Advanced Techniques of Stress Analysis

Simion TATARU*^,1^, Daniela BARAN[2], Dorin LOZICI-BRANZEI[2]

*Corresponding author
*,1 Aerospace Consulting, 061126 Bucharest, Romania
sitataru@incas.ro
[2] INCAS – National Institute for Aerospace Research "Elie Carafoli"
B-dul Iuliu Maniu 220, Bucharest 061126, Romania
dbaran@incas.ro, lozicid@incas.ro

DOI: 10.13111/2066-8201.2013.5.4.7

Abstract: *This article aims to check the stress analysis technique based on 3D models also making a comparison with the traditional technique which utilizes a model built directly into the stress analysis program. This comparison of the two methods will be made with reference to the rear fuselage of IAR-99 aircraft, structure with a high degree of complexity which allows a meaningful evaluation of both approaches. Three updated databases are envisaged: the database having the idealized model obtained using ANSYS and working directly on documentation, without automatic generation of nodes and elements (with few exceptions), the rear fuselage database (performed at this stage) obtained with Pro/ ENGINEER and the one obtained by using ANSYS with the second database. Then, each of the three databases will be used according to arising necessities.*
The main objective is to develop the parameterized model of the rear fuselage using the computer aided design software Pro/ ENGINEER. A review of research regarding the use of virtual reality with the interactive analysis performed by the finite element method is made to show the state- of- the-art achieved in this field.

Key Works: *analysis stress, safety margins, parametric modeling*

1. INTRODUCTION

Worldwide, the current trend in aircraft construction is the creation of parameterized models leading to optimization of projects, reduced costs of design and execution, and ultimately, an increased quality of products.

The aircraft operating conditions lead to disruptive phenomena (vibrations, high inertial forces) whose effects must be known and controlled.

In order to know and control the effects of different dynamic loads software that simulates the behavior of different structures, assemblies and subassemblies can be used. Specific situations that occur in various parts of the structure since the design phase can be identified thus reducing the number of prototypes and necessary subsequent changes.

To this end the ANSYS software is used as it can cover the whole range of problems of stress and vibration analysis for various assemblies and subassemblies.

Using a computer aided design program such as CAD / CAM, geometric patterns of the desired structure of the aircraft are achieved (e.g. fuselage); then this model is utilized to build a finite element model and to perform the static stress analysis which can also be completed with vibration analysis.

The results obtained with ANSYS are accurate and have a high degree of accuracy.

Large aircraft manufacturers use parametric models in their research and design activities. Parametric models allow shortening the design cycles, quick analysis and optimization project, thus reducing costs and increasing the product quality.

This comparison of the two methods will be made with reference to the rear fuselage of a school and trainer aircraft, which is a structure with a high degree of complexity allowing a meaningful evaluation of both approaches.

The improved methodology can be used in many different areas such as research - design and optimization of motor vehicles, agricultural machinery, energy, etc.

1.1 Regulatory Requirements

IAR-99 aircraft was designed to meet the requirements of British military Regulation AvP970; for some issues for which there weren't any provisions the American regulation MIL has been used namely:

i) For the ultimate load which are the calculated operational loads (specified loads) multiplied by the factor of safety (ultimate factor) denoted by j_u

(ii) For the proof loads consisting of specified loads multiplied by the proof factor denoted by j_{pr}

Between the two factors of safety (ultimate factors) there is the following relation:

$$j_{pr} = 0.75\, j_u$$

The structure tested under the above mentioned loads must meet the following conditions:

(i) At the "proof" load level, the emergence of strains that threaten the structure security is not permitted; moving parts essential to the security of the aircraft shall function properly. After removing the "proof" load permanent deformations should not remain.

(ii) Failure / breakage (collapse) of the structure may not occur before achieving the "ultimate" level of load.

The aircraft structure has the following general safety factors in symmetric flight maneuvers (Chapter 201, ref. [2]), asymmetric flight maneuvers (Chapter 202, ref. [2]), in flight through atmospheric turbulence (Chapter 203) and in other cases.

