LOW-VELOCITY IMPACT OF SANDWICH STRUCTURES WITH EMBEDDED CERAMIC PLATES

T. Mayr1, 4, M. Battley2, T. Müller1, A. Ramesh3, J. Weber3 and H. Rapp4

1 BMW AG, Knorrstraße 147, 80788 Munich, DE
2 Department of Engineering Science, University of Auckland, 1010 Auckland, NZ
3 Department of Mechanical Engineering, University of Auckland, 1010 Auckland, NZ
4 Institute of Lightweight Structures, Universität der Bundeswehr München, 85579 Neubiberg, DE

Keywords: Multifunctional, Sandwich, Low-velocity impact, Modelling, Experiments

ABSTRACT

In this study the low-velocity impact behavior of a sandwich panel structure with glass-fibre reinforced composite facesheets, a foam core and an embedded ferrite plate is investigated. The impact response of the sandwich structure is predicted by using a two-degree-of-freedom spring-mass analytical model and by explicit dynamic finite element modelling. The physics of the impact are characterized and the models validated by the results of drop-tower experiments with impact energies from 20 J up to 480 J. During the tests a displacement transducer, high-speed imaging, a force sensor in the impactor and strain gauges on the top and bottom of the ferrite have been used for measurement. The influence of the low-velocity impact event on the strains of the ferrite and the deflection of the sandwich are investigated. The main effects are described and conclusions on the behavior of a sandwich structure with embedded ferrite plate subjected to low-velocity impact are drawn.

1 INTRODUCTION

Current technical developments like the electrification in the automotive sector lead to the need for engineering components that can fulfill a lot of different and complex requirements. These requirements are mostly multi-disciplinary and therefore new material systems and structures have to be developed. Functions like mechanical protection, temperature management or electromagnetic shielding should be taken over by one integrative material system. The use of multifunctional composite materials enable the construction of parts with minimal packaging as well as reduced number of components and joining operations [1].

One field of application with many different multidisciplinary requirements is the inductive charging of electrical road vehicles. To transfer power wirelessly an alternating electromagnetic field is applied between two copper coils. One of the coils generates the field, whilst the other is assembled in the underbody of the car and transforms the electromagnetic field back into electrical power. Additional ferrite plates are used to guide and focus the EM-field and increase the efficiency of the power transfer [2, 3]. A key requirement for parts integrated in the underbody of a car is to withstand low velocity impacts that occur when running over an obstruction [5]. At this events energies in the range of 140 J are reached. Due to its brittle behavior the ferrite has to be specially protected from mechanical damage [4]. By embedding the electrical components into composite sandwich structures good mechanical protection as well as low weight and cost is expected.

To understand how this can be achieved, this study investigates the low-velocity impact response of sandwich structures with embedded ferrite plates. An analytical spring-mass model and a FEA model are developed of the impact scenario. Low-velocity impact tests are conducted in a drop-weight impact tower and the deflection of the panel as well as the strains of the ferrite are measured. Finally conclusions on the low-velocity impact behavior of a sandwich structure with embedded ferrite plate are drawn.

2 PANEL CONSTRUCCION

The dimensions of the sandwich panel and its cross-section can be seen in Figure 1. The facesheets consist of 8 layers of 500 gsm glass fiber unidirectional layers (Colan MU 4500 G) in a symmetric (0/90)n sequence. As matrix the infusion grade epoxy resin Gurit PrimeTM 20LV is used. The
T. Mayr, M. Battley, T. Müller, A. Ramesh, J. Weber and H. Rapp

thermoplastic foam core is a PET material from Gurit (G-PET 90). The embedded ceramic layer is a hard ferrite of grade Y30BH and is situated between the back facesheet and the foam core material. The ferrite layer is composed of panels with 100 mm x 100 mm. The thickness of the front facesheet is \( h_f = 3.1 \) mm and of the back facesheet \( h_b = 3.1 \) mm. The foam core and the ferrite plates have the thickness \( h_c = 7 \) mm and \( h_f = 5 \) mm respectively. All layers have in plane dimensions of 500 mm x 500 mm and are assumed to be perfectly connected for the analytical and numerical model. The uniaxial strain gauges are of type Tokyo Sokki Kenkyujo FLA-3-350-11-1L. They are positioned on top (TC0, TC25, TC75) and bottom of the ferrite (BC0, BC25, BC75) in the positions 0 mm, 25 mm and 75 mm away from the center (see Figure 1).

