Pamukkale Üniversitesi Mühendislik Bilimleri Dergisi

Pamukkale University Journal of Engineering Sciences

Fea-Based verification of mechanical and thermal behavior of PVC coated pur foam substrate used in bus dashboards

Otobüs gösterge panellerinde kullanılan PVC kaplı pur köpük alt yapısının mekanik ve sıcaklık etkileri altındaki davranışının sonlu elemanlar analizi tabanlı doğrulaması

Niyazi Oral[1], Okan Otuz[2], Rafet Kemal Kocabıyık[2], Mustafa Akoğlu[1]*

[1]Smartinject, Bursa, Türkiye.
noral.16@hotmail.com, mustafaocakoglu@vimasys.net
[2]Mercedes-Benz Türk, İstanbul, Türkiye.
okan.otuz@daimlertruck.com, rafet_kemal.kocabiyik@daimlertruck.com

Received/Geliş Tarihi: 06.12.2024 Revision/Düzeltme Tarihi: 01.09.2025 doi: 10.5505/pajes.2025.00810
Accepted/Kabul Tarihi: 24.09.2025 Research Article/Araştırma Makalesi

Abstract

Polyurethane (PUR) foam layers with polyvinylchloride (PVC) coating used in bus instrument panels are subjected to various mechanical and thermal effects. Understanding the behavior of these materials under these effects is critical to improve product performance and durability, and for this purpose, finite element analysis (FEA) was employed to complement experimental testing. The aim of this study is to investigate the mechanical and thermal behavior of PVC-coated PUR specimens by FEA and in particular to investigate the degradation mechanisms and the performance of the material under different conditions. The mechanical behavior of PVC coated PUR specimens was modeled under different loading and temperature conditions and analyzed with finite element software. The simulations were modeled based on experimental data and literature information and included tensile, three-point bending, peel and thermal expansion tests. Strain-displacement ratios were monitored using digital image correlation (DIC) techniques. The results are in high agreement with the experimental data. Strain quantities show small variations due to calculation errors, but 97% convergence in three-point bending tests and 96% convergence in single lap joint tests. Thermal analysis and DIC results confirm the other findings. The work presented here demonstrates the ability to simulate the behavior of PVC-coated PUR specimens through FEA, avoiding material and time waste and contributing to design changes. FEA results were in good agreement with the physical tests, confirming suitability for specimen-level analysis.

Keywords: PVC coating, Polyurethane foam, Material characterization, Bus dashboard, Primed material, Finite element method

Özet

Bu çalışma kapsamında poliüretan (PUR) köpük katmanları ile polivinilklorür (PVC) kaplama kullanılan otobüs gösterge panelleri, çeşitli mekanik ve termal etkilere maruz kalmaktadır. Bu malzemelerin bu etkiler altındaki davranışını anlamak, ürün performansını ve dayanıklılığını artırmak için kritik öneme sahiptir ve bu amaçla deneysel testlere ek olarak sonlu elemanlar analizi (FEA) uygulanmıştır. Bu çalışmanın amacı, FEA kullanarak PVC kaplı PUR numunelerin mekanik ve termal davranışlarını incelemek ve özellikle farklı koşullar altında malzemenin bozunma mekanizmalarını ve performansını araştırmaktır. PVC kaplı PUR numunelerin mekanik davranışı, farklı yükleme ve sıcaklık koşulları altında modellenmiş ve sonlu elemanlar yazılımı ile analiz edilmiştir. Simülasyonlar, deneysel veriler ve literatür bilgilerine dayanarak modellenmiş ve çekme, üç nokta eğme, soyulma ve termal genleşme testlerini içermiştir. Gerinim-yer değiştirme oranları, dijital görüntü korelasyonu (DIC) teknikleri ile izlenmiştir. Sonuçlar, deneysel verilerle yüksek uyum göstermektedir. Gerinim değerleri hesaplama hatalarından kaynaklanan küçük farklılıklar göstermektedir, ancak üç nokta eğme testlerinde %97, tek bindirmeli bağlantı testinde ise %96 yakınsama sağlanmıştır. Termal analiz ve DIC sonuçları diğer bulguları doğrulamaktadır. Bu çalışmada sunulan sonuçlar, FEA kullanılarak PVC kaplı PUR numunelerin davranışını simüle etme yeteneğini göstermekte, malzeme ve zaman kaybını önlemekte ve tasarım değişikliklerine katkı sağlamaktadır. FEA sonuçları fiziksel testlerle iyi bir uyum içindedir ve numune düzeyinde analiz için uygunluğunu doğrulamaktadır.

Anahtar Kelimeler: PVC kaplama, Poliüretan köpük, Malzeme karakterizasyonu, Otobüs gösterge paneli, Astarlanmış malzeme, Sonlu elemanlar yöntemi

1 Introduction

The dashboard is an important component of any vehicle. It is positioned directly in front of the driver and is used to display the indicators and controls necessary for vehicle operation. Dashboards also protect the mechanics of the vehicle against moisture, impact, abrasion, excessive heat and harmful UV rays. In buses, the dashboard is usually in the form of a modular panel with a closed driving area surrounding the driver.

Passenger vehicle dashboards typically consist of three main categories [1]: foam-molded, injection-molded and fiberglass-reinforced plastic dashboards. Injection molded dashboards are formed from materials such as polyvinylchloride (PVC), polypropylene (PP), styrene–maleic anhydride (SMA), polyphenylene oxide (PPO). They are used in large, solid-bodied vehicles. Advantages include superior surface quality and efficient production in large quantities. Foam molded dashboards are composed of PVC plastic, glass fiber and PUR foam. They are divided into two classes: soft and rigid foam molded. It is advantageous due to its medium-high development cost, smooth surface and lightweight

*Corresponding author/Yazışılan Yazar

construction. Fiber Reinforced Polymer (FRP) dashboards are used in unique vehicles with gel-coated and different painting methods. They are portable, versatile and the paint can be peeled off easily.

