Integral development of an innovative concept of dump trailer body for carrying arid materials

Luis Castejon*, Jesus Cuartero, David Valladares and Hugo Malón

Department of Mechanical Engineering, EINA,
University of Zaragoza,
Maria de Luna s/n,
Zaragoza 50.018, Spain
Email: luiscast@unizar.es
Email: jcs@unizar.es
Email: valladar@unizar.es
Email: hml@unizar.es
*Corresponding author

Carlos Martin and Carlos Fabra

Lecitrailer SA,
Cmno. de los Huertos s/n,
Casetas-Zaragoza 50620, Spain
Email: martin@lecitrailer.es
Email: fabra@lecitrailer.es

Abstract: The authors develop a new concept of dump trailer vehicle designed to carry arid materials, with a series of advantages and improvements with regard to equivalent structures. For this reason, design criteria and objectives that must have been carried out in executing different stages to reach the final 'optimum' vehicle have been taken into account. From a detailed study of previous performances for this type of vehicle, it is intended to reach a series of objectives involving redesign and modification. That is the structural lightering and changes must mean the improvement of structural performances, both in terms of rigidity and endurance. The improvement of these variables will make possible the launch of a new product incorporating an improvement over current models. Moreover, the inclusion of sandwich structures in the design of the dump box's floor and door supposes a great novelty, intended to achieve significant weight saving.

Keywords: arid materials; dump trailer body; numerical methods; round box; sandwich structure; strain gages.

Reference to this paper should be made as follows: Castejon, L., Cuartero, J., Valladares, D., Malón, H., Martin, C. and Fabra, C. (xxxx) 'Integral development of an innovative concept of dump trailer body for carrying arid materials', *Int. J. Heavy Vehicle Systems*, Vol. x, No. x, pp.xxx–xxx.

Biographical notes: Luis Castejon received his degree in Mechanical Engineering from the University of Zaragoza, Spain, in 1992 and PhD in Mechanical Engineering at the University of Zaragoza, Spain, in 1998. He is a Full Professor in Transportation Engineering at the University of Zaragoza.

Since 1991, he has been involved in several projects in the areas of vehicle dynamics and new materials for vehicles, buses, railways and ground transportation. He has written over 120 papers in technical journals and international congresses and nine books, published in Europe.

Jesus Cuartero graduated in 1995 with a degree in Mechanical Engineering and in 2001 with a PhD in Mechanical Engineering, both from University of Zaragoza, Spain, where from 1997 to 2005, he worked as an Associate Professor at the Department of Mechanical Engineering, University of Zaragoza, Spain. From 1998 to 2001, he was General Manager of Sistemas y Procesos Avanzados SL, a company manufacturing vehicle components made of composite materials. He has written more than four books and 60 papers focused in transportation and composite materials.

David Valladares obtained his degree in Mechanical Engineering at the University of Zaragoza, Spain, in September 2005. From this time on, he continued his PhD studies in the area of New Materials for Road Transport and worked in research projects involving structural analysis of vehicles by the FEM, covering topics such as weight optimisation and road safety. In March 2011, he gained his PhD in Mechanical Engineering at the University of Zaragoza with the thesis: "Transversal analysis of the rollover phenomenon in heavy and large size vehicles". After that, he participated in the European Projects 'SMART' and 'OPTIBODY'.

Hugo Malón obtained his degree in Mechanical Engineering from the University of Zaragoza, Spain, in 2004 and PhD in Mechanical Engineering at the University of Zaragoza, Spain, in 2010. He is an Assistant Professor in Mechanical Engineering at the University of Zaragoza. Since 2004, he has been involved in several projects in the areas of vehicle dynamics and new materials for vehicles, buses, agricultural implements and machinery. He has written over 30 papers in technical journals and international congresses and four books.

Carlos Martin graduated in 1991 with a degree in Mechanical Engineering from the University of Zaragoza, Spain. Since 1995, he has been Operations Director in Lecitrailer, Zaragoza. From 1992 to 1995, he worked at Lecinena, a semitrailer builder, in Zaragoza, and from 1991 to 1992, he worked for Neumin, a Spanish brakes builder. He has obtained the APICS Master (American Production Inventories Construction Systems) and EWE Master (European Welding Engineer) qualifications.

Carlos Fabra graduated with a degree in Mechanical Engineering from the University of Zaragoza, Spain in 1997. Since 2004, he has worked as Production Director in Lecitrailer, Zaragoza. From 1997 to 2004, he was their Technical Director.

