UDC 539.3

STRESS ANALYSIS OF DELTA FIN STRUCTURE AND DETERMINATION OF DEFORMATION

[1]**Mostafa Essuri,** [2]**Benur Maatug,** [1]**Khalid Kurmaji**

Azawia University, Faculty of Engineering, Libya
Al-Mergib University, Faculty of Engineering, Garaboulli, Libya

Introduction

Usually fin represent as one of the important structures in all type of airplane and the fin structure was greatly improved, especially for modern supersonic speed constructions. The main aim of delta fin construction is to minimize the weight of structure as much as possible and keeping the stiffness of material structural in margin of safety under design load [1]. The primary difference between classical method and finite element are the view structure and the ensuring solution procedure. Classical method considers the structure as a continuum whose behavior is governed by partial or ordinary differential equations [2]. By using finite element method consider the structure to be an assembly of small finite-sized particles. The behavior of the particles and the overall structure is obtained by formulating a system of algebraic equation that can be readily solved by developed methodology, which will be presented in form of software.

Evaluation of stiffness of bearing shells with oblique spars

The finite difference method used for the evaluation of stiffens of bearing shells of delta fin. The wing is loaded from the system of evenly distributed vertical forces. The area of the bearing surface can be treated as a mesh with 24 small identical rectangles (Fig.1) with sides [D_x] and [D_y]. The geometric features of the delta fin as wall as the relevant loading were adopted from the literature [3]. The delta fin structure is composed of exterior composite skin, and interior metal (al-alloy) structure. General expression for the stiffness of the bearing surface (treated as a plate) can be obtained for each node in the following form:

$$D_x = \frac{Eh^2 t_x}{2(1-\upsilon^2)} \quad ; \quad D_y = \frac{Eh^2 t_y}{2(1-\upsilon^2)} \quad ; \quad D_{yx} = \frac{Eh^2 t(1-\upsilon)}{2(1-\upsilon^2)}$$

Assuming that the skin (shell) and the oblique spars are subject to bending and that skin alone is subject to twisting. In expression above, denoted the height by [h] of the profile in each node with the thickness of the skin is not included [4]. Equalizing moments of inertia of the section [D_y] and [D_x] can calculate the effective widths of the skin of the idealized structure of the original wing and the model.
Moments of inertia [Ix] , [Iy], and the effective thickness t_x, t_y, are averaged for the corresponding lengths [D_y] and a [D_x], taking into account constructive elements which convey bending moments.

For example, to obtain the effective thickness for node i (given in Fig. 2) about the axes O_y and O_x, for examine one part of the section at the distance [$D_y/2$] and [$D_x/2$] measured from each side of the node [5]. For the case of oblique spar the effective thickness and moments of inertia is

$$2\frac{\Delta y.t.h_i^{\,2}}{4} + \frac{b(h-t)^3}{12}\cos^3\varphi = 2\frac{\Delta y.t_{x_i}.h_i^{\,2}}{4} = I_y \ , \ t_{x_i} = t + \frac{b(h-t)^3}{6.\Delta y.h_i^{\,2}}\cos^3\varphi$$

$$2\frac{\Delta x.t.h_i^2}{4}+\frac{b(h-t)^3}{12}\sin^3\varphi = 2\frac{\Delta x.t_{y_i}.h_i^2}{4} = I_x\,,\; t_{y_i} = t + \frac{b(h-t)^3}{12}\sin^3\varphi$$

With b (h-t) denoted the area of the cross section which is perpendicular to the axial axis.

