EAN 2018
56th conference on experimental stress analysis

Experimental Verification of FEA Model

MEDŮNA Ondřej[1,a]

[1]VÚTS, a.s. Svárovská 619, 46001 Liberec, Czech Republic

[a]ondrej.meduna@vuts.cz

Keywords: FEA simulation, material test, material model, verification, experiment, 3D scan, deformation, variable test frame.

Abstract. The article deals with our procedure during steps FEA simulation, real test and their comparison which is shown on deformation tests of car bumper. These steps are supported by material tests and sophisticated scanning method.

Introduction

The output verification of the each procedure step during the R&D process is very important. Primarily, it means validation of result correctness for each single step in development cycle mostly related to the conformity of simulation and real test. In our practice we solve this issue with a comprehensive use of our current possibilities in the field of test, simulation and scanning methods. This is part of a cycle of connected activities analysis – optimization – design – prototype – test, which leads from initial analysis to final tested product.

The part of this cycle is verification of FEA simulation model via test results on the real part.

Material model

The material tests are done due to need to determine the real behavior of used materials on analyzed structure. These requirements were met by our own tensile and flexural tests. These types of test are chosen because of expected type of structure load: *finger test* that means local pushing into the bumper with controlled displacement and external load force. Testing machine for static tests, biaxial extensometer and environmental chamber were used for these material tests to get stress - strain curve and Poisson's ratio. The specimens were cut from real parts of bumper (Fig. 1).

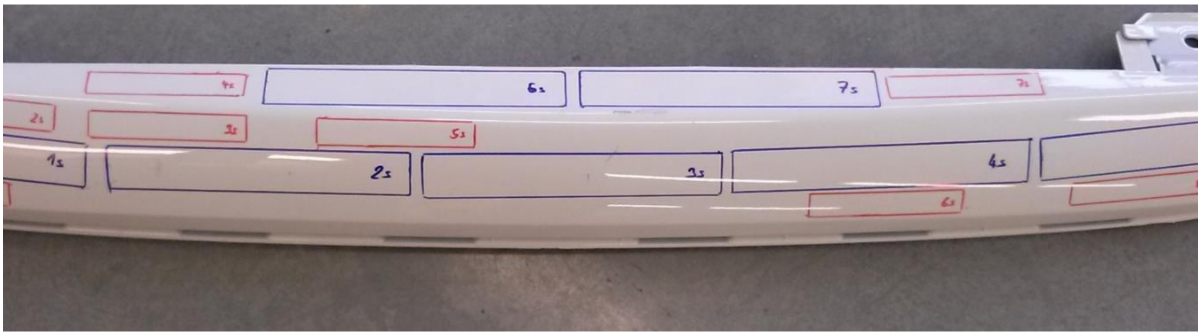

Fig. 1 Preparation of specimens

The stress – strain curve is used as input to material model. Because no applied force release is considered, a nonlinear elastic material model is used. The FEA simulation of both types of material tests is used to validate input data for the selected material model. To achieve compliance, input data of the material model is modified based on the FEA model results of the performed tests.

The results of the original (FEM - orig) and modified (FEM - final) input data for the material model compared to the real tensile and flexural test data (TEST) are shown on Fig. 2 and Fig. 3.

