IMPACT AND NONLINEAR ANALYSIS FOR HIGHLY LOADED AIRFRAMES

Prof. Kravchenko S.; Zhidyaev K.; prof. Dadunashvili S.
Boeing Design Center
Moscow, Russia
sergey.v.kravchenko@boeing.com, kirill.a.zhidyaev@boeing.com, Sergo.dadunashvili@boeing.com

Keywords- nonlinear analysis; impact; airframe structure; structural design; composite panel; finite element analysis; tension fitting

I. INTRODUCTION

Airplane during its life cycle is exposed by the action of various load conditions. To satisfy airplane safety requirements all these load conditions should be considered in strength analysis of the airplane structure. These structures mostly are analyzed by analytical methods using loads obtained from the global linear finite element model of airframe. But this approach is inapplicable for most complex structure commodities and load cases.

The examples of such events are various impacts on structure which occurs during airplane flight. These are bird strike event, tire burst and tire fragment impact to the structure, hail impact etc. It is very difficult to analyze these events because of high nonlinearity and furthermore analysis should be supported by expensive structural test. We have improved the analysis methodology based on nonlinear explicit finite element analysis proved by the test. Our approach allows from one side to reduce tests numbers and from another side raise the analysis accuracy and provide enough data to confirm strength of the structure for all locations where impacts may occur.

Another example concerns highly nonlinear analysis. In modern airplane structures are used various types of structural materials. These are metal alloys, carbon and glass composites, rubbers and hybrid materials. All these materials have different properties which are mostly nonlinear and have a different nature – plasticity, hyper elasticity, viscosity, fracture, etc. Nonlinear finite element analysis based on test data helps to consider wide spectra of physically nonlinear material properties in conjunction with geometry nonlinearity such as bucking and post buckling to get more accurate solution, reduce degree of conservatism and so help to reduce weight of the airplane.

II. IMPACT ANALISYS

Bird strike to carbon composite panel

Every year we see wider and wider using in airframe structures composite materials due to their high strength-to-weight and stiffness-to-weight ratios, corrosion resistance, but one of deficiencies of composites is brittleness. The carbon composite panels are more brittle then aluminum because the plasticity of carbon fibers is neglectable. That’s why during impact composite structure is absorbing much less energy then aluminum one, impact load may become critical for structure crashworthiness and needs to be analyzed more carefully and precisely. For analysis was used nonlinear finite element approach [1].

Bird model

Bird finite element model geometry was generated using a "prolate spheroid" by scaling a sphere of diameter “D” in one direction (Fig. 1). Linear dimensions L and D and aspect ratio L/D were chosen

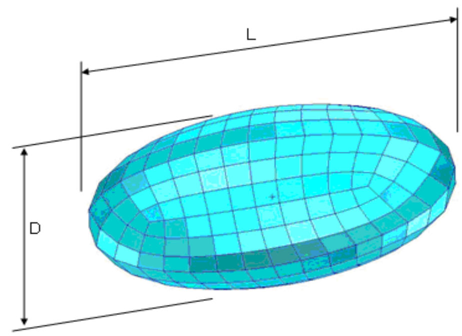

Fig. 1.

to best matching of bird total weight (4 lb) and actual dimensions. Mesh quality was checked by increasing number of elements by a factor of 4 times with no significant difference in the results. Compressible fluid was used to represent the bird material properties [2]. Fluid’s characteristics were inputted into the finite element analysis as an Equation of State material in Mie-Grüneisen form:

$$p - p_H = \Gamma \rho (E_m - E_H), \qquad (1),$$

where p_H and E_H are the Hugoniot pressure and specific energy and are functions of density only, and Γ is the Grüneisen ratio defined as

$$\Gamma = \Gamma_0 \frac{\rho_0}{\rho}, \qquad (2),$$

where Γ_0 is a material constant and ρ_0 is the reference density.
The Hugoniot energy, E_H, is related to the Hugoniot pressure by

$$E_H = \frac{p_H \eta}{2\rho_0}, \qquad (3),$$

where $\eta = 1 - \rho_0/\rho$ is the nominal volumetric compressive strain.

