FATIGUE ANALYSIS OF FIBRE-REINFORCED POLYMERS

S. Vervoort¹

¹Hottinger Baldwin Messtechnik GmbH
Carl-Zeiss-Ring 11-13, 85737 Ismaning, Germany
e-mail: stephan.vervoort@hbmncode.com

Keywords: Fatigue, durability, short fibre composites, polymer, nCode DesignLife, CAE

Abstract. The utilization of fibre composite materials has become more important over the last years, because they can be specifically designed for the requirements of the application. This could be a combination of different material properties of the individual materials or entire new material properties. In particular, a fundamental advantage of short fibre-reinforced polymers is the combination of lower weight with adequate strength.

In modern product development processes more and more traditional metal materials are substituted with short fibre-reinforced polymers. The material-specific design of the final product has to cover durability or reliability aspects in addition to structural strength aspects. A fatigue life prediction of fibre-reinforced components identifies critical areas for the development at a very early stage.

The lecture shows a virtual durability product development process (VDPD) to predict fatigue life of short fibre-reinforced materials. The multi-disciplinary process consists of a manufacturing simulation, followed by a finite element structural analysis and a fatigue calculation.

The lecture focusses on the fatigue analysis of short fibre composites which applies finite element results, cyclic loads and fatigue material properties as inputs. The calculation is carried out using nCode DesignLife, one of the leading commercial durability software tools worldwide.[1,2]

The lecture ends with an example of a correlation between virtual and physical fatigue results.
1 INTRODUCTION

Over recent years, numerous computer-based test and analysis systems have been developed to enhance durability evaluation. The current methods used by the automotive industry for evaluating durability product development include proving ground testing, testing in the laboratory and analytical prediction methods.

By adopting "virtual testing", an approach that integrates physical testing with analytical procedures through integrated durability product development, manufacturers can develop more efficient and cost-effective ways of ensuring product durability [3]. The pay-off is substantial. By integrating the best features and information gleaned through each approach, manufacturers can overcome the limitations of a single approach and develop a comprehensive understanding of durability performance and product behaviour prior to finalizing designs and committing to volume production tooling.

The traditional physical "build - test - fix" approach to optimizing product life is time consuming and extremely costly. Historically, this has led to designs that are far from being optimized. They are either over or under designed. Test and evaluation departments, not design departments, have traditionally carried out durability evaluation late in the design/development process, when changes are expensive to implement.

A durability product development requires a multi-disciplinary team approach where no single activity is more or less important than another. For the automotive industry, critical inter-relationships must be established between original equipment manufacturers and their key suppliers, so tasks could be shared. It is essential that consistent methods and tools be used, so that information can be moved within and between these organizations with minimal confusion and difficulty.

The following document concentrates on the virtual part of the durability product development for lifetime predictions of automotive components. It is built from short fibre-reinforced polymers, but could easily transfer to other materials. The process takes place in the concept design phase to minimize costs and should manage as few physical testing as possible.

2 VIRTUAL DURABILITY PRODUCT DEVELOPMENT PROCESS

The Virtual Durability Product Development Process (VDPD) is shown in figure 1 and consists of multi-disciplinary engineering areas, hot press flow analysis, structural analysis, multi-body dynamics and fatigue analysis.

The aim is to predict the life of a component of short fibre-reinforced polymers, especially short fibre-reinforced thermoplastic. The process devotes resources of an existing CAE environment, which sets limits of the simulations and confines the quality of the fatigue results.

The simulation of the production of the component initiates the virtual durability product development process. It is followed by a structural analysis. The fatigue analysis constitutes the main part of the process. At the end of the process the lifetime is predicted and critical locations in the design are identified. Based on the results the design could be optimized before various prototypes have to be built and physically tested. The use of the virtual durability product development process reduces the number of prototypes.

The single steps of the process are characterized in details below on a basis of an automotive component of a passenger car, the bottom of the spare wheel compartment of a passenger car. In the past this kind of automotive component was built of sheet metal.
The competition on the automotive market requires a more cost efficient solution. In this case the sheet metals were substituted with short fibre-reinforced polymers. The exchange of sheet metal for short fibre-reinforced materials results in different production processes and styling of the design.

2.1 Geometry

![Figure 1: Virtual durability product development process of short fibre-reinforced polymers](image1)

![Figure 2: Design of the automotive component](image2)
The model is created for a number of different purposes. The modelling requirements depend on its intended use and to achieve acceptable levels of accuracy. Fatigue analysis results are very sensitive to the accuracy of the calculated stresses in local regions of a component.