This paper is a revised and expanded version of a paper entitled [title] presented at [name, location and date of conference].

AQ1: If a previous version of your paper has originally been presented at a conference please complete the statement to this effect or delete if not applicable.

1 Introduction

The engineering design tools used for the development of new products have recently experienced a very positive evolution (Larrodé et al., 1996), by obtaining new optimum structures compared to those obtained with traditional methods (Ortiz Berrocal, 1980a, 1980b; Timoshenko, 1961). The results obtained by numerical simulation, according to a series of load cases and boundary conditions, along with the carrying out of trials, is an effective design method that can be applied to new product design (Larrodé et al., 1996).

In this project, the tools used prior to the case of heavy vehicles designed to carry arid materials (Li et al., 2011) (also called ‘dump trailer type’) are also applied. Firstly, in order to make this possible, it is necessary to carry out a detailed study of the structural performances of the current vehicles, paying special attention to a series of critical and decisive variables, such as structural lightering and improvement of rigid and resistant structure performances.

In order to carry out the main objectives of the project, different stages must be overcome. During the first stage, it is necessary to obtain the structural performance of the current dump trailer models. The numerical simulation software used for this task (based on the finite element methods (FEMs)) allows us to obtain the rigidity and endurance levels that the model is subjected to, according to certain manoeuvres.

According to the results obtained in this first stage, new redesigns intending to improve the structural performance according to the above mentioned parameters are proposed. Subsequently, a first prototype will be designed, whose performance must be tested and analysed.

Moreover, the inclusion of sandwich structures (Herranen et al., 2012; Zangani et al., 2007; Ingrassia et al., 2007) in the design of the dump box’s floor and door instead of steel beams and thick steel sheets supposes a great novelty, intended to achieve significant weight savings in the vehicle. Not only must provide these sandwich structures the rigidity and strength values required in the vehicle, but they must also overcome the requirements associated to debris impact.

2 General description of the structure

This vehicle, the dump trailer type and the object of analysis, basically consists of a steel box structure supported over a chassis linked to the driving cab through the front area of the king pin. The importance of this study is the optimisation of the box structure (Li et al., 2015). Figure 1 shows details of this type of vehicle:

Figure 1 Details of a heavy vehicle dumb trailer type to carry arid materials

The box is composed of two lateral plates linked to another steel plate (box floor) strengthened by two jambs and some crosspieces. In the front section, there is a closure plate and over it, a linkage to the lifting hydraulic cylinder. In the back section, there is the dump door.

The weight of the box is 25,358 kg and its volume is 20 m^3. With these data, it is considered that the maximum load carried by the structure is 28 tons, distributed over the whole surface of the inner box.

3 Definition in case of load – experimental determination of the stresses over the structure

Three situations for the box structures have been studied (Kural et al., 2014). These are:

- *'Standing-off' case*: The vehicle stops and the dump trailer is loaded, supported in the chassis and the back track master pins.
- *Initial acceleration case*: The dump trailer starts elevating because of the cylinder action, loses contact with the chassis and finally supports only in the back and the front (the cylinder) track master pins.
- *The 48° ramp case*: The structure is unloading in a position as 48° in the same conditions of the previous case.

In the three situations, there is a maximum load of 28 tons applied as a hydrostatics distribution of stresses. The back door must be closed because the material would fall out of the box.

Because of the complexity of fixing the maximum acceleration value in the 'initial acceleration case' with analytical methods, different tests have been devised with an accelerometer to obtain this value in a more direct and precise way. This value will be taken into account in later calculations.

The instruments used in this test are a piezoelectric accelerometer, data acquisition equipment, a charged battery, a laptop and acquisition and data processing software, as shown in Figure 2.

Applying the appropriate treatment to the signal registered by the accelerometer can be obtained by the temporary evolution of the tangential velocity, as represented in Figure 3.

The angular velocity with which the dump trailer spins in that precise moment has been taken from the tangential velocity, at the moment that the shank starts to go out and from the geometry, as Figure 4 shows.

According to the parameters of the dump trailers and the results obtained in the test, an initial value of initial velocity $V_i = 1.79$ m/s can be obtained, corresponding to a value of angular velocity $w = 0.2672$ rad/s. This value will be taken into account in the later numerical simulations.

