

Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymers

- by Use of Digimat and nCode DesignLife

Master's thesis in Applied Mechanics

JAN-ANDERS LINDHULT
MIRANDA LJUNGBERG

MASTER'S THESIS IN APPLIED MECHANICS

Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymers

- by Use of Digimat and nCode DesignLife

JAN-ANDERS LINDHULT

MIRANDA LJUNGBERG

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2015

Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymers
- by Use of Digimat and nCode DesignLife
JAN-ANDERS LINDHULT
MIRANDA LJUNGBERG

© JAN-ANDERS LINDHULT, MIRANDA LJUNGBERG, 2015-06-08

Master's Thesis 2015:33
ISSN 1652-8557
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:

Experimental data of dumbbells is the basis for the material modelling. Fatigue analysis of a component is performed coupled to the material modelling tool Digimat.

Chalmers Reproservice
Göteborg, Sweden 2015-06-08

Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymers
- by Use of Digimat and nCode DesignLife
Master's thesis in Applied Mechanics
JAN-ANDERS LINDHULT
MIRANDA LJUNGBERG
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology

Abstract

In order to meet demands on reduced fuel consumption in the automotive industry it is necessary to introduce lightweight materials in semi-structural components. A cost efficient example of such material class is injection moulded short fibre reinforced polymers. To accurately analyse such components, the anisotropic material properties induced by the manufacturing process, needs to be accounted for. In this thesis, carried out at Volvo Cars, a framework for fatigue analysis of injection moulded short fibre reinforced polymers is established. The anisotropic material properties are taken into account by using the multi-scale modelling tool Digimat together with the finite element solver Abaqus and the fatigue life prediction tool nCode DesignLife.

A case study of an existing component has been performed in order to outline the workflow of fatigue analyses of short fibre reinforced polymers. A fatigue analysis of the component has been carried out. Also, a fatigue test of the component has been conducted for guidance and evaluation of the analysis.

The software interfaces of the coupling between Digimat, Abaqus, nCode DesignLife are working well and the fatigue analysis workflow is relatively easy to follow. The analysis predicted fatigue failure in the same location as it occurred in the physical test. The number of cycles to failure was however under predicted. This is thought to be caused by the finite element model, limitations of material model, approximation of material properties and uncertainties of the fatigue failure definition. Better correlation between physical tests and CAE analyses can be achieved by additional experimental fatigue data or by adjustments of the linearized material model, though, this is beyond the scope of the thesis.

Keywords: Fatigue, Anisotropy, Short Fibre Reinforced Polymers, Composites, Digimat, nCode DesignLife, Abaqus, Material Modelling

Contents

Abstract	I
Contents	III
Preface.....	V
Acknowledgement	V
1 Introduction.....	1
1.1 Background.....	1
1.2 Objective.....	1
1.3 Delimitations	2
1.4 Method.....	2
2 Short Fibre Reinforced Polymers Modelling.....	4
2.1 Mechanical Properties	4
2.2 Injection Moulding	4
2.3 Fibre Orientation	5
2.4 Digimat	6
2.5 Micro to Macro Scale	6
2.6 Pseudo-Grain based Fatigue Model.....	7
3 Analyses	9
3.1 Calibration of Material Model.....	9
3.2 Fatigue Data.....	10
3.3 Mapping of Fibre Orientations	13
3.4 Coupled Analysis Abaqus-Digimat	13
3.5 Coupled Analysis nCode DesignLife-Digimat	14
4 Case Study – Front End Carrier	16
4.1 CAE Fatigue Analysis	16
4.2 Physical Fatigue Test.....	19
4.3 Comparison of CAE Simulation and Physical Test.....	21
5 Discussion	23
5.1 Material Data	23
5.2 Material Modelling.....	23
5.3 Failure Criteria.....	24
5.4 Discretisation.....	24
5.5 Analysis Method.....	24
6 Suggestions for Future Work	25
7 Conclusions.....	26
References.....	27
Appendix A.....	28

Preface

This master thesis is the final part of our Master's degrees in Applied Mechanics at Chalmers University of Technology. The work was carried out during the spring of 2015 at the department of CAE Durability Body and Trim, at Volvo Car Corporation. M.Sc. Marcus Hammar and Ph.D. Åsa Sällström have been the supervisors at Volvo Cars. Magnus Ekh, Professor at the division of Material and Computational Mechanics at Chalmers, was the examiner.

Acknowledgements

We would like to thank our supervisors M.Sc. Marcus Hammar and Ph.D. Åsa Sällström for their support and commitment throughout the thesis. Also, we would like to thank Ph.D. Mats Landervik at Dynamore Nordic AB for advice and help regarding Digimat. Special gratitude goes to Dr.Ir. Benoît Bidaine and his colleagues at e-Xtream engineering for their expertise and for patiently answering our questions about the material modelling in Digimat. Additionally, we would like to thank nCode for help and support. We also appreciate the valuable feedback and expertise from Professor Magnus Ekh, for which we are grateful.

We would also like to thank Dr.-Ing. Marcel Brandt at Lanxess for giving advice and answering questions regarding processing of material data. Thanks to Röchling Automotive, manufacturer of the case study component, for providing accurate parameters of the injection moulding process. In addition, we also thank M.Sc. Andreas Ramström, Injection mould analysis engineer at Volvo Cars, for performing the process simulations.

Special thanks goes to M.Sc. Malin Bodfors, at Volvo Cars, for guidance of and conducting the physical tests included in this thesis. Last but not least, we would like to thank all our colleagues at Volvo Cars for providing a friendly work environment.

Göteborg, June 2015

JAN-ANDERS LINDHULT
MIRANDA LJUNGBERG

1 Introduction

In order to meet demands on fuel consumption and CO₂ regulations, the automotive industry strives to reduce the weight of the vehicles. One way is to introduce lightweight materials. Short Fibre Reinforced Polymer (SFRP) composites are such materials where a favourable strength to weight ratio can be obtained. A further advantage is that they can be injection moulded, enabling cost efficient high production rates and complex geometries. Introducing SFRPs in semi-structural components with focus on weight reduction puts high demands on strength and fatigue predictions. This thesis, performed at Volvo Cars, concerns fatigue modelling of injection moulded SFRPs, accounting for the process induced anisotropy of the material.

1.1 Background

Methods to predict fatigue of metals have been developed since the mid-1800s. The first well known fatigue testing was reported by the German mining engineer W. A. S Albert in 1829 and was followed by the comprehensive systematic fatigue investigation of August Wöhler [1]. However, fatigue predictions of SFRPs are not as developed as for metals. The degradation of a composite's integrity under cyclic loading is caused by micro-events that are more difficult to predict compared to those occurring in metals. Examples of such micro-events are debonding and cracking, causing damage accumulation and crack growth [2]. Furthermore, the injection moulding process causes a varying fibre orientation throughout the structure. This makes the material properties highly anisotropic and heterogeneous. To perform accurate fatigue analyses of an SFRP, the variation of mechanical and fatigue properties throughout the structure needs to be accounted for.

A tool that accounts for the micromechanical anisotropic behaviour of the material is Digimat. The macroscopic response is predicted by mean-field homogenization of the microscopic response. The basis is the material parameters of the separate constituents and microstructure information, such as mass fraction and orientation of the fibres. Static and dynamic strength studies based on this method have previously been carried out at Volvo Cars with satisfying results and good correlation to tests [3]. The next step to further widen the analysis of SFRPs at Volvo Cars is to implement anisotropic material models into fatigue analyses. This is to be carried out by implementing fatigue data from experiments into Digimat, which generates local material fatigue properties, based on the fibre orientations throughout the structure. The fatigue analysis of the SFRP structure will then be carried out in nCode DesignLife.

1.2 Objective

The objective of this thesis is to investigate the use of Digimat in combination with the CAE fatigue tool nCode DesignLife. More specifically, a methodology to predict the fatigue life of an SFRP in nCode DesignLife is to be formulated. The procedure should include implementing injection moulding simulation results into Digimat, setting up an appropriate material model in Digimat, and running finite element simulations. For evaluation of the methodology and for future improvements, physical fatigue tests will be performed.

1.3 Delimitations

The aim is to establish a framework for fatigue analysis of SFRPs, not to develop a complete, fully developed method.

The simulation of the injection moulding process was carried out in the software Moldflow, and was provided by Volvo Cars. The manufacturer of the SFRP component, Röchling Automotive, provided the parameters of the injection moulding process. Process induced phenomena such as pores, weld lines, residual stresses, variation of fibre content and fibre size will not be accounted for.

Material test data of the SFRP was provided from the supplier of the composite material, Lanxess, and no calibration against the fatigue test was carried out.

1.4 Method

In order to outline the fatigue analysis framework, a case study of an existing SFRP component is performed. The component will be used throughout the analyses and required assumptions and approximations will be investigated. Additionally, a fatigue test of the component will be conducted for evaluation of the CAE results.

The anisotropic behaviour of the SFRP will be modelled using the software Digimat, in combination with the fibre orientations in the structure. The process induced fibre orientations, described in Section 2.3, are estimated by simulations carried out in Moldflow which will be performed by an Injection mould engineer at Volvo Cars. The fatigue analysis tool nCode DesignLife, version 10.0, will be used which requires the results from a static structural analysis. The static Finite Element (FE) analysis will be performed in Abaqus, version 6.13-4. Ansa and Meta will be used as pre- and postprocessor.

Figure 1.1 shows an overview of the established framework for fatigue analyses. First, the model is preprocessed in Ansa where an Abaqus FE model is defined. Additionally, a separate mesh is exported and delivered to the Injection mould engineer for simulation in Moldflow. The results of the Moldflow simulation contains estimated fibre orientations for the component which are mapped onto the structural mesh by the use of Digimat. A material model is defined in Digimat by reverse engineering of the constituent's properties based on experimental data of the SFRP. Implementing the Digimat material model and the fibre orientation in Abaqus, the FE analysis is performed coupled with Digimat. During the analysis, Digimat will compute the material response in each integration point based on the microscopic properties of the constituents. The fatigue data in terms of S-N curves, i.e. stress range, S , versus number of cycles to failure, N , is processed and included in the Digimat material model. After importing the Digimat material and the FE results into nCode DesignLife, the fatigue analysis will be performed coupled with Digimat. The anisotropy will be accounted for by prediction of local S-N curves in each material point depending on fibre orientation. The result of the fatigue analysis will be presented as number of cycles to failure.

