Proceedings of 7th Transport Research Arena TRA 2018, April 16-19, 2018, Vienna, Austria

Verification and Validation of Static Structural and Explicit Dynamic Simulations Respectively for Plastic Products with Especial Care for Automotive Application

Boglárka Zentai[a] and Árpád Veress[b]

[a] *Trainee, Knorr-Bremse R&D Center Budapest, Engineering Calculations Group, Hungary*
[a]*Associate Professor, Department of Aeronautics, Naval Architecture and Railway Vehicles, Budapest University of Technology and Economics, Hungary*

Abstract

Nowadays, the finite element analyses provide beneficial support in many fields of engineering by means of their cost, time and capacity saving characteristics. During the analyses, the one of the key elements is the correctly defined material properties. This is especially true for the plastics due to their orthotropic and nonlinear material properties as plasticity, relaxation or creep for example. Additionally, the applications of the plastics are widely spread in the automotive engineering beside the other sectors of the industry. Hence, the main goal of the present research is to verify static structural and to validate impact dynamic simulation results by the available material data and actually performed measurement results respectively. Tensile test and creep simulations were carried out in Ansys Workbench software for comparing the static structural simulation results with the available material data given by the material manufacturer in case of the used test samples, boundary conditions and material properties. Following the verification, the material models were applied for a specimen and for a model of an existing product in order to complete validation of the explicit dynamic simulation's results. Real vertical hammer tests were set up and performed for this purpose and the same procedure was simulated also in order to be able to compare the results and drawn conclusions about the accuracy of the calculation method.

Keywords: finite element analysis, material model, plastic, explicit dynamics, verification and validation

1. Introduction

Nowadays vehicle manufacturers have to meet more and more demands. One of them is the rigorous environmental laws, but on the other hand higher comfort levels are required by the passengers. That's why weight reduction is a more important perspective in automobile industry. Namely if the weight of the car is less, the manufacturers can reduce of the pollutant emissions. In case of trucks, these characteristics are more essential, considering the fact that these vehicles roll hundreds of thousands of kilometres. The one of the solutions can be found in changing material application. If it is possible, metal parts should be replaced with more lightweight plastic parts. This is supported by an interesting data also published by Lukács that about 7.5 l fuel can be saved with the utilization of 1 kg plastic material over the entire live time of a truck. Considering the whole amount of plastic in a truck (in 2005 this value was 143 kg in an average passenger car), the consumption reduction is more significant.

Beside the relatively low mass, the fiber reinforced plastics have excellent properties, such as high energy absorbing and high thermal resistance. In contrary, parts made of these materials are more sensitive to the environmental conditions (especially for temperature, vapour and radiation) and can suffer from creep, which are definitively needed to be considered and investigated. However, the testing of the different products made of various type of plastics would be very costly. For this problem finite elements analysis can provide solution, because high number of sampling phases can be replaced with creating and the investigating virtual prototypes. With this method significant amount of time, cost, and capacity saving is reachable. But the accuracy of these analyses is indispensable. The average deviation between the measured and the calculated values should not exceed the 5-6 %. This way of the product development can be reliable and so widespread not only in the automotive industry but in the other segments of the economy too.

Short glass fiber reinforced polyamide was examined with multiple virtual testing methods in the recent research. During simulations, the presence of plastic material can cause some challenges due to their nonlinearities and sensitivities. The characteristics of plastics - such as plasticity, relaxation or creep in the function of temperature and humidity - are more complicated to take into account during the finite elements analysis with respect to linear or bi-linear approaches and so the correctly defined material model is one of the key elements.

The main goal of the present work is to verify static structural and to validate impact dynamic simulation results in Ansys environment. For this purpose, available material data and actually performed measurement results were used. The implemented material tests can be divided into two groups. Tensile test and creep simulations were carried out in Ansys static structural module in the verification group. These results were compared with the available material data given by the material manufacturer. The test setups were configured according to the given standards. After the verification, the next goal was to validate the finite element simulation results with the used material model via transient collision simulations. Charpy's impact test of a probe and an existing product's vertical hammer real tests were set up and performed. The same tests were simulated in the explicit module of the Ansys to compare the results.

