The 3rd International Conference on "Computational Mechanics and Virtual Engineering"
COMEC 2009
29 – 30 OCTOBER 2009, Brasov, Romania

COMPUTATIONAL METHODS CONCERNING THE SIMMULATION OF THE MECHANICAL TESTS IN CASE OF SOME COMPOSITE MATERIALS

Assist. eng. C. ITU[1], Assist. prof. dr.eng. C. CERBU[2], Prof.dr.eng. A. C.V. POPA[3],
[1] Transilvania UNIVERSITY, Brasov, ROMANIA, calinitu@unitbv.ro
[2] Transilvania UNIVERSITY, Brasov, ROMANIA, cerbu@unitbv.ro
[2] Transilvania UNIVERSITY, Brasov, ROMANIA, acvpopa@unitbv.ro

Abstract: *The work focus on a simulation methodology of the mechanical tests in case of some composite materials, by using numerical methods based on explicit solvers that involve the long-time analyses. This paper considers only the case of the flexural test applied to some composite materials reinforced with wood flour or / and E-glass woven fabrics. This simulation is validated by using the experimental results obtained in case of the real mechanical tests. The objectives of this methodology consist in the control and the following of the main parameters of the mechanical tests, cost reduction of these tests.*
Keywords: *simulation, finite element analysis, composite material*

1. INTRODUCTION

The actual tendency of the engineers and researchers people from worldwide are focused both on the research-developed-optimization activities for some components or parts of system assemblies or sub-assemblies and on the increasingly quality, reliability and durability of products in order to remain competitive and profitable or to obtain the marketplace.

The manufacturers are pressed to refresh their product models much faster than before. Shortening time to market and reducing costs are simply not good enough anymore. They have to penetrate new markets with innovative new products that support and cultivate strong brand values. This calls for a high performance development process – with high throughput and high precision. A process that delivers the right products – designed right first time. Engineering departments now more than ever are challenged to tune the functional performance characteristics of their new designs.

Creating innovative and attractive products is not just about producing visually attractive designs. Some manufacturers consider safety and reliability not just to be constraints but critical to their image.

Today, the industry evolution towards more virtual prototyping results in fewer prototypes that must be tested in greater detail.

Virtual prototyping method won it an important role in the different high industry domains, such as: automotive, aircraft, biomechanical industry. The truth virtual prototyping is created by means of FEA software, seeing obtained some products which accomplish the request impose by market. Therefore, by means of FEA software, it can be obtained a faithful modeling, both for system components and functional conditions of the system. In this way, it can be eliminated a long stage from experimental testing which represent an expensive process.

FE analyses provide a very efficient way to represent structural flexibility for even the most complicated geometry. The method uses an advanced numerical algorithm based on displacement calculus method. The flexible behavior for any number of parts in the simulation can be represented and visualized by graphing and animating results.

This functionality allows visualizing the stress field of a part undergoing dynamic or static loading. When using the dynamic solver, the resulting stresses are the dynamic stresses that take all transients into account. In this paper, we attempt to evaluate the structural behavior of a composite cantilever that is support on the both ends and is charged with a time transient loading that is increased after a linear law in the middle zone.

2. TECHNICAL REQUIREMENTS

The processes and phenomena that develop in mechanical system are quite complex and difficult to represent faithful in virtual models, which represent in fact an approximation of the real systems.

Theoretical interpretation of the phenomena and processes from mechanical systems gets useful in order to make a final decision with regard to adoption some optimal functional parameters.

The analyzed model was made for a 3 point bending composite cantilever [1], and block diagram of the debate study is presented in the figure 1.

The input values known are mass, material & geometrical properties for cantilever. The other components: supports and nose that acts on the cantilever in the middle zone of it, will be considered as rigid body parts. The external loading is applied as velocity on the nose. The supports have all DOF restraints.

