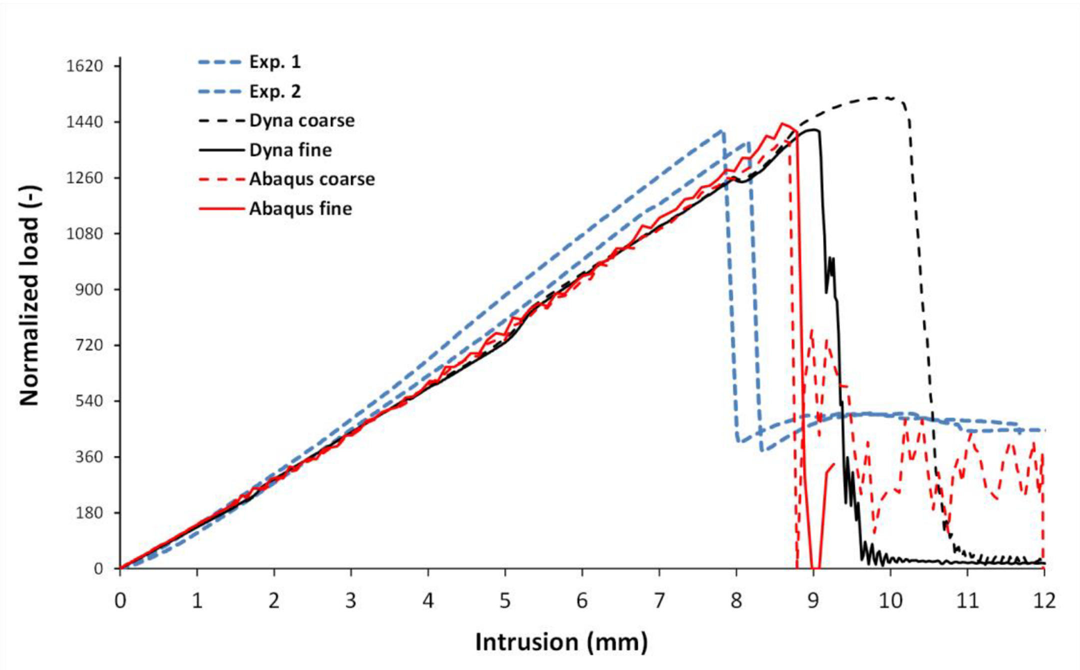

Fig. 11. Load-deflection relations in the three-point bend test.

It was noted that both simulations in Abaqus and LS-Dyna were in good agreement with the experiments, but that LS-Dyna appears to be more sensitive to the FE element mesh size.

The crash tube test, Fig. 12, involved quasi-static axial crushing of a circular CFRP tube with fibres equally distributed in the axial direction (0°) and hoop direction (90°). Figure 13 shows the predicted and measured stress-displacement relation. In general, the simulation of this test was significantly more demanding and required much more computational time. For computational efficiency only half of the tube length was included in the FE-model, which explains the early termination of the stress-displacement curve from LS-Dyna. Delamination initiation and growth was simulated by cohesive elements.

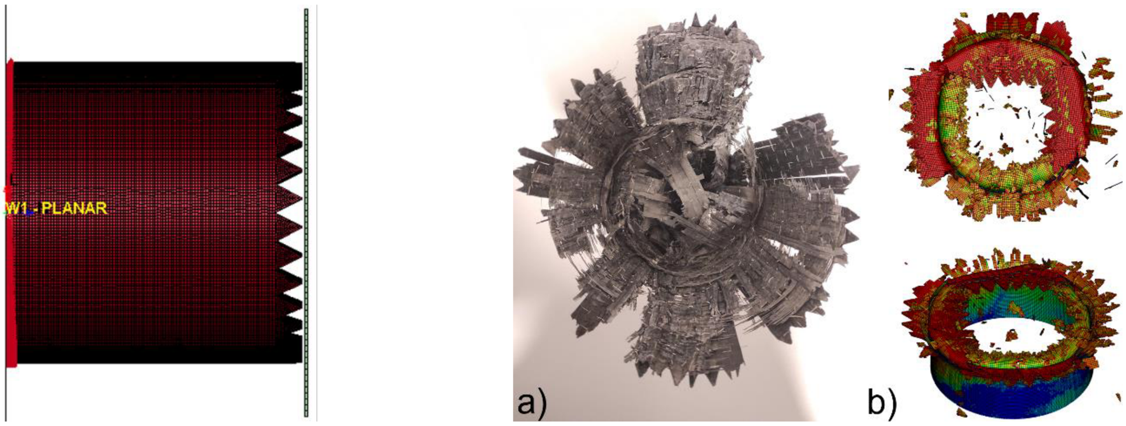

Fig. 12. FE-model of crash tube and images of observed and predicted crushing.