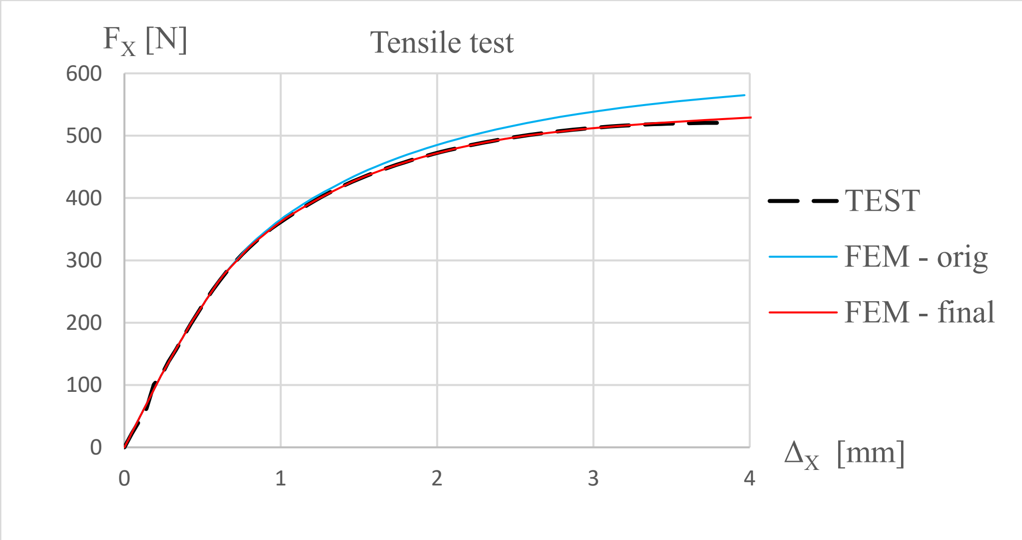

Fig. 2 Comparison of the tensile test and its FEA simulation

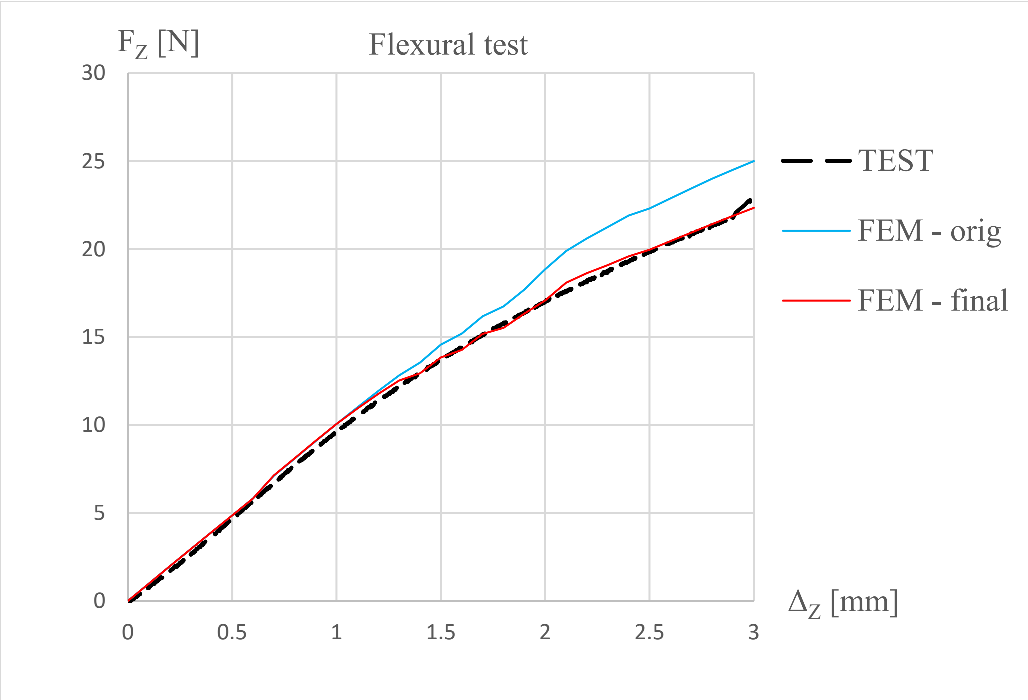

Fig. 3 Comparison of the flexural test and its FEA simulation

Test frame

For simulation and test purposes, a variable test frame [1] is designed, constructed and manufactured (Fig. 4). On the frame, there is a set of reference points for coordinate system definition that is identical to the CAD model coordinate system. It allows us to automatically input scanned data to the CAD/FEA model with the same global coordinate system. The frame consists of variable attachment points. The X, Y, Z coordinates of each one are mutually independent. This allows us to achieve high accuracy of the position of the tested part that is enabled by verification of attachment point positions by 3D scanning. In this way, the error of each attachment point positions is less than 0.5 mm (while the bumper width is over 1500 mm).

Another advantage of this concept is the use of the CAD model of the test frame to find required position of the loading system for the real test frame and define the right direction of applied force in FEA model.

In this way, compliance of the boundary conditions of the FEA model with the real test is guaranteed.