Impact model and test correlation

The analysis simulates the test of bird impact to stiffened composite panel. Composite panel finite element model is shown on Fig.2.

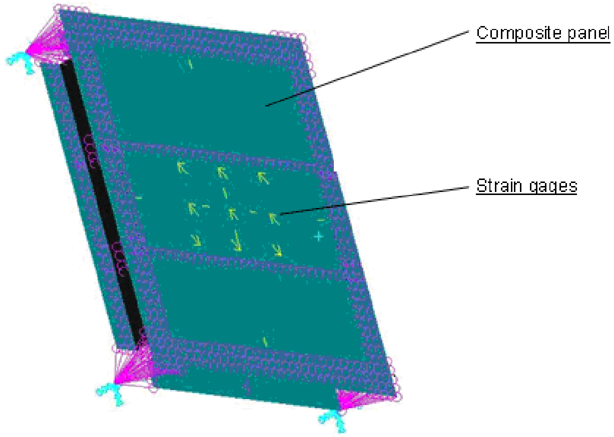

Fig.2

The equations of motion was integrated using the explicit central difference integration rule

$$\dot{\mathbf{u}}^{(i+\frac{1}{2})} = \dot{\mathbf{u}}^{(i-\frac{1}{2})} + \frac{\Delta t^{(i+1)} + \Delta t^{(i)}}{2}\ddot{\mathbf{u}}^{(i)}, \qquad (4),$$

$$\mathbf{u}^{(i+1)} = \mathbf{u}^{(i)} + \Delta t^{(i+1)}\dot{\mathbf{u}}^{(i+\frac{1}{2})}, \qquad (5),$$

where: $\dot{\mathbf{u}}$ is velocity and $\ddot{\mathbf{u}}$ is acceleration; superscript (i)refers to the increment number and $i-\frac{1}{2}$ and $i+\frac{1}{2}$ refer to midincrement values. For strain control strain gages was modeled as a thin bars. Contact interaction was modeled between panel and bird [6]. The results of analysis show good correlation with the test (Fig.3). To achieve such correlation energy absorption properties of bird's body was very important.

Tire fragment impact analysis

One of the critical load cases for airplane takeoff is tire burst with subsequent tire fragment impact to wing low panel. Similar accident led to Concord crash in 2000.

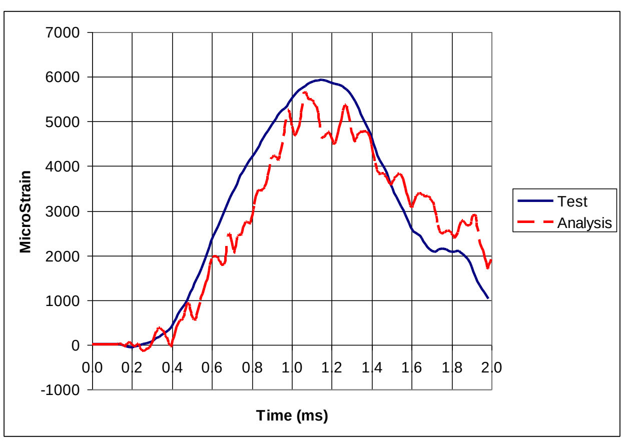

Fig.3

Neoprene material model

Tires of airplanes are usually made from neoprene. It is rubber like material with hyperelastic properties. As a mathematical model of material behavior was chosen reduced polynomial strain energy potential in the form [4]:

$$U = \sum_{i=1}^{N} C_{i0}(\overline{I}_1 - 3)^i + \sum_{i=1}^{N} \frac{1}{D_i}(J^{el} - 1)^{2i}, \qquad (6),$$

where U is the strain energy per unit of reference volume; C_{i0} and D_i are temperature-dependent material parameters; J^{el} is the elastic volume ratio and $\overline{I}_1$ is the first deviatoric strain invariant defined as

$$\overline{I}_1 = \overline{\lambda}_1^2 + \overline{\lambda}_2^2 + \overline{\lambda}_3^2, \qquad (7),$$

where λ_i are the principal stretches.

