EAN 2024 62tn annual conference on experimental stress analysis

Methodologies for Modeling and Testing Bus Hybrid Structure Roll-Over Dynamics

S. Špirk [1,a], T. Kalina [1] , J. Václavík [2]

[1] *University of West Bohemia, Czech Republic*
[2] *research and testing institute Pilsen, Czech Republic*
[a] *spirks@fst.zcu.cz*

Abstract: The realm of bus construction is a multifaceted and intricate domain, characterized by diverse requirements. Among these, the mass of the body frame stands out as a crucial parameter. A lighter body frame often translates to greater occupancy potential, yet it typically compromises the structural rigidity. However, striking a balance between reduced mass and maintained stiffness is achievable through geometric optimization or the integration of cutting-edge materials. This paper delves into a novel structural concept for bus construction, incorporating carbon fiber layers and foam fillings as localized reinforcements throughout the framework. The primary objective is to explore the viability of these hybrid joints within the bus construction landscape, encompassing hybrid beams and structural nodes.[3]

Keywords: bus, hybrid, composite, foam, crash.

1 Introduction

The primary stress on the bus framework predominantly arises from fatigue and collision scenarios. While steel design offers lower raw material costs compared to composite alternatives, the manufacturing time for steel designs significantly exceeds that of composites.[1] This study hones in on crash dynamics, particularly focusing on roll-over scenarios (Ro.66), where composite materials and manufacturing techniques play a pivotal role in reducing vehicle weight for buses and trams.[2] Efforts to streamline bus construction, while maintaining or enhancing stiffness, unfold across two fronts. Firstly, establishing a numerical model, grounded in available material data, serves as the initial framework for experimental endeavors. Secondly, validating this numerical model entails cross-referencing it with empirical measurements. A prudent approach involves validating individual materials independently, followed by more complex assemblies, such as steel beams integrated with composite strips and foam. Though the core of the work revolves around a simplified beam model, the overarching findings will directly inform advancements in bus construction methodologies. More information about the details of the numerical model is provided in an older post [6] that focuses exclusively on validation using three-point bending.

2 Numerical model of a bus segment

The car body segment model was created based on CAD data provided by the manufacturer of the specific bus. As the main software the explicit solver PAM-CRASH. Similar results can be achieved with another dominant explicit solver.[4] The primary simulated part is the load-bearing structure model. Additionally, wheel models were created with a relatively coarse mesh to define the position of the platform and its pivot point. Although the wheels are not essential for the calculation and are deleted in some versions of the calculation, they are crucial for determining the initial state of the platform and creating the surface into which the car body segment impacts. The angle and speed at which the segment impacts are derived from free fall and were calculated using simple analytical relationships, so the simulation can start just before the impact.

The finite element mesh is mainly composed of quadrilateral elements (with one Gauss integration point) and only occasionally isolated degenerate triangular elements (node3 = node4). The characteristic size of the elements is 5 mm, similar to the three-point bending test simulation, ensuring compatibility of the material models used (element size dependence). The meshes are as regular as possible to prevent the occurrence of "phantom" stress waves. The detail of significant mesh areas is shown in the following images (Fig. 1,2).