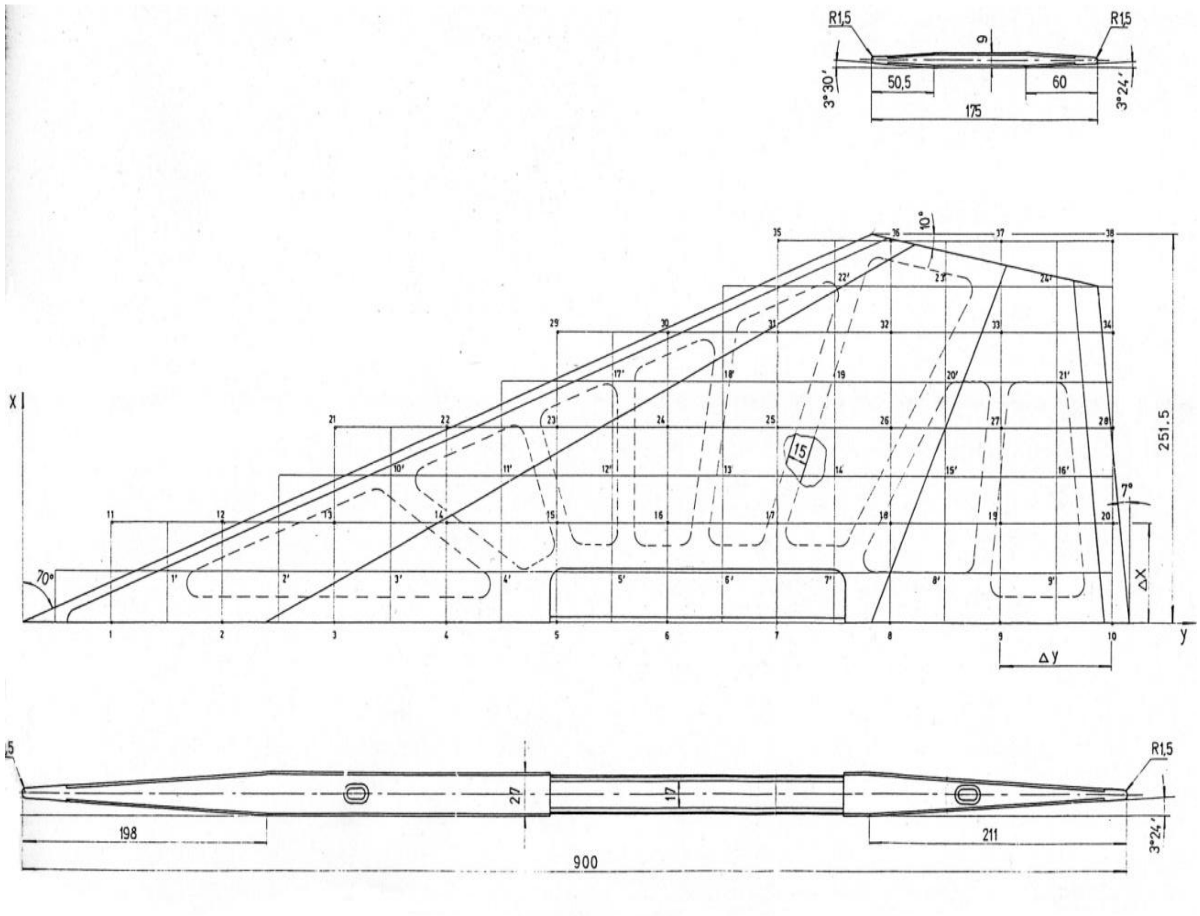

Fig. 1. – Model Geometry with Mesh of Bearing Surface

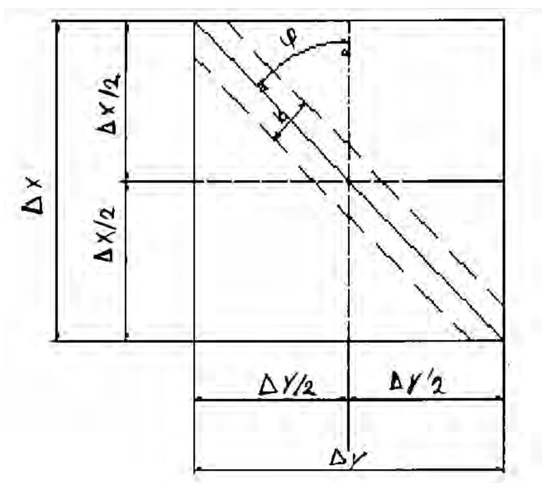

Fig. 2. – Effective thickness for node i

Numerical Result

The finite element method was used for the calculation of stress states in delta fin. As the output the displacement vector, deformation and stress tensor were obtained. The Fig. 3 shows the delta wing displacement in millimeters.