Oral et al. [2] conducted a detailed examination of the mechanical and thermal properties of PVC-coated PUR foams. The study evaluated the structural behavior of PVC and PUR layers under the influence of primer and analyzed material performance under different temperature and loading conditions. Scanning electron microscopy (SEM) and fourier transform infrared spectroscopy (FTIR) analyses were performed to investigate material properties and interactions between layers. According to the results of the study, it was observed that PVC-coated PUR components can be effectively separated, and design modifications could improve the durability of bus dashboards.

Jeyanthi S. et al. [3] aimed to investigate the mechanical properties of rigid PUR foams and numerically evaluate the behavior of these foams using finite element analysis (FEA). In this study, the effect of using castor oil based polyols instead of petroleum derived polyols on the mechanical properties of PUR foams was investigated. In this context, experiments and numerical analyzes were carried out. Experimentally, PUR foams were synthesized using castor oil based polyols. The foams were then tested for their mechanical properties such as compressive strength. The experimental data obtained formed the basis for the numerical analysis. The Kelvin foam model developed using ANSYS software was used for the numerical analysis. These models were used to verify the experimental results. When the numerical analysis results were compared with the experimental data, the results showed that PUR foams using castor oil based polyols had higher compressive strength than foams using petroleum derived polyols.

Marsavina L. et al. [4] studied the mechanical properties of PUR foam under different loading conditions and the effect of impregnation. Experimental studies on PUR foam with a density of 200 kg/m^3 were carried out in three different materials engineering laboratories in Lublin, Bucharest and Timisoara. This study evaluated the possibility of describing foam material behavior through compression testing, micromechanical modeling and FEA and highlighted the need to develop micromechanical models to describe the mechanical behavior of PUR foam materials. An optimization procedure was proposed for the determination of micromechanical parameters obtained from uniaxial compression tests of different foam types. The study concluded the ability of micromechanical modeling and FEA being successfully used to describe the behavior of PUR foams, but the compressive tests should provide reliable material parameters.

Alagarraja K. et al. [5] focused on the use of PVC and PUR foam materials in hybrid composites. In this research, the mechanical and thermal properties of sandwich composite materials were evaluated through various tests. The results showed that PVC foam composites outperformed other materials in properties such as tensile, flexural, and impact strength. Notably, PVC foam with 0° orientation demonstrated the highest tensile strength [5].

Milojević S. et al. [6] explored replacing bus dashboard materials with eco-friendly alternatives, emphasizing design considerations for disassembly, reuse, and recycling to optimize material reusability. The study focused on transitioning from plastic and PUR foam to ecological and carbon-reinforced materials in natural gas city buses, integrating recycling requirements into the new design [6].

Although there are various experimental and simulation studies on PVC, PUR foam and bus dashboard in the literature, there is no FEA of PVC and PUR foam together. Extending the results, obtained by physical tests and FEA of PVC coated PUR component layers under different loading and temperature conditions, aimed to fill the gap in the literature.

FEA has undoubtedly changed the way engineers design and analyze, providing a comprehensive and advanced approach to predict and optimize the behavior of engineering components and systems. FEA provides a powerful method for simulating and evaluating material properties in order to gain a full understanding of a sample's characteristics. FEA can be used to evaluate factors such as the interaction between the foam and the PVC coating, the impact resistance of the component and the overall structural integrity. In this context, a systematic approach has been adopted to perform FEA of PVC coated PUR foam parts.

The dashboard analyzed in this study is primarily composed of three key components: PVC coating, primer, and PUR foam. The PVC coating enhances the dashboard's durability and aesthetic appearance, while the PUR foam contributes to its flexibility and strength. The study encompasses FEA of tensile testing, three-point bending tests, primer characterization, peel testing, thermal expansion testing of PUR material, and thermal evaluations of PVC-coated PUR components both with and without primer. This study aimed to validate the results of FEA of PVC coated PUR foam components by comparing them with previously reported test results [2].

The first step involves creating an accurate model of the component, taking into account geometric complexity and material properties. Once the model is created, various loading scenarios are simulated. The analysis focuses on issues such as durability, behavior under thermal and mechanical effects. It also includes experimental tests and comparative studies to evaluate the performance of PVC coated PUR foam under specific conditions. Before starting the analysis, it is important to understand the basic behaviors of PVC coated PUR foam and their impact on the overall performance. By studying the specific composition of a material and its interactions, a deeper understanding of the material's behavior and properties can be gained. Parameters such as Young's modulus, Poisson's ratio and coefficient of thermal expansion are critical to accurately simulate the behavior of the components in the model. In the modeling, the material properties were determined based on literature data and experimentally obtained information. The experiments were also monitored using the digital image correlation (DIC) technique and the strain-displacement curves obtained were analyzed.

Understanding the response of the material to various force factors and environmental effects is an important aspect of this analysis. In order to evaluate the performance of PVC coated PUR foam components under different environmental conditions, temperature effects as well as mechanical loading were included in the analysis. This provides data on how temperature changes affect the structural integrity and mechanical properties of the components and helps to gain a comprehensive understanding of their behavior.

2 Finite Element Method

Engineering problems can be addressed using three main approaches: numerical method, analytical method, and experimental method [7]. Finite Element Method (FEM) is a numerical technique for analyzing structures. It involves creating a computer model of a material or design, applying stress, and evaluating the results. FEM utilizes a network of points and elements to form a grid known as a mesh. This mesh is programmed with material and structural properties such as modulus of elasticity, Poisson's ratio, and yield strength. These properties determine the structure's response to specific loading conditions. A grid functions like a spider web with grid elements extending from each node to adjacent nodes. The basic principle is to perform calculations only on a certain (finite) number of points and then interpolate the results over the entire area (surface or volume). Continuous objects have infinite degrees of freedom (dofs) and cannot be solved in this format. FEM reduces the degrees of freedom from infinite to finite with the help of meshing (points and elements) and all calculations are performed on a finite number of nodes [8].