The finite element model based on an existing CAD design for sheet metal was quickly modified to consider fibre reinforced plastic material. The model consists of 10140 shell elements. A key requirement is that the finite element model must be of good quality with sufficient mesh refinement (figure 2).

### 2.2 Flow Simulation

The flow simulation represents the manufacturing process of the component. It is a sheet molding compounds (SMC) process technique based on glass fibre-reinforced thermoplastics with good mechanical properties. The initial shape of the basic raw material is a pre-plate. They consist of random glass fibre orientated polypropylene. In the process one or more pre-plates, depending on the thickness of the final component, are pressed in tempered form.

The glass fibres become aligned through the pressing process along the geometric and material flow lines. The orientation of the fibres is simulated by the hot press flow simulation programs. The input parameters of the simulation are production, process and rheological material parameters. Figure 3 shows the location of the pre-plates with random fibre orientation in the pressing mold at the start of the production process. The pressing mold is heated.

![Figure 3: Pre plates location in the pressing mold (model design variation)](image)

The fibre orientation angle is determined for every element. The Poisson ratio and two Young’s moduli depending on the fibre orientation are written in to the output result file. One Young’s modulus is parallel the fibre and the other one is calculated perpendicular to the fibre. Both parameters together with the fibre orientation angle describe the orthotropic material behaviour. An interface of the flow simulation program writes the input deck for the FE solver including the orthotropic material parameters.
2.3 FEM Simulation

The FE solver MSC.Nastran is used for the structural analysis. A linear-elastic finite element simulation is carried out. The anisotropic material parameters of the flow simulation were included in MAT8 and PCOMP cards. The MAT8 card defines the shell element orthotropic material property and the PCOMP card defines layered composite element properties for every single element. The PCOMP card could divide the element into different layers, but only the surface layers were factored into the calculation as top and bottom surface of the component [5].

Every element has its own ID also a unique property ID due to the fact that an element has its own fibre orientation and orthotropic material properties. Additionally the orthotropic material properties can be saved by the use of Digimat [6] into a material orientation tensor of each element in preparation for the fatigue analysis afterwards. If no material orientation tensor could be created by the finite element solver the material orientation information could be saved into an ASCII file for following analyses. Digimat supports different commercial solver like e.g. Abaqus, ANSYS, RADIOSS.

For the loading conditions nine independent, static displacement sub cases are attached to the model. These unit loads are attached to three locations in three directions and balanced on the model. The x- and y-axis describe the plane of the model while the z-direction describes perpendicular axis of the model. Built in the passenger car the z-axis is vertical to the ground surface.

![Figure 4: VON MISES stress distribution (load case: z-axis, top)](image)

Figure 4 shows the VON MISES stress result of one sub case (z-Axis loading) as an example. A complete stress tensor was calculated for every shell element. To achieve the best fatigue results the finite element analysis must provide stresses that are as accurate as possible, particularly with critical locations at free surfaces where fatigue cracks are expected to initiate.

Anything that compromises the accuracy of the stresses in critical areas can have a significant effect on the calculated life. According to a rule of thumb: 10 % stress tolerance induces 100 % difference in life.
2.4 Fatigue Simulation

The fatigue analysis represents the main part of the virtual durability product development process. It presumes a finite element geometry and stress result. In principle a modern CAE fatigue analysis is based on three primary input parameters (figure 5):

- FE geometry and results
- Cyclic loading time histories
- Material properties

The accuracy of all three inputs influences life or damage results and can be optimized during post processing to extend the life of a part or system, independently of each other up to a point [7].

The following chapters below describe only the fatigue details of the analysis parameters, loading and material properties; the geometry and FE results were already described before.

![Figure 5: CAE based fatigue analysis](image)

2.4.1 Analysis parameters

The FE-based total life, or S-N, method of fatigue analysis is executed for predicting life and damage. Total life methods are typically more applicable to high cycle fatigue situations. The fatigue program nCode DesignLife superposes the nine displacement time histories for every element and transforms the FE stress tensor into principle stresses by an internally critical plane method [2]. Next rainflow cycle counting [8] and GOODMAN mean stress correction [9] are applied. Finally the damage is calculated by use of the linear damage accumulation theory of PALMGREN and MINER [10, 11].
The most popular result of stress based CAE life prediction is life (unit: repeats of applied load spectrum) or damage (no unit) result for every element diagrammed in a fringe plot or a table form. The critical locations are highlighted and can be eliminated by local optimization of shape.