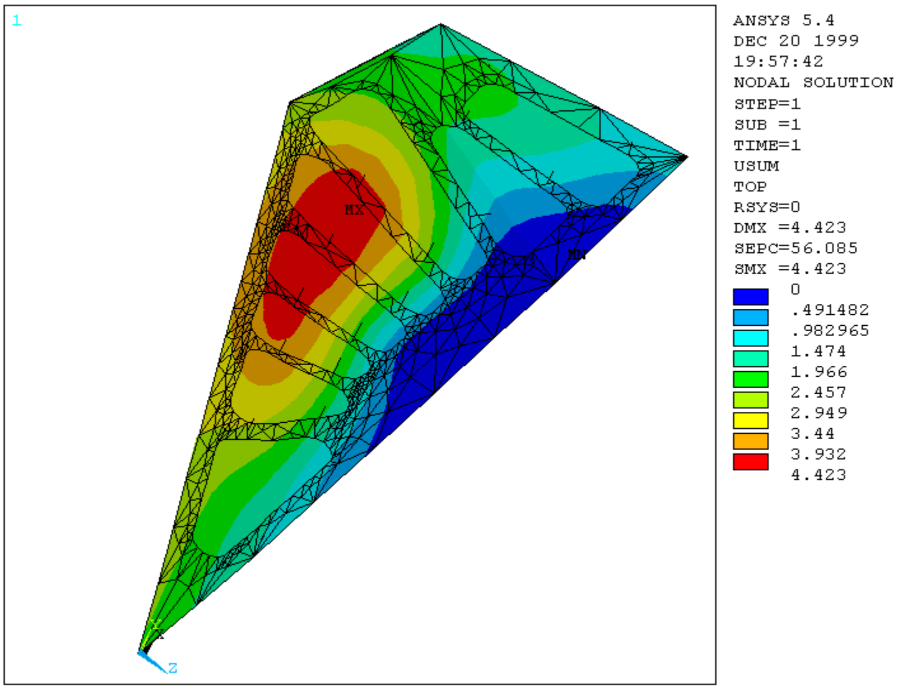

Fig. 3. – Delta fin displacement

Usually, the most useful stresses is Von Misses stress and the graphical results of the stress various in the form of colored area and from color area we can see distribution of Von Misses stress over skin and distribution of Von Misses stress in interior structure Fig. 4.