1.1. Description of the used approaches

The beneficial effect of the physical modelling and simulations has been proven in many different areas. They can be an effective tool in the industrial struggles or in research and development as well. Depending on the application fields, many researches have been published already, (e.g.: Hargitai, Beneda, Rohács et al. and Bera et. al.), which demonstrate the utilization of these approaches in the vehicle engineering.

In the application area of the explicit simulation, there are many outstanding researches; several of them achieved promising results. At ANSYS Conference the same tensile test was investigated to compare different material models by Schmailzl. In this study the deviation was varied between 3 and 10.2 % depending on the material model. In another conference paper two types of material model (MISO and BISO) were tested with experimental data from a tension test by Gaertner et al. Here, good correlation between the numerical and experimental results was found (differences ware from 12 to 16 %) by the authors.

Out of these researches, there are many examples are available in the scientific literature in relation with dynamic behaviour too. As regard of the explicit simulations, several different impact tests were implemented with finite elements analysis. In automotive industry one of the most often tested part in this concern, is the car bumper. The behaviour of this object was simulated by Liu and Uddandapu too. In the Liu's article, experimental data of the impact tests were collected, and then the same tests were carried out in Ansys. The maximum of the found deviation was 5.9 %. In the second article the aim was to study the structure and material which was fiber reinforced poly-ether-imide and acrylonitrile butadiene styrene, employed for a bumper. The material model was the subject of another project by Behrens et al. In this article material modelling of the short fiber reinforced thermoplastic sheets was presented in particular the orientation of the fibers. Four different fiber reinforced composite plates was studied after being impacted by a standard drop weight with different impact energies and moments. After comparison of the experimental and the test results it was declared, that Ansys was able to predict the threshold of damage but not the damage zone shapes.

1.2. Method and material

Two different simulation methods were used in the present research. Both of them, the implicit and the explicit ones refer to different time integration methods, by which static, quasi-static and dynamic simulations can be solved respectively. The application area of these methods aren't the same, the main difference between them is in the length of the tested event. The implicit schema can be applied with greater accuracy and short simulation time for rather static or quasi-static problems and creep. In the other hand, the explicit method is suitable for short time transient processes (less than or equal with 1-2 s) as collision, drop or ballistic tests, detonation and hypervelocity impacts. To simulate creep or low speed problem (tensile test) implicit method, but in case of vertical hammer or Charpy's test explicit method was used.

1.2.1. Implicit time integration (ANSYS Training manual)

$$m\ddot{x} + c\dot{x} + kx = F(t) \tag{1}$$

The main equation in dynamics (1) represents a set of "static" equilibrium equations at any time *t*. Ansys solver uses Newmark or HHT (Hilber-Hughes-Taylor) solution method at discrete time points, where the time increment between the successive time points is called the integration time step. For linear problems, the implicit time integration is unconditionally stable for certain integration parameters. The time step size will vary to satisfy accuracy requirements. In case of the non-linear problem the solution is obtained by using a series of linear approximations by Newton-Raphson method. Iterations might be needed to achieve convergence in each time step depends on the complexity of the problem. The solution requires inversion of the nonlinear dynamic equivalent tangent stiffness matrix. Small, iterative time steps may be required to achieve the convergence and accuracy.