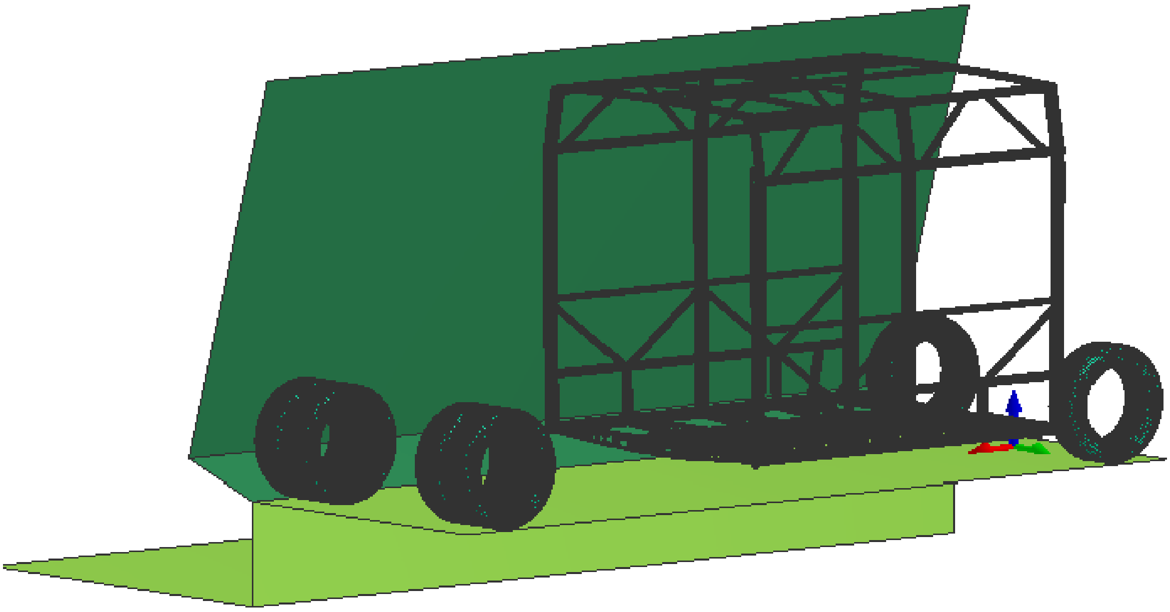

Fig. 1: The numerical model of a bus segment

Welds in the model are idealized with a rigid "Tide" connection. This simplified model does not consider weld stiffness or the possibility of its failure, but elements surrounding the joint partially compensate for this function. This idealization is sufficient for the case in question, and there is no need to use a "Plink" connection method or to model the welds with volume elements. In the model, a single contact is defined between the corner of the segment and the surface it impacts. It is a symmetric node-segment "33" type contact, where the coefficient of friction was set at 0.1[-].If necessary, the model can be supplemented with self-contact (type 36) in areas of significant plastic deformation. The floor into which the segment impacts is created with very coarse elements and serves only to define a rigid body. The center of mass of this body has all degrees of freedom removed. The mass of the segment is increased by non-structural masses. This mass represents the additional equipment of the bus that is not modeled in the simulation as its contribution to stiffness is negligible. The mass increase was made considering the mass of the entire bus and the ratio of the segment length to the length of the whole bus.

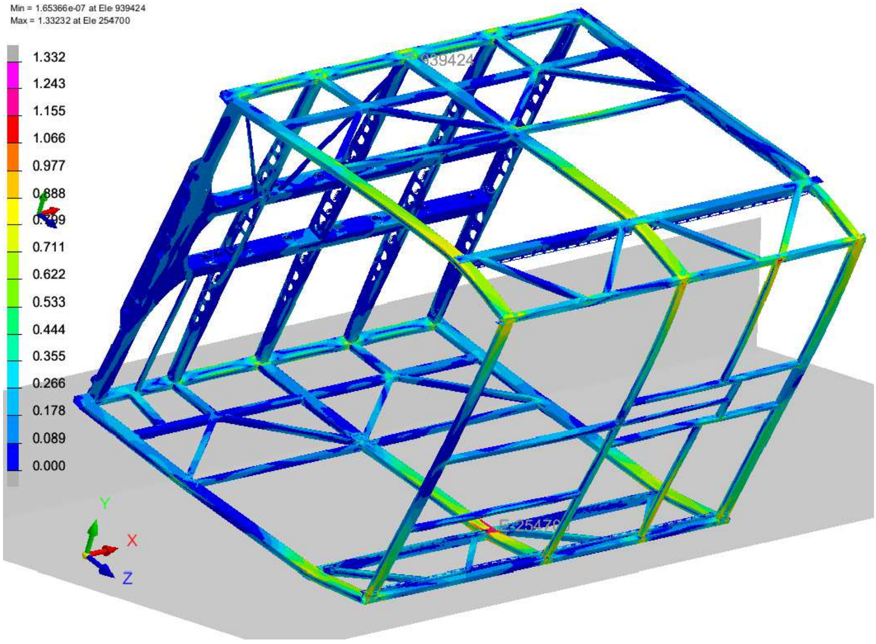

Fig. 2: The Segment during roll-over test Von Mises Stress [GPa] (80 ms)

3 Numerical model of a ring-type sample

Based on the discussion of experimental possibilities and representativeness, the manufacturer proposed a "RING" type sample. This sample represents the smallest possible section of the bus body segment and is suitable for static or quasi-static tests, the results of which will serve well for validating the numerical model. It should be noted that such a narrow segment does not meet the standard requirements, and this was anticipated in advance. On the other hand, the approval process can be supported by numerical calculation. A segment consisting of two rings (to meet the standard requirements) would pose a risk of uneven loading during the collapse of one of the beams. Therefore, such a sample may not be entirely suitable for validating the numerical model (considering the relatively unconventional reinforcement methods). A sample with at least three posts is already relatively large, and its testing would be considerably more demanding. Furthermore, as the previous numerical calculation shows, even such a part may not accurately represent the behavior of the entire bus body. An undeniable advantage of the "RING" type sample is also the possibility of conducting many more tests, which allows, for example, testing the structure with and without CFRP strips, or obtaining more detailed results for a single post (in the segment, only the average deformation characteristic for all posts would be determined).