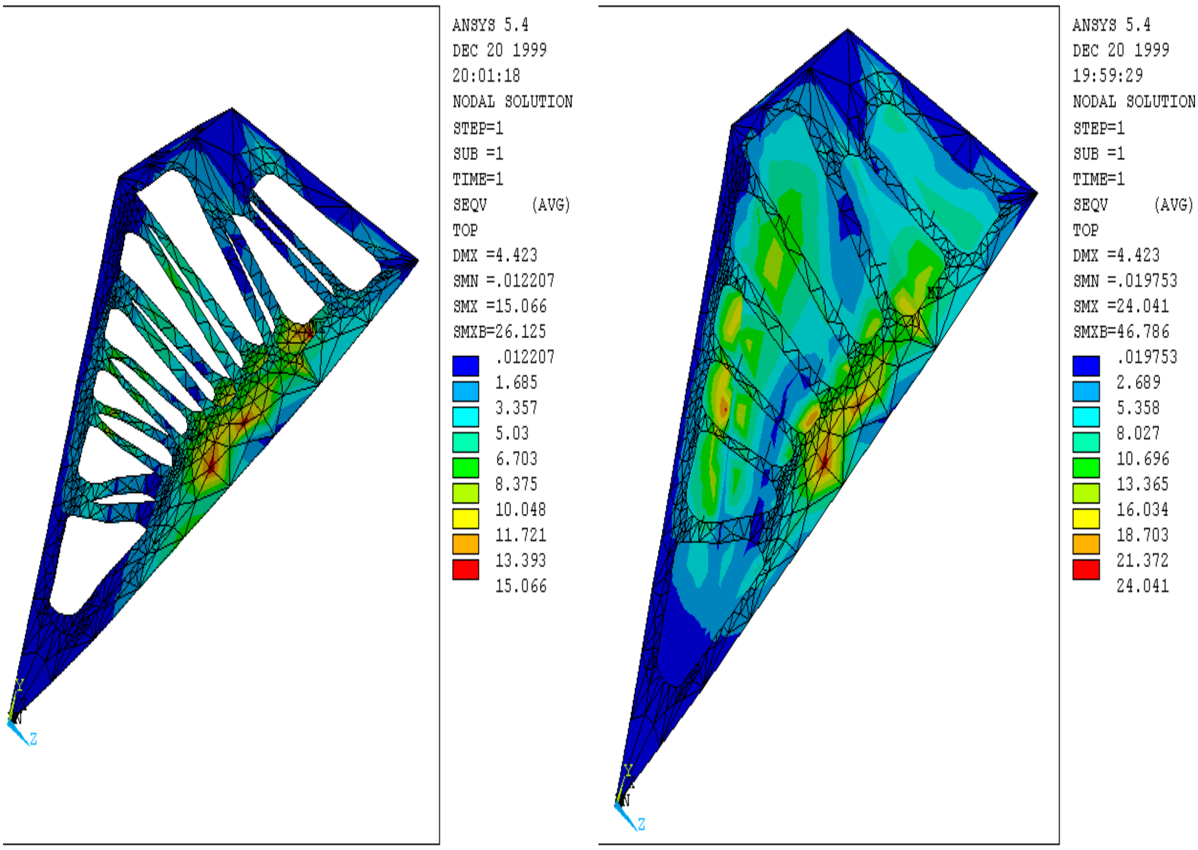

Fig. 4. – Distribution of Von Misses stress over skin and interior

By comparing numerical results, calculation and experimental tests, the high correlation degree of the results can be established.

Test Model

Usually before starting experiment, checking the test surface fixing of strain gages on that surface where was glued way or not and other accessories like cables, wire connection.. The experiment verification of that model with proper loading in laboratory conditions will be given in detail, as static test, the strain gages have been selected in such way distribute on the surface of model and the strain gage must be small enough comparing with model size and that will allow us to glued many strain gages over the model as vortices of triangle as shown in fig.5, Load is transferred to the fin surface through 18 points by means of 6 triangle sheets coupled with two bigger triangle sheets connected to the spar. Sideways the model is fixed in the same as for the model computation (the same boundary condition) for accuracy the surface where the strain gages was fixed must be cleaned by special material, the strain gage connection with cable to the A/D converter and to the computer. Arrangement of strain gages that placed on the upper structure surface should be compatible with sheet of triangular shape, which used to develop forces of uniform load distribution on the model. And the load introduced gradually

