SIMULATION OF DELAMINATION AND COLLAPSE OF A FIRE EXPOSED FRP COMPOSITE BULKHEAD

Kim Olsson1, Johan Sandström2, Joakim Albrektsson1 and Johan Anderson1

1Department of Fire Research, SP Technical Research Institute of Sweden
Box 857, SE-501 15 Borås, Sweden
Email: kim.olsson@sp.se
     joakim.albrektsson@sp.se
     johan.anderson.br@sp.se

2Department of Structural and Solid mechanics, SP Technical Research Institute of Sweden
Box 857, SE-501 15 Borås, Sweden
Email: johan.sandstrom@sp.se

Keywords: Debonding, FEM simulation, Fire resistance, FRP composite, Lightweight design

ABSTRACT

In marine applications, Fibre Reinforced Polymers (FRP) sandwich composites are used for designing bulkheads with low weight and high strength. The FRP composites can be constructed with an inner core made of a PVC (polyvinyl chloride) foam and face sheets with glassfiber-reinforced polyester. A point open for investigation is the performance of FRP composite bulkheads exposed to fire. During fire resistance tests of FRP-composites bulkheads, both delamination and collapse towards the unexposed side usually occurs. The load carrying capacity of such structures start to degrade at temperatures below 100 °C and can therefore be extra susceptible to fire, compared to maximum structural core temperature rise of 200 °C for aluminum bulkhead according to the HSC-code [1].

This study aims at investigating the delamination and collapse of fire exposed FRP composite bulkheads, which are insulated on the exposed side. The geometry, material data, boundary conditions, temperature and deflection measurements from a large scale fire resistance test conducted at SP Fire Research are used to validate the simulations.

A 2D FE-model has been developed which captures the delamination process and the collapse towards the unexposed side. The 2D FE-model is a simplified model where the adhesive layer in the interfaces between the laminates and the core is modeled using cohesive surface-to-surface interactions. This is followed by a 3D FE-model with similar cohesive modelling that also captures the temperature-induced deformation and point of collapse. The 3D FE-model uses a mix of shell elements and continuum-shell elements to limit the number of elements, allowing simulation of more complex geometries with reasonable computational effort. The models are able to capture the two dynamical characteristics, delamination and collapse towards the unexposed side. The results in this paper support the hypothesis that the fire resistance is mainly dependent on insulation.

1 INTRODUCTION

Various modelling techniques for sandwich composites exist. One common approach is to, through homogenization, establish an equivalent single layer that can be used in e.g. beam and shell analyses. A thorough review of computational models for sandwich plates and shells can be found in [2] and additional other work is discussed in [3]. A homogenized model cannot model localized behavior in the interaction between different constituents of the composites which in this study is the temperature induced loss of adhesion between face-core and degradation of the material them self. A more advanced modelling is presented by Frostig and Thomsen [4] with modelling using nonlinear high-order sandwich panel theory to describe the delamination and material degradation. Kinematically their models allows, in addition to Euler-Bernoulli bending of the faces, normal (out-of-plane) deformation and shearing of the core.
In Figure 1, a FRP-composite structure bulkhead from a test conducted at SP Fire Research is seen. The bulkhead is mounted to a vertical furnace and is exposed to fire on one side and a vertical mechanical load is applied on the top.

Figure 1: A FRP-composite bulkhead from a test conducted at SP Fire Research is seen. It is mounted to a vertical furnace and is exposed to fire on one side and a vertical mechanical load is applied on the top. As the test terminates, it has collapsed towards the unexposed side. Left: Unexposed side. Right: Exposed side.

The temperature in the furnace is controlled to correspond to the standard time-temperature curve in ISO 834-1 [5]. The exposed specimen face is protected by an insulating layer making the temperature increase in it smaller and delayed. Thermocouples are positioned between the insulation and laminate on the fire exposed side, see Figure 2.