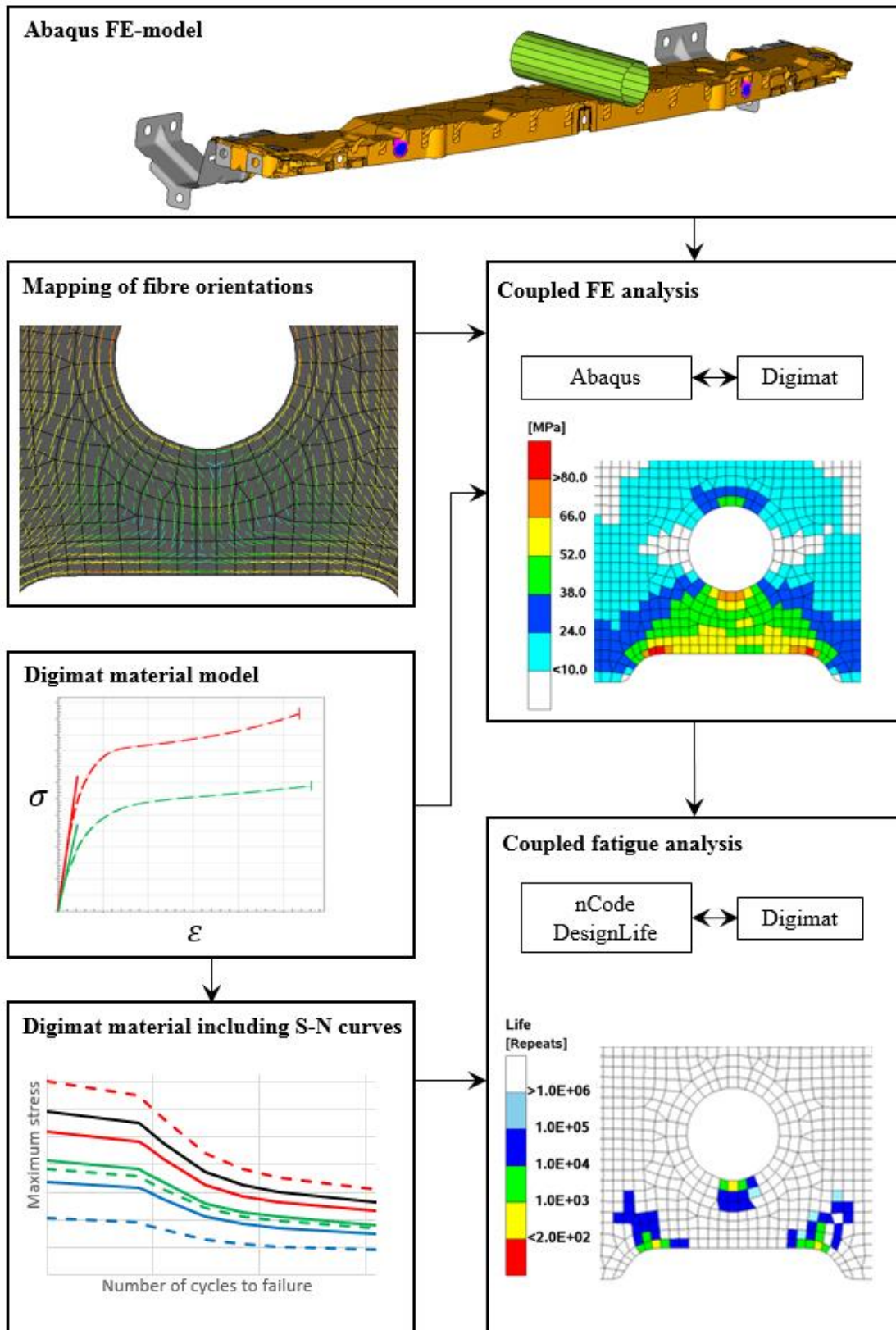


Figure 1.1 Workflow of fatigue analysis of an SFRP. The material is modelled in Digimat. Static FE analyses and fatigue analyses are performed coupled with Digimat.

2 Short Fibre Reinforced Polymers Modelling

Short fibre composites consist of a continuous phase, the *matrix*, in which one or more discontinuous phases, the *fibres*, are included. The fibres act as a reinforcement. Composites can be divided into many groups, such as unidirectional continuous fibre composites or short fibre reinforced composites. In this study, the composite is a polymer reinforced with short fibres. Common for most types of composites is the anisotropy of the material caused by the reinforcement.

For continuous fibre composites classical laminate theory is often used to account for the anisotropic behaviour. For a component of SFRP, the fibre orientation needs to be estimated throughout the structure to account for the anisotropic behaviour.

2.1 Mechanical Properties

Load applied to a composite is transferred through the matrix to the fibres. In a short fibre composite the load transfer is built up from the fibre ends. The fibre dimensions and the mechanical properties of the constituents have a strong influence on the total load transfer. This will affect the stiffness and strength of the composite [2]. They will also affect the failure mode, such as; fibre breakage, yield and rupture of matrix or debonding between fibres and matrix [2, 4]. The fibre dimensions can be represented by the aspect ratio, defined as the fibre length divided by the fibre diameter. Studies show that the longitudinal stiffness and strength increase with increased fibre aspect ratio and approach that of a continuous fibre composite. In the transverse direction, the stiffness is often assumed to be independent of the aspect ratio [2].

Injection moulded thermoplastics in cyclic loading often show progressive softening and creep strain accumulation until failure. The strain at which failure occurs in Low Cycle Fatigue (LCF) is often close to that of monotonic failure. By adding fibres to thermoplastics the fatigue crack threshold K_{th} is increased. Therefore the composite is less sensitive to loading and initial cracks. However, a crack in the composite tends to propagate faster and behave more brittle than a crack in pure thermoplastics [4].

2.2 Injection Moulding

The dominant manufacturing process for SFRPs is injection moulding. This is a process that enables complex cost efficient structures. The process is most suitable for thin structures since a significant part of the process time is the cooling time, which is approximately proportional to the square of the thickness [5].

Prior to the injection moulding of a fibre reinforced thermoplastic, a compounding of fibre and resin is carried out in order to separate the fibres from each other such that there is total enclosure of the fibres. To maintain an appropriate aspect ratio, caution is taken such that a low fibre breakage occurs, since this would have a negative effect on the matrix to fibre stress transfer [2].

The quality of the injection moulded product will be dependent not only on the rheological properties of the material but also on process variables such as pressure, flow velocity, time and temperature. These will affect the residual stresses and the fibre orientations in the structure. Furthermore, when two mould fronts meet, a so

called *weld line* is created, causing a reduction of strength, see Figure 2.1. Weld lines at critical areas can often be avoided by appropriate positions of the injection gates.

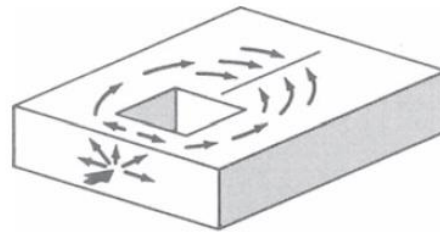


Figure 2.1 Two mould fronts meet, causing a weld line. Reproduced from [6] with permission of Springer Science+Business Media.

Residual stresses, fibre orientation and weld lines can be estimated by simulation of the injection moulding process. In this study, the fibre orientation is predicted by using the simulation tool Moldflow.

2.3 Fibre Orientation

The process induced fibre orientation depends on geometrical features of the component such as thickness, ribs and injection mould gate location. Other influencing factors are processing variables from the injection moulding such as fill time, melt- and mould temperature. The fibre orientation in injection moulded structures varies through the thickness and can usually be divided in a skin-core-skin layup. In the skin layers the fibres are oriented in the flow direction and in the core transversally to the flow [7]. Since the material properties vary along and transverse to the fibre direction, this results in a complex anisotropic material with different stiffness, yield, ultimate and fatigue strength in each material point and in each direction. The fibre orientation is in many analysis tools, such as Moldflow and Digimat, described by the orientation tensor, a_{ij} , see Figure 2.2.

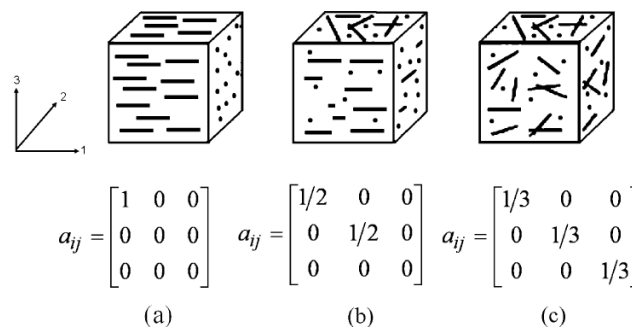


Figure 2.2 Examples of orientation tensors a) Unidirectional 1-direction b) Randomly oriented in 1-2 plane c) Randomly orientated in 3D. Reproduced from [8] with permission of Springer Science+Business Media.

The orientation tensor describes the fibre orientation distribution for a specific material point and is based on the orientation distribution function. The diagonal terms describes the fibre orientation intensity in the 1, 2 and 3 direction while the off-diagonal terms express a redistribution of the intensities in the angular space [9].

2.4 Digimat

In most cases, it is not possible to perform a full multi-scale simulation of an SFRP structure using the microscopic properties directly due to the complexity of the problem and computational efforts [10]. Instead, the composite structural analyses are performed on the macroscopic material scale, where the material is considered to be locally homogeneous. Nevertheless, it is of importance to accurately predict the influence of the microscopic properties on the macroscopic scale. Digimat is a multi-scale material modelling software that can be used for this purpose. The software platform consists of several tools for modelling the anisotropy of composites and calibrating the material parameters to experimental data.

In this study, Digimat is used to model the anisotropic material, including fatigue parameters, and to map the fibre orientations from Moldflow simulations onto the structural mesh. To model the SFRP in Digimat, separate models for the matrix material and the fibre material are implemented, along with additional information of the microstructure. In order to predict the macroscopic material properties, the microstructure is homogenized in Digimat using a Mean Field Homogenization (MFH) method.

2.5 Micro to Macro Scale

The transition between the micro and macro scales of a material can be made by introducing a *Representative Volume Element*, RVE, which should be small compared to the structural dimensions and sufficiently large to represent the microstructure. In Digimat, each macro material point is the centre of an RVE. At the micro scale level the RVE contains a sufficient number of short fibres with directions that give a representative macroscopic response [9].

First, a relation between the micro and macro scales needs to be defined. By adopting a homogenization scheme that fulfils the Hill-Mandel condition, stating the equivalence of the strain energy at macro and micro scales, the macroscopic stress and strain of the RVE can be associated to the microscopic as the volume averages of the corresponding microscopic quantity [2].

To represent the constitutive properties at the macro scale, a MFH of the RVE microstructure is applied in Digimat. The objective of the homogenization is to find a homogeneous material with the same effective stiffness as the actual microstructure. The MFH results in an approximation of the volume average of the stress and strain fields over the RVE.

There are different types of MFH methods, all making approximations of the microscopic behaviour. In this study, the Mori-Tanaka model will be used for the MFH. The model assumes that each inclusion behaves as if it was isolated in the matrix, and the RVE is treated as a single inclusion problem subjected to a strain corresponding to the average strain in the matrix phase. This indicates that the model is suitable for composites of lower fibre volume fractions, up to approximately 25 %. However, it can give accurate predictions for composites of fibre volume fractions beyond this limit [9].