$$j_u = 1.5 \text{ si } j_{pr} = 1.125$$

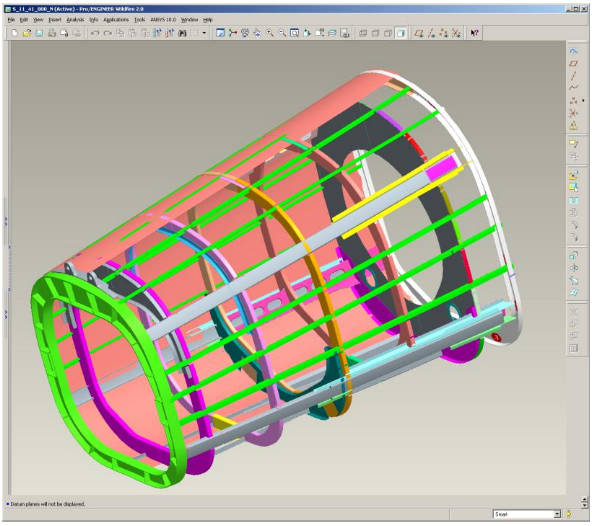
1.2 Materials Utilised in Stress Calculation of the Rear Fuselage Structure

Crt no	Material Specifications	Preform	Finished part	Mechanical properties				
				σ_u [daN/cm^2]	τ_u [daN/cm^2]	$\sigma_{0.2}$ [daN/cm^2]	E [daN/cm^2]	δ_5 [%]
1	3.1354.T351 LN9073	Laminated boards	Milled frames ribs	3900	2600	2600	735000	3.5
2	3.1364.T42 LN9073	Skin plate	Shape frames coatings	4000	2400	2400	735000	14
3	3.1354.T3 LN9073	Plate	Floor	4300	2900	2800	735000	14
4	3.1354.T351 LN9496	Rolled profiled L, I, U	Ledges, spars, stiffeners	4000	2100	2700	735000	5 ÷ 12
5	1.7220.5	Bars	Bolts,nuts	9000-11000	5400	7500	2.1×10^6	14
6	1.7734.6	Bars	Assembly parts, metal fittings, bolts	10500÷11000	6700÷7000	9000÷9500	2.1×10^6	10
7	3.3214.7 LN9073	Plate	Careening n of tank covering	3000	1900	2500	696000	10

8.	1.7734.4	Sheet, Plate	Spars,frames	9800	5800	7800	2060000	10
9.	1.7784.6	Bars and forging	Metal fittings frames	18000-20000	10800-12000	15000	2060000	7

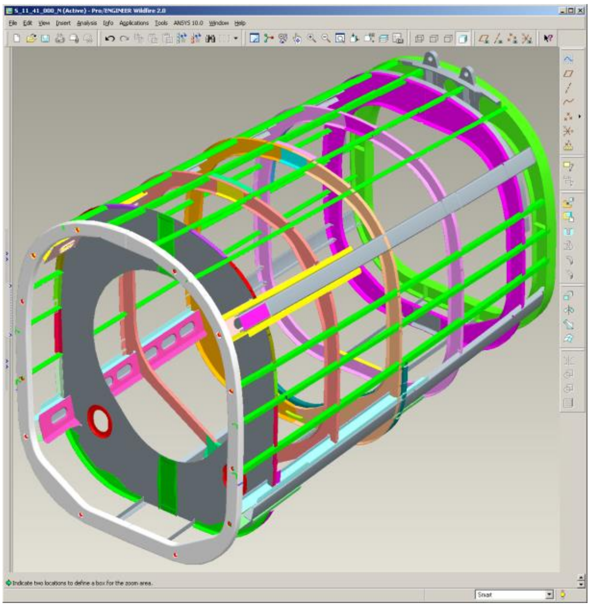
1.3 Parametric Modeling of the Rear Fuselage

The modeling process is characteristic for such a structure; the starting point is the established shape of the studied fuselage sector. Once defined the fuselage “skin” the internal components are defined and shaped while keeping the connection with the outer surface. The first step consists in introduction of the curves defining the frames, after which the surface modeling is performed with a feature like "The Boundary Blend Tool" similar to the model shown in Figure 1 and 2.



Datum planes will not be displayed.