Figure 2 Schema of the components used and details of the test

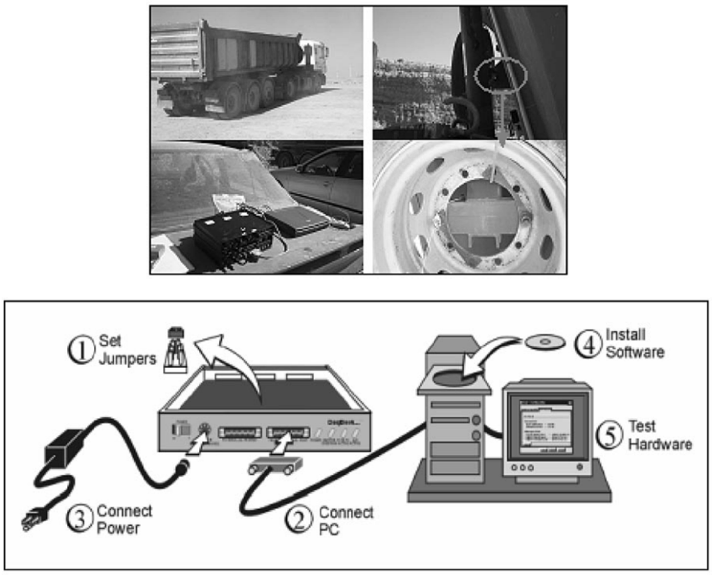


Figure 3 Results obtained in the 'initial acceleration' test

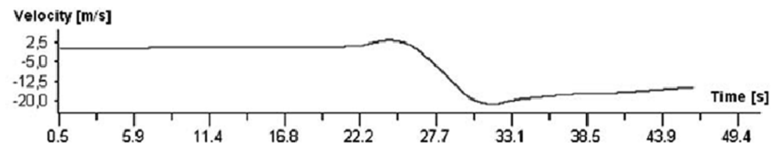
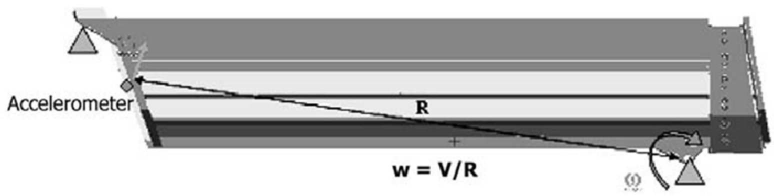


Figure 4 Obtaining the spinning of the angular velocity



4 Obtaining the numerical models of finite element

4.1 Application of the finite element method (FEM)

At this stage of the project, the analytical technique to be applied in the different numerical calculations has already been specified. In this case, the numerical analysis method used is modelled on the use of computer packages based on the FEM.

The procedure followed to achieve an accurate simulation of this method is summarised below:

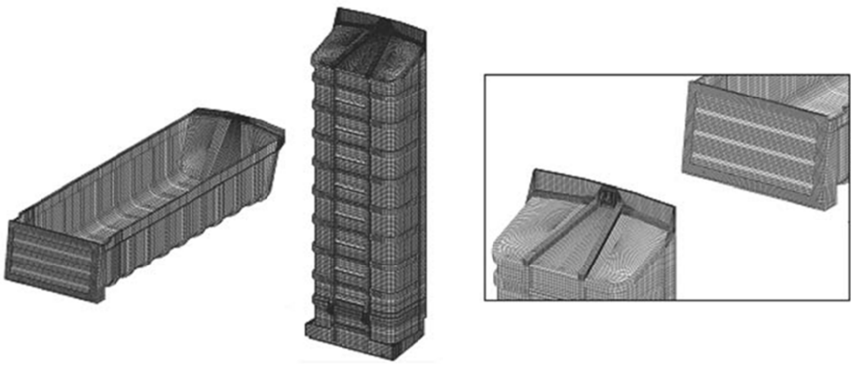
- deciding the geometry required to model the software correctly
- selecting the type of element

- decreeing the geometry in a mesh
- defining the local properties of the element
- defining the contour conditions applied to the system, and formulating the nodal forces vector {F}, as well as defining displacements, fixed nodes, contacts, charges, temperature, etc.
- resolving a group of linear algebraic equations resulting from the appropriate methods
- resolving and calculating stresses and strains.

4.2 Finite element model of the box structure

The numerical model developed for the box structure is composed of 136.000 elements, most of them the 'Shell' kind (S4R and S3R), although it has been necessary to use 'volumetric' (C3D8) elements in some components (Figure 5).

Figure 5 Finite element model in the box structure



4.3 Property assignment

After obtaining the finite element model of the box, the properties must be assigned to each component, depending on the material and the corresponding thickness in each case.