Figure 2: A schematic of the thermocouple placement between the insulation and the laminate on the exposed side of the bulkhead.
The average temperature of these thermocouples is used as temperature boundary condition in the simulations. A typical average temperature on the exposed side of the bulkhead together with the ISO 834-1 standard time-temperature curve is given in Figure 3.

The hypothesis is that the bulkhead will collapse when the bond between laminate and core reaches a critical temperature and detaches, i.e. this is basically an insulation problem. The geometry and material parameters from the bulkhead shown in Figure 1 are used in the FE-models in this report.

![Figure 3: Furnace temperature and typical average temperature between the insulation and laminate on the exposed side of the bulkhead.](image)

This paper concerns 2D and 3D modeling of delamination of composite structures under thermal and mechanical load with the Finite Element (FE) method. The two models consist of a core with a laminate on each side. The composite is subject to thermal loading on one side, referred to as the hot side and a mechanical vertical load applied on the top using a rigid body. The materials and geometry are defined such that they resemble a fibre reinforced polymer (FRP) sandwich composite bulkhead. Fire exposed bulkheads displays two dynamical characteristics during fire testing. First of all delamination occurs, i.e. the laminate separates from the core on the hot side. Secondly, the structure deflects towards the hot side but after delamination is complete the core undergoes relaxation and collapses towards the cold side. At this stage, the structure has lost its load bearing ability.
2 MODELLING

The modeling consists of a thermal model and a mechanical model implemented in the commercial FE-package Abaqus [6]. The thermal model is used to compute the thermal load and how the temperature propagates through the structure in time. The thermal load is applied as a boundary condition using the typical bulkhead temperature in Figure 3. The output from the thermal model is used as input in the mechanical model, where the mechanical load is applied to the top of the structure. Thus, this is not a fully coupled problem. Due to the thermal and mechanical load the composite will deform.

The density of the laminate is set to 1870 kg/m$^3$ and the core to 80 kg/m$^3$. The temperature dependent thermal conductivity and specific heat capacity properties are given in Table 1 for various temperatures [7]. Values for intermediate temperatures are interpolated by the FE solver.

<table>
<thead>
<tr>
<th>Temperature [°C]</th>
<th>Thermal conductivity [W/m°C]</th>
<th>Specific heat capacity [J/kg°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Laminate</td>
<td>Core</td>
</tr>
<tr>
<td>0</td>
<td>0.04</td>
<td>0.028</td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>0.05</td>
<td>0.03</td>
</tr>
<tr>
<td>150</td>
<td>0.051</td>
<td>0.045</td>
</tr>
<tr>
<td>1200</td>
<td>0.052</td>
<td>0.05</td>
</tr>
</tbody>
</table>

Table 1: Thermal properties of the laminate and core material.

2.1 2D FE model

In the 2D model, cohesive surface-to-surface interaction is used. Hence there is no adhesive layer in the 2D model but instead a surface pair with cohesive interaction where it is possible to describe how the surfaces delaminates, using temperature dependence.

2.1.1 Geometry and mesh

A schematic of the 2D FRP composite is seen in Figure 4 together with the boundary conditions. The average mesh element size of the core is 5 mm everywhere except at the boundary on the hot side, where it is of size 1 mm. The average mesh element size of the laminates is 1 mm. The mesh properties for the thermal and mechanical models are given in Table 2.

<table>
<thead>
<tr>
<th>Abaqus element type</th>
<th>Properties</th>
<th>Number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mechanical model</td>
<td>Thermal model</td>
<td>Mechanical model</td>
</tr>
<tr>
<td>CPS4R</td>
<td>DC2D4</td>
<td>4-node bilinear</td>
</tr>
</tbody>
</table>

Table 2: Mesh properties for the thermal and mechanical model.
2.1.2 Mechanical simulation

The load on the rigid body pressing down on the composite is represented by a load of 12 kN/m distributed evenly over the surface area of the rigid body with dimensions (width x depth) 60 x 1 mm. A cohesive surface to surface interaction is applied on the boundary between the laminates and the core. In this paper there is no available data regarding at which stresses the surface bounds delaminates. However, an estimation of the temperature for which the cohesive bound loses its cohesive abilities is known, around 120 °C. In order to impose this behavior, a temperature dependence of the maximum nominal (contact) stress the surface bound can withstand is introduced. This is called damage initiation. The cohesive behavior properties are listed in Table 3.