The following image shows the geometry of the sample. The discussed reinforcements are visible, as well as the indentations on the inner part of the segment, which are (according to the manufacturer) necessary to create space for the electrical installation. Since it is not possible to show the entire mesh due to its fineness and the dimensions of the geometry, the following images show at least the details of the finite element mesh (Fig. 3). The mesh is created with similar parameters as mentioned in the previous chapter. It is a relatively regular mesh with a characteristic element size of 5 mm. The welds are idealized similarly to the previous case using a rigid "Tide" connection. It is a node-segment connection where a constant mutual distance is maintained during the calculation by a penalty algorithm. A preview of the individual welds is shown in the following image (Fig. 4).

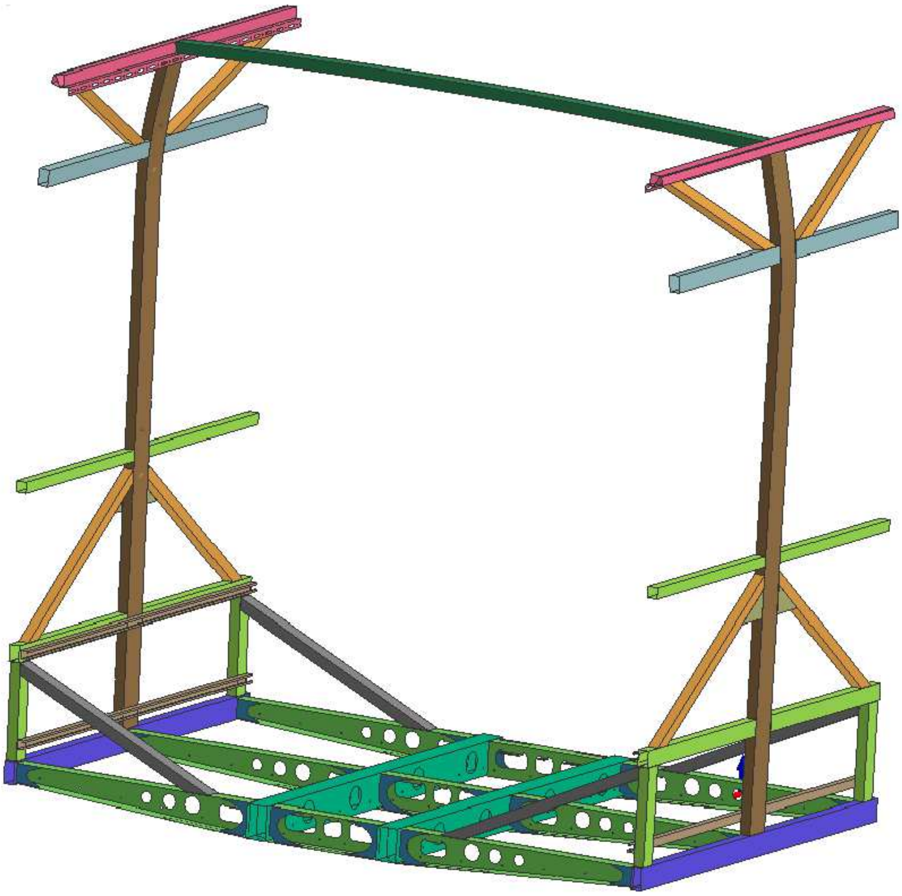

Fig. 3: The numerical model of sample named as “RING”

4 Proposal of a Test Procedure Supported by Numerical Simulation

Due to better contact conditions, the possibility of transferring forces to the structure using a generally planar element with sufficient rigidity was discussed and simulated. Loading by a plane requires significantly complicated mounting to enforce displacement, so the displacement of the sheet was gradually enforced again using a rope.

Since, unlike the real car body, the corresponding rotation of the car body (occurring during a rollover impact) cannot be simply ensured in the case of a simplified test, there is unrealistic contact of the surface with the structure in the lower part of the sheet. The structure essentially rests on the sheet and is shaped by it from the point of contact to the bottom edge of the sheet. Initially, the sheet was gradually shortened for this purpose, so that in the end, a cutout for the beam was made in it. This prevents unrealistic straightening of the beam by the sheet.