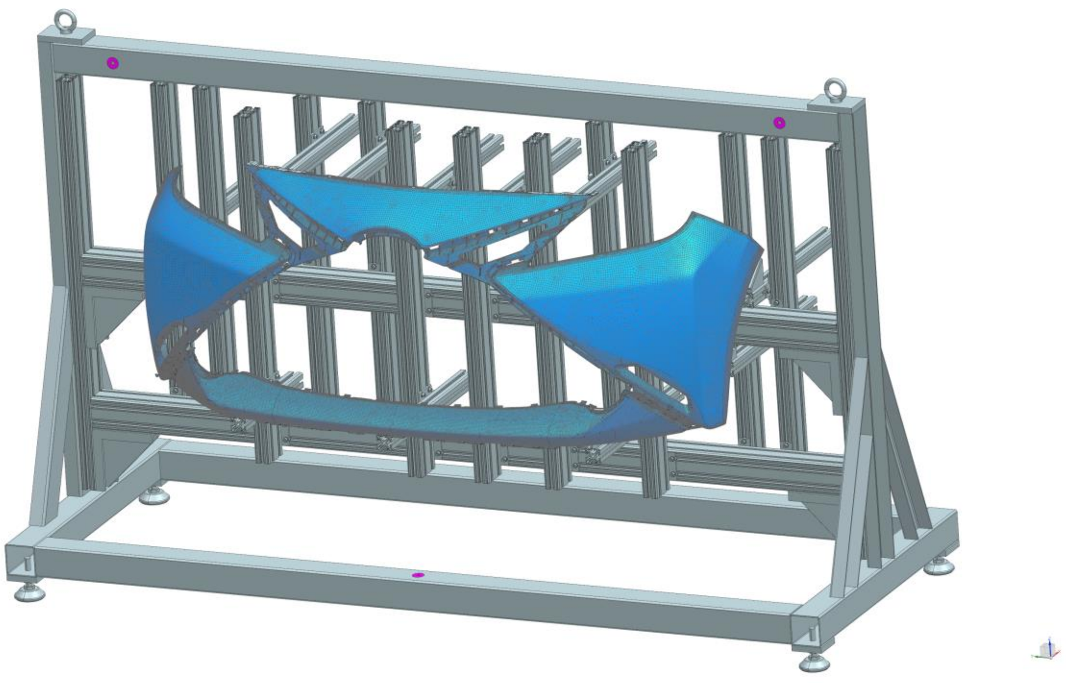

Fig. 4 Test frame with FEA mesh of the bumper

Test

A suspended weight with a pulley is used as a loading system. As mentioned above, the CAD model of test frame is used for determination of its desired position. The load force value is not determined solely by the weights used due to the influence of the friction on the pulley. Therefore, a force sensor located between the bumper and the pulley is used to determine the exact value of the load force. Such a measured load force value is then used in the FEA simulation. When a load force is applied to the bumper surface, its surface is scanned by the optical 3D scanner (Fig. 5) [2]. The STL data thus obtained are used for comparison with the results of the simulations.

Fig. 5 Scanning the bumper while the load force is being applied

Simulation

The main part of the car bumper with the auxiliary components is considered to be the deformable structure of the FEA model. This geometry is constrained to the rigid frame using preloaded bolts. 3D mesh of the whole analyzed structure is created using NX Siemens. It consists of 20-nodal hexahedral elements on large smooth faces and 10-nodal tetrahedral elements on more curved geometry. These two types of mesh are joined together by 13-node pyramidal elements. The whole FEA simulation is completed in pre/postprocessor MSC.Mentat and solved in MSC.Marc solver.

The contact between all deformable parts is considered the same as between bumper parts and rigid frame. The assembly is loaded in three stages. In the first one, the fastening of the joint is simulated by applying a pretension to each of the bolts. In the second one (after the bolts have been loaded), gravity is applied, and in the third one, the load force is applied to the top surface of the bumper to simulate finger test. Value of this force is measured in the previous real test on the bumper. The direction of this applied force is determined using CAD model of the test frame.

The STL file that represents deformed shape of FE mesh under external load is prepared from simulation results (Fig. 6). This file is then used for comparison with deformed shape of the bumper from the real test.

