STRESS AND MODAL ANALISYS REPORT FOR AVERT PROGRAM

Eng. Bogdan CALOIAN, INCAS, caloianb@incas.ro
Eng. Dorin LOZICI, INCAS, lozicid@incas.ro
Eng. Radu BISCA, INCAS, biscar@incas.ro

DOI: 10.13111/2066-8201.2009.1.1.6

Abstract

This report presents a modal and a static analysis of the assembly wing-flap ,using the finite element software's ***ANSYS 11 PATRAN-NASTRAN 2007****. The geometry was created in* ***Catia V5 R18*** *and imported. The results from the modal analysis refer to the natural frequencies and represent the displacement vector sum for each frequency. For the static analysis, 17 zones of pressure were chosen. For each zone the pressures values were calculated using the pressure distribution resulted from the INCAS subsonic wind tunnel experiments. For this analysis the results consist in the displacement vector sum and the Von Mises equivalent stress.*

Introduction

In the design of wing airfoils for transport aircrafts, achieving different requirements for distinct phases of the flight namely cruise flight on one side and takeoff and landing on the other side arises as a necessity. In cruise flight, where characteristic is $0.75 \leq Ma\infty \leq 0.85$, a large glide ratio for a prescribed lift coefficient is optimal, while the drag coefficient should be small in the sense of fuel savings. This requirement is achieved by slim profiles with small curvature. By contrast, for landing and especially for take-off, at low velocity magnitude, $Ma\infty \approx 0.2$, enough lift has to be produced in order to compensate the airplane weight. For this a considerable larger lift is necessary, hardly to be fulfilled by airfoils with large curvature. The disagreement may be solved through the usage of high lift systems as particular profiles at a certain offset.

Modeling

The geometry of the wing and flap was created in Catia V5 R18 [1].

Fig.1
Isometric View of the wing-flap assembly

FEM Idealization

The Finite Element Model was realized using ANSYS11 [2] and Patran 2007 [3].