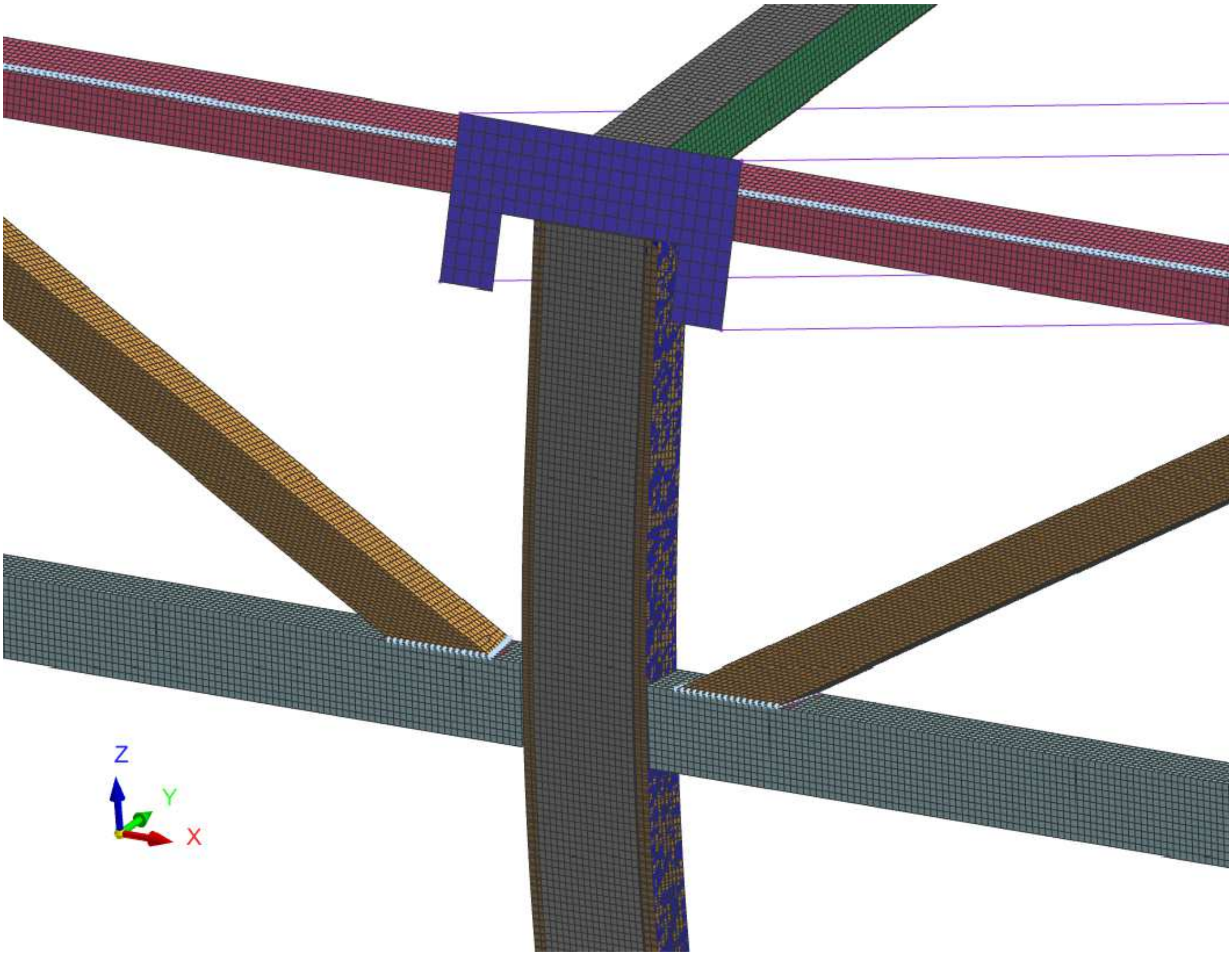

Fig. 4: The loading method of the sample demonstrates the testing procedure

5 Simulation results

The simulation results serve as important support for the experiment. Some significant results are shown in the images below. These primarily include the lateral deformation of the structure, the field of plastic stresses in the most exposed parts of the structure, and the indication of the angle by which the vertical beam bends due to plastic deformations. The image below clearly shows the overall deformation of the "ring" type sample. Two critical points are clearly visible in the overall structure where structural stability collapse occurs and where the largest local plastic deformations appear. These are the vertical main beam, which collapses at the point above the diagonal bracing, and the horizontal roof beam, which collapses at the corner of the roof structure. For better illustration, a polygon representing the crew's survival space was added in the numerical simulation. The polygon is doubled because the bus has two levels of floor height and thus two levels of passenger seats.