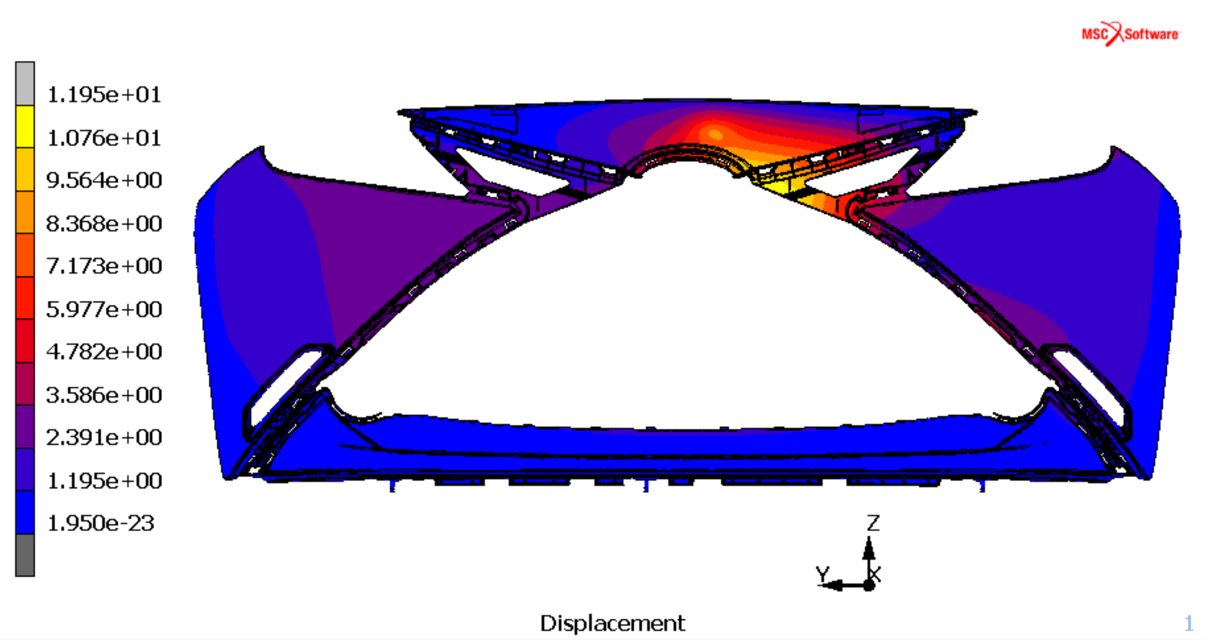

Fig. 6 FEA simulation results

Comparison of the FEA results with the real test

As mentioned above, the simulation results and the scanned deformed surface of the real bumper are compared using GOM Inspect software. Using the same coordinate system for both scanning and simulation, the deviations of both STL surfaces were computed and displayed (positive value means distance of scanned surface in positive normal direction from simulation model).

The first comparison (Fig. 7) is between the simulation results and the real test with only bolts preload and gravity load applied. There are noticeable deviations at some edges of the geometry: about +5 mm on the side areas, about -6 mm on the bottom area. The deviation in area close to the applied load force is under +1 mm.

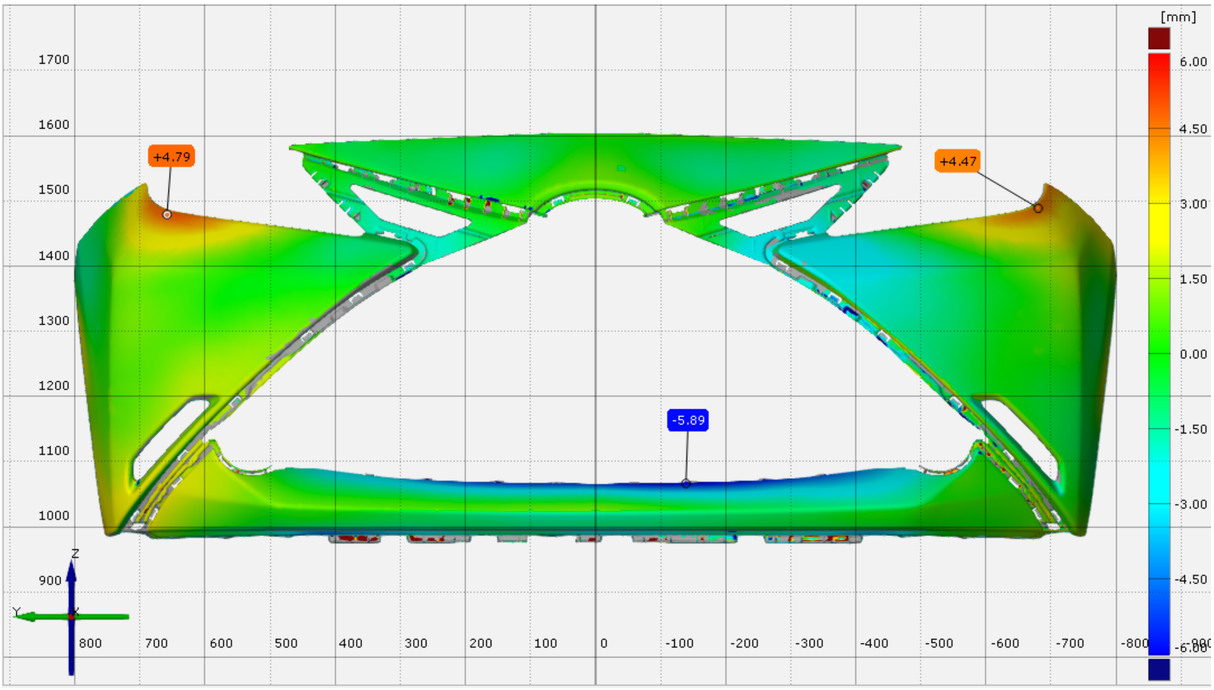

Fig. 7 FEA simulation results compared to the real deformed bumper (force load not applied)

The second comparison (Fig. 8) is between the simulation results and the real test with all loads applied (external load force included). The deviation at the point of the applied load force is now about +2 mm, the deviations on the side and bottom areas are very similar to the previous case. These appreciable deviations are mainly caused by wider bumper against its nominal CAD geometry (deformed due to its long-term storage) during mounting to the frame.