In the MFH procedure of an SFRP, the RVE is decomposed into a number of grains, called *pseudo-grains*, with unidirectional fibre alignment. Each pseudo-grain is expressing a unique segment in space, representing a specific angular increment. The number of angular increments allowed is 6 to 16, Digimat recommends 12 for good compromise between accuracy and computational time. In the case of randomly oriented fibres in the SFRP, all pseudo-grains are of the same importance. However, if the fibres are oriented in a specific direction, as for injection moulded components, the pseudo-grains with corresponding directions have a larger influence on the homogenized response. Accordingly, the pseudo-grains are given different weighting factors, with greater weight for pseudo-grains with the directions of the existing fibre alignment, given by the orientation tensor, in the RVE [9]. In this way a realistic response of the RVE is achieved.

The homogenization of the unidirectional pseudo-grains is then performed in two steps. First, each pseudo-grain is homogenized according to the Mori-Tanaka model. In the second step, the collective homogenized response of the pseudo-grains is computed resulting in the final response of the RVE. Here, the Voigt model is used which assumes that the macro stiffness is equal to the volume average of the micro stiffness. The concept of pseudo-grains and the two-step homogenization are illustrated in Figure 2.3.

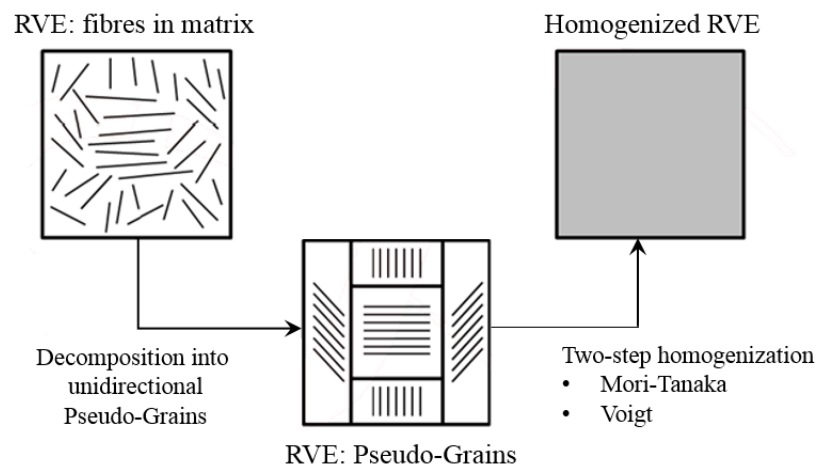


Figure 2.3 Principle of homogenization. The RVE is decomposed into a finite number of unidirectional pseudo-grains. Each pseudo-grain is homogenized according to the Mori-Tanaka method. In a second step, the RVE of homogenized pseudo-grains are homogenized by the Voigt model into a fully homogeneous material. Reproduced from [9], with permission from e-Xstream engineering.

2.6 Pseudo-Grain based Fatigue Model

Digmat uses a high cycle pseudo-grain based fatigue model to estimate the fatigue life of SFRPs. As previously described in the homogenization process, each RVE is decomposed into pseudo-grains, in which the fibres are aligned. By using a pseudo-grain failure model, some pseudo-grains are allowed to fail without causing failure of the whole RVE.

For a given stress state, the number of cycles to failure for a unidirectional composite can be predicted by the 3D transversal isotropic Tsai-Hill fatigue criterion, FC , as [9];

$$\begin{aligned}
 FC(N_c) = & \frac{\sigma_L^2}{S_L(N_c)^2} - \frac{\sigma_L(\sigma_{T1} + \sigma_{T2})}{S_L(N_c)^2} + \frac{\sigma_{T1}^2 + \sigma_{T2}^2}{S_T(N_c)^2} + \left(\frac{1}{S_L(N_c)^2} - \frac{2}{S_T(N_c)^2} \right) \sigma_{T1}\sigma_{T2} + \\
 & + \frac{\sigma_{LT1}^2 + \sigma_{LT2}^2}{S_{LT}(N_c)^2} + \left(\frac{4}{S_T(N_c)^2} - \frac{1}{S_L(N_c)^2} \right) \sigma_{TT}^2 \quad (2.1)
 \end{aligned}$$

where σ_L , σ_{T1} and σ_{T2} are the longitudinal and the two transversal stress amplitudes, respectively. Furthermore, σ_{LT1} and σ_{LT2} are the shear stress amplitude between the longitudinal and transverse directions and σ_{TT} is the shear stress amplitude in the plane normal to the longitudinal direction. For each RVE, these stresses are obtained from the macroscopic stress tensor combined with the orientation tensor. $S_L(N_c)$ and $S_T(N_c)$ are the stress amplitudes for the critical number of cycles, N_c , in the S-N curves of longitudinal and transverse loading of unidirectional composites, respectively. The value of $S_{LT}(N_c)$ is obtained through the Tsai-Hill failure criterion in combination with the stresses $S_L(N_c)$, $S_T(N_c)$ and $S_\theta(N_c)$. $S_\theta(N_c)$ corresponds to the S-N curve of a unidirectional composite with an arbitrary loading angle θ . Failure is assumed to occur when the criterion, FC , reaches the value 1 and the number of cycles to failure is found as the corresponding value of N_c .

In this fatigue model, the failure criterion in equation 2.1 is applied to each unidirectional pseudo-grain of the RVE. For a given stress state, the fatigue failure criterion of the RVE is obtained as a weighted average of the failure criteria of the pseudo-grains, based on the orientation distribution in the RVE.

3 Analyses

The coupled fatigue analysis requires preparation of the Digimat material model and fatigue parameters by calibration to experimental data and processing of the fatigue data. Also, the fibre orientations of the component need to be implemented in the analyses. In this section, the method of performing the coupled analyses are described and the case study is used as an example. More specific details and results of the case study are presented in Chapter 4.

3.1 Calibration of the Material Model

The material used in the case study is Durethan BKV 30, which is an SFRP of polyamide PA6, reinforced with 30 % mass fraction of glass fibres. To model the SFRP in Digimat, the matrix and the fibre material models of the SFRP are defined separately. Additionally, to fully define the microstructure, the mass fraction and aspect ratio of the fibres together with the fibre orientation tensor are needed. In the previous study of static strength carried out at Volvo Cars, an elastoplastic material model is used to model the matrix phase and a linear elastic model is used for the fibres [3]. However, in the fatigue analysis the material model of the SFRP has to be linear. Therefore, a linear elastic model is used also for the matrix. The linear elastic material parameters are identified by a least-square fit of the material response to experimental data [9].

The experimental data for the SFRP material is provided by the material supplier Lanxess. The strength data is based on tests of two types of specimens; direct injection moulded dumbbells and dumbbells cut out from injection moulded plates. They are from now on referred to as moulded and milled specimens, respectively. In the moulded specimens, the fibres are highly oriented in the loading direction [3]. The milled specimens are cut out from the moulded plates in two different directions; longitudinal and transverse to the flow direction, see Figure 3.1. The fibre orientations of these specimens are of mid-level, with different orientation in the core and skin layers.

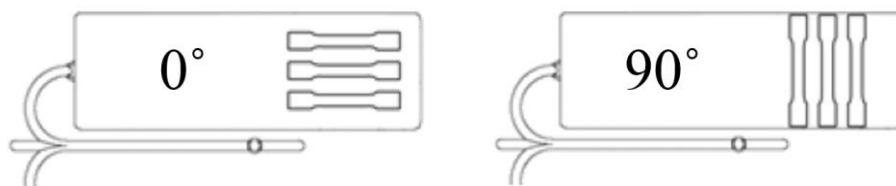


Figure 3.1 Dumbbells cut out of plates, angle with respect to flow direction. Reproduced from [9], with permission from e-Xstream engineering.

True stress-strain data from uniaxial tensile tests are given for the three different types of specimens. In the Digimat-MX module, reverse engineering (RE) is performed to calibrate the material parameters of the SFRP to the tensile tests. The RE iteratively optimizes selected material parameters to fit the prediction of the stress and strain relation to the experimental material response [9]. By varying the Young's modulus of the matrix material and aspect ratio of the fibres, an elastic model of the matrix is estimated for a selected elastic range. The range is chosen such that the elastic model gives good approximation of the analysed stress range. For the calibration, the fibre

orientation tensors of milled and moulded specimens are specified as in the previous strength study, see Table 3.1 [3].

Table 3.1 Fibre orientation of milled and moulded test specimens

Orientation of milled specimen			
	a_{11}	a_{22}	a_{33}
Skin, $0.8 \times$ thickness	0.75	0.2	0.05
Core, $0.2 \times$ thickness	0.2	0.75	0.05
Orientation of moulded specimen			
	a_{11}	a_{22}	a_{33}
Constant through thickness	0.9	0.1	0

The results from the RE is shown in Figure 3.2. The graphs show true stress-strain curves for the elastic model of the SFRP after calibration for the different microstructures of milled and moulded specimens, along with the corresponding experimental curves from the tensile tests.

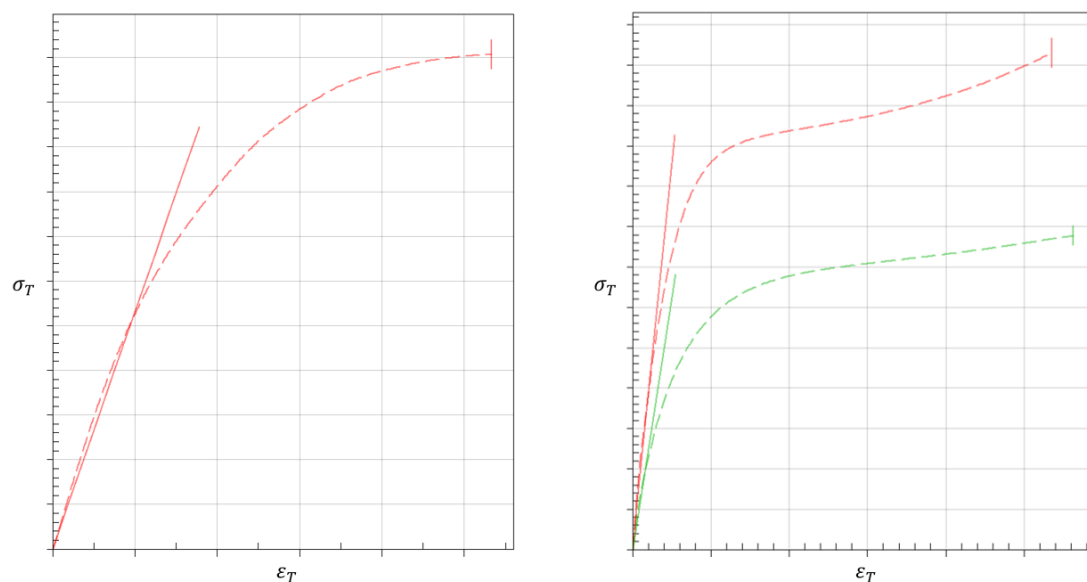


Figure 3.2 Calibration of linear elastic material model to tensile tests expressed in true stress-strains. Dashed lines correspond to experimental data and solid lines represent the calibrated elastic material response of the SFRP. To the left: moulded specimen. To the right: milled specimen, red and green lines correspond to longitudinal and transverse loading, respectively.