1.2.2. Explicit time integration (ANSYS Inc. Mechanical APDL Theory Reference)

The Explicit module of Ansys is suited for nonlinear problems. Contrary to the implicit method, in the explicit module the equations (1) are solved directly. In explicit simulations equations are calculated for each time step, and input values are based on the value of the previous step's end. With this method the inversion of the tangent stiffness matrix can be skipped out but the mass and damping matrices cannot be neglected. Here, there is no need to check the convergence since the equations are uncoupled. The convergence is guaranteed in almost every case, but the solution can be unstable. The time step size is limited according to Courant-Friedrichs-Lewy condition, to assure the stability. This is the explanation why the stable time step and the element's length have a direct relationship. Central difference time integration scheme is used by the solver. Fig. 1. shows the solution cycle with the successive steps according to the ANSYS Inc. Mechanical APDL Theory Reference. To solve a problem mesh had to be generated, then material model, loads constrains and initial conditions had to be adjust. In this cycle, the motion of the mesh nodes is produced by integration in time. After the deformation of the elements caused by this motion, in each element change in volume and density of the material befall. In the next step deformation rate is used to derive strain rates from which physical laws derive stresses. Nodal forces are calculated from these stresses. External nodal forces can be computed from boundary conditions, loads and contacts. Nodal accelerations are calculated from nodal forces by dividing total nodal force with mass. New nodal forces can be specified with the explicit integration in time of the total nodal forces. Then the explicit integration in time of accelerations produces new nodal velocities. Finally, new nodal positions is determined by a last integration of the velocities. And the same cycle will be repeated, if the end time isn't reached.

Despite of the useful quality of explicit scheme, the time increments set the limit of the utilization. If the increments are small enough, the results will be accurate, but appropriate accuracy requires very small increments, and already a few seconds test will take long time.

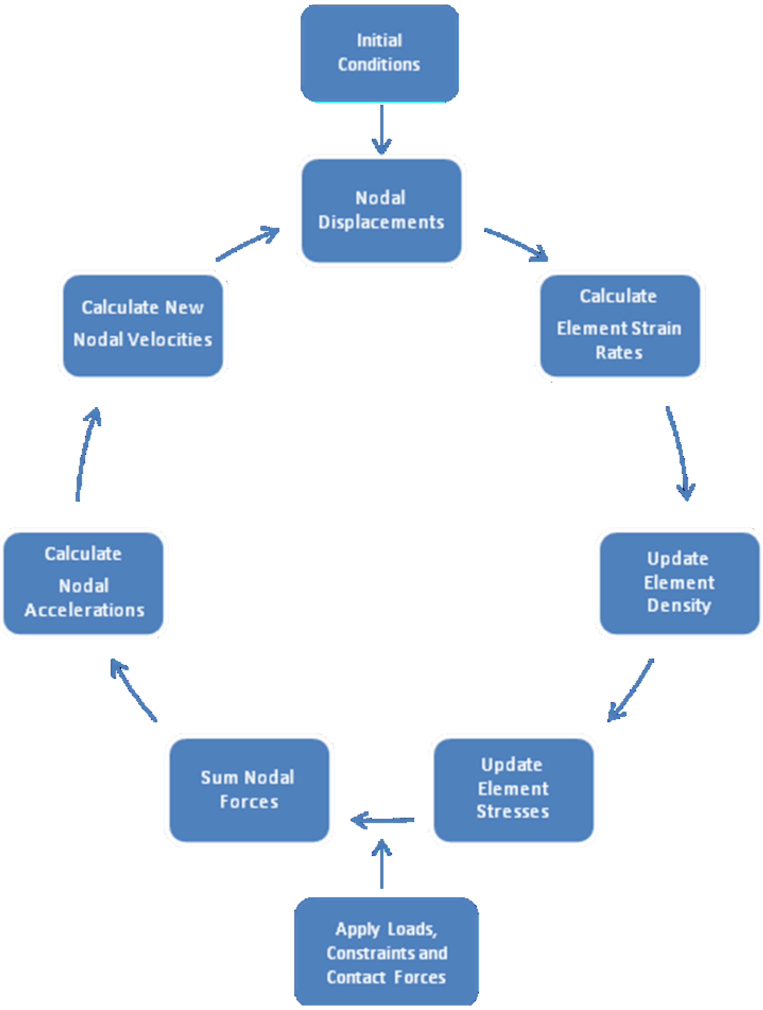

Fig. 1 Explicit Solution Cycle (ANSYS Training manual)

1.2.3. Material

In the tests, the probes and the real product was made from polyamide with 30% short glass fibers reinforced. These fibers increase the strength of the material, the effect of fiber's quantity was investigated by A. Güllü et al. In this present work, our goal was also that the material model have to be not just accurate but simple, so the orientation of the fibers wasn't examined.