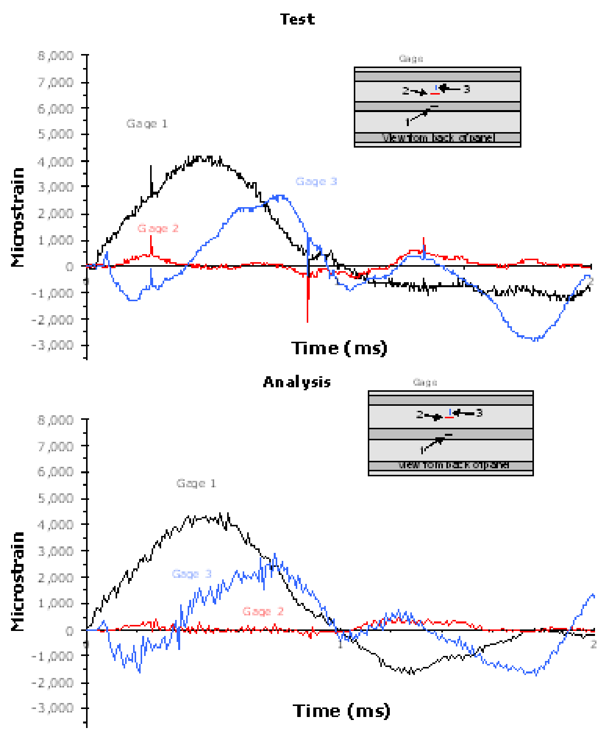

Fig.4

Material model was validated by neoprene ball to composite panel impact test results. Finite element analysis vs. test comparison shown on Fig.4 confirmed quite good correlation.

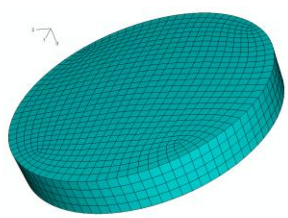
Fig.5

Tire fragment to fuel tank impact analysis

To simulate tire fragment impact to the wing box fuei tank low surface was performed explicit dynamic finite element analysis. Neoprene tire part was modeled as a cylinder with diameter 8 inch and height of 1 inch (Fig. 5). The wing box was modeled as a three bay stiffened segment. To rise the dynamic analysis accuracy in FEM segment was included acoustic media. The result of analysis for time T=3 milliseconds is shown on Fig.6.

Based on the results of this analysis some minor changes were made in wing box design, which make structure safer and lighter.

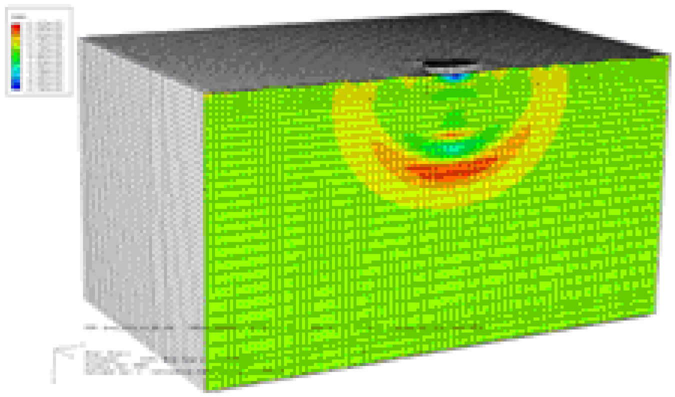

Fig 6.

III. NONLINEAR ANALYSIS

In the airframe structures are widely used different types of tension fittings. These details are usually used in critical locations and have to carry out significant loads. But strength analysis of such parts conjugates with significant difficulties due to their geometry complexity. Classical methods of strength analysis for such fittings may be applied only with big conservatism, so solution accuracy very often is unacceptable. That is why the strength of almost all new tension fitting configuration needs to be proved by test. The main problem is in fact that in some areas of the fitting complex stress state is realized, but at the same time fitting material has significant anisotropy in grain directions. Such conjunction of multiaxial stress state and material properties anisotropy make strength analysis really challenging. The target of our investigation is to resolve mentioned problem by finding appropriate parameters of material model and material failure criteria and developing finite element based method of tension fitting strength analysis which will be able exactly describe tension fittings behavior starting from linear elastic domain, including yielding and up to fitting failure.