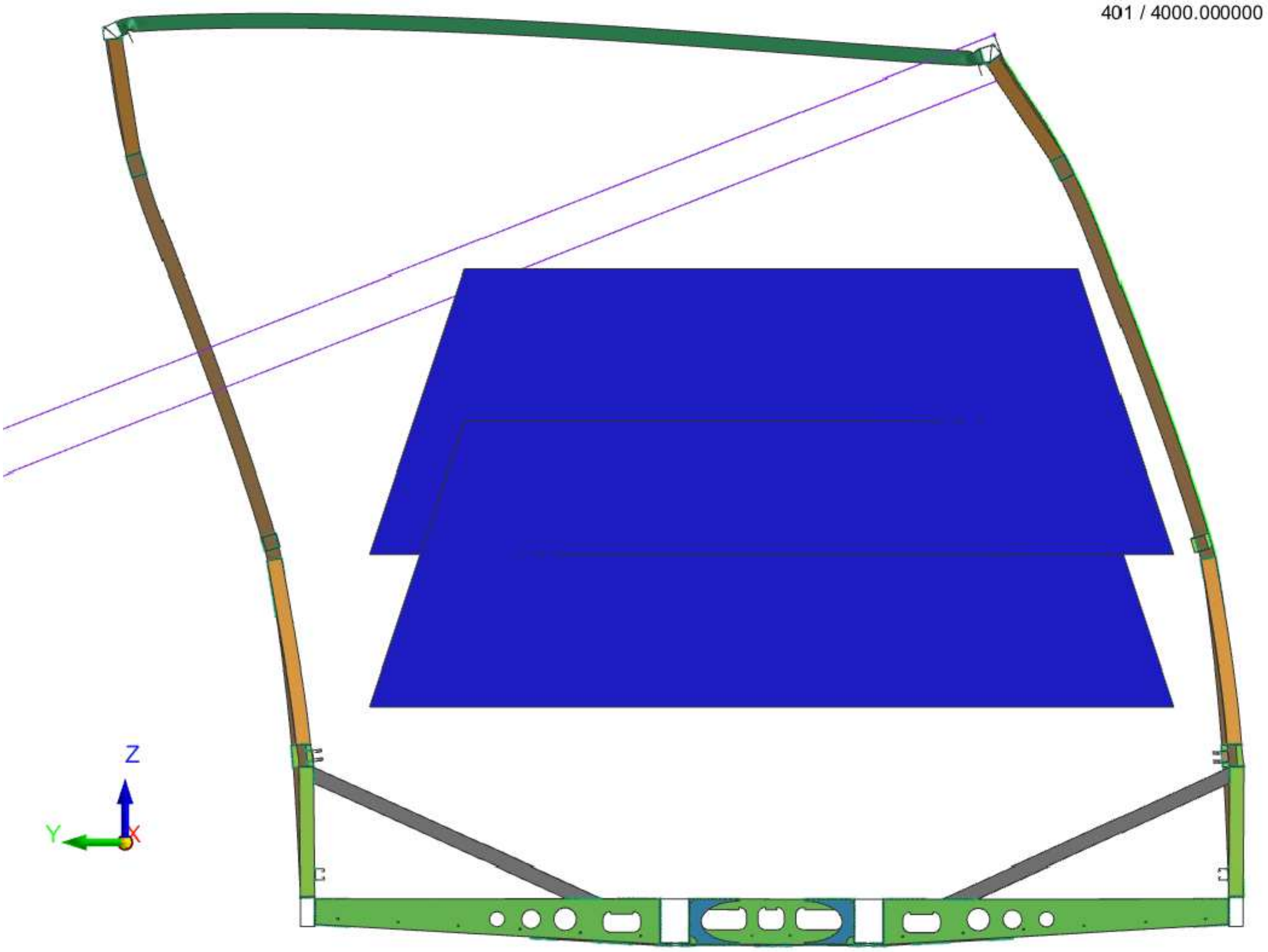

Fig. 5: The deformation of whole structure with illustrated survival space, time 4000 [ms]

The two images below very precisely illustrate the plastic deformations at the two critical points described above. In the top image, the collapse of the structure at the indentation is clearly visible. This geometric element (according to the manufacturer) is used for cable routing. It is evident that the area where the diagonal braces end will always appear as the weakest point. The behavior is similar to a cantilever beam. The indentation itself significantly influences the manner and occurrence of plastic deformations. However, during the experiment, the plastic deformations at this point were somewhat smaller. This can be explained by a certain geometric discrepancy, where the provided CAD data corresponded more to embossing, while the transverse section of the physical profile exhibited a more V-shaped curve. By partially adjusting the geometry of the numerical model, at least a somewhat better match was achieved.