To define a material model one of the most common way is to give stress-strain pairs. Since the stress-strain curve of this material isn't linear, the model has two part, a linear and nonlinear, these sections were approximated with LISO (Linear Isotropic Elasticity) and MISO (Multilinear Isotropic Hardening) models. During the model making, the key element is the correction of the engineering data which are the result of a tensile test. Another important part of the model, which used only during creep simulations, is the definition of creep-strain and time points at the given temperature. In case of the explicit tests, a failure criterion was applied like in the research of X. Li et al.

2. Implicit tests

For the tensile test, the test specimen was created according to the DIN EN ISO 1874 and DIN EN ISO 527 standard by Ansys Design Modeler. Separate surfaces were situated at the end of the specimen for the fixation. This test was executed also by Schmailzl, A and A. Arriaga too. Fig 2. illustrates the results of the simulations, the colors changes according to different equivalent stresses. The values of the tensile test are shown in the Figure 3. Next to the normalized simulated results, the normalized data from the material's manufacturer are presented too. The highest deviation between the stress values is 2.44%.

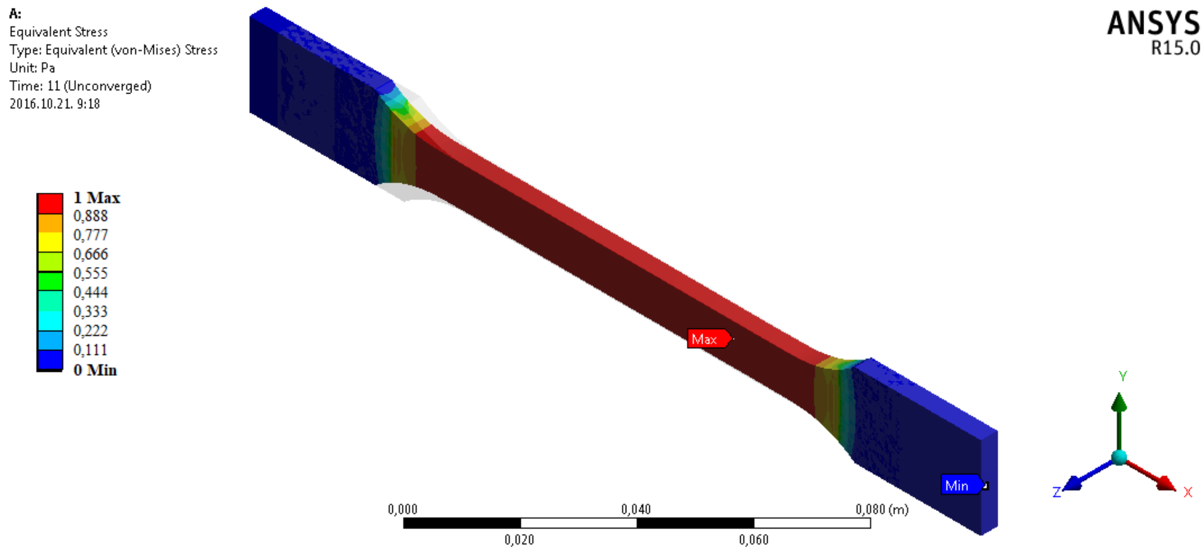

Fig.2 Normalised equivalent stresses on the specimen at the tensile test

Fig. 3 Normalised stress-strain curve of the specimen at tensile test

As regard the creep test, Figure 4. illustrates the normalized creep strain values. Here the difference between the computed and available data in the specification is higher (~12%), but at the end of the test time, it reduced under 4%. The concrete analysis was different in two points from the tensile test. The material model was completed with a creep model (Generalized Time Hardening), on the other hand the test time was longer and thanks to this, more separate load steps were specified. The creation of the correct creep model can be a difficult question, because in most of the cases, the curve fitting procedure requires long time. So the accuracy of the creep model should depend on the investigated problem.