![Figure 1: Dimensions of the sandwich structure (left figure) and ferrite plates with position of the strain gauges (right figure).](image)

Table 1 summarizes the material properties used for the modelling. The GFRP material property data was based on experimental tests on the same material corrected using micromechanics for slight differences in fiber volume fraction. The foam data listed in Table 1 is from the manufacturer’s datasheet. The input data for the FEA model was based on uniaxial compressive tests of the actual foam used. The ferrite material properties were sourced from datasheets for the same grade material.

<table>
<thead>
<tr>
<th>Material</th>
<th>Type</th>
<th>Young’s Modulus</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>[MPa]</td>
</tr>
<tr>
<td>Facesheet</td>
<td>Bi-axial GFRP</td>
<td>( E_{face} = 21000 )</td>
</tr>
<tr>
<td>Foam Core</td>
<td>Gurit G-PET 90</td>
<td>( G_{face} = 3200 )</td>
</tr>
<tr>
<td>Ferrite</td>
<td>Y30BH</td>
<td>( v_{face} = 0.3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \rho_{face} = 1900 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E_c = 95 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( G_c = 19 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( v_c = 0.3 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \rho_c = 94 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E_f = 180000 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( G_f = 70000 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( v_f = 0.28 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \rho_f = 5000 )</td>
</tr>
</tbody>
</table>

Table 1: Material Properties.

The GFRP/epoxy facesheets were made individually by laying up the unidirectional cloth onto a glass plate, then were resin infused with the epoxy resin. The facesheets underwent an initial room temperature cure before cutting to size. Following bonding of the resistance strain gauges to each side of the ferrite the different layers were joined using a toughened epoxy resin. Then the assembly post-cured under vacuum at about 80 °C for 6 hrs.

3 ANALYTICAL MODEL

Composite sandwich structures under low-velocity impact have been investigated in several studies. Besides experimental tests also simulations and analytical studies have been conducted [6]. One analytical approach is the use of a spring-mass model. Results of Ferri et al. show that this quasi-static approach can be used for low-velocity impact [7]. The model is limited to the elastic response of the material without damage. The analytical spring-mass model is depicted in Figure 2. The effective mass
of the impactor $M_{\text{Imp}}$, of the top facesheet $m_t$ and of the whole sandwich $m_{\text{Sand}}$ are represented. The calculation of the local stiffness $k_{\text{loc}}$ and global stiffness $k_{\text{glo}}$ will be described in the following parts. For the development of the analytical part the deformation mechanism of the sandwich is split in two stages. At stage 1 the impactor indents the sandwich at the front face with the rest of the structure not moving. Stage 2 starts when the local indentation of the sandwich reaches its maximum and the rest of the sandwich structure starts to deflect. In Figure 2 the deformed cross section of the sandwich at the transition point between stage 1 and 2 can be seen. There the thickness of the foam reaches its minimum $h_{c, \text{comp}}$. The x-axis of the coordinate system in the figure is congruent to the neutral fiber of the deformed sandwich.

![Diagram](image)

Figure 2: Spring-mass model (left figure) and deformed cross-section of the sandwich at the transition point between stage 1 and 2 (right figure).