The calibration resulted in an aspect ratio of 15. This was the chosen lower bound in the RE, usually short fibres have aspect ratios in the range of 15 to 30 [9]. The Young's modulus of the matrix material from calibration was lower than the value of the elastoplastic matrix material.

3.2 Fatigue Data

The pseudo-grain based fatigue model, described in Section 2.6, requires the S-N curves of unidirectional composites, loaded longitudinally, transversally and in an arbitrary angle θ to the fibre direction. For an injection moulded SFRP, it is not

possible to achieve perfect fibre alignment. However, S-N curves of unidirectional composites can be estimated by internal reverse engineering of three S-N curves for milled specimens with varying fibre orientations. Specifically, the longitudinal and transverse loading directions relative to the flow direction are required to perform RE, along with loading at an arbitrary angle θ [9]. An overview of this procedure is shown in Figure 3.3.

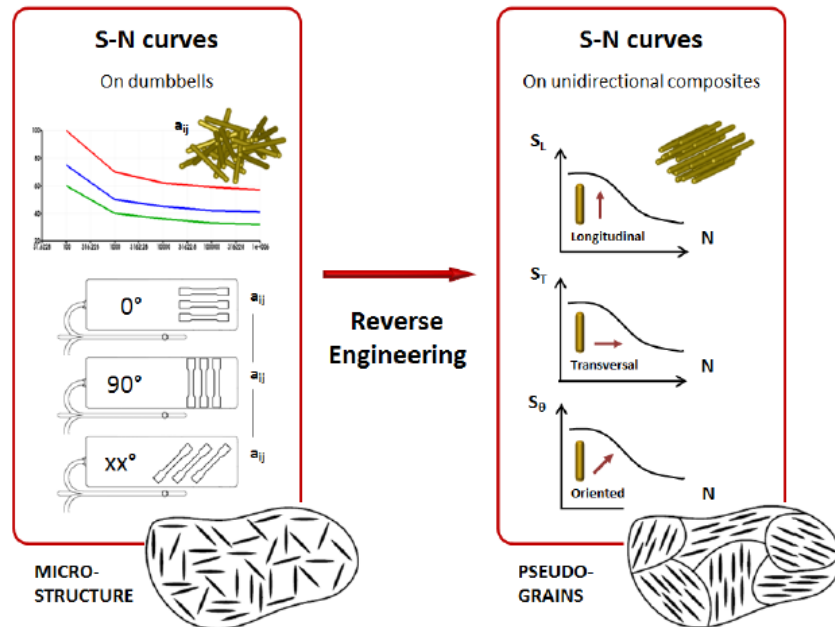


Figure 3.3 The S-N curves of unidirectional composites can be obtained by reverse engineering of milled specimen's S-N curves. Reproduced from [9] with permission from e-Xstream engineering.

The experimental fatigue data of Durethan BKV 30 is given as an S-N curve for moulded specimen, i.e. with assumed orientation as in Table 3.1. The S-N curve is based on cyclic fatigue tests with load ratio $R = \frac{\sigma_{\min}}{\sigma_{\max}} = 0$. It is presented as maximum nominal stress versus number of cycles to failure. The fatigue tests are performed on moulded dumbbells of standard ISO527 1A, conditioned according to standard ISO1110. The cyclic loading is performed at 5 Hz, temperature 23 °C.

As only one S-N curve of moulded specimens is available, the three S-N curves for the milled specimens are estimated from the available curve. It is assumed that the fatigue stress is changed for the milled specimens in the same way as the Ultimate Tensile Strength (UTS) is changed. The S-N curves for the milled specimens, loaded longitudinally and transversally to the flow, can then be estimated from the data of the moulded specimen [12]. In other words, the UTS ratios of milled and moulded specimens, $R_{UTS} = \frac{UTS_{\text{milled}}}{UTS_{\text{moulded}}}$, are used as scaling factors of the S-N curve.

The UTS of milled specimens, loaded longitudinally and transversally to the flow direction, are estimated from the true stress-strain data of corresponding specimens. First, the engineering stresses are computed from the true stress strain data, by assuming incompressibility [11]. The UTS is then obtained as the highest engineering stress for each specimen.

There are no additional stress-strain data available to predict the UTS of an arbitrary load angle, θ . A load angle of 45° is chosen for the third S-N curve and the UTS value has to be predicted from the three tensile tests of milled specimens, loaded at 0° and 90° , and the moulded specimen. This can be done by a more advanced procedure where the static failure parameters of the SFRP is derived from experimental data and applied to the 45° specimen to obtain the UTS. It is based on the assumption that failure occurs in the fibres for the longitudinally loaded specimens and in the matrix phase for specimens loaded at 45° and 90° . In this study, this prediction was performed by Benoît Bidaine at e-Xstream engineering [13]. The results for the estimated ratios of UTS that is used for scaling of the S-N curve are presented in Table 3.2.

Table 3.2 UTS ratios of milled specimens at different load angles with the moulded specimens as reference.

	Milled, 0°	Milled, 45°	Milled, 90°
R_{UTS}	0.881	0.7	0.574

As a recommendation, if the procedure described above is not performed, the UTS ratio for a 45° loaded specimen should be lower than the average value of the longitudinal and transverse loaded specimens. It can be estimated by;

$$R_{UTS,45} = R_{UTS,90} + 0.25(R_{UTS,0} - R_{UTS,90}) \quad (3.1)$$

Here, $R_{UTS,45}$, $R_{UTS,0}$ and $R_{UTS,90}$ correspond to the UTS ratios of milled and moulded for load angles 45° , 90° and 0° , respectively [13].

With the three S-N curves of milled specimens loaded at different angles, the curves of unidirectional SFRPs are obtained by internal RE in the Digimat-MF module. The microstructure of the milled specimen is defined as in Table 3.1. Figure 3.4 shows the S-N curves of moulded and milled specimens and the unidirectional curves from RE.

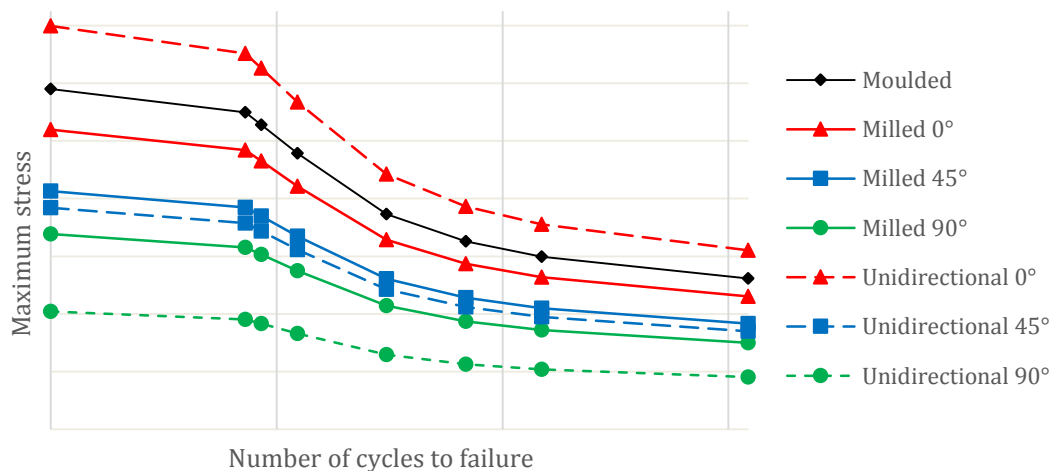


Figure 3.4 Experimental S-N curve of moulded specimen and predicted S-N curves of milled and unidirectional specimens.

In the figure, the S-N curves are given as maximum stress as a function of number of cycles to failure with a load ratio $R = 0$. However, nCode uses the stress amplitude for evaluation of the fatigue life and consequently, the curves are scaled by a factor of 0.5.

3.3 Mapping of Fibre Orientations

In order to implement the process induced fibre orientation in the fatigue analyses, the results from the injection moulding simulation have to be mapped onto the structural mesh. In the Digimat-MAP module the results of an injection moulding simulation can be mapped onto a structural analysis mesh dissimilar to the process simulation mesh. It is possible to map simulated process induced properties such as fibre orientations, residual stresses and weld lines [9].

The Moldflow results contain the fibre orientation, stored as a symmetric orientation tensor, as described in Section 2.3, for each integration point of the Moldflow mesh. The mesh for the Moldflow simulation consists entirely of triangular elements while the structural mesh preferably consists of mainly quadratic elements. The elements are discretized into layers, capturing the through the thickness variation of the fibre orientation. In Digimat-MAP, the orientation data is mapped onto the integration points of the structural mesh from the integration points in the Moldflow mesh.

3.4 Coupled Analysis Abaqus-Digimat

The fatigue analysis requires the results from a static coupled Abaqus-Digimat FE analysis. Digimat-CAE is the module that enables the coupling of the anisotropic material properties to the structural analysis. There are different types of solution procedures in Digimat-CAE, using strong or weak multi-scale coupling. In order to perform fatigue analyses, a strongly coupled solution procedure, *Micro solution*, is chosen. In this solution, for each iteration in the structural analysis, Digimat interactively computes the macroscopic stress response and updates the tangent stiffness of the composite. This results in high computational effort, but allows for microscopic outputs, such as the orientation tensor, needed in the fatigue analysis. There is another solution method, *Hybrid solution*, available in Digimat. For the linear elastic material model, this method assumes a constant stiffness response and uses a pre-computed tangent stiffness tensor. The hybrid solution was used in the previous static strength study and is assumed to be sufficient. However, using the micro solution gives a more straight-forward workflow of the fatigue analyses.

Figure 3.5 schematically shows the workflow of the coupled analysis. Firstly, the material parameters are implemented and calibrated to experimental data in Digimat-MF and Digimat-MX according to Section 3.1. This only needs to be performed once for each material. The mapping is performed in Digimat-MAP. The resulting orientation file is implemented in Digimat-CAE, along with the elastic material model. In Digimat-CAE, interface files are generated containing the Digimat material and Abaqus user material definitions. The interface files are included in the Abaqus input file, enabling the interaction between Abaqus and Digimat during the analysis. The Digimat material is assigned to each composite shell element. The generated Digimat material file points to the fibre orientation file.