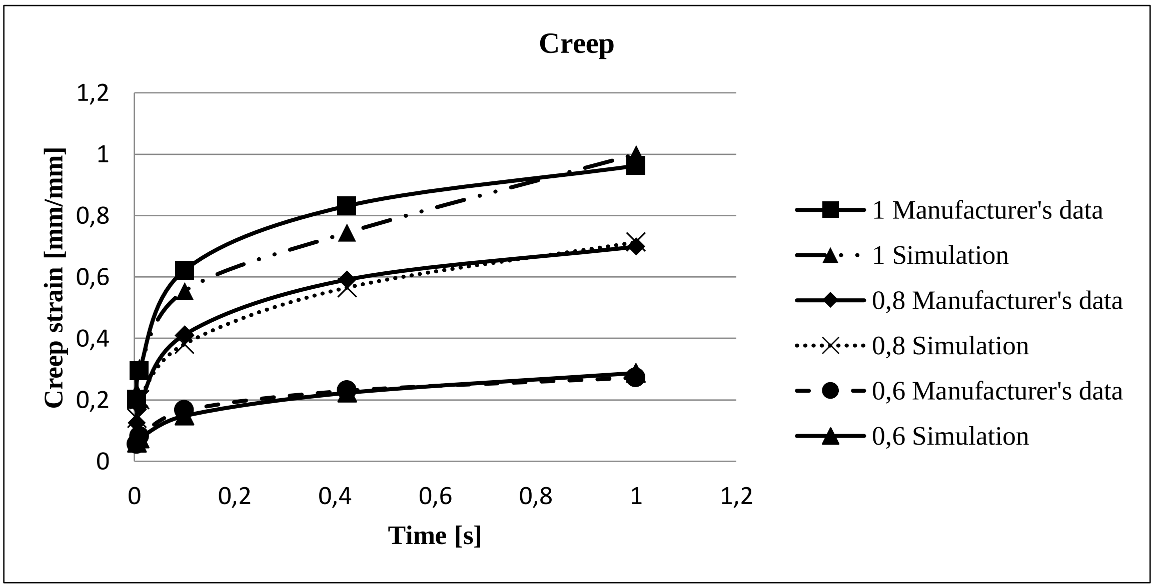

Fig. 4 Normalised creep strain curve in the function of stress levels at given temperature and moisture content

3. Explicit tests

The Charpy's impact test was performed in the reality and in Ansys Explicit module too, and three values, manufacturer's data, own measured data and simulation results were compared. Concerning the Charpy's impact test, the test specimen was created according to the DIN EN ISO 179 and DIN EN ISO 294 standards in the measurement and in the simulation (as it was relevant) together with the definition of the hammer. The size of the specimen and the geometry of the hammer were the same in the manufacturer test, measurement and simulation versions. There was only one difference in the radius of the specimen's V notch. In case of the measurement the radius was 1 mm counter to the 0.25 mm radius, which was used by the manufacturer test. The reason of the higher value in the own measurements is to make the geometry less rigid with expecting higher impact strength. An important part of the simulation is the failure criteria, which was the maximum value of the plastic strain. The results of the analysis are found in Table 1. and Fig. 5-6 represent the fracture of the test specimen. In the simulations, two conditions were examined, when the material is dry and conditioned. The maximum difference between the measured and the virtual results is 5.4 %, the deviation between the measured and the manufacturer's data is 13.5 %. This relatively high value is due to the case that the geometry was less rigid in case of measurement due to the higher notch radius and the material producers provide the lowest value of the impact strength in general.