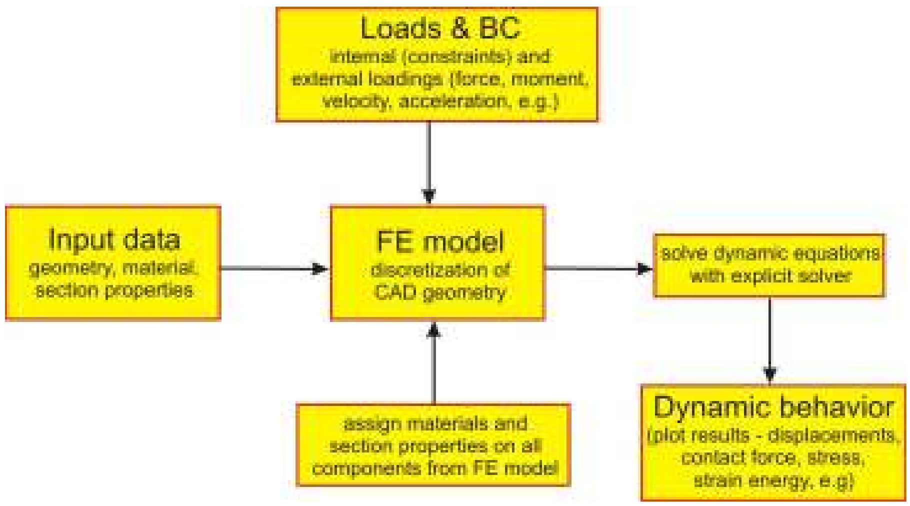

Figure 1 - Bloc schema for analysis

Based on input kinematics parameter adopted and the restraints, it can be obtained the equation solutions of the mechanical system, resulting the reactions force in the nodes that have restraints all DOF and based on them the force loading that charge the composite cantilever. The motion equations obtained from kinematics behavior of the mechanical system along with external and internal loading applied on the study model leads on to obtained the dynamic model and implicit to the dynamic behavior of the system [2], [3].

Finally, based on summing kinematics and dynamic behavior effects of the mechanical system, it obtained the dynamic stress and strain state of the marked component.

3. Dynamic stress analysis of the composite cantilever 3-point bending

In the figure 2, it is shown the FE (virtual) model of composite cantilever. According with virtual schema, the external load is transmitted from nose to composite cantilever.

The external loading is applied on nose as velocity on a total time period of explicit FEA analysis. The velocity imposed is 1.5 [mm/min] on the total time period of analysis (see figure 4) [4]. Also, the experimental data was made in the same time conditions.

Between all components from FE model have been applied surface to surface contact in order to simulate as possible the reality behavior of composite cantilever under external load-time varying.

The both end supports and nose component have been simulated as rigid component in assembly.

Only the composite cantilever was simulated as part deformable. The stress - strain curve material used in FEA analysis for the cantilever is shown in figure 3.

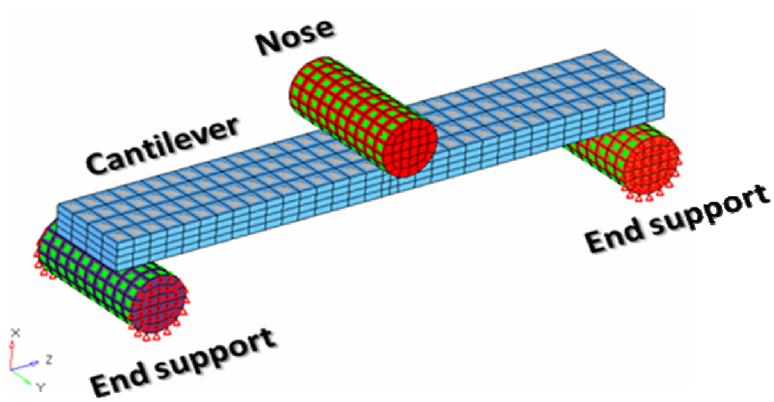

Figure 2 - FE model of composite cantilever

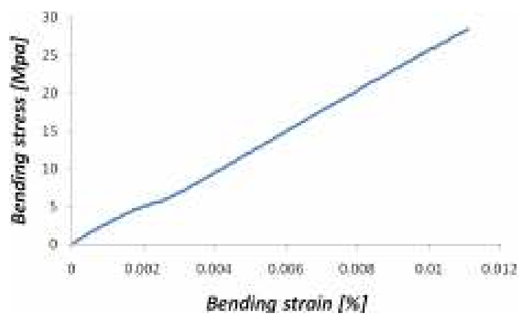

Figure 3 – The bending stress – strain curve material used in the explicit analysis

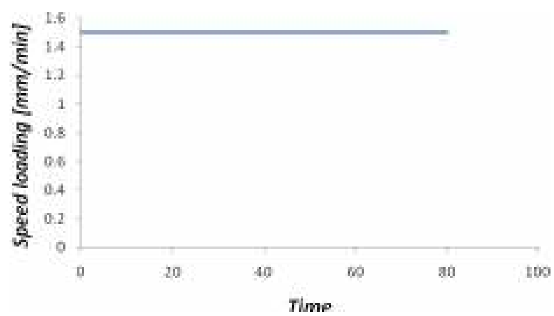

Figure 4 – Speed loading curve vs. time

In figure 5 can be observed the comparison force obtain on the FEA analysis vs. experimental data. The difference between these are lower than 10%.