In the bottom image, significant plastic deformations in the roof corner are visible. Neither during the simulation nor during the experiment did the structure fail at this point. It is clear that this part represents a "bottleneck" significantly affecting the strength of the entire structure. Unfortunately, no reinforcement is possible here due to surrounding equipment and installed cabling. CFRP strips placed on both the top and bottom sides of the roof significantly improve the overall condition. However, it is a major drawback that composite reinforcements generally drive plastic deformations increasingly into the corner, which is not desirable.

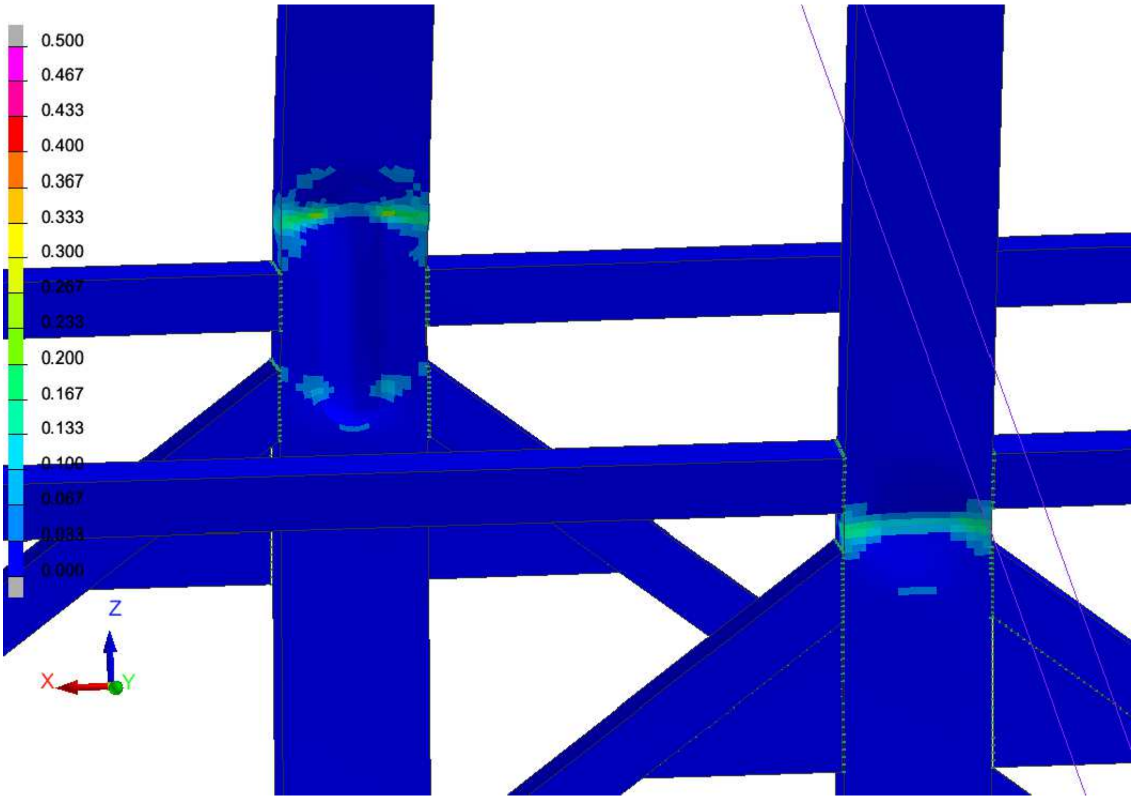

Fig. 6: Local plastic deformation of the main beam [-]