Material properties used in finite element analyses are shown in Table. 1. For PVC and PUR foam. It is well established that both PVC and PUR foams exhibit viscoelastic and rate-dependent behavior under mechanical loading. However, since the validation tests in this study were conducted under quasi-static conditions, the influence of rate dependency was assumed to be negligible. Therefore, a rate-independent elastoplastic material model was employed to ensure sufficient accuracy without increasing computational cost. Abaqus was used as finite element software [9].

Table 1. Material Properties Used in Finite Element Analysis [10, 11]

Material Property	*PVC*	*PUR Foam*
Elasticity Modulus (MPa)	1000	66.1
Poisson Ratio	0.429	0.316
Density (kg/m^3)	1400	192

2.1 Mesh Convergence Study

To ensure that the finite element analysis results were independent of the mesh density, a mesh convergence study was performed. Due to the significant stress concentration observed under the loading pin, the three-point bending test model was selected for this study. Various mesh densities were generated using C3D8R elements. The maximum von Mises stress at the center of the bottom surface of the specimen was monitored as the key convergence parameter. The results of the study are presented in Figure 1.

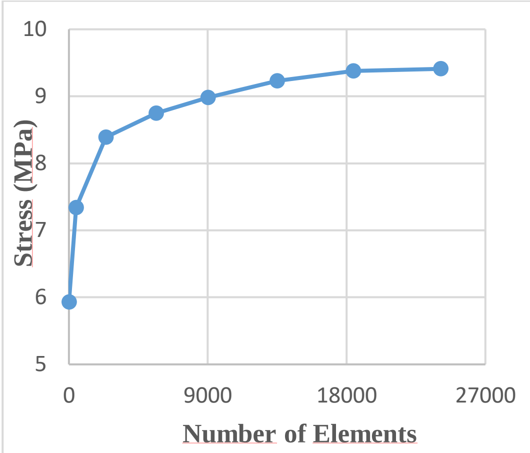


Figure 1. Mesh Convergence Analysis for Finite Element Analysis

2.2 Finite Element Modeling of Three Point Bending Test

The three-point bending test finite element model was created as a 3D dynamic/explicit model. The parts representing the upper and lower test equipment shown in Figure 2 are drawn as 'discrete rigid' elements. The intermediate parts representing the specimens were created as deformable elements.

As material properties, PVC and PUR parts are considered as elasto-plastic materials and 'ductile damage' is defined. The analysis time was 0.001 seconds. Since there was no separation between PVC and PUR during the test, perfect adhesion between these two parts was defined in the model.

Rotational and translational movements in all directions were prevented by defining the lower support parts as anchored. The moving part at the top was only allowed to move in the Y direction and the displacement data was modeled as 34 mm displacement in order to reach the displacement value obtained during the test. Finally, the parts were meshed using C3D8R element type and the analysis was solved.

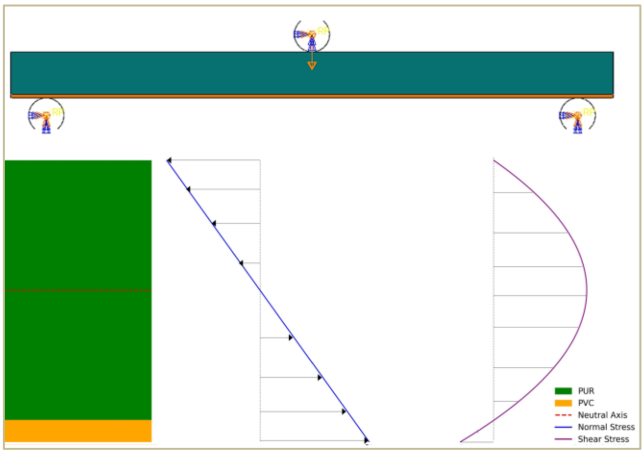


Figure 2. Finite Element Model Constraints and Associated Stress Distributions

2.3 Finite Element Modeling of Peel Test

The peel test finite element model was created as a 2D dynamic/explicit model as shown in Figure 3. Since there was no fracture on the PVC surface during the test and fractures occurred from the PUR material, only elasto-plastic material properties were entered for the PVC material. The PUR material was defined by entering damage properties in addition to elasto-plastic properties as in other finite element applications. The test duration was set as 1 second.

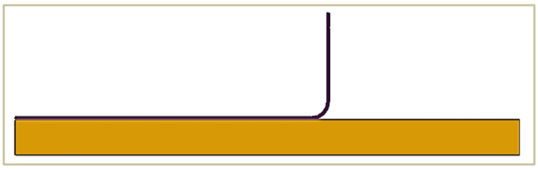


Figure 3. Model Created for the Peel Test

When tension is applied to the PVC material, the separation is between the PVC and PUR materials and since the PUR material breaks, perfect adhesion between these two materials is defined on the model. In the constraints in Figure 4, PUR material movement is prevented in all directions and PVC material is allowed to move only in Y direction. For the tensile process, a displacement of 60 mm was given from the top of the PVC material to create movement. The parts were meshed using CPE4R element type and the analysis was solved.