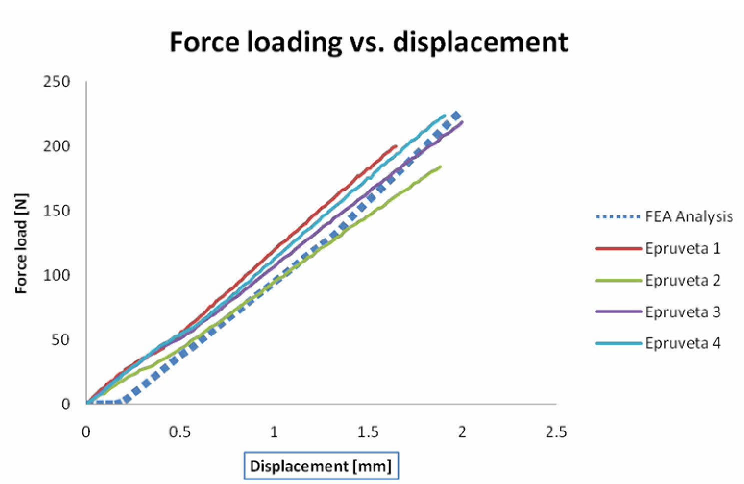

Figure 5 – Comparison between forces obtained from FEA analysis and forces obtained from experimental data

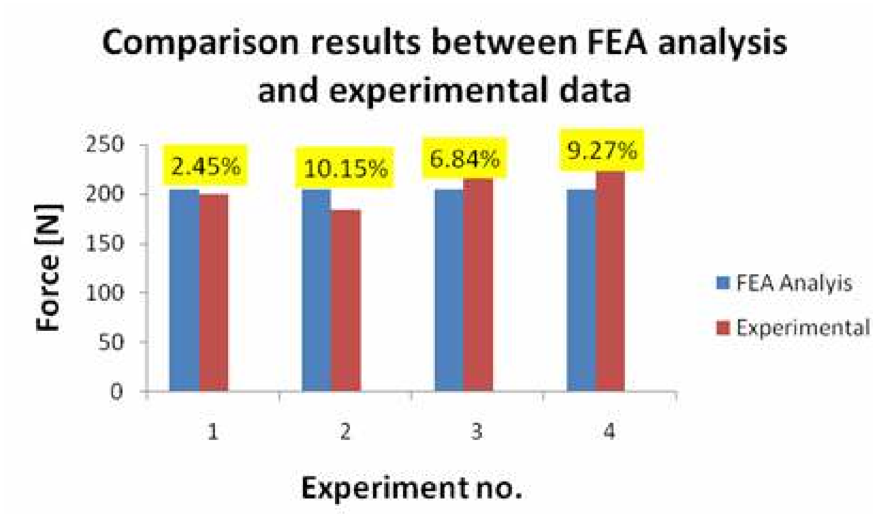

Figure 6 - Error percentage between FEA analysis and testing

3. CONCLUSION

In the figure 6, it can be observed the comparison force obtain on the FEA analysis vs. experimental data. From all experiment (with except no 2 that is slightly higher than 10%), the error percentage between FEA analysis and testing is lower than 10% and this value can be considered acceptable taking into account to possible errors that can be caused by analytic error calculus, error reading from the real stand that is depending on calibration stand, non-linearity of material e.g.

The dynamical stress evaluation method based on virtual simulation offers a quickly and better information about stress of state for a part under a dynamic transient loading comparison by evaluation method based on static analysis because the inertial loading is determined by means of motion equation established by the soft.

Also, the dynamical stress evaluation method of the state stress is very advantageous for research-developed-optimization activities of the product and is applicable for any complex mechanical system.

Based on dynamic FEA analysis, it can make appreciation with regards to factor of safety for a component desired on a complete duty cycle, known material properties of the part studied.

REFERENCES

[1] CERBU Camelia - *Aspecte privind degradarea caracteristicilor elastice şi mecanice de încovoiere ale materialelor compozite sticlă / polimer din cauza absorbţiei de umiditate*, Revista Materiale Plastice (ISI), ISSN 0025-5289, vol. 44, iunie 2007, p. 97-102;

[2] OCHA O.O., REDDY J.N. - *Finite element analysis of composite laminates,* Kluwer Academic Publisher, Dordrecht / Boston / London, 1992, ISBN 0-7923-1125-6;

[3] TENEK Lazarus Teneketzis, ARGYRIS John – *Finite element analysis for composite structures*, Kluwer Academic Publisher, Dordrecht / Boston / London, 1998, ISBN 0-7923-4899-0;

[4] *** LS-DYNA manual keyword.