The local stiffness $k_{\text{loc}}$ represents the relation between contact force and front face deflection. Therefore the contact between impactor and top facesheet, the stiffness of the top facesheet and the strength of the foam core have to be taken into account. These local effects have been investigated in several studies [8-12]. The results can be used for the sandwich structure of this study, as the ferrite will have no influence on the local stiffness. As the approach of Zhou et al. includes the highest number of effects like the membrane and bending stiffness of the top facesheet as well as a perfectly plastic material model for the core, the results from this studies are used [12]. There the relationship between front face deflection $x_1$ and contact force $P$ is represented by the equation:

$$P = \frac{16 \pi}{3} \sqrt{\frac{D_{f} q x_1 (1+\frac{x_1^2 L}{26460 H D_f})}{D_f}}.$$  \hspace{1cm} (1)

Here $q$ is the core crushing strength of the foam core. As the core crushing strength is often not available, the compression strength can be used. This simplification underestimates the influence of the core to the local stiffness, but the effect on the behavior of the structure is expected to be negligible [13]. $H$ and $L$ represent the elements of the laminate in-plane stiffness submatrix and can be looked up at Zhou et al. [12]. The equivalent bending stiffness $D_f$ of the top facesheet is given by [14]:

$$D_f = \frac{1}{8} (3 D_{11} + 2 (D_{12} + 2 D_{66}) + 3 D_{22}).$$ \hspace{1cm} (2)

The elements of the bending stiffness $D_{ij}$ for the front facesheet are calculated using the classical laminate theory. Equation 1 shows a complex relationship between contact force and local indentation. So a nonlinear spring would be needed to represent such a behavior. For simplification the formula is linearized in a way that the deformation energy of the linear spring will be equal to the one that would be reached by a nonlinear spring. For the global stiffness $k_{\text{glo}}$ of the sandwich panel springs representing the bending stiffness $k_b$ and shearing stiffness $k_s$ are connected in series:

$$k_{\text{glo}} = \frac{k_b k_s}{k_b + k_s}.$$ \hspace{1cm} (3)
The bending stiffness $k_b$ for a square plate with fully clamped edges is calculated by [14]:

$$k_b = \frac{D}{0.0056 a^2}. \quad (4)$$

The equivalent bending stiffness $D$ of the sandwich can be calculated comparable to equation 3, but by using the elements of the bending stiffness matrix $D_{ij}$ of the whole structure. For the shear stiffness $k_s$ no formula for fully clamped square plates could be found. So a formula for a round sandwich with fully clamped edges is used [12]:

$$k_s = \frac{4 \pi G_c (h_{face} + h_c)^2}{h_c (1 + 2 \ln \frac{R}{R_c})} = \frac{4 \pi G_c (h_{face} + h_c)^2}{h_c (1 + 2 \ln \frac{4 \sqrt{\pi} \sqrt{0.00560}}{R_c})}. \quad (5)$$

Here the contact radius $R_c$ is being assumed to be 0.4 times the radius of the impactor $R_{Imp}$ [11]. $G_c$ is the shear modulus of the core and $h_{face}$ the averaged thickness of the facesheets. The radius of the corresponding circular plate $R$ is being chosen, so that the bending stiffness of a round and a square plate with both clamped edges would be equal [14, 15].

With the calculated local and global stiffness the equation of motion for the spring-mass model become:

$$\left( M_{Imp} + m_t \right) \ddot{x}_1 + (x_1 - x_2) k_{loc} = 0, \quad (6)$$

$$m_{sand} \ddot{x}_2 + k_{bs} x_2 - (x_1 - x_2) k_{loc} = 0. \quad (7)$$

The effective masses of the top face sheet $m_t$ and of the sandwich $m_{sand}$ are about 10% of their real mass [12, 13]. So they are considerably smaller than the mass of the impactor $M_{Imp} = 24.77$ kg and can therefore be neglected. By doing this the system of equations becomes analytically solvable. The local indentation $(x_2 - x_1)$ is limited to 6 mm which was shown in a static compression test on the sandwich structure to be the maximum foam compression before consolidation.