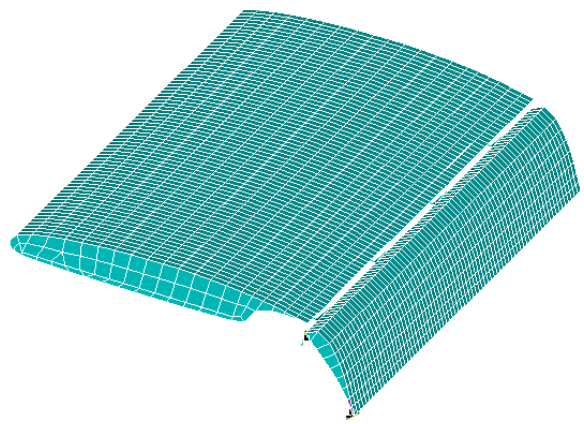

Fig.2
Wing-flap FEM idealization using ANSYS 11

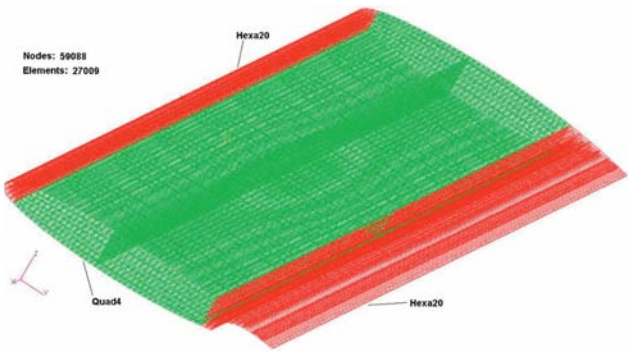

Fig.3
Wing FEM idealization using Patran 2007

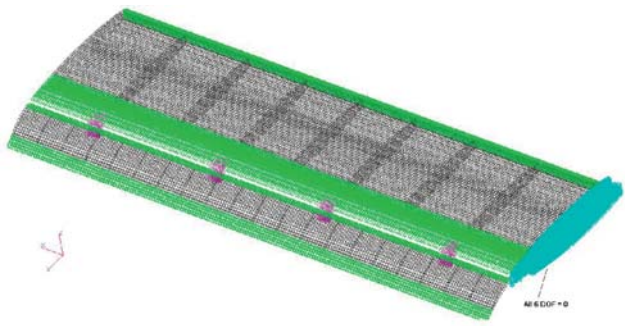

Fig.4
Flap FEM idealization using Patran 2007

In **ANSYS** the types of elements used: SHELL63 Element - Elastic Shell and BEAM4 Element Description - 3D Elastic Beam. SHELL63 have both bending and membrane capabilities. Both in plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included.
A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses. BEAM4 is a uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. Stress stiffening and large deflection capabilities are
Included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses.
Number of nodes used: 5270.
Number of elements used: 5847.
Real constants: R1 = 3 mm R2 = 2 mm R3 = 5 mm R4 = 12 mm R5 = 24 mm R6:= 480 mm² , 16000 mm$_4$, 16000 mm$_4$.
SHELL63 ELEMENT ON: R1,R2,R3,R4,R5
BEAM4 ELEMENT ON: R6
In Patran elements HEXA20, QUAD4 and RBE2 Have been used.

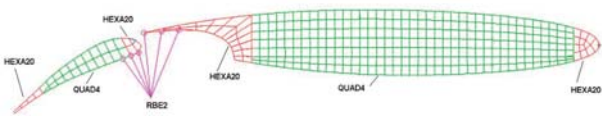

Fig.5
Element type distribution

Material used. Characteristics.
Type of material: Aluminum alloy 7075, T651 – for wing
Density = 2800 (kg/m³)
Poisson ratio = 0.33
E = 70000 N/mm²
Type of material: X39CrMo17 – 1 - for flap and hinges
Density = 7700 (kg/m³) and Poisson ratio = 0.3
E = 215000 N/mm²

MODAL ANALYSIS

We use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient
dynamic analysis, a harmonic response analysis, or a spectrum analysis. Modal analysis can be made on a prestressed structure, such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, enabling to review the mode shapes of a cyclically symmetric structure by modeling just a sector of it. Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, are ignored even if they are defined. There is the possibility of choice from several mode extraction methods: Block Lanczos (default), subspace, PowerDynamics, reduced, unsymmetrical, damped, and QR damped. The damped and QR damped methods enable to include damping in the structure. The QR Damped method also allows for unsymmetrical damping and stiffness matrices.

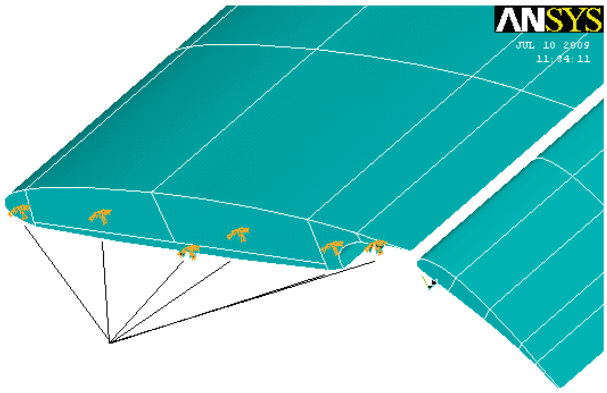

Fig.6
Boundary conditions in ANSYS

All DOF constrained with 0 displacement. The left side of the wing has been blocked by constraining it with 0 degrees of freedom. Mode extraction method: Block Lanczos
Number of modes to extract: 10.
Number of modes to expand: 10.
Start Frequency: 0 Hz
End Frequency: 1000 Hz