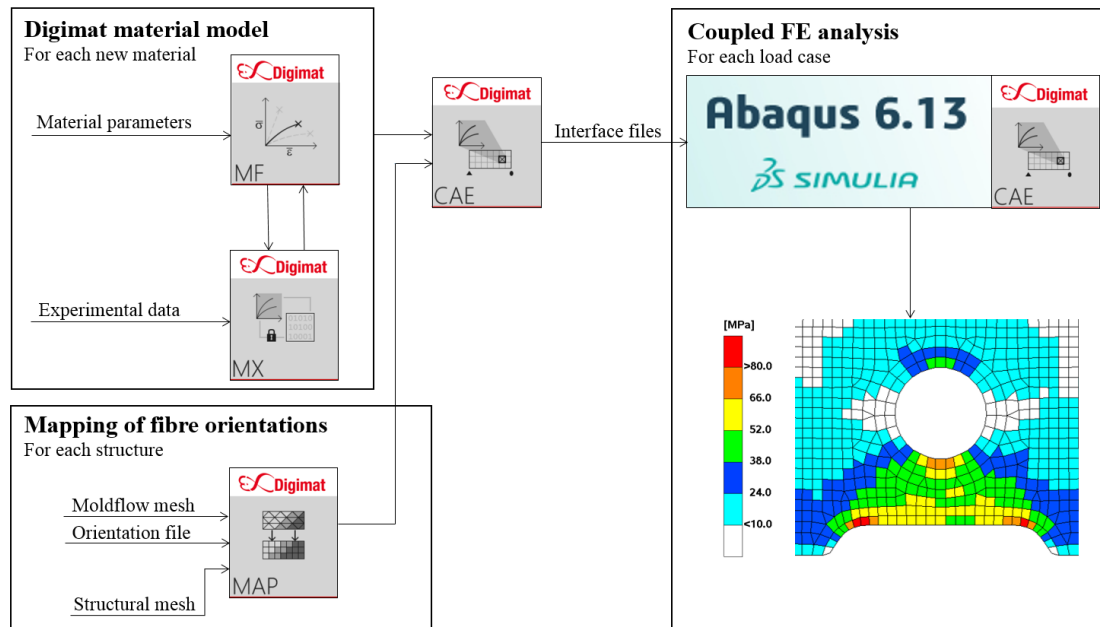


Figure 3.5 Workflow of coupled FE analysis. The material model of the SFRP and the fibre orientations are implemented in Abaqus. The FE analysis is performed coupled to Digimat.

During the coupled analysis, Abaqus will interact with Digimat to compute the macroscopic stress response for each integration point of all composite shell elements. In each iteration of the analysis, Digimat evaluates the macroscopic response using the Mori-Tanaka MFH model. The data from the orientation file is extracted for each integration point in all layers of the shell elements and used in the computation of the properties of the underlying composite [9].

The coupled FE analysis results is stored as an Abaqus output database file where the macroscopic stresses and strains from Digimat are included, along with the fibre orientations.

3.5 Coupled Analysis nCode DesignLife-Digimat

nCode DesignLife is a fatigue analysis software that allows for fatigue life prediction of a structure given results from a finite element analysis. In order to perform a coupled nCode DesignLife-Digimat fatigue analysis, the result file of a static FE analysis is needed, such as the Abaqus output database file obtained in the Abaqus-Digimat coupled analysis.

The results from the FE analyses are imported into nCode DesignLife where a cyclic loading of constant amplitude is defined. DesignLife uses an average of the stresses and fibre orientations, stored in section points, one for each layer of the composite shell elements.

Due to the anisotropy of the SFRP, the fatigue strength varies throughout the component depending on the fibre orientation. To estimate the fatigue life, a local S-N curve for each section point is created by Digimat during the coupled analysis. The local S-N curve is predicted from the fibre orientation and the stress state or stress principal direction of the section point, according to the pseudo-grain fatigue model

described in Section 2.6. At pseudo-grain level, the extended Tsai-Hill criterion in equation 2.1 is applied and weighted into an S-N curve for the specific section point. In DesignLife, the stress tensors are reduced to an equivalent scalar value to be compared to the local S-N curves. This equivalent stress is defined by the method *Absolute Maximum Principal*, i.e. it is defined as the principal stress with largest magnitude [14].

During the coupled fatigue analysis, the amplitude of the loading is compared to the generated local S-N curves, expressed in amplitude stresses, in each section point. No mean stress correction is assumed to be necessary in the analysis since the S-N curves are based on the same load ratio as in the current analysis. It should be noted that it is currently not possible to define the load ratio for the S-N curves in Digimat, it is by default set to $R = -1$. Consequently, in the case of $R \neq -1$, the S-N curves and the cyclic loading should be of the same load ratio, allowing life predictions without any further adjustments [13].

The workflow of the coupled fatigue analysis is shown in Figure 3.6. From the experimental S-N curves, the S-N curves of unidirectional composites are predicted in Digimat-MF, according to Section 3.2. New Digimat material files of the elastic material model, including the S-N curves, are created and imported into Digimat-CAE along with the orientation file of the structural mesh. Interface files are generated which enables the coupling of Digimat and nCode DesignLife. By interaction with Digimat, the damage is evaluated in nCode DesignLife for each layer through the thickness of all elements.

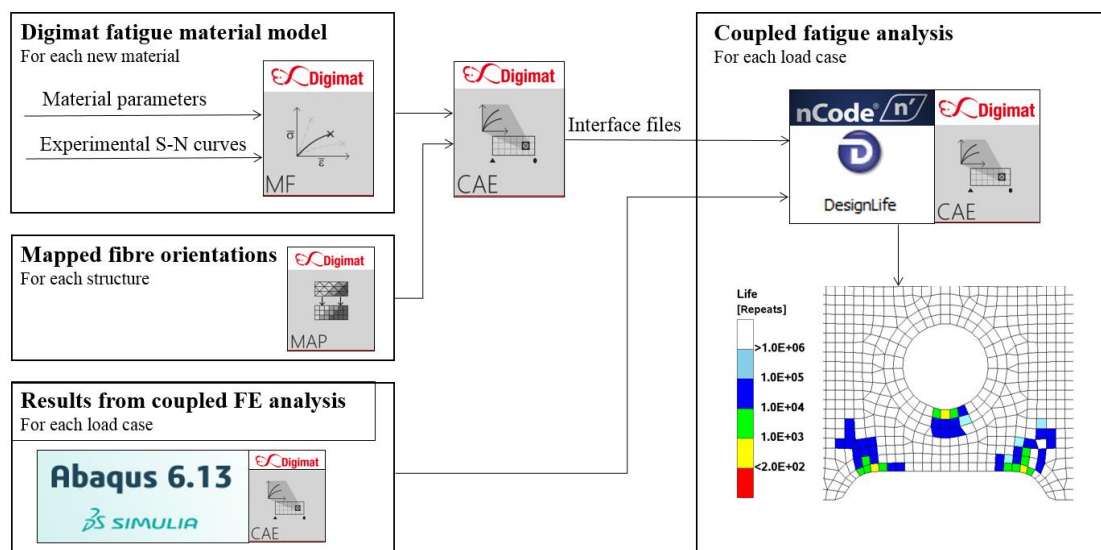


Figure 3.6 Workflow of coupled fatigue analysis. The material model, including fatigue parameters, and fibre orientations are implemented in DesignLife along with the FE results. The fatigue analysis is performed as a coupled analysis where DesignLife and Digimat are interacting.

4 Case Study – Front End Carrier

An existing car component was used in order to investigate the workflow of fatigue analysis of SFRPs using Digimat. The component is the Front End Carrier (FEC) of the new Volvo XC90, located as shown in Figure 4.1. It is the component previously analysed in the static strength study [3]. The component is injection moulded with Durethan BKV 30, which is a PA6 reinforced with 30 % mass fraction of glass fibres. During the injection moulding process, two metal brackets are fastened in each end of the FEC. The injection moulding Moldflow simulation, which was performed by Volvo Cars, showed that a weld line occurs in the area of the hole in the middle of the FEC, Figure 4.1. It is accounted for in terms of fibre orientations, but not its influence on the polymer properties. A weld line weakens the polymer, which would imply a reduction of the predicted life. It is possible to account for this in the analyses, but it was not included in this study since the polymer properties in the weld line are unknown.

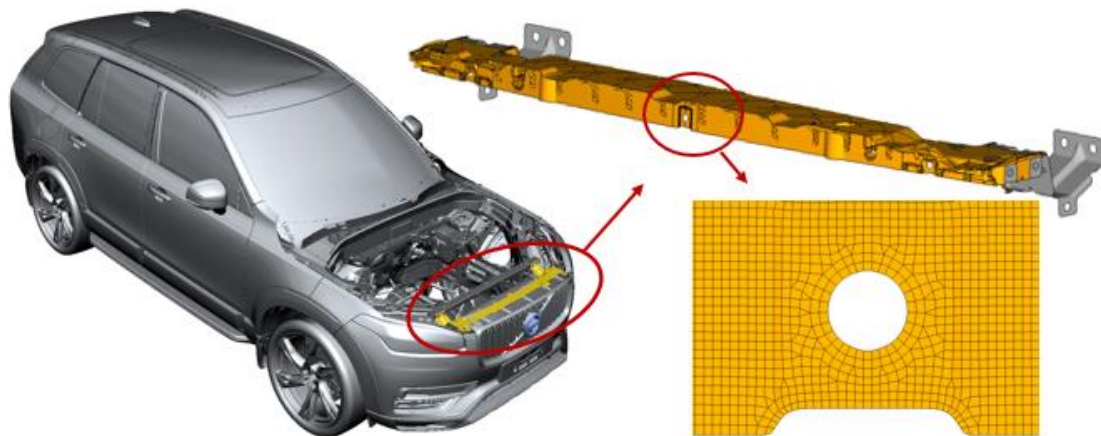


Figure 4.1 Front End Carrier, position in the XC90 and the area of interest.

For the fatigue analysis and physical testing of the FEC, a hypothetical load case was used. The component was pinned in both ends and a force was applied via a cylinder at the middle of the FEC. The load was cycled at constant amplitude with load ratio $R = 0$, to correspond to the load ratio of the material fatigue data. The load case was chosen due to its simplicity and its possibility to be set up in a test rig. This load case has previously been used in the static strength analysis, with good correlation to tests [3].

4.1 CAE Fatigue Analysis

The geometry of the FEC was meshed in Ansa for the structural analysis in Abaqus. Composite shell elements were used to model the mid-plane of the thickness with SFRP. In the previous static strength study, 10 layers of the composite shell element were used, and therefore also used in this study [3]. The metal brackets were modelled with conventional shell elements and connected to the composite with Abaqus tie contacts. Additionally, the force is applied to a modelled cylinder and transferred to the FEC by contacts. Three different load levels were chosen to be evaluated in the fatigue analysis; 210, 260 and 310 N.

To model the pinned boundary conditions, a distributed coupling is used to couple the nodes of each boundary to a node free to rotate around the x -axis. This is a simplification made in order to avoid convergence problems during the coupled FE analysis. The FE model is shown in Figure 4.2.