The effect of the impact on the ferrite is estimated by looking at the resulting strains. Therefore a narrow strap of the sandwich plate with a width of 1 mm is considered as a composite beam. The highest strains will be reached when the sandwich has the maximum of its global deflection $x_2$. The corresponding force that is needed to get the same deflection with the composite beam is:

$$F = x_{2\text{max}} / \left( \frac{a^3}{192 \, EI_{id}} + \frac{a}{4 \, GA_s} \right). \quad (8)$$

$GA_s$ is the shear stiffness and is calculated analytically by using the formula:

$$\frac{1}{GA_s} = \frac{1}{(EI_{y})^2} \int_{z_1}^{z_b} \frac{1}{G(z)} (ES(z))^2 \, dz,$$

$$ES(z) = \int_{z_t}^{z} E(z') z (z') \, dz'. \quad (9)$$

The coordinate $z$ is defined in the coordinate system of Figure 2. The strains $\varepsilon$ in the composite beam can then be calculated by using:

$$\varepsilon = \varepsilon_n \kappa = z_n M_{bending} = z_n \frac{F \, a}{8 \, EI_{id}}. \quad (11)$$

Here $\kappa$ is defined as the curvature of the panel strip and $z_n$ the distance from the top or bottom of the ferrite to the neutral axis.
4 NUMERICAL SIMULATION

Explicit Dynamic Finite Element Models were developed of the panel impact using Abaqus 6.14-2. One quarter of the panel was modelled with symmetry boundary conditions at the centerlines. The outer regions of the panel were clamped as in the test fixture.

The GFRP/Foam/Ferrite/GFRP sandwich panel was constructed as a single part using C3D8R elements and partitioned through the thickness for each layer. Three elements were used through the thickness for the facesheets, five elements for the foam and four elements for the ferrite. The facesheets were defined as orthotropic elastic and the ferrite as isotropic elastic. For the foam the Abaqus HYPERFOAM constitutive model was used, based on experimental uniaxial compressive data.

The impactor was modelled as a spherical rigid body with defined mass and initial velocity and constrained to only move vertically. General contact was defined between the impactor and the panel.

Figure 3: Finite element mesh plot.

5 DROP-WEIGHT IMPACT TESTING

Two panels were tested in an Imatek IM10 Impact testing System with a custom designed test fixture used to clamp the outer edges of the panel, as shown in Figure 5. The exposed region of the panel has dimensions of 340 x 340 mm. The Impact Testing System measures impact velocity and force and calculates impactor acceleration and displacement. An LCIT displacement transducer was used to measure the deformation of the underside of the panel at the center. A high-speed camera Phantom V210 with a frame rate of 3000 fps was used to image the deformation, validate the displacement measurements and to characterize any fixture deformations.

Figure 4: Impact Testing System (Left) and Test Fixture (Right).
The impactor has the mass $M_{\text{imp}} = 24.77$ kg and the radius $R_{\text{imp}} = 90$ mm. The test of the first panel started at an impact energy of 20 J (1.25 m/s) and increased in 20 J increments to 140 J (3.29 m/s). The second panel was tested starting at an impact energy of 140 J and increasing in 20 J increments to 480 J (6.23 m/s).

### 6 RESULTS AND DISCUSSION

In this section the results of the impact tests are shown and compared to the analytical and numerical calculations. The time-histories of the measured force and energy of the striker, the deflection of top and back face and the strains at the six positions are presented for panel #2 at 140 J. The maximum of the displacement and the strains reached for all tested energies are then compared and used to validate the analytical and numerical models.

As can be seen in Figure 5 the impact event at 140 J takes about 14.5 ms and the measured values show an approximately sinusoidal curve. The striker force reaches a maximum of almost 19 kN and shows small oscillations. The energy transferred from the impactor to the system goes up to almost 140 J, when the panel is deflected the most and the striker therefore has no velocity. At the end of the event 48 J are not returned to the impactor.

![Figure 5: Front and back face deflection, energy and force over time for 140 J and panel #2.](image)

The front face deflects a maximum of 13.8 mm while the back face only deflects 8.31 mm for panel #2 at 140 J. Calibrated high-speed photography was used to validate these deflection measurements and to quantify deformations of the test fixture edges at each impact energy. When corrected for the fixture deformations the front face deflects a maximum of 12.0 mm and the back face 6.58 mm. The difference of 5.42 mm is the through-thickness compression of the test specimen at the impact position. This is believed to be predominantly due to the compression of the polymeric foam. The foam has an initial thickness of 7 mm, meaning that it is almost completely compressed during impact.