Table 1. Charpy's test results

	Manufacturer' data		Measurement		Simulation	
Specimen's type	1eA		1eB		1eA	
Condition	Dry	Conditioned	Dry	Conditioned	Dry	Conditioned
Normalized impact strength	0.476	0.865	-	1	0.458	0.839

Finally a vertical rod (hammer) test was carried out on a real product, on a park shunt valve by means of measurement and simulation. With this examination, the actual benefit of a finite element simulation can clearly demonstrate. Since this part is placed on the outside of the trailer, it must resist the smash of stones. A special, unique measuring equipment has been used to perform a test. The test was carried out several positions; two of them were selected and repeated in Ansys. Only qualitative evaluation was available by this technique, quantitative results were not determined and compared.

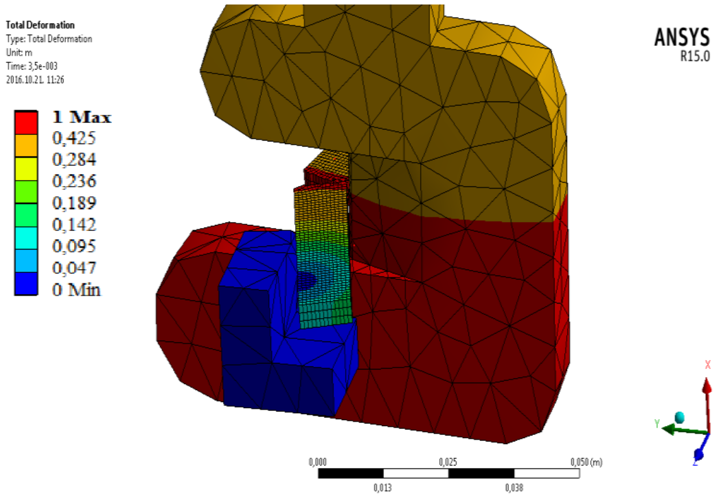

Fig. 5. Simulation results of the Charpy's test with the normalized total deformation values

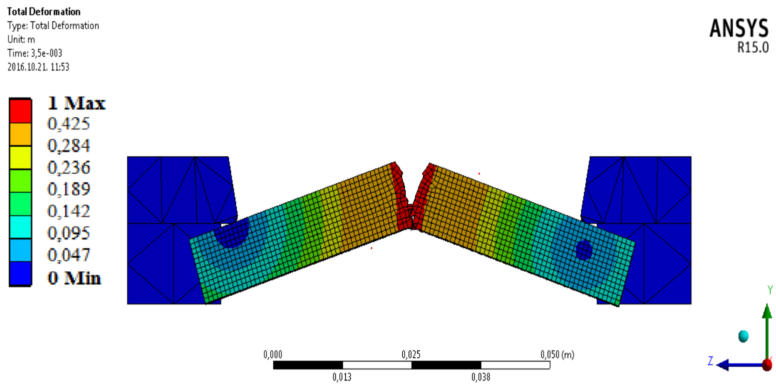

Fig. 6. Simulation results of the Charpy's test. The broken specimen with the normalized total deformations.

Two models were created and complemented with two kinds of failure criteria. These results were compared with the real test results. In the first failure criteria (case a,), the geometric strain limit was defined, and in the second scenario (case b,), the maximum of the plastic strain was the given data. The results of the simulations together with output of the real test are shown in Fig. 7. At position 2, the results are shown in Fig. 8. It shows that there is no significant difference between the two results, but in case a, the impact size is closer to the reality. In this case, a finer mesh was applied which also helped to approximate the measured results more precisely. However, further measurements and simulations are needed for continuing the validation at different geometrical, material and test conditions.