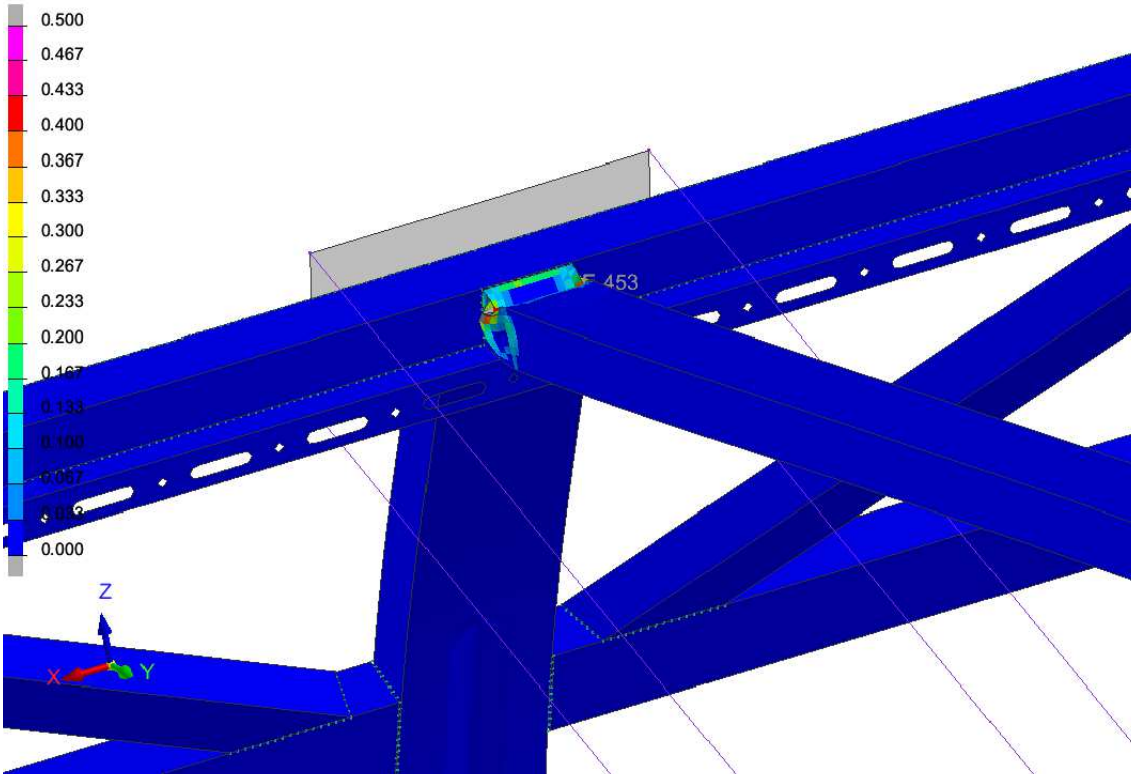

Fig. 7: The local plastic deformation in critical corner of the roof [-]

6 Results discussion

Probably the most significant results are summarized in the graph in the image below. The graph demonstrates a fairly good agreement between the results of numerical simulations and experiments. The numerical simulation achieves the most favorable result, where both the outer profiles (on the outer sides) and the horizontal profile of the roof (on the top and bottom sides) are reinforced. This was expected. A less expected result is the relatively small benefit of increasing stiffness for the foam-filled profile, both for a simple steel profile filled with foam and for a combination of reinforcement with foam and composites. A very interesting result from the numerical simulation is the finding that the reinforcement of the upper strip has a dominant influence on the roof. This reinforcement certainly significantly contributes to the overall increase in stiffness in the variant with the maximum number of reinforcements. The contribution of the reinforcement of the side profiles is very close to the contribution of the reinforcement of the roof horizontal beam. These results were achieved despite being suboptimal and without using more detailed optimization methods as described in the article [5].