The main innovation concerning the materials used in the project is the incorporation of specific high endurance steels, which present mechanical properties four times better than the usual steels. For example, XAR 450 (used in inner plates in contact with the material) has an elastic limit of 1300 MPa and ultimate stress of 1550 MPa, opposed as St-52, with 360 MPa and 520 MPa, respectively.

4.4 Load cases and boundary conditions

Finally, the charge cases and boundary conditions corresponding to critical conditions of manoeuvrability for the dump trailer (described in Section 3) must be applied to the model. A dynamic impact calculation over the floor structure has also been carried out.

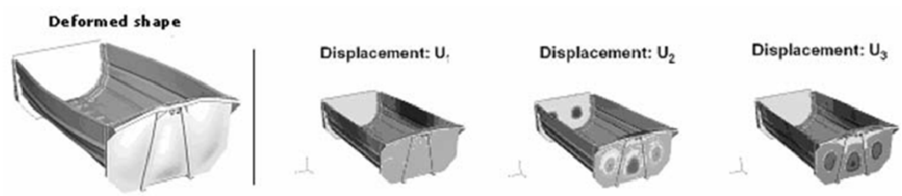
5 Preliminary analysis of the box structure

Once the finite element model has been defined, it is necessary to proceed to its analysis (simulation), after which a series of results are obtained, both in rigidity and endurance terms.

5.1 Static calculation

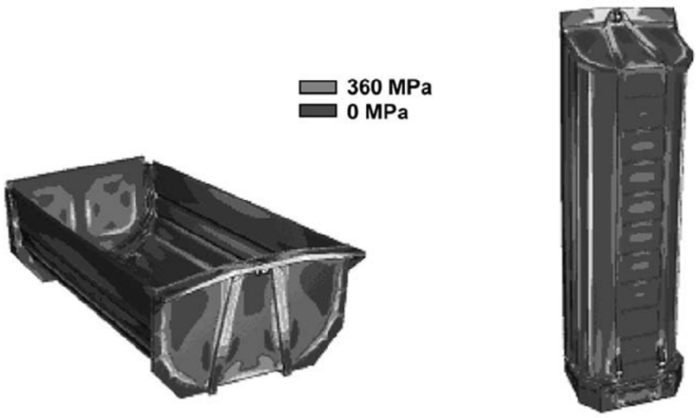
The obtained results in the simulation of the dump trailer are shown below in the ‘initial acceleration’ case because, as already been proved, this is supposed to be the most unfavourable case from a structural point of view. This has been taken as a ‘critical case’. The resulting deformed shape and the displacement in this case are shown in Figure 6.

Figure 6 Deformed shape and displacement [mm] for the ‘initial case’



The maximum transverse displacement (U_1) is 20.7 mm, located in the middle part of the lateral plates. As far as longitudinal displacements (U_2) are concerned, the approximate results are 5.7 mm, in the back part and in the closure front plate. The maximum vertical displacement is located in the floor plate of the dump, with approximate values of 6.5 mm. Figure 7 shows the Von Mises equivalent stress map [MPa], obtained in the simulation for this load case.

Figure 7 Von Mises equivalent stress [MPa] for the ‘initial acceleration’ case

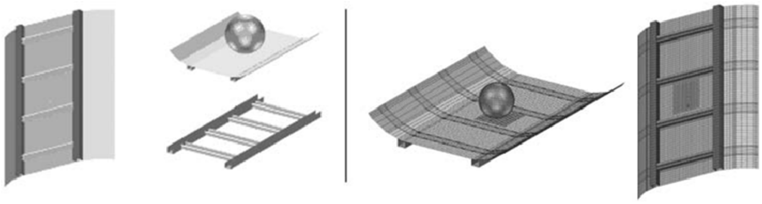


In this case, the maximum stress value is 360 MPa. The front part of the dump trailer (front shield) and the turning back section are the parts of the box, which present the more unfavourable performance, from an endurance point of view. In any of the cases, it exceeded the elastic limit of the material.

5.2 Dynamic impact calculation over the floor plate

The floor plate is one of the more critical substructures from the required design point of view. It is therefore necessary to carry out a particular analysis of it, above all an impact analysis (Kural et al., 2014), for different combinations of materials and thicknesses (Figure 8).

Figure 8 Finite element components and models taken into consideration for the impact analysis



These simulations have been carried out by using explicit calculations in the following charge cases:

- generic impact case: a rigid sphere of 60 kg thrown from 1.5 m with no initial velocity
- rubble impact case (most unfavourable case): a rigid sphere of 500 kg thrown from 2 m with no initial velocity.