Tension fittings nonlinear finite element analysis and test correlation

Development of tension fitting strength analysis methodology was started with series of nonlinear finite element analyses performed for a variety of tension fittings. These analyses were the simulation of static tests performed for aluminum tension fittings with different configurations.. Typical finite element model of tension fitting is shown on Fig.7.

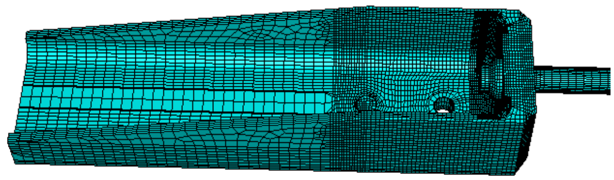

Fig.7

Fitting finite element mesh density is varying thru the model. Zones with high loading and with stress concentrators were meshed finer. Besides that to get more accurate results on the surface of detail where maximum stresses are realized additional thin layer of elements was used. It is very important for accurate plasticity zones determination and makes failure analysis more reliable. Since simulation was performed up to the failure of the fitting all effects due to ultimate material loading had to be taken into account. One of such effects is material plasticity. Standard Von Mises plasticity model could be used for isotropic material only, but in our case significant material anisotropy in plasticity zone was observed. To take this effect into account for material anisotropy description was used Mises – Hill plasticity model based on Hill's potential function [3,5]. Hill's potential function can be expressed in terms of rectangular Cartesian stress components

as: $$f(\boldsymbol{\sigma}) = \sqrt{F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 2L{\sigma_{23}}^2 + 2M{\sigma_{31}}^2 + 2N{\sigma_{12}}^2} \quad (8),$$

where ***F, G, H, L, M*** and ***N*** are constants obtained by tests of the material in different orientations.
The flow rule for this model is

$$d\varepsilon^{pl} = d\lambda\frac{\partial f}{\partial \boldsymbol{\sigma}} = \frac{d\lambda}{f}\mathbf{b}, \quad (9)$$

where

$$\mathbf{b} = \begin{bmatrix} -G(\sigma_{33}-\sigma_{11}) + H(\sigma_{11}-\sigma_{22}) \\ F(\sigma_{22}-\sigma_{33}) - H(\sigma_{11}-\sigma_{22}) \\ -F(\sigma_{22}-\sigma_{33}) + G(\sigma_{33}-\sigma_{11}) \\ 2N\sigma_{12} \\ 2M\sigma_{31} \\ 2L\sigma_{23} \end{bmatrix}.$$

Failure criterion for material was used in form the same as in (8), but with different material constants, which were determined based on failure stress in each orientation [7]. During the analysis was assumed that fitting failure load is reached when failure criterion is satisfied at least at one integration point of the fitting finite element model.

Suggested approach assures good correlation between results of finite element analysis and tests. Analysis has predicted the same failure modes as was observed during the tests for all considered fitting configurations (Fig.8) and with small deviation in loads magnitude.