Despite the large through-thickness deformation of the panel during the impact, post-test sectioning of the test specimen (see Figure 6) showed that there was less than 0.5 mm permanent through thickness deformation of the specimen at the impact position. It appears that the relatively stiff glass-fiber facesheets pull the crushed foam back to almost its initial thickness after the impact. Some delamination between the back facesheet and the ferrite was observed at the center of the panel opposite the impact location and at the positions of the joins between the ferrite blocks closest to the clamped support edges. This corresponds to areas of high transverse shear force due to the shear lag effect. There was no damage visible in the ferrite and the top facesheet, but some micro cracking of the 90 degree layers in the back facesheet.
The strains at the different positions at the top and bottom of the ferrite can be seen in Figure 7. The measured data showed significant noise, especially during the first half of the impact event. For a better comparison of the data, a running average with a window size of 40 was used. At the top of the ferrite (TC0, TC25, TC75) compression strains occur, although the strain at TC75 changes to tensile at approximately 5 ms, presumably due to damage occurring to the panel. The maximum compression strain of approximately -1500 με is reached at the strain gauge 25 mm away from the center. At the central impact position the high radius of the impactor and the plastic deformation of the foam is expected to result in an almost uniform pressure on the ferrite. This explains the smaller strains at TC0 in comparison to TC25.

At the bottom of the ferrite (BC0, BC25, BC75) the strains are generally tensile, with a maximum of 2800 με at the panel center. The absolute strains are higher at the bottom of the ferrite than at the top, due to the higher distance to the neutral axis at the bottom. At all bottom ferrite strains a temporary drop occurs at the first half of the impact event, although not at the same time. Again this effect is suspected to occur due to damage in the panel. This initial plateau in the strains does not occur at higher energies, where the panel has already been impacted several times. A possible reason is that the foam core compresses abruptly due to crushing at the first impact event. During the following tests, the predamaged foam does not show an abrupt compression.

![Diagram](image-url)

**Figure 7:** Filtered strains over time for 140 J and panel #2.

The deflections of the impact face and of the back face for all energies up to 200 J are shown in Figure 8. At 140 J two data points are available, one for each panel tested at this energy. As panel #1 was already tested at 20 J to 120 J, the higher deflections at 140 J in comparison to panel #2 is presumably related to the accumulated material damage. Measured deflections with already impacted panels are as suspected higher than the ones reachable with new panels. The front face deflects from 5.06 mm at 20 J to 16.1 mm at 200 J. At the higher energies, the deflection increases at a smaller rate than at lower energies. This increased stiffness of the panel at higher energies is expected due to the
large deflections which will lead to significant membrane forces in the facesheets. Panel #2 was impacted up to 480 J, with a back face deflection of 22.5 mm and the panel staying structural intact. But due to increased delamination and expected damage in the ferrite and facesheets the measured data are not included in the figure.

The back face deflects from 1.75 mm at 20 J to 10.1 mm at 200 J. The difference between impact face and back face deflection is due to the compression of the foam. Starting from 3 mm at 20 J the compression of the core increases up to 6 mm at 120 J. For the higher energies the local indentation of the front facesheet stays at 6 mm, as this is the maximal possible compression of the foam. Static compression tests were conducted to characterise the behavior of the core. These demonstrated that the foam can be compressed up to 5 mm without an increase of the compressive stress. Then the compressive stress starts to increase dramatically up to the maximal compressive extension of about 6 mm.

Figure 8 shows that the analytical model predicts both deflections quite well and correlates well with the shape of the experimental curve. The measured deflections of panel #2 at 140 J are about 2 mm smaller at the back face and about 1.5 mm smaller at the front face then calculated analytically. So the real stiffness of the undamaged panel seems to be higher than the one used in the spring-mass model. In comparison to the tests, the results of the FEA-analysis show almost the same front face deflections up to approximately 120 J, but under predict the deflections above this. The back face deflections of the FEA-analysis are lower than the experiments over all energies. When using undamaged panels for each energy a smaller deflection is expected. Then the measured data would be smaller and nearer to the FEA-simulation. For the energies above 200 J the deflection is underestimated by the FEA-model. But as increasing damage is expected within the structure above 200 J, the results of experiment and simulation are hardly comparable.