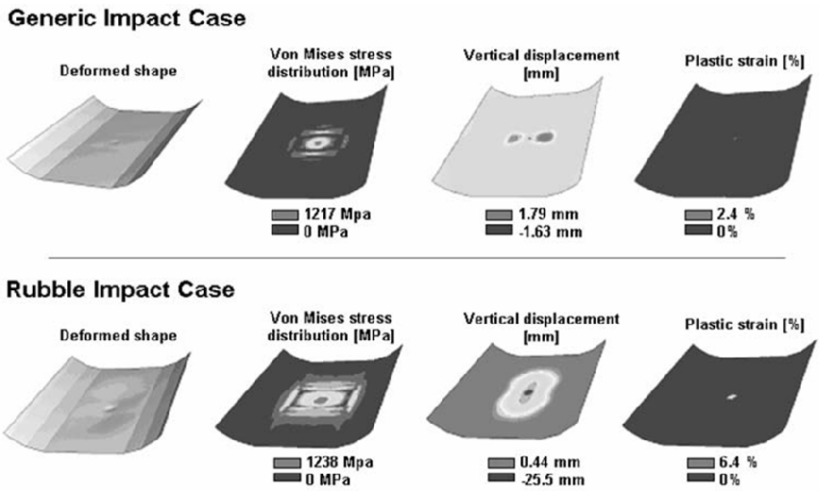
Eight different calculations have been made, plates of 6, 5, 4 and 3 mm thickness with Hardox 500 and XAR 450 steels. The results for a generic configuration can be seen in Figure 9 and Table 1.

For both steels, and for an impact of 60 kg mass, the tendency is the following: if the thickness of the plate is reduced, the value of the plastic strain diminishes. However, in the 'rubble case', the level of maximum plastic strain remains constant. Also, when reducing the thickness, the vertical displacement is increased, but in the case of 3 mm and 4 mm thickness, it is reduced a little when the material used is Hardox 500.

Table 1 Summary of the impact results obtained for different configurations

		XAR 450 steel			*HARDOX 500 steel*		
Thickness	*Weight*	U_3 *max* (mm)	*Von Mises stress* (MPa)	*Plastic strain* (%)	U_3 *max* (mm)	*Von Mises stress* (MPa)	*Plastic strain* (%)
e = 6 mm	m = 60 kg	1.8	1217	5.2	1.9	1325	4.8
	m = 500 kg	25.5	1243	6.4	24.8	1368	6.1
e = 5 mm	m = 60 kg	3.1	1218	5.3	3	1326	5
	m = 500 kg	26.2	1244	6.5	25.5	1365	6
e = 4 mm	m = 60 kg	6.2	1228	4.4	5.8	1344	4.2
	m = 500 kg	30.5	1243	6.6	29.8	1367	6.2
e = 3 mm	m = 60 kg	9.5	1222	3.8	9.3	1335	3.5
	m = 500 kg	34.3	1243	6.5	33.3	1367	6.1

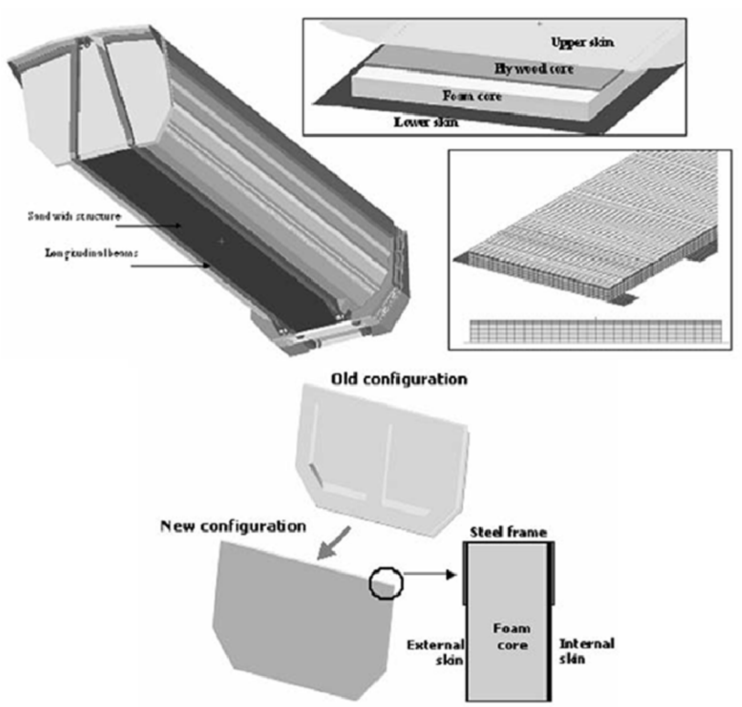
Figure 9 Comparison of the obtained results for both impact cases