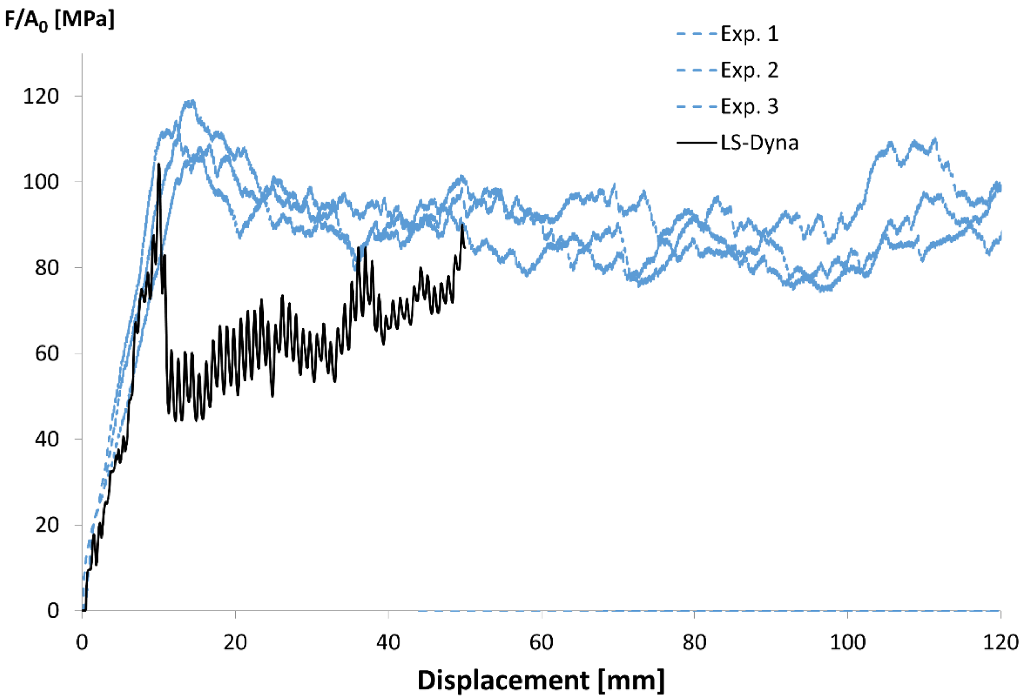

Fig. 13. FE-model of crash tube and images of tested specimens

The predicted sudden load drop in Fig. 13 at the end of the trigger (at 10 mm) was not observed in the experiments. The reason for this instability in the computational model is not clear, but may be due to a perfect symmetry, which is not present in real crash tubes.

It was found very essential to include delaminations in the model, as no delaminations resulted in a significant overestimation of the actual crush load. Delaminations in too many interfaces may, on the other hand, result in an underestimation of the load. For a cross-ply laminate, physical arguments and fractographic evidence indicate that an out-of-plane shearing load will result in delamination in every second interface, as illustrated in Fig. 14. The LS-Dyna predictions in Fig. 13 were based on 11 delaminations (one at every interface), but the response with 6 delaminations (in every second interface) was virtually identical.