2.4.2 Loading

The loading of the fatigue analysis is represented by displacement time histories of the component. An example of three time histories, out of nine applied, attached to one node is shown in figure 6. The cyclic displacements come from a multi-body dynamics simulation (MSC.Adams). A simulation of a real existing test rig is created in the multi-body dynamics program. Figure 7 shows the assembly of the test rig.

![Figure 6: Displacements time histories at one attachment point (x-, y- and z-axis)](image)

The input loading of the multi-body system analysis is extracted from a real durability test of a fibre-reinforced prototype component.

![Figure 7: Simulation of a physical durability test](image)
2.4.3 Materials

The third input parameter of the fatigue analysis is the material property. In this case at least one WÖHLER curve (SN curve or BASQUIN curve) is required to define it for the total life method. To limit the time and costs of material characterization, curves are measured at a zero mean and adjust by a mean stress correction equation.

As the reinforced polymer is fibre orientation dependent, one SN curve is not sufficient to describe the material properties and feed the fatigue analysis. For fibre reinforced polymers the fatigue strength of different fibre orientations was investigated.

Special plates are manufactured with fixed fibre orientation at which the base material and the production process was the same as for the automotive component. The flow of the material is controlled during the pressing process in defined directions to produce material with a dominant fibre orientation. Specimens are cut out of these plates in different directions to get unidirectional fibre orientation in the specimens. Stress based fatigue tests are carried out on specimens with 0°, 20°, 45° and 90° degrees fibre angles to loading direction at room temperature. Figure 8 shows the BASQUIN curves for each fibre angle of the short fibre-reinforced polymer.

The fatigue software program nCode DesignLife has an intelligent internal interpolation method for the fibre orientation dependent SN curves. A fibre share parameter takes the angle between fibre orientation and loading direction and material orientation tensor into account for the interpolation.
3 RESULTS

The result of the fatigue analysis is a life time contour plot, which is shown in figure 9. A life value is calculated for every element and the red color indicates the minimum life and the critical areas in the fringe plot. Blue coloured areas indicate values below the endurance limit.

The plot shows in details possible failures in different areas, mostly on the peripheral regions or surface pleats of the component. In general the z-axis motion, causing the component to flex, dominates the fatigue damage.

4 CORRELATION

To interpret the quality of the fatigue calculation the result has to be correlated. The results of the damage calculation based on orthotropic material properties (shown in figure 9) can compared to the FE structural analysis results (figure 4), damage calculation using isotropic material properties (figure 10) and real fatigue test results (figure 11).

The comparison between the Von Mises stress result of the dominant static load case (z-direction) of the FE structural analysis (figure 4) and life time calculation (figure 9) based on orthotropic material properties as described in this paper is valuably, because often CAE departments are carrying out only structural analysis and make their decision based on their results only. Compared to the fatigue results, which gives detailed information about various local failures the FE analysis is missing this detailed information. A decision based on the FE results may lead to wrong decisions.

Figure 10 shows a results of a fatigue analysis base on isotropic material properties. For the isotropic material calculation one SN curve is used. The fatigue tests are carried out with specimens of random fibre orientation. A fibre orientation is not considered.

A comparison between the results the of fatigue analysis using isotropic (figure 10) and orthotropic (figure 9) show differences in critical locations. The orthotropic fatigue analysis is
detecting more details and show clearer the critical locations. Decisions based on an analysis based on isotropic material parameter may fail.

An final comparison between simulation and reality is performed. A prototype of the bottom of the spare wheel compartment was manufactured, placed in a passenger car and the car was tested on the proving ground. According to a defined test schedule the component was afterwards inspected for cracks and failures.
Figure 11 shows the comparison between the orthotropic fatigue simulation and cracks found in components tested on the proving ground. The comparison shows a good correlation between simulation and reality.

5 CONCLUSION

The virtual durability product development process helps to indicate weakness or critical locations of the design at the early state in the concept phase. An optimized fatigue simulation

- Gives additional model information, which is previously not available.
- Gives more realistic results using fibre orientation material parameter compared to simple random material properties.
- Illustrates the advantages of the design.

The process combines standard software tools and is easy to integrate in existing CAE environments. It will save costly re-designs and re-tests. Beside the information of structural analysis and multi-body system analysis fatigue information based on fibre orientation dependent material properties gives new insight into material performance.

REFERENCES