6 Redesign and optimisation of bodywork components

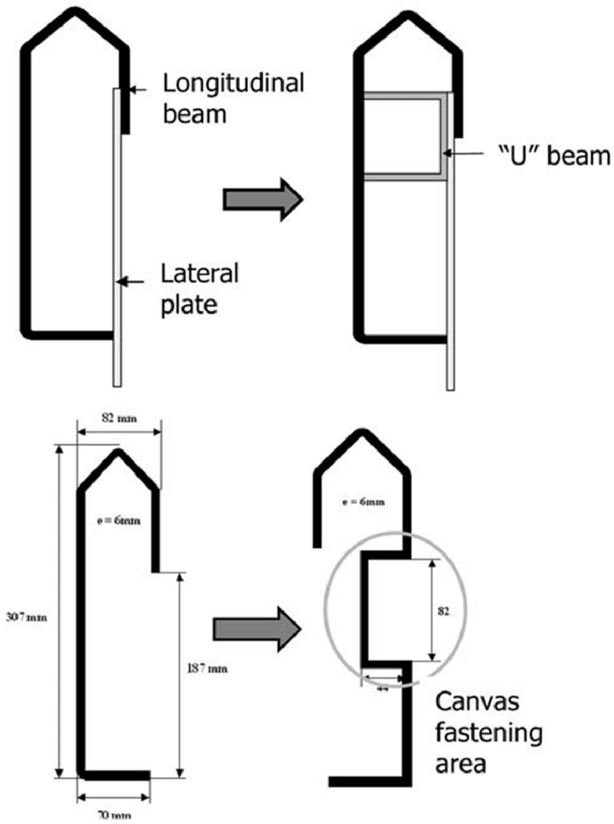
In order to reduce the vehicle tare and to improve its rigidity in flexion, certain sandwich-type structures have been developed for the floor and for the design of a new door, as shown in Figure 10.

Figure 10 New configurations for the floor (top) and door (bottom)



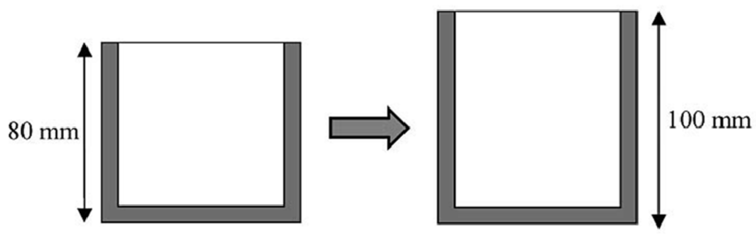
In the first case, the tare can be reduced to 8.3% (211 kg) and the redesign of the door reduces to another 4% (108 kg). The other two redesigns affect the longitudinal beam: the first is intended to stiffen across the vehicle in flexion and the second redesign allows the canvas to fasten to it, see Figure 11.

Figure 11 Stiffen of the beam (left) and canvas fastening (right)



In order to diminish the stress state of the front crossbar, next to the plastic load in basculation manoeuvres, the height of the profile has been modified, increasing the tare 0.2%, as shown in Figure 12.

Figure 12 Increase of the profile height in the front crossbar



Finally, the front rib of the vehicle has been suppressed. In Table 2, the weight variations linked to the most important modifications proposed can be seen.

Table 2 Summary of the different modifications proposed

Modification	*Original weight* (kg)	*Modified weight* (kg)	*Weight variation* (kg)	*Percentage* (%)
Sandwich floor	2.538	2.327	–211	–8.3
Sandwich door	2.538	2.430	–108	–4
Canvas fastening	2.538	2.580	+42	+1.7
'U' beam insertion	2.538	2.559	+21	+0.8

7 Production of a lighter prototype

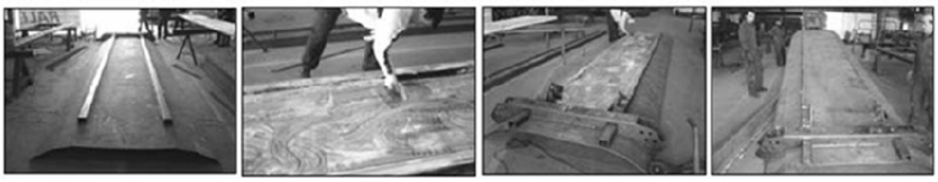
From the obtained results in the different redesigns, a prototype has been developed. This prototype incorporates the following new features: a sandwich-type floor structure, transversal stiffening by increasing the profile of the longitudinal beam and the soldering of a U-profile to this, and an increase in height in the front UPFs.