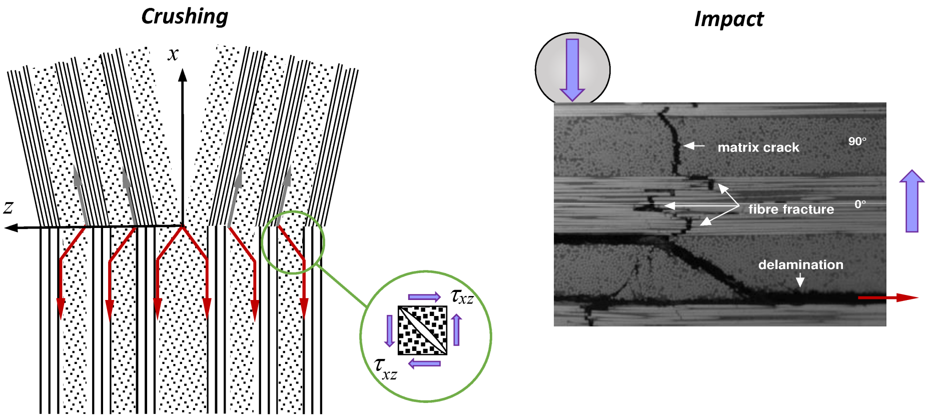

Fig. 14. Generation of delaminations in cross-ply laminate by out-of-plane shear load

Using LS-Dyna with 64 CPU:s the computational time for the three-point bend test was 6 h for the coarse mesh and 46 h for the fine mesh. The corresponding time for the crash tube was about 62 h to reach a 40 mm displacement, while 38 h was required to reach 20 mm displacement and only 10 h to reach 20 mm when delaminations were excluded.

In addition to the simulations originally planned in the project the new model was also validated by comparison with micromechanics-based FE simulations performed by Dr Miguel Herráez at University Rey Juan Carlos in Madrid. The micromechanics model included a representative wavelength of a single elastic fibre embedded in an elastic-plastic resin and considered both square and hexagonal array of the fibres. This collaboration was initiated through personal contacts by Dr Costa and the work in Madrid was not funded by the project.

Figure 15 illustrates the computational micromechanical models (CMM) for a square and hexagonal fibre array that were developed in Madrid.

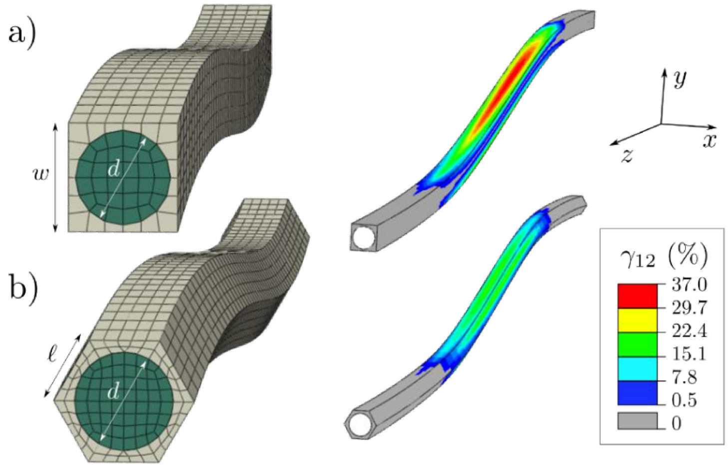

Fig. 15. Micromechanical FE models for (a) square and (b) hexagonal fibre arrays

Figure 16 gives a comparison between the current homogenised model and the CMM for the square and hexagonal fibre array for two initial fibre misalignment angles. The strain of the CMM was evaluated at the cross-section with maximum misalignment angle in the centre of each CMM, and a virtually perfect agreement is observed with the current model. A significantly poorer agreement in the post-peak region was obtained when the average strain of the CMM was used, i.e. when dividing the end displacement with the original length of the CMM [14]. In this case the CMM showed significantly smaller (average) strains, which is due to strain localisation after the onset of fibre kinking.