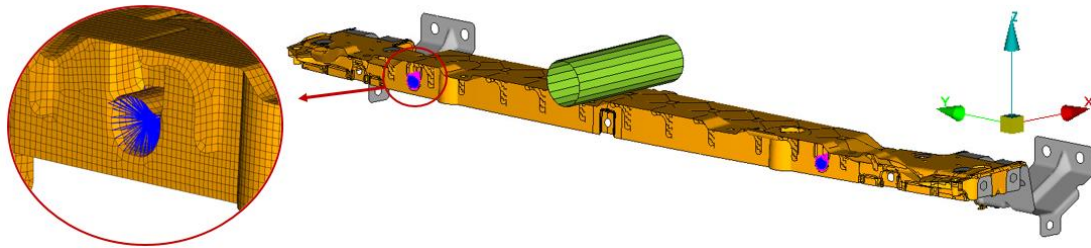


Figure 4.2 FE-model of the FEC with load cylinder. Enlarged region shows distributed coupling used to model the pinned boundary conditions.

In order to obtain good accuracy of the injection mould simulation, each element is divided into 20 layers through the thickness. The fibre orientation from the simulation is mapped onto the structural mesh, where it is reduced to 10 layers through the thickness.

Figure 4.3 shows the first principal fibre direction of the top layer and the middle layer of the area of interest in the structural mesh, after reduction to 10 layers. The vector field corresponds to the first eigenvector of each integration point's orientation tensor, with the colour corresponding to the normalized eigenvalue. The value can be interpreted as the probability of the fibres being oriented according to the direction of the eigenvector. A comparison of top and bottom layer are presented in Figure A.1, showing the symmetry through the thickness.

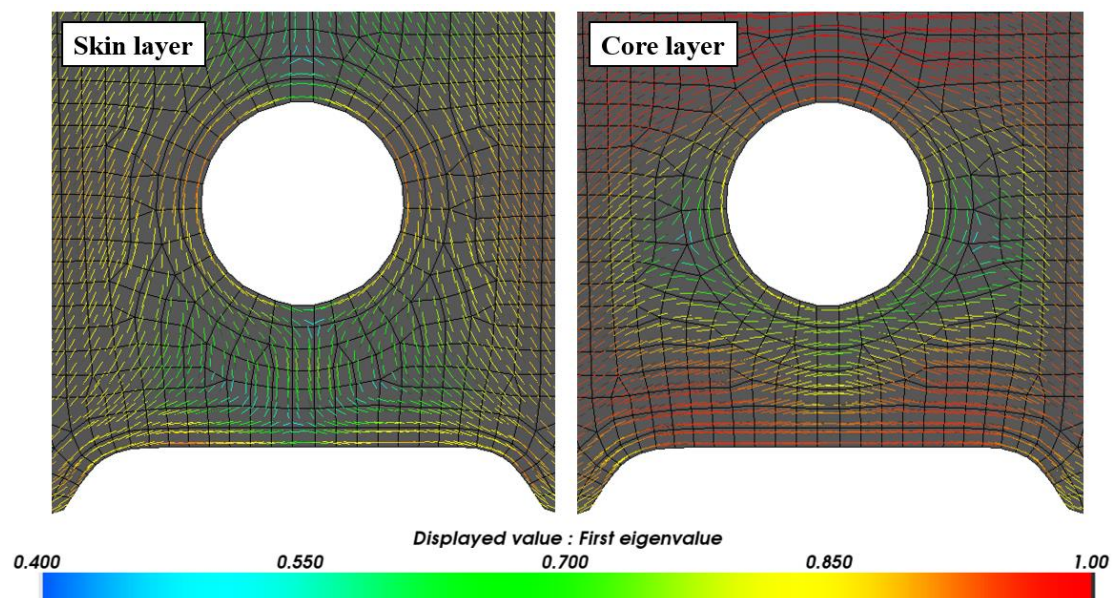


Figure 4.3 Fibre orientations of the top and middle layer of each integration point. The vectors show the first eigenvector of the orientation tensor and the colour its eigenvalue.

For the fatigue analysis, a static analysis using the linear elastic material model was performed. For evaluation of the stress field, a static analysis was also performed using the elastoplastic material model that was used in the previous static strength study [3]. Figure 4.4, shows the first principal stress at the area of interest for the mid

load level, 260 N. The colour plots show the result for the layer with the highest principal stress of each element. It can be seen that the results are similar, with highest stresses located in the same areas, but with different magnitudes.

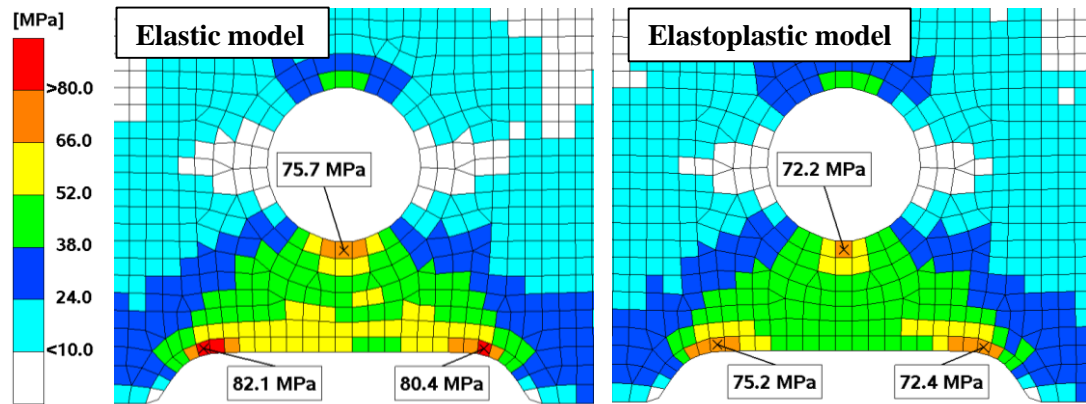


Figure 4.4 First principal stresses, at load 260 N, of the layer with the highest stress in each element. To the left: Elastic material model. To the right: Elastoplastic matrix material model.

The three load levels were evaluated and the fatigue life was predicted for each layer of all shell elements, the result can be seen in Figure 4.5. The figure show the results for the layer of shortest life for each shell element.

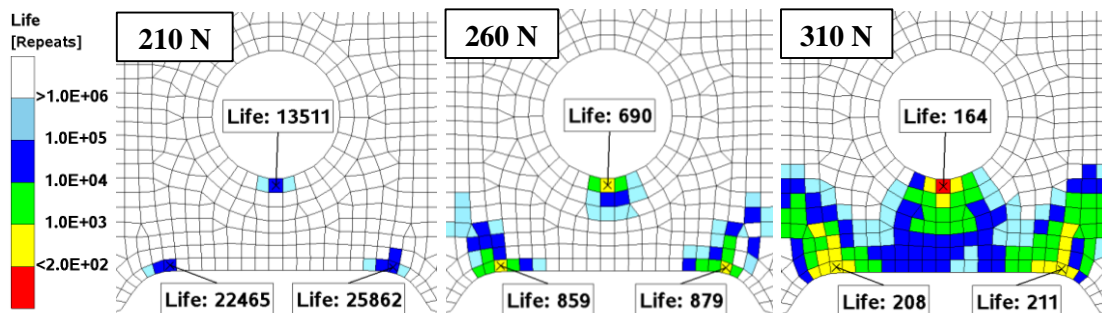


Figure 4.5 Predicted life given as number of cycles to failure. The life is given for the layer of shortest life for each element.

Comparing the stress field of load level 260 N, in Figure 4.4, to the corresponding life, in Figure 4.5, it is seen that the element with the shortest life does not coincide with the highest stress. The fatigue life is not only dependent on the stress level, but also the stress direction in combination with the fibre orientation. Table 4.1 shows the life, first principal stress and the *fibre share* for all layers of the element with the layer of shortest life at load level 260 N. The fibre share is a measure of the fibre alignment in the loading direction, where the value 1 indicates perfectly aligned fibres with the dominant stress direction [14]. As can be seen, the fibre share is symmetric through the thickness, but the stress magnitude is not.

Table 4.1 The predicted life, first principal stress and fibre share of all layers of the element with the most damaged layer at load 260 N.

Layer number	1	2	3	4	5	6	7	8	9	10
Number of cycles [-]	4018	3019	2004	1249	779	690	776	811	814	758
First principal stress [MPa]	50.4	54.1	60.5	67.0	74.4	75.7	70.4	65.7	60.8	58.6
Fibre share [-]	0.511	0.545	0.608	0.668	0.734	0.734	0.668	0.608	0.545	0.511

4.2 Physical Fatigue Test

A fatigue test of the FEC was performed in order to have a reference for the outcome of the fatigue analysis. The test was set up as a three point bending test, subjected to cyclic loading. Totally 18 test objects were fatigue tested in both the low cycle and the high cycle fatigue range.

The three point bending test was set up as shown in Figure 4.6. The fatigue testing was performed with a cyclic force controlled load of 0.8 Hz, applied through the large cylinder with a load ratio $R = 0$. At the attachments, two small pinned cylinders are pressed through holes allowing the FEC to rotate around each cylinder axis. For more specific details, see Figure A.2-A.4. The tests were performed at room temperature and no conditioning of the FECs was carried out. Prior to the fatigue test, the stiffness of each FEC was tested by a force-displacement test with a maximum load of 110 N, which is sufficiently low not to influence the fatigue test.



Figure 4.6 Three point bending test with force controlled cyclic loading and pinned boundary conditions.

The load case aimed to obtain failure in the area marked in Figure 4.7. Failure was defined as a deviation of 2 mm from the nominal displacement when the maximum load was applied, which corresponds to a visible crack. The resulting number of cycles to failure for each physical test are shown in Table A.2.

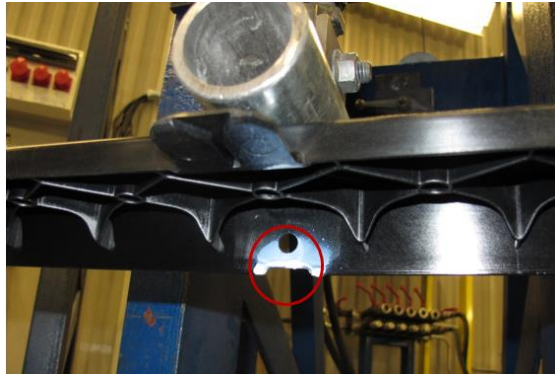


Figure 4.7 Expected failure area underneath the load.

The temperature of the material in the area of interest was measured to ensure the validity of material properties, however no significant deviation from room temperature could be noticed. Moreover, in order to more easily detect cracks, some of the components were painted, see Figure 4.7. The paint had no significant influence on the results.

For all test pieces, except one, the crack propagated from the hole to the outer edge and resulted in failure, as is shown in Figure 4.8. This location correlates well with the most critical area in the CAE life prediction in Section 4.1.