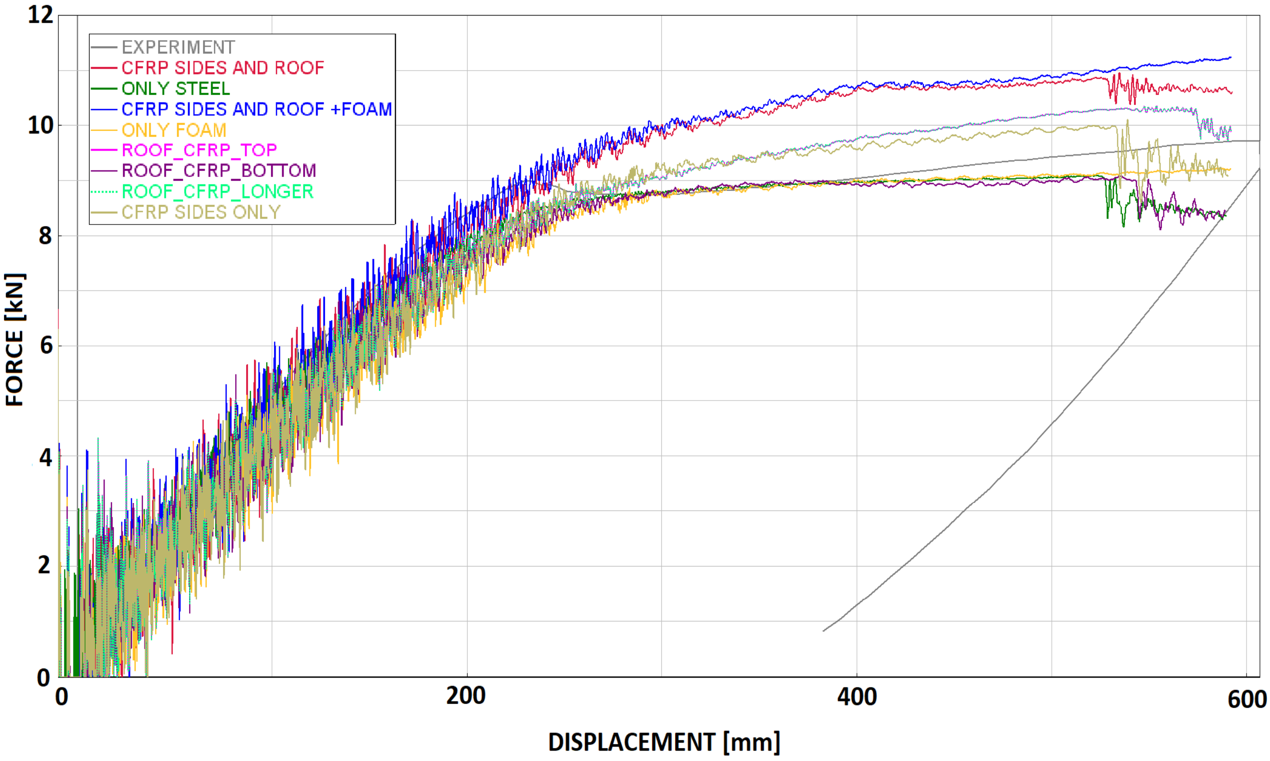

Fig. 8: The deformation characteristic of all simulated variants and its comparison with experiment

7 Conclusion

The paper briefly describes the numerical simulations created in the field of passive safety. First, the simulation of a bus segment impact on a rigid surface is presented. This involves a rollover test of the segment. A relatively low stiffness was found in the corners of the structure, which led to the modification of the "RING" type sample. The "RING" type sample represents the smallest possible section of the bus segment and is ideal for validating the numerical model. This sample was physically manufactured and is expected to be tested.

To support the conducted tests, several calculations representing different methods of substitute loading were created. Since static loading of the structure with a force at an angle (corresponding to the load during a bus rollover) is assumed, various possibilities of transferring the force to the structure were primarily addressed without artificially increasing its stiffness. Loading with an inclined plate would be relatively simple (and its simulation was very easy); however, this loading method seems to be practically difficult to implement. On the other hand, a relatively simpler method might be loading using a steel rope. Within these calculations, a variant with a 15° lateral deflection of the rope was also simulated. The results of this calculation do not indicate any danger of stability loss (unintended deflection of the structure in the direction of the vehicle's movement).

The paper also presents preliminary findings and methodologies for numerically simulating hybrid beam structures. Initial insights are gleaned from a partially validated model featuring a steel hollow profile, augmented with a composite strip and foam filling. Observations indicate that optimal performance is achieved when the composite material closely matches the Young´s modulus of steel, highlighting the importance of effective collaboration between composite and steel components. The integrity of the glue connection emerges as a critical factor in ensuring the structural robustness. Validation of material models can be augmented through three-point bending tests and their corresponding simulations. However, for comprehensive validation, it is recommended to conduct numerical simulations alongside full-scale experiments on bus segments. While such experiments are slated for the future, they promise to offer invaluable insights into the performance and behavior of the hybrid beam structure under real-world conditions.

Acknowledgement

This work has been supported by the project TH71020003 “Composite reinforcement in a light stainless steel bus structure” provided by the Ministry of Industry and Trade of Czech Republic.

References

[1] E. Larodé at al. A new concept of a bus structure made of composite materials by using continuous transversal frames. Composite Structures 32 (1995), 345-356.

[2] K. Hee-Young at al. A study on the crashworthiness and rollover characteristics of low-floor bus made of sandwich composites. Journal of Mechanical Science and Technology 23 (2009), 2686-2693.

[3] M. Kriescher at al. Possibilities for the use of metal-hybrid-structures for vehicle crash load cases. Proceedings o 1. International Conference Euro Hybrid Materials and Structures 2014, 10-11. April 2014, Stade, Germany.

[4] E. Acar, B. Yilmaz, M.A. Güler, et al. Multi-fidelity crashworthiness optimization of a bus bumper system under frontal impact. J Braz. Soc. Mech. Sci. Eng. (2020), 42, 493.

[5] P. Kunakorn-ong, K. Ruangjirakit, P. Jongpradist, S. Aimmanee, Y. Laoonual. Design and optimization of electric bus monocoque structure consisting of composite materials. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science. (2020), 234(20):4069-4086.

[6] S. Špirk, J. Václavík. New Structure Concept of the Bus Construction with the Addition Carbon Fiber Layers and Filling Foams. EAN 2021 - 59th International Scientific Conference on Experimental Stress Analysis - Book of Full Papers, pp. 218 – 221.