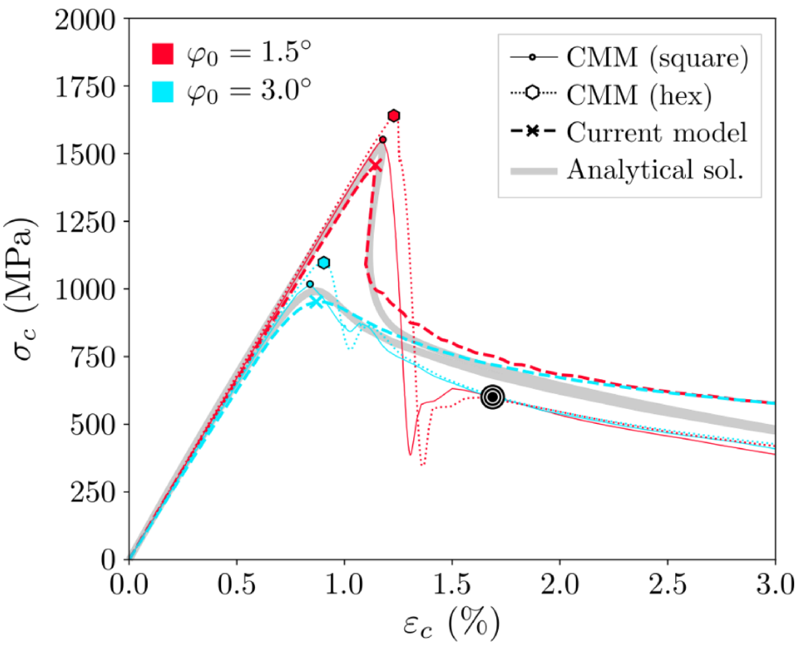

Fig. 16. Comparison of for fibre compression response in the current model and the micromechanical (CMM) model for two different fibre arrays and misalignment angles.

Further details on the LS-Dyna simulation of the three-point bend test and the crash tube test may be found in [15], while an overview of all simulations is given in [16].

6. Summary of progress and need for further work

A unified computational material model for composites under triaxial stresses has been developed by merging two previously developed models, using a unified finite strain formulation. The resulting material model has been implemented in Abaqus and LS-Dyna and extensive single element tests have been performed for different load cases to verify stability, accuracy, and agreement for the two software packages. Furthermore, the material model has been validated by simulations of two completed lab tests on composite specimens, including a three-point bend test and axial crushing of a crash tube. The unified finite deformation formulation increases the computational speed and eliminates some of the iterations that have been necessary in previous versions of the model.

Overall, the model appears stable and with only minor differences for single element models in Abaqus and LS-Dyna. Both FE packages provide good agreement with a three-point bend experiment, although LS-Dyna appears to be more sensitive to mesh size. Simulation of a crash tube experiment reveals several challenges and uncertainties. It is clear that this kind of test is computationally demanding and is sensitive to many details in the model, e.g. mesh geometry, interlaminar toughness and number of delaminations. In this case large differences were also observed between Abaqus and LS-Dyna. In general, neglect of delaminations will result of an overestimation of the crush load, while a too low interlaminar toughness or allowing delaminations in all interfaces will result in an underestimation. In fact, physical reasoning and experimental evidence reveals that delaminations only appear in a certain fraction of the interfaces.

For an efficient simulation of crash tube response it may be advisable to perform a detailed local simulation of the crush zone with all relevant delaminations using the current material model, and then use a single layer of shell elements with homogenised stress-strain behaviour for larger structural simulations. The "homogenised crush stress" will then represent the average behaviour of several plies with crushing, delamination and local buckling and bending.

Need for further work includes the following issues:

- Methods for increased computational efficiency by mesh adaptivity and by combining solid elements with shell elements, and possibly also minor improvements of the computational algorithms.
- Further studies of the difference between nominal strain ("element deformation") and the different measures of logarithmic strain ("true strain") in LS-Dyna and Abaqus.
- Further studies of the observed difference between material coordinates of FE elements and the actual orientation of fibres in composites.

7. List of publications and reports

Journal articles

A.1 Costa S, Herraez M, Olsson R, Zrida H, Östlund R (2021). A unified physically-based finite deformation model for damage growth in composites. Composites A; Submitted for publication. *(Deliverable D4.2)*

Conference papers

C.1 Costa S, Zrida H, Herraez M, Olsson R, Östlund R (2021). Modelling damage growth using a physically-based and finite deformation model. Proc. 8th ECCOMAS Thematic Conf on Composites. Gothenburg, Sweden. 10.23967/composites.2021.073

Reports

R.1 Costa S (2021). Material model for crush - description and verification: version 3.0. RISE SICOMP confidential report CR21-005, RISE SICOMP, Mölndal. *(Deliverable D2.1)*

R.2 Zrida H (2021). Testing and validation of a new material model for prediction of damage development in composite structures under crash. Report T010-21. Gestamp Hardtech, Luleå. *(Deliverable D4.1)*

R.3 Olsson R (2021). IMKO final project report. RISE SICOMP open report TR21-001, RISE SICOMP, Mölndal. *(Deliverable D1.2)*

8. List of deliverables

	Title	Planned*	Completed	Author
D1.1	Detailed project plan	2020-10-30	2020-10-30	RO
D1.2	Final project report	2021-09-30	2021-12-15	RO
D2.1	Software implemented in LS-DYNA	2021-02-26	2021-03-29	SC
D3.1	Internal report on model testing and adaptation	2021-05-31	Incl. in D4.1	HZ
D4.1	Internal report on simulations and validation	2021-08-31	2021-12-08	HZ
D4.2	Journal article on model development and validation	2021-09-15	2021-12-06	SC

*) According to plan at actual start of project.