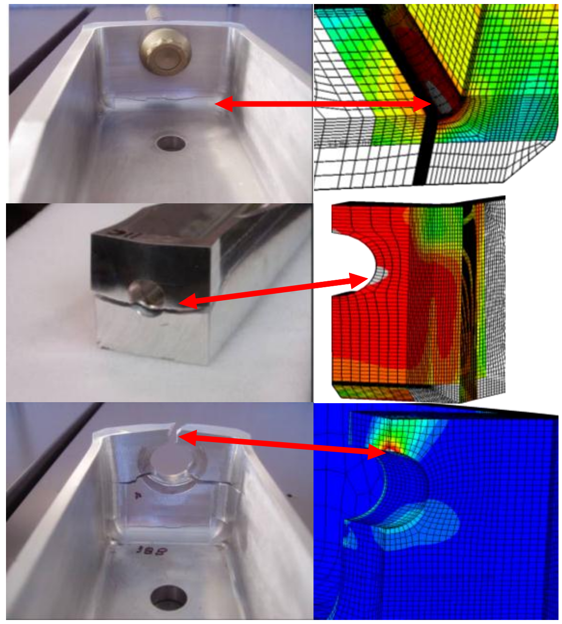

Fig.8

Fig, 9 illustrates test-analysis failure loads correlation for 5 different fitting configurations. This diagram shows that for some fitting configurations failure loads predicted by finite element analysis are slightly different in comparison with failure loads obtained from the test. It happened because of insufficiency material data available. In some cases material constants roughness may bring out even the unacceptable errors. To adjust material constants and avoid errors additional tests on multiaxial stress state need to be performed. Based on the results of these tests material model will be updated and finite element analyses will be rerun.

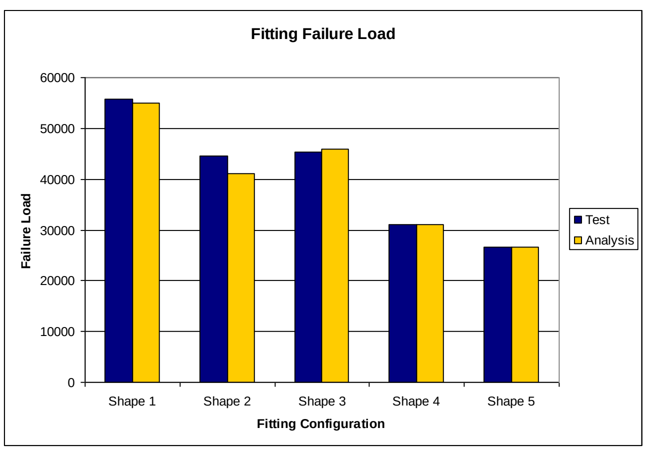

Fig.9

IV. CONCLUSION

Three examples of highly nonlinear finite element analyses were considered in this paper. Bird strike simulation helps to confirm airplane structure ability to resist such ultimate case in all areas of airplane where it is possible to occur based on results of only few real tests. This analysis helps engineers to choose optimal structural parameters and thus to reduce the weight of the structure. Numerical analysis of tire fragment impact allows to improve airplane structure and make it safer based only on the results of structural component tests. It minimizes the number of very expensive airplane full scale tests. Numerical derivation of tension fittings failure loads helps to improve existing analytical methods and allow reduce numerous static tests which were required before.

Thus numerical simulation makes airplane design process faster, cheaper and more accurate, and so airplanes structure become safer, cheaper and lighter.

REFERENCES

[1] Abaqus Analysis User's Manual (ver. 6.10) 2010

[2] A. Airoldi, B. Cacchione “Modelling of impact forces and pressures in Lagrangian bird strike analyses”, Int J Impact Eng, 32 (2006), pp. 1651–1677.

[3] Chu, E. “Generalization of Hill's 1979 anisotropic yield criteria”. Journal of Materials Processing Technology, vol. 50, 1995, pp. 207-215.

[4] Gendy A.S., Saleeb A.F. Nonlinear material parameter estimation for characterizing hyperelastic large strain models.Comput. Mech., 2000, v.25, pp.66–77.

[5] R. Hill. (1979). “Theoretical plasticity of textured aggregates”. Math. Proc. Camb. Phil. Soc., 85(1), 1979, pp.179–191.

[6] A.F. Johnson, M. Holzapfel “Modelling soft body impact on composite structures”, Compos Struct, 63 (2003), pp. 103–113.

[7] Zhu, Y., Dodd, B., Caddell, R. M. and Hosford, W. F. “Limitations of Hill's 1979 anisotropic yield criterion”. International Journal of Mechanical Sciences, 1987, vol. 29, pp. 733.