Fig. 1 Structural model of the rear fuselage

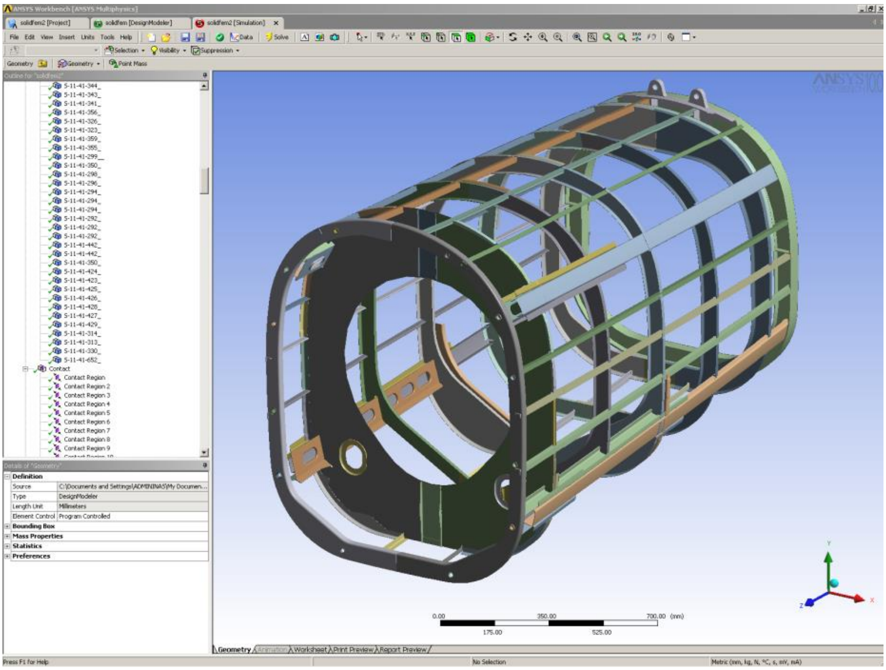


Indicate two locations to define a box for the zoom area.

Fig. 2 Parametric structural model of the rear fuselage

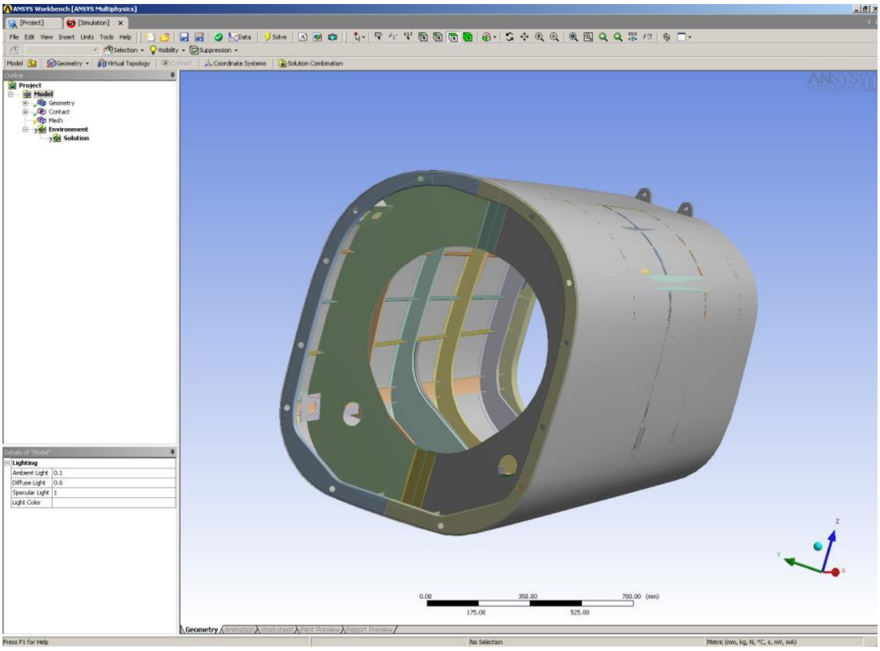
1.4 Transfer of Geometrical Model into ANSYS

The integration of ANSYS Workbench v10.0 into modeling software CAD Pro/ ENGINEER Wildfire v2.0 allows launching the analysis directly from the CAD environment. Transferring the geometry is easy and can be performed in a relatively short time. (Figure 3, Figure 4)



Definition	
Source	C:\Documents and Settings\ADMININAS\My Documen...
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Bounding Box	
Mass Properties	
Statistics	
Preferences	

Fig. 3



Lighting	
Ambient Light	0.1
Diffuse Light	0.6
Specular Light	1
Light Color	

Fig. 4

1.5 Safety Margin

Effective stresses occurring in the rear fuselage strength structure are calculated at the "proof" level (j = 1).

In this paper the effective equivalent stresses (SEQV) are compared with allowable stresses (σ_{adm}) for the materials used.

The equivalent stress is given by relation:

$$SEQV = \left\{ \frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma 3)^2 + (\sigma_3 - \sigma_1)^2 \right] \right\}^{1/2}$$

where σ_1, σ_2, σ_3 are the main stresses along the three directions.

The safety margin is calculated using the relation:

$$MS = \frac{\sigma_{adm}}{SEQV} - 1.$$

The summary table gives the minimum values of safety margins (MS) for the main elements of structural strength for the rear fuselage, from which it can be concluded that the regulatory computing requirements are satisfied, meaning that the positive safety margins (MS) are obtained, namely MS > 0.