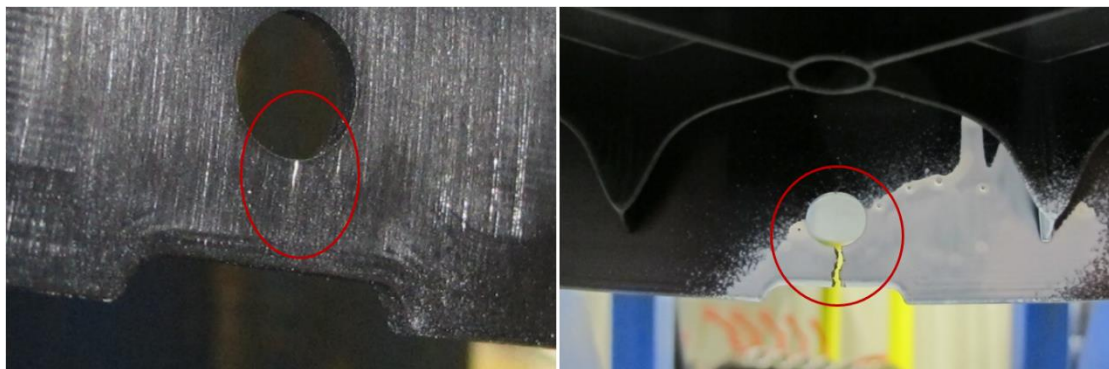


Figure 4.8 Left: Crack after 198 000 load cycles, failure after 200 696 load cycles, load 220 N. Right: Failure at predicted location.

For the test object with longest life, a crack propagated from the outer edge, see Figure 4.9. It should be noted that also this region was identified as critical in the life prediction analysis in section 4.1, although not the most critical region.



Figure 4.9 Deviating failure location at outer edge, failure at 727 594 cycles, load 210N.

It was observed that the displacement below the cylinder was accumulated during the cyclic loading, see Figure 4.10. The red cycle is measured in the beginning of the test while the blue is closer to failure. Whether this is due to ratcheting or accumulation of creep, or a combination of both, has not been examined.

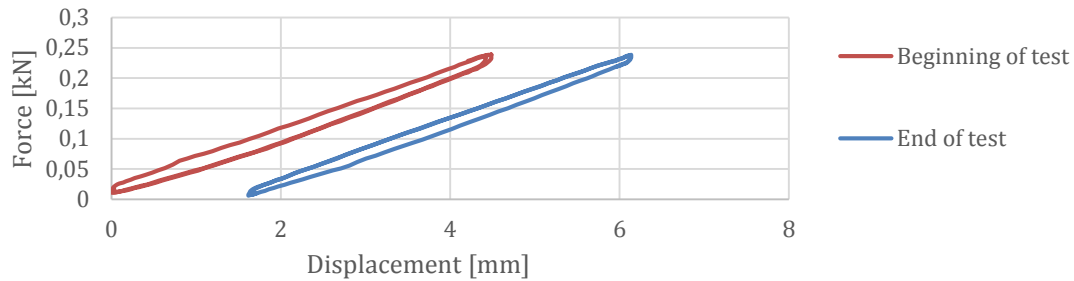


Figure 4.10 Accumulation of displacement during cyclic loading. Test object FEC 9, loading at 240 N with failure at 77 115 cycles.

4.3 Comparison of CAE Analysis and Physical Test

The fatigue test results are presented in the F-N curve in Figure 4.11. The results from the physical tests can be estimated as a linear line in the semi-log plot in the high cycle fatigue range, which is a generally known phenomenon and thus expected. The results from the CAE analysis are also included in Figure 4.11. Results for both the worst layer and for the average of the layers in the most critical element are shown.

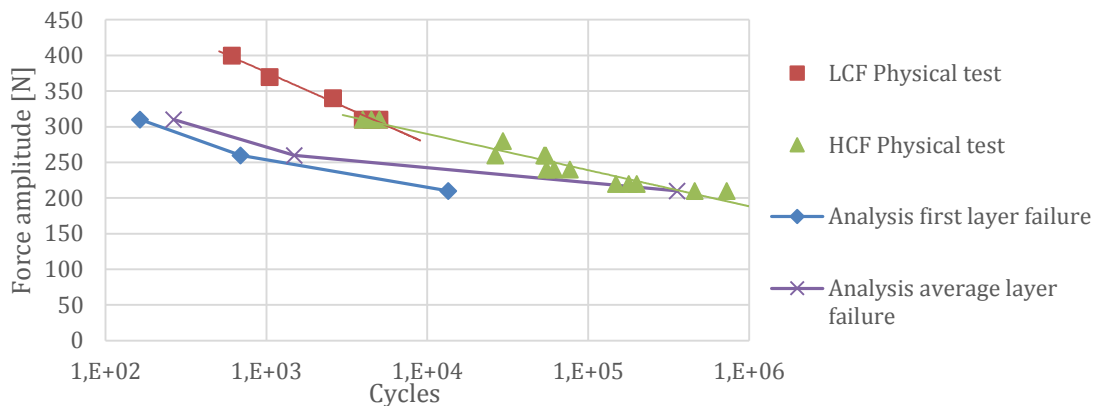


Figure 4.11 F-N curve of FEC. Comparison of physical test and CAE analysis.

Figure 4.12 shows the results from two physical stiffness tests of the FECs, along with simulation results for three different material models. The isotropic model is based on a homogeneous approximation currently used by Volvo Cars. One can see that the response of all the CAE models are stiffer than the physical tests. The physical tests show a slightly nonlinear response.

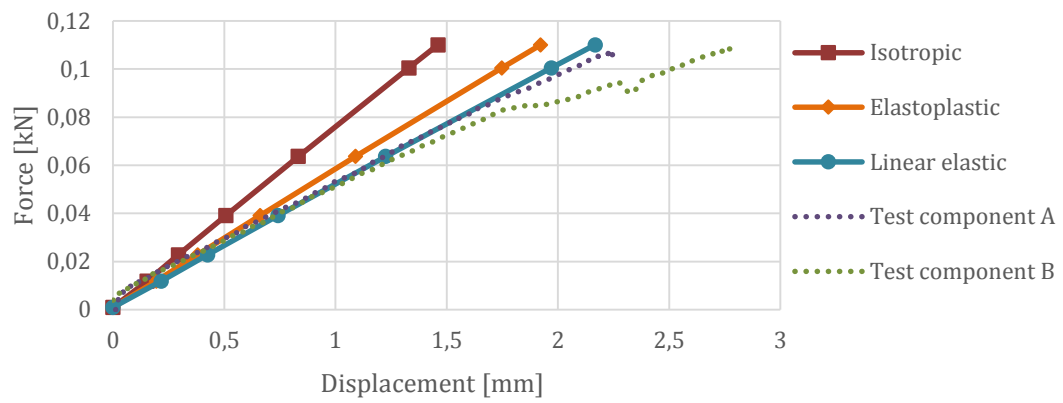


Figure 4.12 Stiffness tests, comparison of physical tests and simulations with different material models.

5 Discussion

The objective to carry out a fatigue analysis of an SFRP was successfully achieved in a case study of a car component, the FEC in the new XC90. The fatigue failure was predicted in the same region as where the failure initiated in most of the physical tests. However, the correlation between test and analyses of the FEC showed that further work and deeper knowledge are needed before satisfying analysis results can be obtained. As can be seen in the F-N diagram in Figure 4.11 in Section 4.3, the analyses predict shorter life than the life obtained in the physical tests. This could be due to several reasons, which will be discussed in the following sections.

In the stiffness test in Figure 4.12, it is shown that the elastic model gave better correlation to the test results than the elastoplastic model. This was not expected and a possible explanation is that poorly defined boundary conditions have made the FE-model too stiff.

5.1 Material Data

The provided S-N curve of the injection moulded dumbbell is based on the average stress from the cross section of the specimen. Stress concentrations, caused by the fibres, may exist in the dumbbells and may initiate failure. In that case, the stress is larger than the average stress and the S-N curve will then underpredict the fatigue life. Moreover, only one S-N curve was provided from the material supplier. Approximations had to be made to obtain the S-N curves for the milled specimens.

The response of an SFRP is in general strain rate dependent, which has not been accounted for. The material data is from tensile tests with very slow loading, i.e. approximately zero strain rate. It is used for calibration of the material model and for estimation of the UTS. The UTS in turn is used to scale the S-N curves, as described in Section 3.2. However, the fatigue testing of both the dumbbells and the FECs are carried out at fixed frequencies, 5 and 0.8 Hz, respectively. This results in different strain rates in the dumbbell and the FEC, which also vary dependent on load level.

Conditioning influence the material properties of a polyamide. In contrast to the test components, the experimental fatigue data of the material was given for a conditioned dumbbell. This may have a significant influence on the correlation between physical test and CAE analysis.

5.2 Material Modelling

The fibre orientation used when deriving the material model, see Table 3.1, has not been validated, and may have a negative effect on the correlation between physical test and CAE analysis. Furthermore, the linearization of the matrix material behaviour has a strong effect on the results and may be further optimised. Since the material behaves non-linear, the linearization will always be limiting the correlation to real tests. Cyclic hardening or viscous effects such as ratcheting or accumulation of creep could be captured with a more advanced material model. In addition, the stiffness degradation is not captured by a linear model. Currently no non-linear material model is available for the coupled fatigue analysis of Digimat and nCode.

A solution to obtain a more suitable material model could be to calibrate a new elastic model for each load level and aim to obtain similar stress levels as for the elastoplastic model. This procedure is time consuming and is not applicable in daily work projects, why this has not been further investigated.

5.3 Failure Criteria

Fatigue failure can be defined in different ways in the fatigue life simulation. Also, there may be a difference in failure definition between CAE analysis and physical test of component as well as of dumbbell. The fatigue life of a finite shell element may be defined as when the first layer in the element fails. Alternatively, all layers could be accounted for when evaluating the fatigue life. Also, it may be too conservative to define failure when only one element has failed, especially in regions with large stress gradients and fine mesh.

During the fatigue testing, the crack was allowed to propagate from the hole to the outer edge before failure was detected. It was not possible to observe exactly when the crack was initiated. However, no crack was observed for any of the components before 85% of the fatigue life.

5.4 Discretisation

Several spatial discretisations have been made during the analyses such as the Moldflow mesh, the structural mesh and the composite shell sections at which each element was divided in layers. The choice of mesh sizes and number of layers that has been used was based on previous studies and recommendations from Volvo Cars. The sensitivity of the solution with respect to discretisation has not been studied.

5.5 Analysis Method

The method for fatigue analysis using Digimat to model the material can be divided into three main tasks; material modelling, coupled FE analysis and coupled fatigue analysis. The most time consuming part is the material modelling, which includes processing and implementing experimental material data and calibrating the material model and fatigue parameters. As previously discussed, the material model is limited to elastic materials due to limitations in the software and it is thereby difficult to obtain a good correlation, see Figure 3.2. However, once a satisfying material model is obtained it can be used for analysis of different structures.