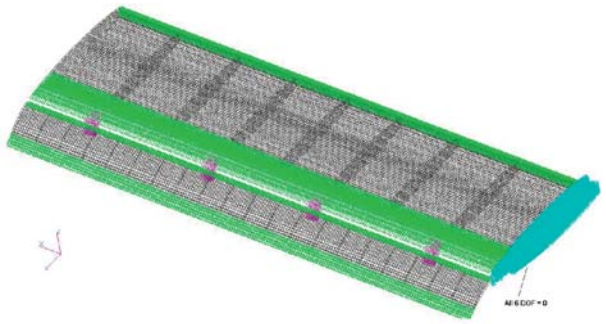

Fig.7
Boundary conditions in Patran

Results for the Modal Analysis

1. Wing & flap general bending mode deformation, at 13.929 Hz frequency in ANSYS.

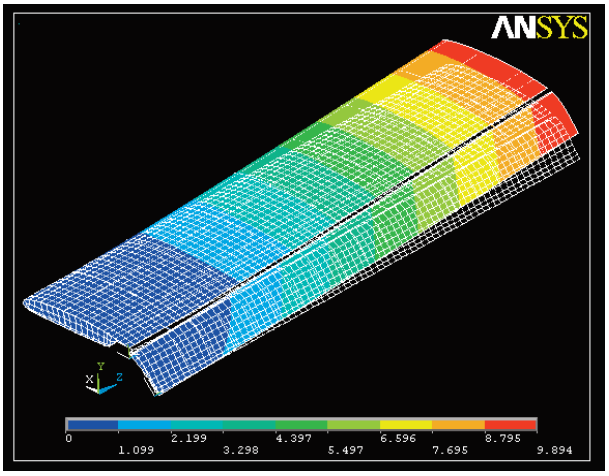

Fig.8
Mode 1 – General bending in ANSYS

Wing & flap general bending mode deformation, bending mode deformation, at 13.997 Hz frequency in Patran
The frequency obtained in ANSYS for this mode 13.929 Hz, is almost identical with the one obtained in Patran: 13.997 Hz.

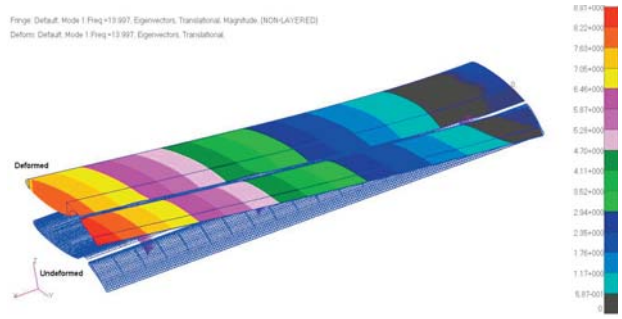

Fig.9
Mode 1 – General bending in Patran

2.General flap rotational mode deformation, at 2nd 37.271 Hz frequency in ANSYS 11.

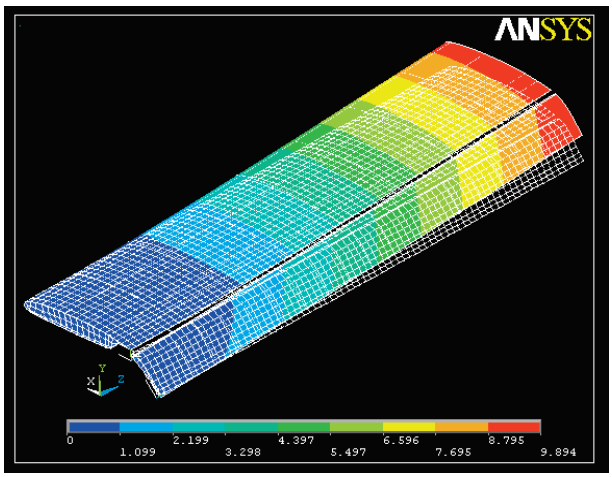

Fig.10
Mode 1 – General flap rotation in ANSYS11

General flap rotational mode deformation, at 2nd 38.652 Hz frequency.

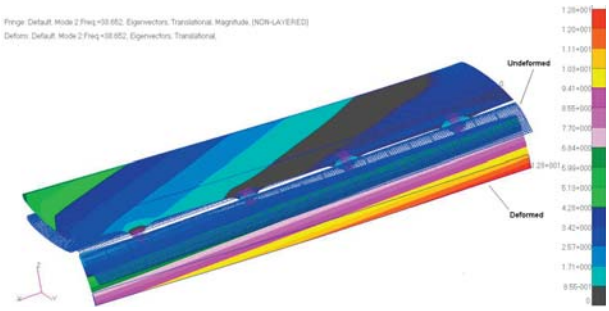

Fig.11
Mode 1 – General flap rotation in Patran

As in the previous mode, in this mode also the Frequency obtained in Patran and the frequency obtained in ANSYS are very alike.