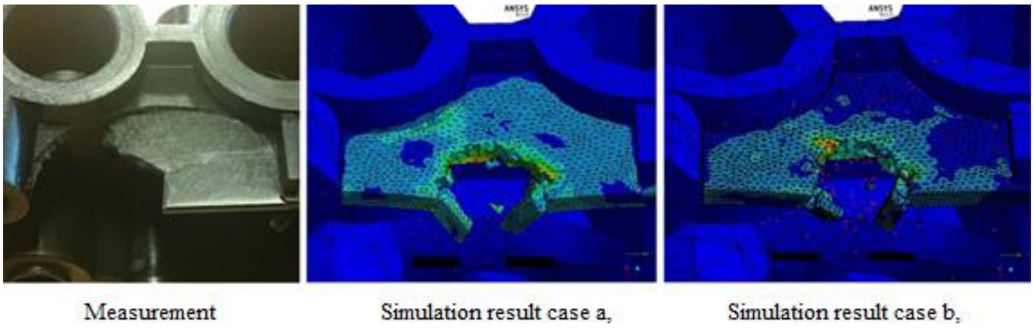

Fig. 7. The measured (left) and the calculated results (middle and right) at position 1. (impact rod test)

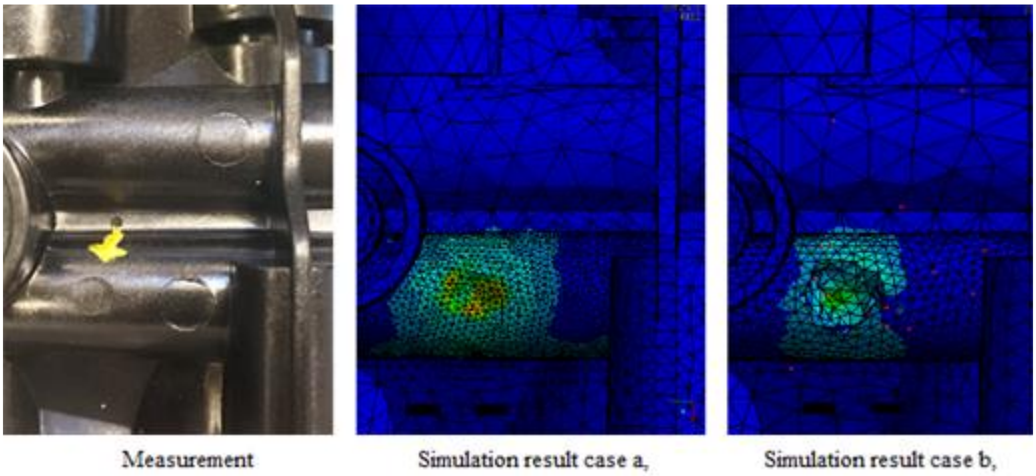

Fig. 8. The measured (left) and the calculated results (middle and right) at position 2. (impact rod test)

4. Conclusion

The performed verification with tensile test confirmed that the results of the simulation are accurate, because the highest deviation of the manufacturer’s data and the simulation results were 2.44 %. The verification with creep test was more difficult, the highest difference between the manufacturer’s data and the results of the analysis was 12 %, but at the end of the test time this value was reduced below 4 %. The applied creep model can be suitable depends on the requirements. The last verification proved that the explicit method was accurate from an engineering point of view. The maximum of the disparity was 5.4 % during these Charpy’s impact test on a test specimen.

On the other hand, the validation of the Charpy’s test showed higher difference between the manufacturer’s and the measured data on a specimen. This 13.5 % deviation was caused by fact that the material producers provide the lowest value of the impact strength in general and the higher radius of the specimen’s V notch in the measurement. In case of the measurement the radius was 1 mm counter to the 0.25 mm radius which was used by the manufacturer test so the material was less rigid by the larger radius.

The vertical hammer (rod) test of the real product was reproduced in Ansys with two failure criteria and two different mesh sizes for the case where the maximum of the plastic strain was considered as the failure criterion. Although the evaluations of the results were made by qualitative manner only, the agreement between the reality and the simulation results were satisfied.

Finally, it can be declared, that the measured results were approximated correctly by the applied material models and the other computational settings. However, more analysis of the 3D parts and their validations are required to increase the confidence level of the method.

Acknowledgements

The publication of the work reported herein has been supported by ETDB at BME.