The software interfaces of the coupled analyses work well. Although, in the coupled FE analysis, convergence issues occurred. These were not present for a corresponding FE analysis using isotropic materials, hence, simplified boundary conditions were adopted.

In the current available versions of the software, there is a limitation on the load ratio of the coupled fatigue analysis. It is not possible to define the load ratio of the S-N curves in Digimat, it is by default defined as $R = -1$. If the S-N curves are given for any other load ratio, it is only possible to perform fatigue analysis of structures subjected to the same load ratio, disregarding of the mean stress correction in nCode DesignLife. This will be updated in future versions of Digimat.

6 Suggestions for Future Work

In order to achieve better correlations with physical tests, deeper knowledge of the material modelling and sources of errors are needed. A good way to start would be to perform fatigue analyses of injection moulded dumbbells. Uncertainties regarding the physical testing would be minimized. A more straightforward comparison of analysis results to the experimental fatigue data used to define the material model would then be possible. The dumbbell analysis could be used to evaluate the elastic material model and the prediction of the unidirectional S-N curves. The results could further be used to determine a suitable failure criterion.

For more accurate results, it is of interest to investigate how weld lines and residual stresses from the injection moulding process affect the calculated fatigue life. This is supported by Moldflow and Digimat and should be possible to include in the analyses.

In the case study, which was based on the micro solution method in Digimat, convergence issues appeared. The hybrid solution method could be an alternative. Investigating the use, computational time, accuracy and stability of the hybrid solution is thus recommended.

The fatigue modelling is under development in Digimat. The company behind Digimat, e-Xstream engineering, is working on extending the options for e.g. load ratios $R \neq -1$ and on including more advanced material models. It is of great interest to follow how this work will proceed to be able to perform more accurate fatigue simulations in the future.

7 Conclusions

A framework for fatigue analysis of an SFRP taking the anisotropic material into consideration has been outlined. The interaction between the multi-scale material modelling tool Digimat, the finite element code and the fatigue analysis software nCode DesignLife, is working satisfactorily. Extensive knowledge of mechanics of materials and the Digimat software is needed in order to set up the material model with reasonable assumptions. However, once the material model is obtained, the coupled analyses can relatively easily be carried out.

The results of the case study show lack of correlation between the life of fatigue tests and fatigue analyses. The difference is believed to be caused by limitations in the material modelling, along with approximations and assumptions of the material parameters. More accurate results could be obtained with additional experimental S-N curves. Also, more experience of setting up the material model in Digimat and making the right approximations will probably improve the correlation. Nevertheless, the predicted failure location was consistent with the physical test, why the analysis is considered to be a good starting point for future studies.

To get a good correlation between physical testing and CAE simulations of fatigue, it is important to define a fatigue failure criterion in a consistent way. The criterion for physical component testing should correspond to the CAE analysis failure criterion. It must be determined whether the CAE failure criterion should account for one or more layers of one or more elements in the evaluation of the fatigue failure. Also, the S-N curve needs to account for stress concentrations in the dumbbells.

References

- [1] Collins JA. Failure of materials in mechanical design: analysis, prediction, prevention. New York: Wiley; 1993.
- [2] Agarwal BD, Broutman LJ, Chandrashekhara K. Analysis and performance of fiber composites. Hoboken, N.J: John Wiley & Sons; 2006.
- [3] Landervik M, Jergeus J. Digimat Material Model for Short Fiber Reinforced Plastics at Volvo Car Corporation. European LS-DYNA Conference. Würzburg, Germany; 2015.
- [4] Reifsnider KL. Fatigue of composite materials. Amsterdam: Elsevier; 1991.
- [5] Klason C, Rigdahl M, Kubát J, Boldizar A. Plaster: materialval och materialdata. Stockholm: Sveriges verkstadsindustrier; 2001.
- [6] Rosato DV, Rosato DV, Rosato MG. Injection molding handbook. Boston: Kluwer; 2000.
- [7] Fu S, Lauke B, Mai YW, ScienceDirect (e-book collection). Science and engineering of short fibre reinforced polymers composites. Cambridge, UK; Boca Raton, FL: Woodhead Publishing; 2009.
- [8] Zheng R, Tanner RI, Fan X, SpringerLink (e-book collection). Injection molding: integration of theory and modeling methods. Berlin; Heidelberg; New York: Springer; 2011.
- [9] e-Xstream engineering: Digimat documentation 5.1.2. Mont-Saint-Guibert, 2014.
- [10] Wu L, Noels L, Adam L, Doghri I. A multiscale mean-field homogenization method for fiber-reinforced composites with gradient-enhanced damage models. *Comput Methods Appl Mech Eng.* 2012;233-236:164.
- [11] Dowling NE, Siva Prasad K, Narayanasamy R. Mechanical behavior of materials: engineering methods for deformation, fracture, and fatigue. Harlow: Pearson Education; 2013.
- [12] Jain A, Verpoest I, Hack M, Lomov S, et al. Fatigue Life Simulation on Fiber Reinforced Composites – Overview and Methods of Analysis for the Automotive Industry. *SAE International J Mater. Manf.* 5(1):2012, doi: 10.4271/2012-01-0730.
- [13] Bidaine, Benoît; Dr. Ir. Software Solution Analyst at e-Xstream engineering. Personal communication, April, 2015.
- [14] HBM nCode: DesignLife Theory Guide. Rotherham, HBM United Kingdom Limited; 2013.

Appendix A

For clarification, some additional figures and information about the fibre orientation and the physical test are presented here.

Fibre Orientations

The fibre orientations of the top and bottom layers are shown in Figure A.1. It can be observed that the top and bottom skin has similar fibre orientations.

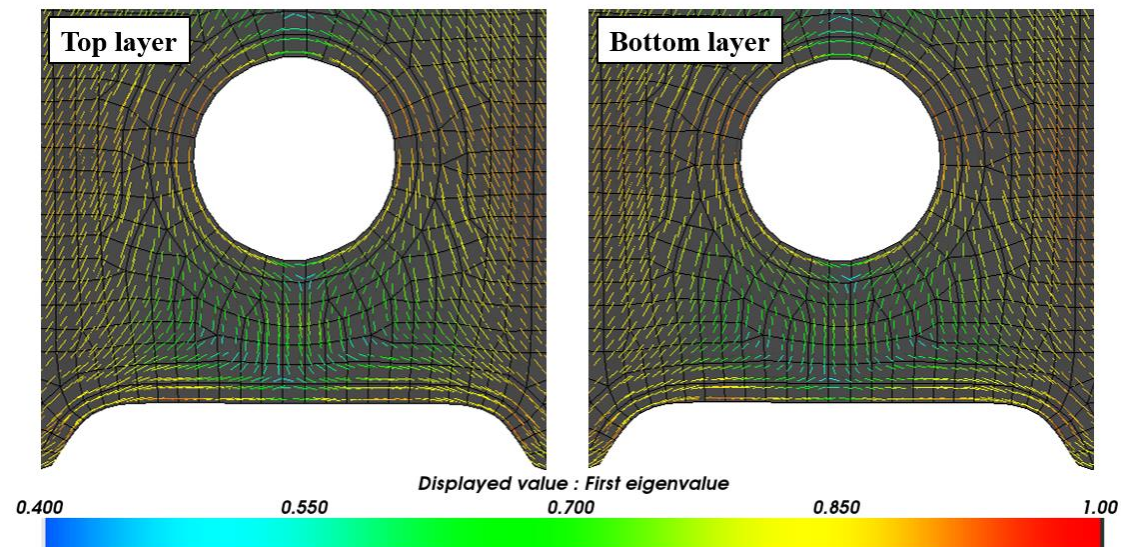


Figure A.1 Fibre orientations of top and bottom layer.

Fatigue test

Tolerance parameters of the test equipment are given in Table A.1, along with the diameter of the beams at the boundary.

Table A.1 Fatigue test tolerance and geometry parameters.

Tolerance force sensor	± 4 N
Tolerance position sensor	± 0.5 mm
Circular beam diameter at boundary	12 mm

In the following figures, details of the tests are shown. Figure A.2 shows the cylinder, to which the load is applied, and the areas of contact between the cylinder and the FEC. In Figure A.3, the contact region at the boundaries are highlighted. The contact areas in both of the figures are not changed during the load cycle.

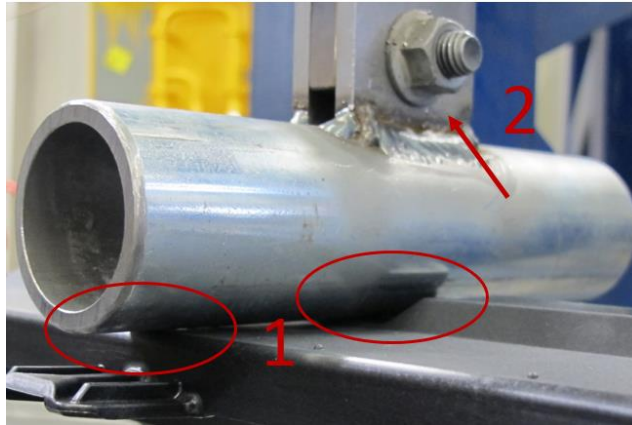


Figure A.2 The cylinder to which the load is applied. The areas marked with 1 show the regions of contact between the cylinder and the FEC. The cylinder is clamped at 2.

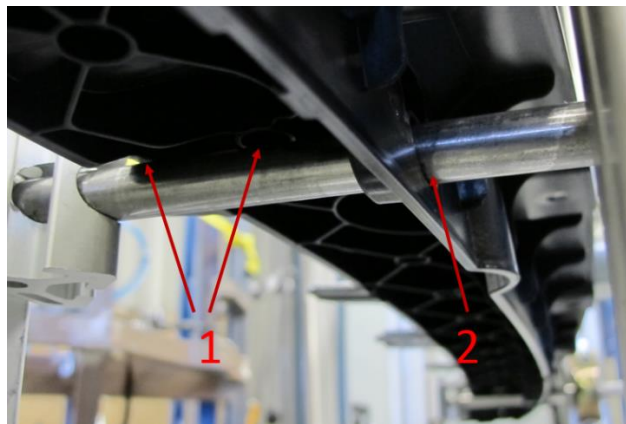


Figure A.3 Areas of contact at boundary. The FEC is pinned to the small cylinder with contact at area 2. No contact occurs in area 1.

The maximum displacement of the cylindrical beams at the boundaries was measured to approximately 0.1 mm when applying a load of 210 N, see set-up in Figure A.4.



Figure A.4 Measurement of maximum displacement of beams at boundary.

The results from the physical fatigue test are presented in Table A.5 as applied load level versus number of cycles to failure.

Table A.5 Results from physical tests, applied load vs. number of cycles to failure.

Load [N]	Cycles 10³
400	0.6
370	1.0
340	2.6
310	4.5
310	5.0
310	4.0
280	29.6
260	53.3
260	54.7
260	26.5
240	77.2
240	61.5
240	55.2
220	200.7
220	178.7
220	148.9
210	727.6
210	460.3