STATIC ANALYSIS

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes). Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects . Steady loading and response conditions are assumed; that is, the loads and the structure response are assumed to vary slowly. The boundary conditions for the Static Analysis are the same both in ANSYS and Patran and in the previous Modal Analysis.

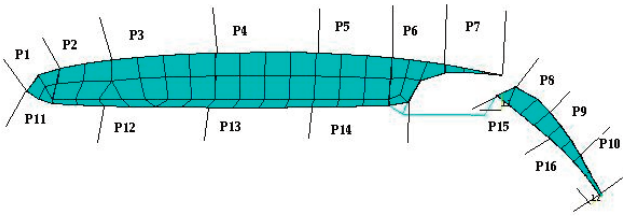

Fig.12
The defined areas to apply the pressures

Pressure Distribution at 50m/s and an angle of attack of 8.12° :

P1 = 0.002295 MPa P2 = 0.0016065 MPa
P3 = 0.00153 MPa P4 = 0.00153 MPa
P5 = 0.00153 MPa P6 = 0.00153 MPa
P7 = 0.002142 MPa P8 = 0.001377 MPa
P9 = 0.001224 MPa P10 = 0.001071 MPa
P11 = 0.001224 MPa P12 = 0.000918 MPa
P13 = 0.000918 MPa P14 = 0.001224 MPa
P15 = 0.001377 MPa P16 = 0.001071 MPa

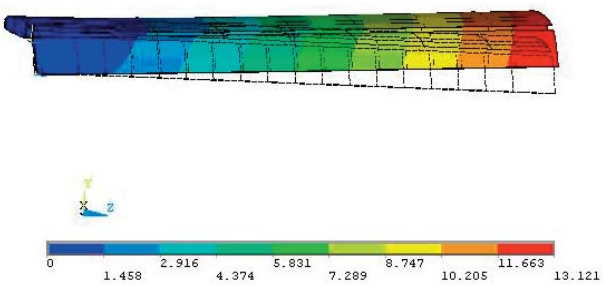

Fig.13
Displacements Vector Sum.

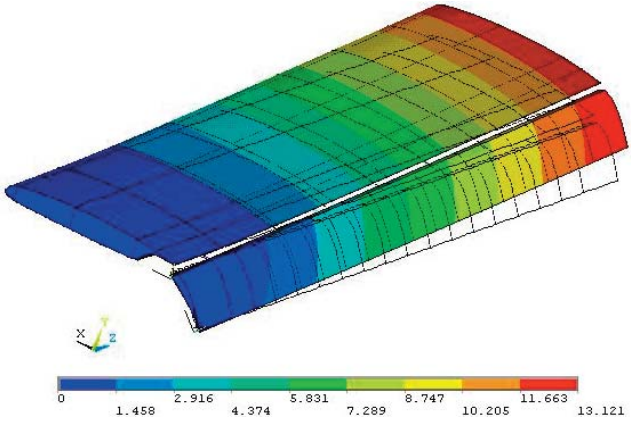

Fig.14
Maximum displacement : 13.121 mm.

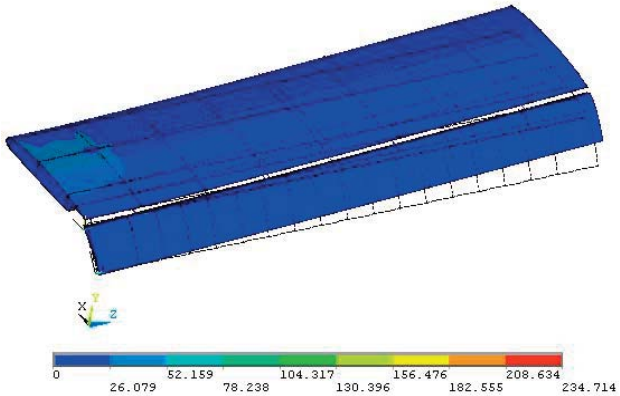

Fig.15
Equivalent Von Mises stress for flap.