<table>
<thead>
<tr>
<th>Temperature [°C]</th>
<th>$S_{11}$</th>
<th>$S_{22}$</th>
<th>$S_{12}$</th>
<th>Damage evolution</th>
<th>Damage stabilization</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>40</td>
<td>60</td>
<td>60</td>
<td>$5 \cdot 10^{-8}$</td>
<td>$5 \cdot 10^{-8}$</td>
</tr>
<tr>
<td>27</td>
<td>20</td>
<td>40</td>
<td>40</td>
<td></td>
<td></td>
</tr>
<tr>
<td>77</td>
<td>10</td>
<td>20</td>
<td>20</td>
<td></td>
<td></td>
</tr>
<tr>
<td>127</td>
<td>5</td>
<td>10</td>
<td>10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>177</td>
<td>3</td>
<td>5</td>
<td>5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>227</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Cohesive behavior properties.
2.2 3D FE Model

An FE-model for simulation the collapse of real sandwich composites in 3D is presented. The model simulates the tested case with a with vertical loaded wall-type FRP sandwich composite depicted in Figure 3.

2.2.1 Geometry and mesh

A half-model is used to exploit the symmetry of the problem. The dimensions and mesh are presented in Figure 5. Geometry in the thickness direction is given by Figure 4. The faces are modelled with shell elements and the core with solid elements. Solid thin interface-elements are used to connect the shell elements to the core elements.

![Geometry and mesh of the simulated sandwich structure.](image)

2.2.2 Thermal simulation

A heat conduction simulation is performed to establish a realistic temperature field with varying temperature through the thickness of the wall. The structure is thermally loaded by applying the typical test specimen temperature in Figure 3 to the outer side of the shell elements on the hot side of the sandwich. On the outer side of the shell elements on the cold side, a natural convection of 5W/m²K is applied. No boundary condition are set on the sides (including the symmetry cut) of the sandwich (i.e. effectively modelling the side boundaries as adiabatic).

2.2.3 Mechanical simulation

A stress-displacement simulation of the sandwich structure is performed where it is affected by temperature and mechanical loading by a vertical force. Non-linear geometry (i.e. large displacements analysis) is used. Time integration is of implicit type and both dynamic and static simulations are used, although the static case is primary reported here. For the static simulation, stabilization is utilized to aid convergence during the buckling phase. The stabilization is configured so that the solver automatically adapts the dissipated viscous energy to 0.001 of the strain energy [6]. The validity of this is briefly discussed in the results section.

A symmetry boundary condition is imposed on the symmetry cut of wall. In the test setup the wall is loaded between two stiff plates. These plates are modelled as two rigid planes where the upper is free to translate in the vertical direction and rotate in the horizontal in-plane direction while otherwise...
fixed. The lower plate is fixed in all directions (translation and rotation). This is modeled with contact interactions between the wall and rigid planes. A low friction coefficient of 0.1 and also setting maximum allowed elastic slip to a large value of 2 mm is used to restrict large net movement relative to the rigid plane while still allowing local movements. This is necessary to allow thermal expansion of the sandwich material which otherwise would cause artificial bending of the sandwich due restricted thermal expansion. The vertical in-plane loading is ramped up to 31.05 kN (corresponding to the line load 20.7 kN/m on the upper edge of the wall) and then held constant during the heating phase. The rise in temperature during the heating phase is simulated by reading the temperature results from the thermal simulation for the integration point positions in the stress-displacement simulation.

The laminate is modelled with quadrilateral first order shell elements with finite strain formulation (Abaqus designation S4) and linear elastic isotropic material with modulus of elasticity 50 GPa and Poisson’s ratio 0.4 for temperatures up to 80 °C. For higher temperatures, the modulus of elasticity is reduced linearly to 8 GP until the temperature 90 °C and then held constant. The coefficient of thermal expansion is $1.5 \times 10^{-5} \degree C^{-1}$.