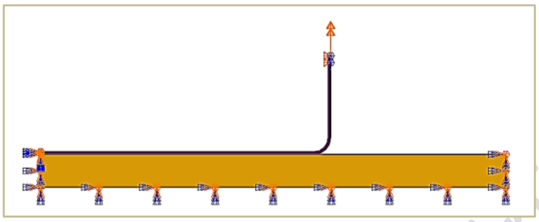


Figure 4. Constraints and Movement on the Created Model

2.4 Finite Element Modeling of Thermal Test

For the thermal test finite element application, the model shown in Figure 5 was created in a 2D deformable structure. The analysis was solved statically and the solution time was set as 1 second. Only elastic properties and thermal expansion coefficients [PVC: $70x10^{-6}$ (1/K) and PUR: $57.8x10^{-6}$ (1/K)] were entered in the material definition.

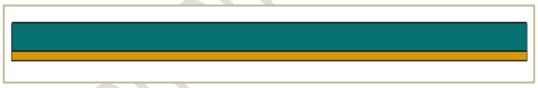


Figure 5. Finite Element Model Created for Thermal Test

A thermomechanical analysis definition was made by entering the ambient temperature on the model. In the definition shown on Figure 6, the temperature was increased from 25 °C to 170 °C in the analysis as well as during the test. The parts were meshed using CPS4R element type and the analysis was solved.

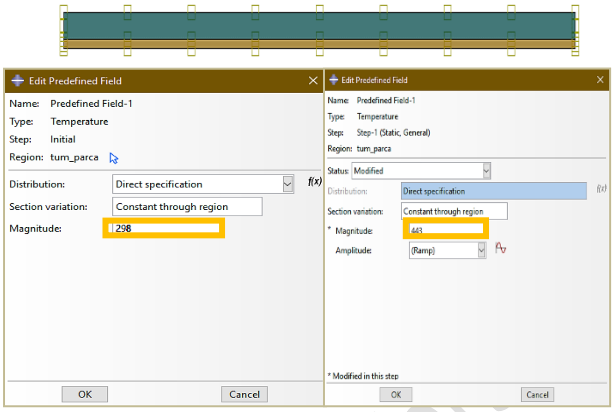


Figure 6. Temperature Values Given for Thermomechanical Test

2.5 Finite Element Modeling of Single Lap Joint (SLJ) Test

It was mentioned that there is a primer used in production between PVC material and PUR material. However, this primer could not be observed in any test other than the thermal test described above and therefore no information about its mechanical behavior could be obtained. In order to know the exact properties of the primer used, a finite element model was created with reference to the SLJ test results performed by the company. The company used aluminum material for the SLJ test.

The SLJ test was modeled as 2D dynamic/explicit as shown in Figure 7. The parts were created as deformable bodies. For the aluminum material, an elasto-plastic material definition was made by entering properties that can be easily found in the literature. Epoxy material properties were given using the 'cohesive section' definition in the Abaqus program. The analysis time was set as 0.01 seconds.

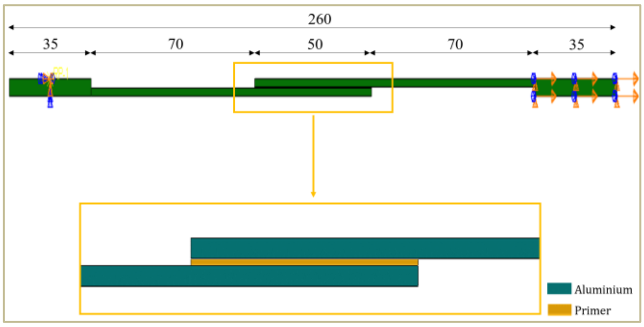


Figure 7. Finite Element Model Created for SLJ Test

Since we want the stresses and fractures that occur as a result of the analysis to be epoxy material, perfect adhesion was defined between aluminum and epoxy. The reference point on the left side was defined as anchored and its movement was restricted in all directions. The reference point on the right side was allowed to move only in the X direction and a displacement value of 2.2 mm was given. Finally, the parts were meshed using CPE4R element type and the analysis was solved.

2.6 Finite Element Modeling of Tensile Test

For the finite element tensile analysis, the model shown in Figure 8 was created in a 3D deformable structure. The analysis was solved statically and the solution time was set as 1 second. Only elastic properties were used in the material definition.

The part was meshed using C3D8R element type and the analysis was solved.

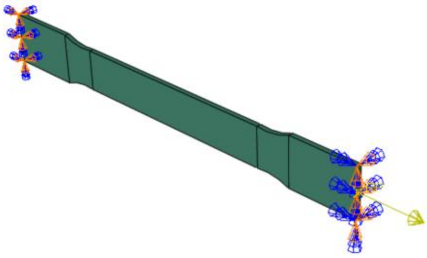


Figure 8. Finite Element Model Created for Tensile Test

3 Digital Image Correlation (DIC)

The primary objective of DIC is to determine the displacement and strain fields within a region of interest (ROI) on a deformed material specimen. This is achieved through image processing techniques. Images of the specimen are captured during deformation and serve as input for a DIC program. The process involves establishing a one-to-one correspondence between material points in the reference image (the initial undeformed state) and the current images (deformed states). DIC accomplishes this by dividing the reference image into small regions, known as subsets, and identifying their corresponding positions in the deformed images [10].

Among the tests, DIC was performed especially for the finite element tests. The results were verified by both tests and finite elements. All the steps described below for the bending test are the same as the DICs for the other tests.

Step 1: While the specimens were being prepared, speckling was done on them and the test was videotaped. After the test was performed, the videos were converted into photographs depending on the second.

Step 2: Matlab application is opened and Ncorr extensions are opened as a file as shown in Figure 9. Then the Ncorr extension is opened by typing the command 'handless_ncorr=ncorr' on the command screen.