9. Acknowledgements

This project was funded by the Swedish Energy Agency (Energimyndigheten) under Project number 50179-1, and by Gestamp Hardtech AB. The work was a joint effort by Sergio Costa and Robin Olsson at RISE SICOMP and Hana Zrida and Rickard Östlund at Gestamp Hardtech. We are also grateful for discussions with Daniel Berglund at RISE SICOMP and for the micromechanical FE modelling by Dr Miguel Herráez at University Rey Juan Carloz in Madrid.

10. References

[1] Grauers L, Olsson R, Gutkin R (2014). Energy absorption and damage mechanisms in progressive crushing of corrugated NCF laminates: Fractographic analysis. Compos Struct;110:110-117.

[2] Bru T, Hellström P, Gutkin R, Ramantani D, Peterson G (2016). Characterisation of the mechanical and fracture properties of a uni-weave carbon fibre/epoxy non-crimp fabric composite. Data in Brief;6:680–695.

[3] Costa S (2016). Material model implementation – version one. Confidential report CR15-037. Swerea SICOMP, Mölndal.

[4] Gutkin R, Costa S, Olsson R (2016). A physically based model for kink-band growth and longitudinal crushing of composites under 3D stress states accounting for friction. Compos Sci Technol;135:39-45.

[5] Costa S, Gutkin R, Olsson R (2017). Mesh objective implementation of a fibre kinking model for damage growth with friction. Compos Struct;168:384–391.

[6] Bru T, Olsson R, Gutkin R, Vyas GM (2017). Use of the Iosipescu test for the identification of shear damage evolution laws of an orthotropic composite. Compos Struct;174:319–328.

[7] Bru T, Waldenström P, Gutkin R, Olsson R, Vyas GM (2017). Development of a test method for evaluating the crushing behaviour of unidirectional laminates. J Compos Mater;51(29):4041-4051.

[8] Costa S (2017). Material model description and validation: Version 1.2. Confidential report CR17-021. RISE SICOMP, Mölndal.

[9] Bru T, Asp LE, Olsson R, Vyas GM (2018). Biaxial transverse compression testing for a fibre reinforced polymer material. Paper 3.10(5)-24. ECCM-18. Athens.

[10] Bru T, Olsson R (2019). Friction tests for simulation of energy absorbing composite structures. CompTest 2019. Paper ID 114. Luleå, Sweden.

[11] Costa S, Bru T, Olsson R, Portugal A (2019). Improvement and validation of a physically based model for the shear and transverse crushing of orthotropic composites. J Compos Mater;53(12):1681-1696.

[12] Costa S, Fagerström M, Olsson R (2020). Development and validation of a finite deformation kinking model for crushing of composites. Compos Sci Technol;197:108236. doi.org/10.1016/j.compscitech.2020.108236.

[13] Costa S (2021). Material model for crush - description and verification: version 3.0. RISE SICOMP confidential report CR21-005, RISE SICOMP, Mölndal. (Deliverable D2.1)

[14] Costa S, Zrida H, Herraez M, Olsson R, Östlund R (2021). Modelling damage growth using a physically-based and finite deformation model. Proc. 8th ECCOMAS Thematic Conf on the Mechanical Response of Composites. Gothenburg, Sweden. 10.23967/composites.2021.073

[15] Zrida H (2021). Testing and validation of a new material model for prediction of damage development in composite structures under crash. Report T010-21. Gestamp Hardtech, Luleå. (Deliverable D4.1)

[16] Costa S, Herraez M, Olsson R, Zrida H, Östlund R (2021). A unified physically-based finite deformation model for damage growth in composites. Composites A; Submitted for publication. (Deliverable D4.2)