The core is modelled with elements of the type continuum shell elements (Abaqus designation CS8R). These elements have the same geometry as regular hexahedral solid elements but internally use kinematics of shells in their “shell-plane directions”. In their out-of-plane direction they behave like regular solid elements (the elements have thickness strain as a “degree of freedom”). This allows the modelling of stiffness in the thickness direction of the core and the bending mode displacements with elements that are very flat which would cause problems for regular solid elements. Like conventional shells, continuum shell elements have a number of integration points along their thickness. Since the temperature gradient through the thickness can be large, 11 integration points along the thickness is used to capture the thermal degradation in the core. This is rather many integration points and in further work it should be investigated how many actually are needed.

The core material is modelled as a linear elastic isotropic material with Poisson’s ratio 0.0 and a temperature dependent modulus of elasticity as given in Figure 6 derived from [4]. At temperatures above 80 °C, the modulus of elasticity is set to 0.1 MPa.

![Figure 6: Temperature dependent modulus of elasticity of the PVC foam core material.](image)

Cohesive elements are used to connect the faces to the core. The failure of the bond (delamination) is modelled with damage in the cohesive behavior. The damage modelling is set up so that the normal and shear strength is 20 MPa. Damage is then initiated when either normal or shear strength is exceeded. Damage is then modelled so the fracture energy in pure normal and shear failure is 740 J/m² and 2100 J/m² respectively. Combined mode fracture is achieved by normalizing damage energy components to the fracture energies. Failure is then reached when the sum of the squared normalized damage energies reaches 1.0 [6]. To account for thermal degradation of the adhesion, the strength 20 MPa is scaled down proportional to the lowering of the temperature dependent modulus of elasticity of core material depicted in Figure 6.
3 RESULTS

3.1 2D simulations

In Figure 7, the overall structural behavior of the bulkhead is seen. The dynamics of the model can be divided into three phases. First delamination starts, see second picture from the left. This phase is characterized by the structure deflecting towards the hot side. In the second phase, the core starts to undergo relaxation, see third figure from the left. In the third phase, delamination is complete and the core collapses towards the cold side, see fourth picture from the left.

In Figure 8, the delamination and deformation at the upper part (300 mm from the top down) of the composite is seen. The time steps, left to right, are 44.8, 48.2 and 60 minutes respectively. The delamination starts at the top, where the rigid body is tied to the composite, and propagates downwards. At time 60 minutes it is seen how the structure has collapsed. The colors indicate nodal temperature where red is hot and green is cold.

Figure 7: The overall behavior of the structure at times: 0, 48.7, 49.5 and 60 minutes ranging from left to right respectively.

Figure 8: Pictures of the upper part of the structure (300 mm from the top down) after 44.8, 48.2 and 60 minutes respectively. The delamination starts at the top and propagates downwards.
Two different kinds of time integration are investigated, static and dynamic. The static step type works until delamination have propagated for roughly 48 minutes and the core undergoes relaxation i.e. complete delamination, and the core changes its direction of deflection from the hot to the cold side. This happens so rapidly, see Figure 9, that the time increment reduces to a magnitude of $1/10^6$ s and computation becomes extremely demanding. However, using the dynamic step type the solver will compute beyond 48 minutes and solve the relaxation of the core. There are drawbacks using a dynamic step type; first of all the solution is not as smooth as the static step type; secondly when compared to experiments the results from the static step type is more reasonable; third and last there is some numeric fluctuation that are unphysical. Both step types predict the same time for the collapse.

![Figure 9: Horizontal displacement of the core at mid height using dynamic and static step type in the simulations. Positive values indicate displacement towards the hot side. The y-axis is removed as this data is not officially released yet.](image-url)

A comparison between the displacement of the core at mid height, from the fire test and a simulation using the static step type is seen in Figure 10. Positive values indicate displacement towards the cold side. Test values originate from un-disclosed reports and cannot be given in more detail.