Figure 8: Deflection of the top and back face from experiment (EXP), analytical (ANA) and numerical (FEA) simulations.

The maximum values of the measured strains at all impact energies can be seen in Figure 9. Due to the impact damaging the strain gauges, the strains at the top of the ferrite could only be measured up to 180 J. The absolute values of the strains at the top of the ferrite increase with higher energies. The maximum is measured at position TC25 with -2252 με at 180 J. For the tests of panel #1 the strains at TC0 are similar to the ones at TC25. For panel #2, the strains at TC0 are significantly smaller than at TC25. As the strain gauges fail at 180 J, damage at 140 J is quite possible and the data are therefore difficult to interpret. Both strains at 75 mm (BC75, TC75) are much smaller than the strains at the other positions.

The strains at BC0 and BC25 both increase with higher energies and go up to 4941 με for 200 J at BC0. The absolute maximum is reached at 440 J with 7544 με. The deflection of the back face reaches about 15 mm at this energies. This is the limit that could be reached in 3 point bending tests on sandwich beams with the same composition before failure. Therefore a lot of damage is expected at this energies.
This was confirmed by cutting the panel after it was impacted with 480 J, where cracks in the ferrite center were observed. Again the results above 200 J are not shown in the figure, as a lot of internal damage is expected in the panel and the data therefore not comparable.

![Figure 9: Measured strains at the top (TC0, TC25, TC75) and bottom (BC0, BC25, BC75) of the ferrite.](image)

As can be seen in Figure 10 the strains at the center of the panel are overestimated by the analytical spring mass model. As the deflections are modelled quite well, the reason therefore must be in the calculation of the strains with equations 8 to 10. Presumably the shear stiffness $G_A$, is calculated too high, as the deformed cross-section in Figure 2 is used for equation 9. A higher ratio of the shear deformation on the total deflection would decrease the strains in the ferrite. The FEA-simulation predicts the strain at the back of the ferrite quite well. At the top of the ferrite the absolute values of the compression strains are calculated comparable to the tests for energies up to 60 J, then both models overestimate the strains. It is possible that the accuracy of the strain gauges is being affected by the high through thickness compression in the impact region. Additionally both models are not capable of modelling plastic deformation and damage in the structure. But as the tensile strains in the back face are higher and assumed to be more critical for the embedded ferrite, the exact prediction of the back face is more important.

![Figure 10: Strains in the panel center from experiment (EXP), analytical (ANA) and numerical (FEA) simulations.](image)
7 CONCLUSIONS

The low-velocity impact response of a foam core sandwich panel with glass-fibre reinforced facesheets and an embedded ferrite plate has been investigated analytically, numerically and experimentally. Deflections of the back face from 1.75 mm at 20 J up to 10.1 mm at 200 J were measured in the experiments. The strains at the top of the ferrite reached maximum values of up to 2251 με for 180 J at 25 mm from the center of the panel. Strains at the bottom of the ferrite up to 4941 με were reached in the panel center at 200 J. The analytical model calculates the front face deflection quite well, but over predicts the deflection of the back face. The strains are calculated to high for top and back face. The numerical simulation accurately models the deflections of the front face at lower energies, but under predicts the front face deflection at higher energies and the back face deflections. The strains in the back of the ferrite are numerically modelled quite well, whereas the strains at the front are lower in the tests then in the simulation. The panel design with the ferrite bonded to the back facesheet and a foam core between the ferrite and the front facesheets was successful at protecting the ferrite from damage at the anticipated impact energy level of 140 J.

ACKNOWLEDGEMENTS

The support of BMW AG Munich and the Centre for Advanced Composite Materials, University of Auckland for this work are gratefully acknowledged, as is the assistance of Mr. Samuel van Oosterom CACM with the manufacturing the test samples.

REFERENCES