References

Lukács, P., 2010. Autóipari műanyagok újrahasznosítása a vonatkozó EU-s direktívák tükrében (Recycling of Plastics in Automotive Industry by means of EU's directives), 3P Műanyagipari, Csomagolástechnikai, Nyomdaipari Klaszter Konferencia (Conference about 3P as Plastics, Packing and Printing), Kecskemét, Hungary. https://issuu.com/autopro/docs/prezentacio_lukacs_20101124 (10.09.2017.).

Hargitai, Cs. 2012. Effect of parameters of inland vessel semi empirical motion equations on transient motion phenomena. Proceedings of the TVL: Second Scientific Workshop on Transport, Vehicle and Logistics organized by the PhD Schools of the Faculty of Transportation Engineering and Vehicle Engineering. ISBN:978-963-313-070-4 pp. 1-6.

Beneda, K. 2008. Dynamic Nonlinear Mathematical Model of Active Compressor Surge Control Devices. 11th Mini Conference On Vehicle System Dynamics, Identification and Anomalies. ISBN: 918 963 313 011 7, 2010. pp. 583-591.

Rohács, D. and Rohács J., 2016. Magnetic levitation assisted aircraft take-off and landing: (feasibility study – GABRIEL concept). Progress in Aerospace Science 85: pp. 33-50.

Bera, J., Pokorádi, L., 2015. Monte-Carlo Simulation of Helicopter Noise, Acta Polytechnica Hungarica, Vol. 12, No. 2, pp. 21-3.

Schmailzl, A., Amann, T., Glockner, M., Fadenelli, M. 2012, Finite element analysis of thermoplastic probes under tensile load using LS-DYNA compared to ANSYS WB 14 in correlation to experimental investigations, ANSYS Conference & 30th CADFEM Users' Meeting – Kongress Palais Kassel.

Eduardo Luís Gaertner, Marcos Giovani Dropa de Bortoli, 2006, Some Aspects for the Simulation of a Non-Linear Problem with Plasticity and Contact, 2006 International ANSYS Conference.

Liu, Y., 2008, Impact Experimental Analysis and Computer Simulation, International Journal for Computational Methods in Engineering Science and Mechanics, Volume 9, Issue 1

Uddandapu, P., K., 2013, Impact Analysis on Car Bumper by Varying Speeds Using Materials ABS Plastic and Poly Ether Imide by Finite Element Analysis Software Solid works, IJMER, Vol.3, Issue.1, Jan-Feb. 2013 pp-391-395 ISSN: 2249-6645

Behrens B., A. et al, 2014, Material modelling of short fiber reinforced thermoplastic for the FEA of a clinching test Procedia CIRP 18, 250 – 255, International Conference on Manufacture of Lightweight Components – ManuLight2014

ANSYS Training manual; Introduction to ANSYS Explicit Dynamics, Release 17.0, 2016.05.24

ANSYS, Inc., Mechanical APDL Theory Reference , Release 15.0, ANSYS, Inc. Southpointe, 275 Technology Derive Canonsburg, PA 15317, ansysinfo@ansys.com, http://www.ansys.com, USA, April 2014

Güllü, A. et al., 2006. Experimental investigation of the effect of glass fibres on the mechanical properties of polypropylene (PP) and polyamide 6 (PA6) plastics, Materials and Design 27, 316–323

A. Arriaga et al, 2007 Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: experimental and numerical analysis results correlation with ANSYS, Polym. Test. 26 284-305.

X Li et al, 2014 Impact behaviors of poly-lactic acid based biocomposite reinforced with unidirectional high-strength magnesium alloy wires, Progress in Natural Science: Materials International 24 472-478

DIN EN ISO 527: Plastics - Determination of tensile properties

DIN EN ISO 1874: Plastics - Polyamide (PA) moulding and extrusion materials

DIN EN ISO 179: Plastics - Determination of Charpy impact properties

DIN EN ISO 294: Plastics - Injection moulding of test specimens of thermoplastic materials