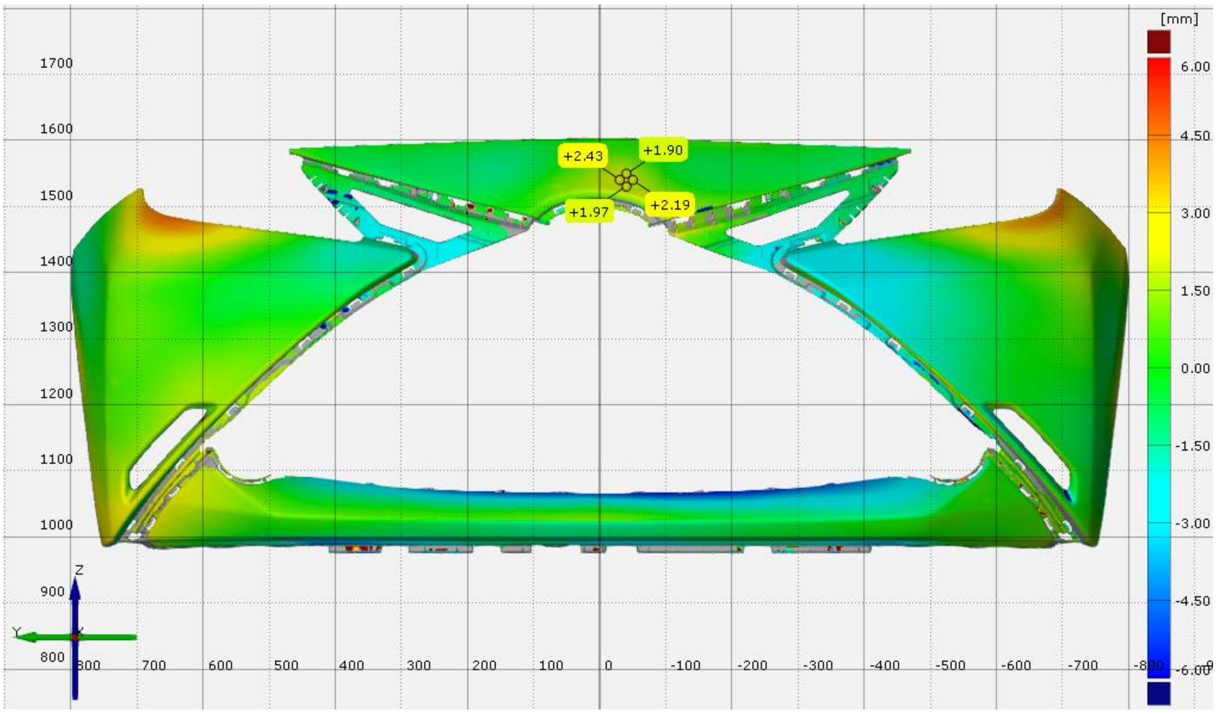

Fig. 8 FEA simulation results compared to the real deformed bumper (force load applied)

Conclusions

For validation of the simulation model of a main part of the car bumper, the deformed mesh of the FEA model with the scanned geometry of the real part with applied load force is compared under the same boundary conditions. Material tests, FEA simulation, a newly developed variable test frame and optical 3D scanning were used for this task. Resulting values of deviations may be due to various causes: manufacturing imperfections, deformation of the bumper due to long-term effect on plastic parts under gravity load, wear of contact clips due to repeated disassembling of bumper.

Acknowledgement

The publication was created with the support of the project LO1213 - Excellent Engineering Research, the grantor - MŠMT, beneficiary – VÚTS, a. s., in support of the National Sustainability Program, with the financial contribution of the Ministry of Education, Youth and Sports of the Czech Republic.

References

[1] Z. Braier, O. Medůna, P. Půlpán, M. Pustka , “Unifikovaný systém ukládání a výměny dat”, VÚTS, a.s. Liberec, 2017.

[2] K. Prokeš, “Možnosti a využití měřicího ramene Romer Absolute Arm RA-7525SI-2 ve VÚTS, a.s.“, VÚTS, a.s. Liberec, 2016.