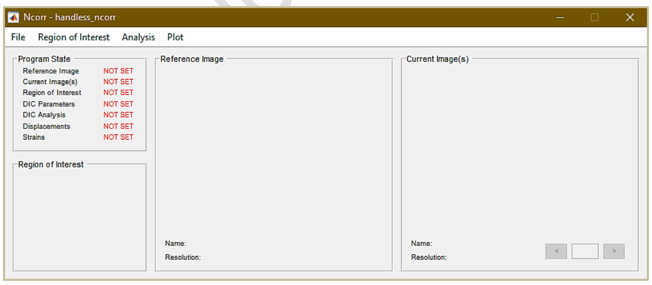


Figure 9. Opened Ncorr Extension

Step 3: Select the first image during the test with the 'File-Load Reference Image' option on the window that opens as shown in Figure 10.

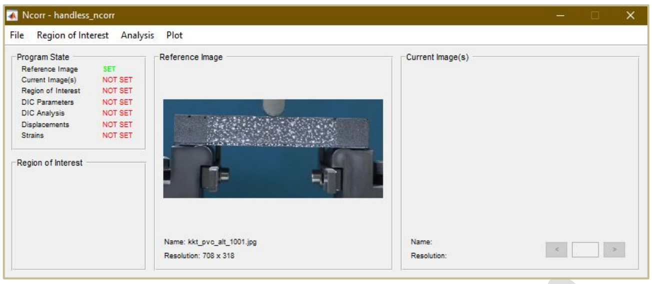


Figure 10. Reference Photo Selection

Step 4: Again 'file-load current image(s)' is used to select other images from the video as shown in Figure 11.

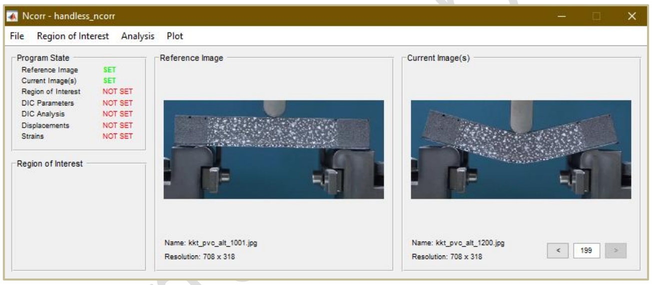


Figure 11. Selection of Other Photos

Step 5: Select the area where the operations will be performed and continue the process for displacement calculation by selecting the diameter of the area to be considered.

Step 6: When the 'Analysis- format displacement' option is selected, it is important to note that when the value of the line drawn on the reference photograph is requested, the length of the line on the real sample should be entered. If the actual value is not entered in this section, the results should not be expected to converge to the actual value.

Step 7: Finally, the strain values are calculated with the 'analysis- calculate strain' command. After the calculation, the displacement values in x and y directions and the strain values E_xx, E_yy and E_xy are displayed with the 'plot' option.

The described process steps were also performed for other tests. The results obtained are given in the results section as headings.

4 Results and Discussion

4.1 Finite Element Bending Analysis

The stress distribution and fractures on the part as a result of the analysis are shown in Figure 12. Stress and strain values were taken and compared with the test data. As a result of the comparison, when the convergence rate on the curves is analyzed, it is observed that there is approximately 97% convergence as shown in Figure 13.

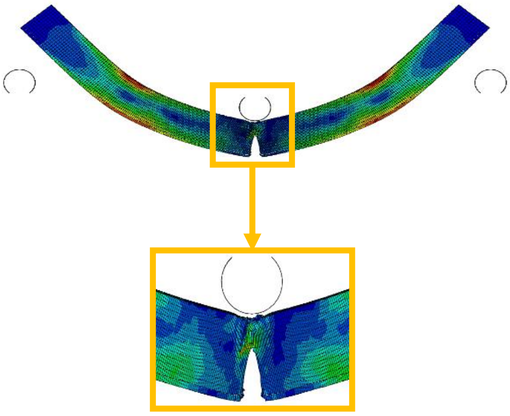


Figure 12. Stress Distribution and Fractures Formed on the Part as a Result of the Analysis

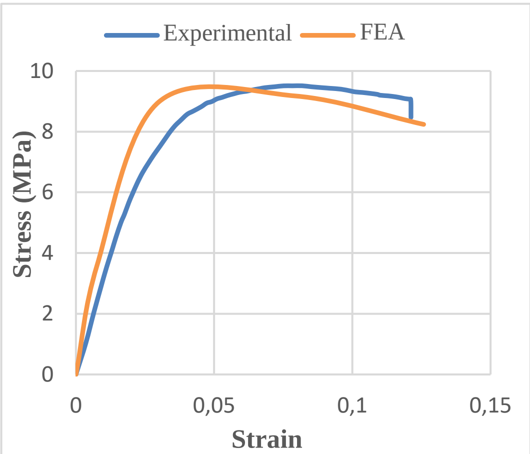


Figure 13. Comparison of FEA and Experimental Data

4.2 Finite Element Peel Analysis

As a result of the analysis, it is seen in Figure 14 that the PUR material fractures as observed in the tests. However, a convergence that can be considered correct could not be achieved in the load-displacement curves shown in Figure 15. The fact that the behaviors in the curves are close shows that the model is correct. However, full verification could not be performed due to the lack of material properties or primer definition assumed to be in between.