The fabrication of the sandwich-type has been carried out by assembling the different components with the 'vacuum bag' technique, as can be seen in Figure 13.

Figure 13 Fabrication process of the sandwich-type

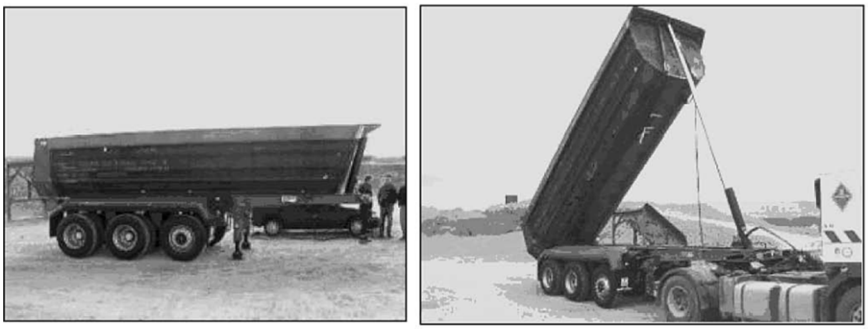
Later, the sandwich-type was incorporated into the floor plate using a structural adhesive. The whole (floor plate + sandwich-type) is incorporated in the next stage into the prototype, as can be seen in Figure 14.

Figure 14 Assembling process of the panel over the floor plate



Once the structural adhesive process is finished and the other modifications have been introduced, the new floor design is welded to the rest of the structure. The prototype is completely finished (Figure 15).

Figure 15 Finished and assembled dump trailer prototype



8 Carrying out tests in the prototype

As the prototype has been completely finished according to the previous directives, the new structure must be tested to check for performance, thus a strain test was carried out (Figure 16). Specifically, the deformations in some critical areas have been measured, like those which were taken into consideration in the numerical simulation of previous stages.

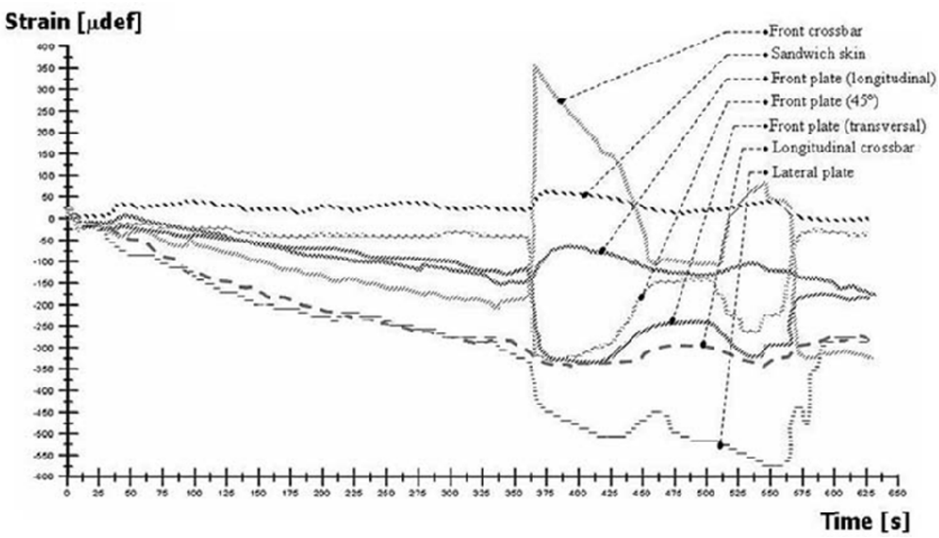
Figure 16 Strain tests in dump trailers

Table 3 Summary of the results obtained in the test

Test point	*Gage direction*	*Strain value (microdeformation)*	*Stress value (MPa)*
Lateral plate	Longitudinal	60	12
Longitudinal crossbar	Longitudinal	570	120
Lower skin of floor sandwich	Longitudinal	350	75
Front transversal crossbar	Vertical	340	72
Front upper plate	Longitudinal	160	33
	Transversal	350	75
	45°	330	70

Each channel in the following graph registers the time evolution of the deformations in the gauge placement point that can be graphed by the corresponding acquisition and data processing software. After this process, the curves (representing the time evolution of the strain values obtained in each line channel) can be seen (Figure 17).