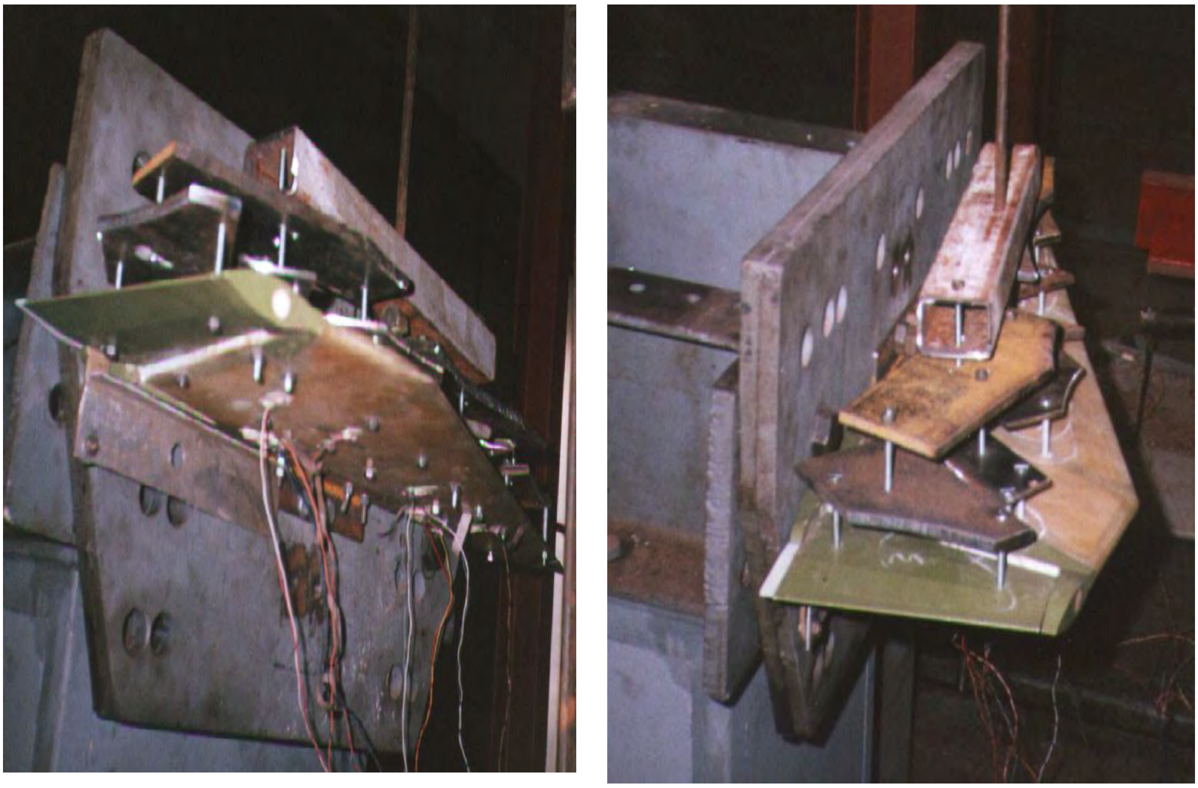

Fig. 5. – Test Model showing a mechanism for load distribution over fin surface

Experimental Result

The experiment has been done on the same model construct used for analysis, when the experimental incorrect start where the first reading taken by strain gages was with out any external loads, that means only the weight of the model with the accessories tools, which used for providing the experiment. By adding 125.4 daN the effected force will increase due to that the strain gages will take different reading according to their different position, as shown in Fig. 6 and then follow the same procedure by adding 125.4 daN in first four steps and then adding 31.35 daN until force becomes 627 daN.