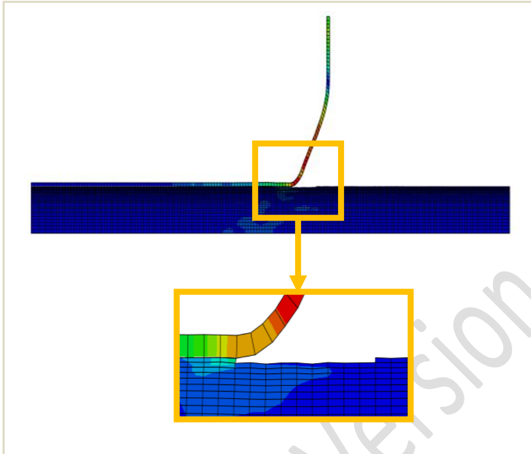


Figure 14. Stress Distribution and Fractures on the Part from the Analysis

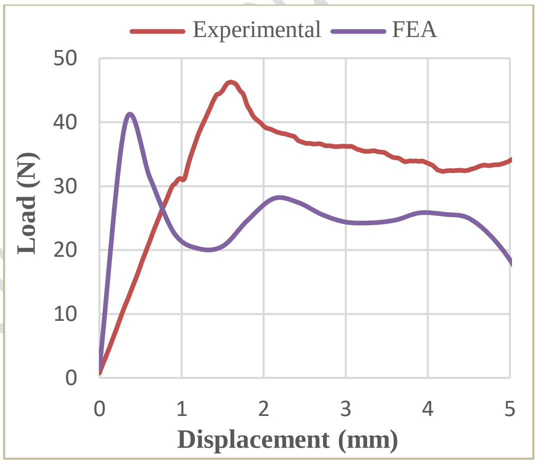


Figure 15. Comparison of Experimental and FEA Results

4.3 Finite Element Thermal Analysis

Since the thermal tests were performed manually to observe the primer behavior, the results shown in Figures 16-18 are not available for comparison. The results of the finite element implementation of the thermal test were compared with the DIC results.

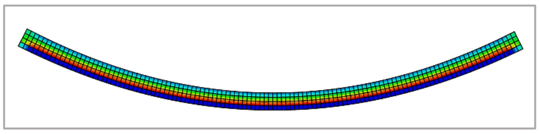


Figure 16. Stress Distribution on the Part as a Result of Analysis

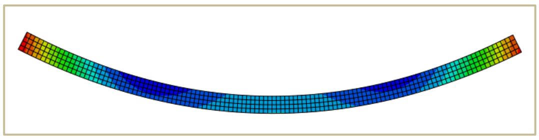


Figure 17. Displacement Distribution on the Part at the End of the Analysis

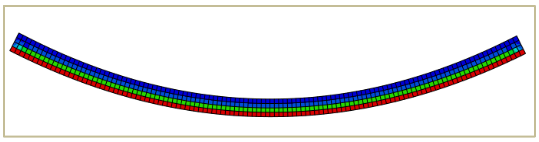


Figure 18. Strain Distribution on the Part as a Result of Analysis

4.4 Finite Element SLJ (Single Lap Joint) Analysis

As a result of the analysis, no fracture was observed in the epoxy material, but shear stress was generated as shown in Figure 19. The company test data shown in Figure 20 is compared with the FEA data in Figure 21. As a result of the analysis, it was seen that the curves showed approximately 96% convergence. The epoxy material values obtained as a result of the analysis are given in Table 2.

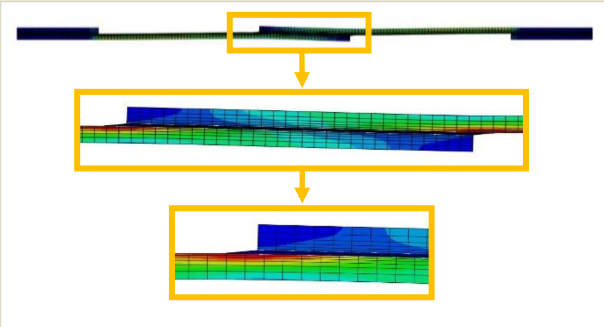


Figure 19. Shear Stresses Resulting from Analysis

Table 2. Primary Material Properties Used in Finite Element Analysis

Material Properties	*Value*
Elasticity Modulus (MPa)	1513.84
Poisson's Ratio	0.33
Density (kg/m^3)	1800
Maximum Stress (Max Principal Stress) (MPa)	18.47
Damage Displacement (Displacement At Failure) (mm)	2.2

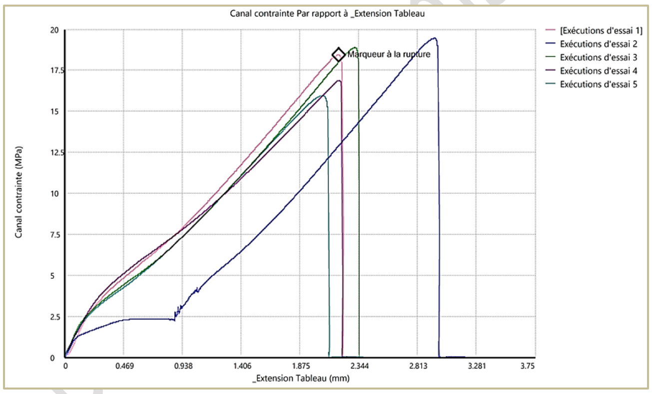


Figure 20. Company Experimental Data

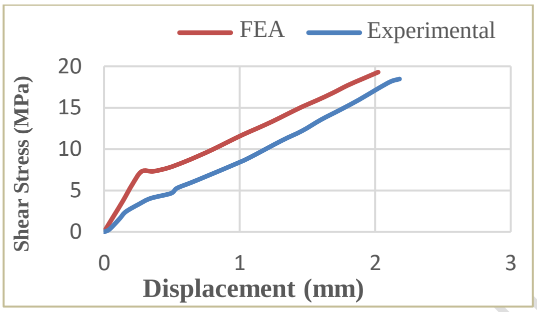


Figure 21. Comparison of Stress-Displacement Data Resulting from Analysis

4.5 Finite Element Tensile Analysis

The results obtained from the tensile test finite element application were compared with the results obtained from the laboratory experiments in the elastic region and a great convergence was obtained as shown in Figure 22. The fact that the elastic region in the curves were close to each other showed that the model is correct.