Figure 17 Time evolution of the obtained strain in each measured point



In Table 3, the results of strain and stress in each point and measure direction can be seen.

With these results, the improvement of the redesigned elements can be checked.

This is the case with the front drag strut. The new design presents maximum stress values of 350 micro-deformations in opposition with 735 micro-deformations of the old design.

9 Conclusion

A vehicle of the 'round box' type has been developed and improved with regard to the two previous models. That means a technological innovation in the field of heavy vehicles for carrying arid materials, because of the incorporation of steels with specific high endurance and sandwich-type configurations in the vehicle structure. Tools of MEF-based numerical simulations and experimental techniques have been combined to achieve this concept.

This new vehicle incorporates sandwich structures in its box's floor and door, which allowed a total weight saving of 300 kg, at the same time that structural rigidity and strength requirements as well as those associated to debris impacts are fulfilled.

Strength and rigidity levels have been taken into consideration when proposing modifications for this new design. In addition, the assembly of the new components must be easy and viable, and the materials should be available in the appropriate formats and measures. Finally, the production process must be viable from a technical and economic point of view. It has to be noted that a prototype incorporating the proposed design modifications was manufactured and its performance was tested during a reasonable time period. Nowadays, this new vehicle has already been launched onto the market.

Acknowledgement

To the Government of Aragon and European Funds FEDER, for funding this project.

References

Herranen, H., Pabut, O., Eerme, M., Majak, J., Pohlak, M., Kers, J., Saarna, M., Allikas, G. and Aruniit, A. (2011) 'Design and testing of sandwich structures with different core materials', *Conference: 20th International Baltic Conference on Materials Engineering*, Kaunas, Lithuania, Vol. 18, No. 1, 27–28 October, pp.45–50.

Ingrassia, T., Alaimo, G., Cappello, F., Mancuso, A. and Nigrelli, V. (2007) 'A new design approach to the use of composite materials for heavy transport vehicles', *International Journal of Vehicle Design*, Vol. 44, Nos. 3–4, pp.311–325.

Kural, K., Voskuijl, M., Fengnian, T. and Pauwelussen, J. (2014) 'Determination of representative loading conditions for effective semitrailer design', *Transport*, Vol. 29, No. 4, October, pp.363–375.

Larrodé, E., Miravete, A. and Castejón, L. (1996) 'Analysis of large structures for ground transportation applications', *29th ISATA International Symposium on Automotive Technology and Automation*, 3–6 June, Florence, Italy, Road and Vehicle Safety, pp.345–351.

Larrodé, E., Miravete, A. and Castejón, L. (1996) 'Design and optimisation of a lightweight frigorific semitrailer', *International Journal of Vehicle Design. Heavy Vehicles Systems*, Vol. 5, No. 1, pp.45–67.

Li, H.G., Si, J.P., Han, L. and Zhang, B.W. (2011) 'Finite element analysis of heavy-duty dump truck subframe based on ANSYS', *Advanced Manufacturing Systems, PTS 1-3. Book Series: Advanced Materials Research*, Vols. 201–203, pp.518–523. Part: 1–3. DOI: 10. 4028/www.scientific.net/AMR. 201-203. 518, Published: 2011, *2nd International Conference on Manufacturing Science and Engineering*, 9–11 April, Guilin, China.

Li, W.P., Tao, J.J. and Liu, M.Y. (2015) 'Nonlinear analysis and uncertainty optimisation of mining dump truck's ROPS based on TPS-HDMR', *International Journal of Heavy Vehicle Systems*, Vol. 22, No. 1, pp.73–92.

Ortiz Berrocal, L. (1980a) *Elasticidad, Madrid: Escuela Técnica Superior de Ingenieros Industriales*.

Ortiz Berrocal, L. (1980b) *Resistencia de Materiales, Madrid: Escuela Técnica Superior de Ingenieros Industriales*.

Timoshenko, S.P. (1961) *Theory of Elastic Stability*, 2nd ed., McGraw-Hill, New York.

Zangani, D., Robinson, M., Gibson, A.G., Torre, L. and Holmberg, J.A. (2007) 'Numerical simulation of bending and failure behavior of z-core sandwich panels', *Plastics Rubber and Composites*, Vol. 36, No. 9, pp.389–395.