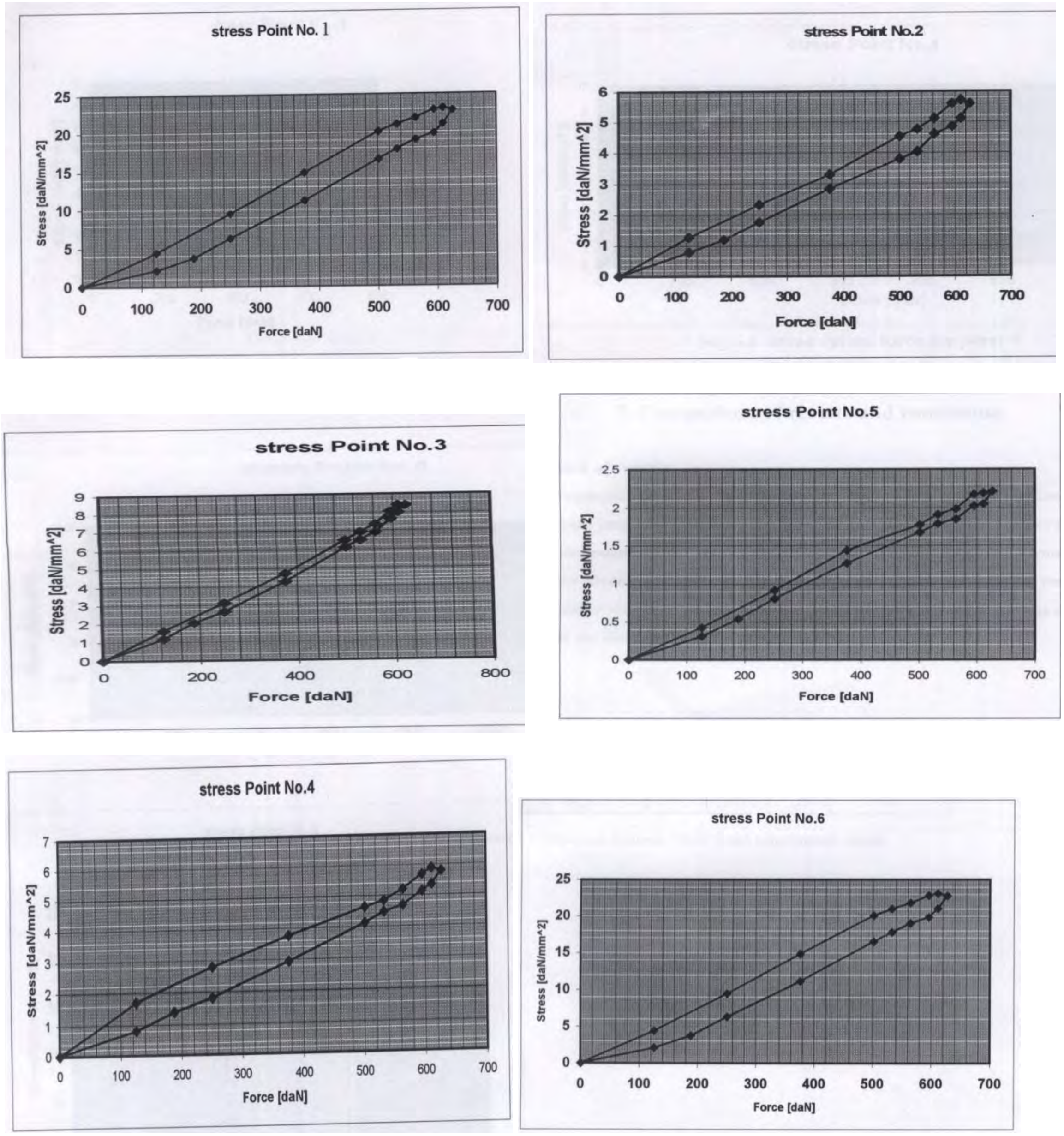

Fig 6. – Variation of measured stresses at the 6 strain gauges with applied load

To make a comparison between the experimental stress and the result obtained from the (ANSYS) software program, we chose the point of maximum load (627-daN). Table 1 shows the experimental results together with the numerical results. It is clear from the table that numerical values of stresses are in good agreement with experimentally measured stresses. At stress point 6 it is probably suffering from strain gauge malfunctioning.

Table 1. – Comparison between numerical analysis (ANSYS) and experiment

Strain gage	**1**	**2**	**3**	**4**	**5**	**6**
Experiment result	4.7	5.59	8.38	5.78	2.21	22.7
ANSYS result	4.7	5.56	8.6	4.9	2.3	16

Conclusion

The numerical and experimental work done in this paper for delta fin model can be concluded in the following points:

- Finite element method together with experimental testing are successfully implemented for the evaluation of stiffness of bearing shells of delta fin in supersonic speed.
- The ANSYS stress results obtained from modeling the structure have maximum value near fixing position.
- From the stress results the developed structural model is free out from any type of failure under applied loads and there is close agreement between numerical and experimental results.
- The most concentrated elements were observed at element area near fixing position so that area should consider as critical elements.
- The value of displacement increment out word from root chord of the model to the tip chord.
- It is worth to note that numerical and experimental results show good agreement.

REFERENCES

1. Constantine C. Spyrakos, 1994 - Finite element modeling in engineering practice.
2. Dr Mirko Josifovic, 1979 - osnovi strukturalne analize aerotehnickih konstrukcija.
3. B.E.Gatewood - virtual principles in aircraft structures volume 2 : Design, plates, finite elements.
4. Simic V. Srbuoljub, 1984 - contribution to estimate stresses state of rocket carrying surface using finite difference method.
5. Zoran C. Petrovic,1995 "Konstrukcija I opterecenje delta krila" Beograd.

SUMMARY

Finite element method is successfully used for the evaluation of stiffness of bearing shells of delta fin in supersonic speed. The wing is loaded from the system of evenly distributed vertical forces the bearing surface is of monolithic type, manufactured from homogenous and isotropic material, with solid shell and strong oblique spars. The fin structure is studied numerically using finite element method (ANSYS) and results are verified by comparison with experiment. Six strain gages were used to measure stresses over expected critical locations. Experimental and numerical results are in good agreement. It is also found that the model withstands the applied loads.

РЕЗЮМЕ

В проекте исследуется прочность ответственных элементов конструкций современного авиастроения. Рассматривается деформирование крыла сверхзвукового самолета. Крыло нагружается системой равномерно распределенных вертикальных сил, материал принят однородным и изотропным. С помощью метода конечных элементов численно исследуется напряженно-деформированное состояние ребра крыла. Установлено, что модель выдерживает приложенные нагрузки. Представлено сравнение полученных численных результатов с результатами натурных испытаний с помощью тензорезисторов. Экспериментальные и численные результаты хорошо согласуются (качественно и количественно). Погрешность для всех испытаний составила менее 3%.

E-mail: mos479@gmail.com
saif_matug@yahoo.com
K_kurmaji@yahoo.com

Поступила в редакцию 01.11.2014