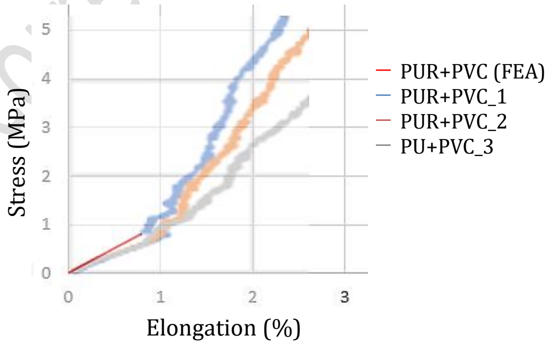
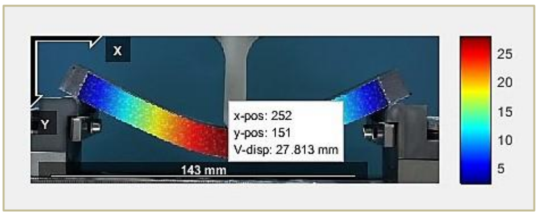


Figure 22. Comparison of linear FEA Results and Experimental Data

4.6 DIC (Digital Image Correlation) Results

The data obtained by DIC technique during the three-point bending test and thermal test are shown in Figures 23 and 24. The x-axis and y-axis represent the horizontal and vertical position of the specimen. v-disp (vertical displacement) represents the degree of deflection of the specimen during the test. Exx and Eyy represent the strain values of the specimen in the x and y directions. The maximum bending and stretching of the specimen occurs at the center. The color scale in the image shows the magnitude of the v-disp, exx and eyy values. The labels "x-pos" and "y-pos" in the image indicate the position of the specimen during the test. The displacement values obtained from the results are very close to the displacement values obtained during the test. Strain values, on the other hand, although close in order, could not be found exactly the same as the actual values due to the difference in Ncorr calculation.



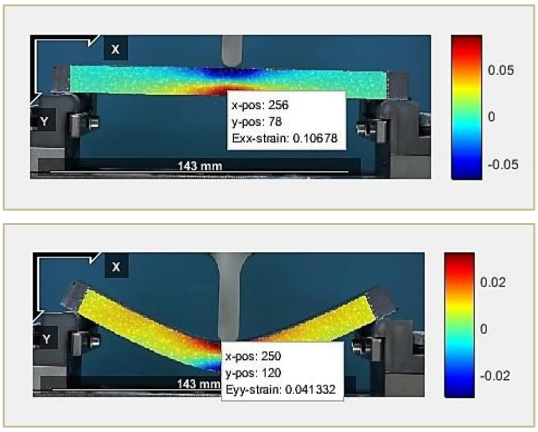


Figure 23. Three-Point Bending Test DIC Image Results

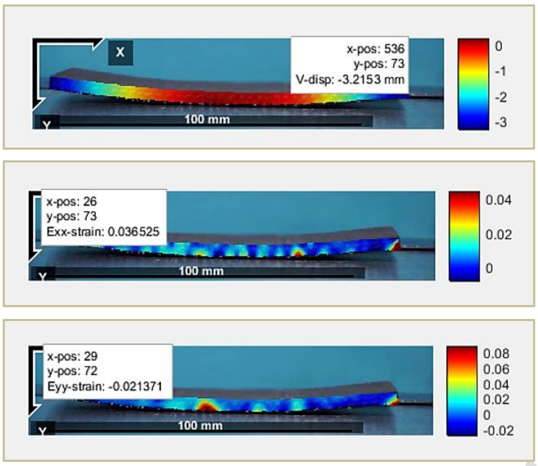


Figure 24. Thermal Test DIC Image Results

4.7 Discussion

Across all coupons and test types, our FEA captured the global response trends of the PVC-coated PUR system with high agreement in three-point bending and SLJ, while the peel response showed the largest gap. Similar levels of agreement between carefully calibrated FE models and foam experiments are widely reported: micromechanical/FE approaches can reproduce polyurethane foam stress–strain behavior provided that reliable test-based parameters are used [4].

In the same vein, recent work that combined experiments with finite-element modeling on rigid PUR foams underscores that FEA is a suitable vehicle for verifying mechanical characteristics when the constitutive inputs reflect the tested material [3].

A central reason our quasi-static bending and SLJ predictions converged is that the loading rates fell in a regime where rate effects on elastomeric polymers are modest; however, the literature is clear that polyurethane's response is strongly strain-rate dependent overall, with hyper-viscoelastic behavior and marked increases in stiffness/strength as rate rises. Accordingly, models that assume rate-independence may remain adequate for quasi-static verification but will need extension (e.g., hyper-viscoelastic laws) for rate dependent scenarios [13].

Temperature likewise plays a decisive role for PUR foams. Experimental campaigns show mechanical properties vary systematically with both temperature and density [14]. In our thermal simulations we employed elastic moduli with CTEs to represent expansion from 25 °C to 170 °C; the literature implies that including temperature-dependent stiffness (and, when relevant, viscoelasticity) can better represent thermo-mechanical coupling in PUR, especially as the matrix approaches temperature regimes where stiffness degrades [14].

The largest discrepancy in our study arose in peel. We assumed perfect adhesion at the PVC–PUR interface to match observed gross separation behavior under some tests; however, perfect tie constraints can suppress interface failure modes and concentrate damage within the compliant substrate. A common remedy in similar polymeric components is to represent the interface with a traction–separation law (cohesive elements) so that partitioning and debonding can emerge naturally [15]. Because our SLJ calibration already delivered ~96 % agreement, those data could be used to identify an interface cohesive law (e.g., strength and displacement at failure) and then transferred to the PVC–PUR peel model. This workflow mirrors prior finite-element practice where cohesive-zone modeling and appropriate interactions are introduced alongside hyperelastic/foam laws for the polymeric parts [15].