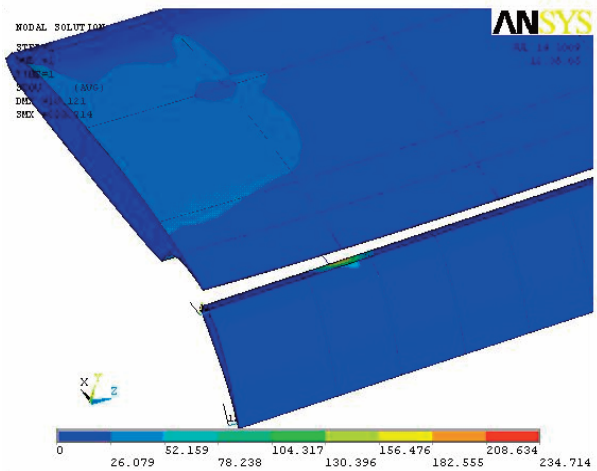

Fig.16
The maxim Von Mises stress of 234.714 MPa for flap.

High value stresses are in hinge locations, but the highest value is in the first hinge from the clamped base.

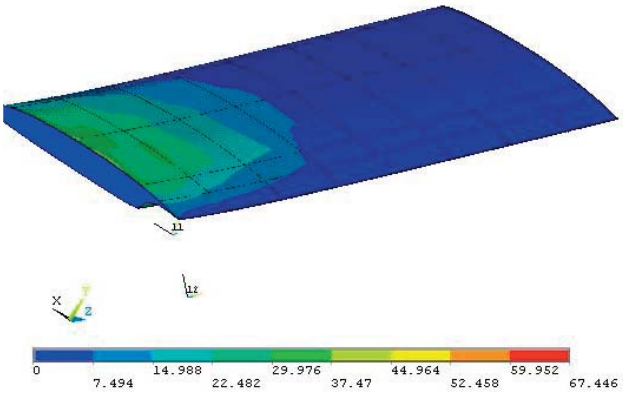

Fig.17
Von Misses stress for wing.

The maximum von Misses Stress for the wing is 67.446 MPa.
Using **Nastran - Patran** we obtained the Von Misses stress as it follows :

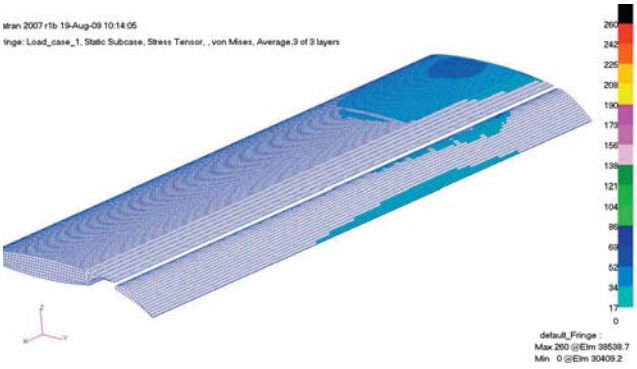

Fig.18
Von Misses stress for wing using Nastran-Patran

Conclusions

Margin of safety for flap will be:
MS= 1200/234-1= 4.12.
The margin of safety for wing is :
MS = 567/67.45-1=7.4 .
By a visual inspection of the resulted and allowable stresses, we conclude that the margin of safety is very high.

REFERENCES

[1] **CATIA V5 R18.** ***PROVIDES AN INTEGRATED SUITE OF COMPUTER AIDED DESIGN (CAD), COMPUTER AIDED ENGINEERING (CAE), AND COMPUTER AIDED MANUFACTURING (CAM) APPLICATIONS FOR DIGITAL PRODUCT DEFINITION AND SIMULATION***

[2] **ANSYS 11.** ***A SOFTWARE PACKAGE USED IN FINITE ELEMENT ANALYSIS***

[3] **NASTRAN - PATRAN 2007.** ***A SOFTWARE PACKAGE USED IN FINITE ELEMENT ANALYSIS***

[4] ***RESULTS OF THE EXPERIMENTS PERFORMED AT INCAS WIND TUNNEL***