![Figure 10: Comparison between the displacement of the core at mid height, from the fire test and a simulation using the static step type. Positive values indicate displacement towards the cold side. The y-axis is removed as this data is not officially released yet.](image-url)
3.2 3D Simulations

Resulting simulated temperatures distributions along the thickness of the sandwich during different points in time are presented in Figure 11. In the in-plane directions of the sandwich, the temperatures are essentially uniform. These results will then be used in the stress-displacement simulations below.

![Figure 11: Temperatures from the thermal simulation along the depth in the thickness direction for different times.](image1)

The out-of-plane deflection of the wall is taken as primary result as this measure is closely related to structural integrity with buckling type of collapse. In Figure 12 are the deflections of two points given. The points are the center (mid) point and a point centered vertically on the horizontal edge. Both simulated and typical measured deflections are given. For simulations without static stabilization, the results are almost identical (maximal difference in deflections less than 3 %) up to the time 37 min when the non-stabilized simulation terminates due to loss of convergence.

![Figure 12: Deflection of the wall for the middle point and edge point where positive values implies deflection towards the cold side. The y-axis is removed as this data is not officially released yet.](image2)
In Figure 13 is the amount of delamination at the end of the simulation given. The delamination starts to appear at 36 min which can be noticed as the time when the deflection starts to reverse.

4 CONCLUSIONS

It is shown that FE modelling implemented in Abaqus is able to model delamination of a fire exposed FRP composite subject to mechanical load. The whole structure deflects towards the hot side and starts to delaminate. As the delamination propagates, the core starts to detach from the laminate and collapses towards the cold side. This behavior has been observed during multiple fire resistance tests of composite structures conducted at SP Fire Research throughout the years.

Although explicit time integration (e.g. Abaqus/Explicit) is more common for dynamic simulations, the cohesive surface-to-surface interaction is not applicable to 2D surfaces in Abaqus/Explicit. Using the standard solver (implicit time integration), a dynamic step type is preferred in order to capture the full behavior of the structure i.e. beyond the point where the structure deflects towards the cold side. The static step type works even better and give more reasonable results up to the point where the core undergoes relaxation but is unable to capture the structural behavior after that point. This shows that although it is tempting to treat the behavior of the model as static, due to the slow overall time scale, there are competing time scales to consider due to the partially rapid movements of the core when delamination occurs.

There are several questions regarding the cohesive contact stresses and in particular how the delamination is modelled. So far the delamination can be simulated qualitatively but in order to quantify this behavior, a more in depth look of the model is required. A study of the temperature dependence of the material parameters, in particular elasticity, is required.

The proposed 3D FE-modelling shows how collapse of a thermally and mechanically loaded sandwich composite can be simulated. The current model accurately captures the initial response and qualitatively the first stage of buckling. With the static modelling the final collapse with deflection towards the cold side is not seen in the simulations. The 3D-modell has also been implemented with dynamic implicit time-integration that gave correct buckling towards the cold side similarly as in the 2D modelling. Although that appears promising, static modelling is here presented as it is not expected that dynamic modelling is a viable way of simulating structural integrity of real fire affected structures.
in designing.

Further work regarding the 3D-model is to better determine relevant material properties, which could allow accurate prediction of collapse of components and structures. This could be achieved by performing thermo-mechanical tests on smaller samples together with detailed simulations (e.g. the presented 2D-modelling). The geometry and mesh of the 3D model is simple and it should not be a complicated task to build more complex structures. The observed simulation times for the 3D cases are limited (around 10 min on a laptop), further mesh optimization should reduce this.

The results in this paper support the hypothesis that the temperature induced degradation of the adhesive and the subsequent delamination dictates the point of collapse. The fire resistance is mainly dependent on insulation.

REFERENCES

[1] Rules for High Speed, Light Craft and Naval Surface Craft, DNV, Pt.0 Ch.6, January 2011