From a system viewpoint, the literature on sandwiches with polyurethane cores shows that multiscale or homogenized descriptions can reproduce global responses efficiently while respecting the microstructural mechanics [16]. Adopting such a framework in future iterations could reduce computational cost for dashboard-scale models while keeping fidelity to foam-core behavior.

Overall, our findings—97 % convergence in three-point bending and 96 % in SLJ—are aligned with the broader evidence that FE models of PUR/PVC structures are reliable once (i) the constitutive law is chosen for the loading regime (rate-independent elastoplastic for quasi-static; hyper-viscoelastic under dynamic), (ii) temperature dependence is represented when excursions are large, and (iii) the interface is modeled with a cohesive law when peel/debonding is central (Marsavina et al. [4]; Somarathna et al. [13]; Horák et al. [14]; Bergström [15]).

5 Conclusions

This study focused on the verification of the basic strength behavior of PVC coated PUR foam substrate used in bus dashboard under mechanical and temperature effects through FEA and experimental test results.

The study evaluated the behavior of PVC coated PUR foam substrate through three-point bending, peel, single lap joint (SLJ) and thermal tests. Comparisons between the FEA and test results showed that the model accurately simulates the strength properties of the specimen and gave results close to the actual test results. FEA and DIC are powerful tools that can be used to validate test results. Through the modeling study, the debonding mechanism between PVC and PUR materials was understood and the influence of the thermal behavior of the primer was observed.

Overall, this study demonstrated the accuracy of the test results of the mechanical and thermal properties of PVC coated PUR foam substrate. These findings will contribute to a better understanding and optimization of the design of a complex component such as a bus dashboard. In the future, these findings can be confirmed and extended with more detailed experiments and analysis. These studies will contribute to the design and development of composite materials used in automotive and other industries. In the future, more convergent results can be obtained by modeling more complex geometries and loading conditions, investigating the effects of different materials and primers, developing more precise experimental test methods, and using multi-scale modeling techniques.

6 Author contribution statements

In the conducted study, Author 1 contributed to the formation of the idea, creation of figures, validation of results, data curation, and writing review; Author 2 contributed to the content review of the manuscript, project management; Author 3 contributed to the provision of references; Author 4 contributed to the methodology.

7 Ethics committee approval and conflict of interest statement

"There is no need to obtain permission from the ethicscommittee for the article prepared".

"There is no conflict of interest with any person / institution in the article prepared".

8 References

[1] Xiamen Eco Import And Export Co. Ltd.. "Knowledge of bus dashboard – Knowledge" https://www.ecobusparts.com/info/knowledge-of-bus-dashboard-74281041.html (11.12.2024).

[2] Oral N, Otuz O, Kocabiyik RK, Akoglu M. "Characterization of mechanical and temperature effects on delamination and basic strength behaviors of PVC coated PUR foam substrate used in bus dashboard". *Heliyon,* 10(20), 2024.

[3] Jeyanthi S, Nivedhitha DM, Jeyamani E, Lazar PJL, Krishnaswamy P. "Experimental and numerical study of rigid PUR foams for mechanical characteristics using FEA". *Journal of Physics: Conference Series*, 2601(1), 2023.

[4] Marsavina L, Sadowski T, Constantinescu DM, Negru R. "PUR foams behaviour. Experiments versus modeling". *Key Eng Materials*, 399(1), 123–130, 2009.

[5] Alagarraja K, Vijaya RBM, Chinnadurai E. "Mechanical and thermal properties of PVC and PUR foam hybrid composites". *Materials Testing, 62(5), 544-552, 2020.*

[6] Milojevic S, Pesic R, Lukic J, Taranovic D, Skrucany T, Stojanovic B. "Vehicles optimization regarding to requirements of recycling example: Bus dashboard". *IOP Conference Series: Materials Science and Engineering*, 659, 2019.

[7] Tajima K, Chen K, Takahashi N, Noda N, Nagamatsu Y, Kakigawa H. "Three-dimensional finite element modeling from CT images of tooth and its validation". *Dental Materials Journal,* 28(2), 219-226, 2009.

[8] Geramy A, Sharafoddin F. "Abfraction 3D analysis by means of the finite element method". *Quintessence International*, 34(7), 526–533, 2003.

[9] Dassault Systemes. *Abaqus 2016 Analysis User's Guide Volume II: Analysis*, Rhode Island, United States, Dassault Systemes, 2011.

[10] Manjusha M, Althaf M. "Numerical analysis on flexural behaviour of GFRP sandwich roof panel with multilayer core material". *IOP Conference Series: Earth and Environmental Science*, 491(1), 012025, 2020.

[11] Naga S, Gadelrab RM, Bekheet N, AM A. "Effect of temperature, cross-head speed and duration of loads on the mechanical properties of PVC polymeric material". *8th International Conference on Aerospace Sciences and Aviation Technology*, Cairo, Egypt, 4–6 May 1999.

[12] Ncorr. "DIC Algorithms" http://www.ncorr.com/index.php/dic-algorithms (02.12.2024).

[13] Somarathna HMCC, Raman SN, Mohotti D, Mutalib AA, Badri KH. "Rate dependent tensile behavior of polyurethane under varying strain rates". *Construction and Building Materials*, 254, 119203, 2020.

[14] Horak Z, Dvorak K, Zarybnicka L, Vojackova H, Dvorakova J, Vilimek M. "Experimental measurements of mechanical properties of PUR foam used for testing medical devices and instruments depending on temperature, density and strain rate". *Materials*, 13(20), 4560, 2020.

[15] Bergström M. Product Development and Finite Element Analysis of Polyurethane Press Shoe. MSc Thesis, Karlstad University, Karlstad, Sweden, 2021.

[16] Belinha J. "Multiscale analysis of sandwich beams with polyurethane foam core: A comparative study of finite element methods and radial point interpolation method". *Materials*, 17(18), 4466, 2024.