Summary table [T1] giving the minimum values of safety margins (MS) for the rear fuselage structure of IAR-99 SOIM (Hawk) aircraft.

Table T1

Crt No	Name of the structural element	SEQV [daN/cm^2]	σ_{adm} [daN/cm^2]	$M_S = \frac{\sigma_{adm}}{SEQV}$ -1
1	Frame C29 (X=902.9 cm)	667	2700	3,04
2	Frame C30 (X=903,5 cm)	4027	7800	0,94
3	Frame C31 (X=924,5 cm)	820	2700	2,29
4	Frame C32 (X=957cm)	1286	2700	1,09
5	Frame C33 (X=989,5 cm)	921	2700	1,93
6	Frame C34 (X=1023cm)	873	2700	2,02
7	Frame C35A (X=1056,5 cm)	1414	2700	0,90
8	Frame C35 (X=1069,5 cm)	1151	2700	1,34
9	Frame C36 (X=1097,5 cm)	574	2700	3,70
10	Frame C37 (X=1131 cm)	576	2700	3,68
11	Frame C38 (X=1157,1 cm)	1905	2700	0,42
12	Frame C39 (X=1184,1 cm)	4374	7800	0,78
13	Frame C40 (X=1211,5 cm)	1952	2700	0,38
14	Rear fuselage skin	2268	2700	0,19

2. ANALYSIS OF STRESS AND STRAIN OF REAR FUSELAGE STRENGTH STRUCTURE

2.1 Development of the idealized model of the strength structure for the rear fuselage. Calculation method

The analysis of stress and strain of the rear fuselage strength structure under the action of inertial aerodynamic loads and of those induced by the horizontal and vertical tail group in the specified calculation cases is based on the displacements method (FEM method) and on the engineering theory of materials strength .Using the displacements method involves the idealization of the rear fuselage strength structure in finite elements so that its continuous structure is replaced by a mechanical system with a finite number of degrees of freedom. Performing the analysis using the finite element method (ANSYS software) implies the following steps:

(a) To determine the coordinates of idealized fuselage nodes the following convention was used: each frame nodes were "named" by the frame number, the last two digits representing the number of the point on the frame (e.g. 3010 node is node 10 on the C30). The exceptions to this rule are nodes of C35A frame for which the first two digits are 45 (e.g. 4523 node is node 23 on the C35 A frame).

In the case of the rear fuselage three types of finite elements were used which connect these structural nodes.

SHELL 63. This type of item absorbs bending and membrane stresses being allowed both the forces in the element plane and the normal forces. Each node has six degrees of freedom: translations along the nodal directions X, Y, Z and rotations about the X, Y, and Z axes. These elements were used for coating panels, frames cores, spars, etc.

(b) LINK 8 is a uniaxial element that absorbs tensile and compressive stresses, with three degrees of freedom per node. It was used for ledges, structural base of frames, diaphragms and spars. Depending on the application, the element can be thought of as part of the bar, junction or arc element, etc.

(c) 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.

The element is defined by two or three nodes, the cross-sectional area, two moments of inertia (IZZ and IYY), two thicknesses (TKY and TKZ), an angle of orientation (θ) about the element X-X axis, the torsional moment of inertia (IXX), and the material properties.

If IXX is not specified, it is assumed equal to the polar moment of inertia (IYY+IZZ). IXX should be positive and is usually less than the polar moment of inertia. The element torsional stiffness decreases with decreasing values of IXX.

The following gives the meshing of the strength structure basics of the analyzed rear fuselage, together with the geometrical features and structural nodes coordinates needed for describing each element (see PREP 7).

Using parametric models allows:

Determination of stresses that occur in different parts of the structure to be statically analyzed (tension, deformation). Models were developed to perform the stress calculation: a 3D model performed with a computer aided design program such as Pro/ ENGINEER, which was then transferred into a stress analysis program (ANSYS) and a model directly realized into ANSYS.

We aimed to create models as complex as possible and similar to the rear fuselage structure of a school and trainer aircraft.

Computing loads have been established and stress analyzes were developed in parallel, the results were compared and advantages and disadvantages of each approach were stated in order to recommend the path to be followed for future projects depending on their objectives for each product.

At this stage special phenomena are also analyzed such as the stress concentration or the way of quick checking the project changes.

The project leads to the development of the parameterized models in aviation, concept utilized world-wide in the aviation industry and within the European research programs.

Based on the existent documentation parameterized models have been elaborated utilizing a CAD/CAM and ANSYS transfer program; definition of loading cases and of the stress on the fuselage have been established.

By a model directly built within the program utilized for the strength analysis, the technology based on the parameterized models has been verified and compared to the traditional strength analysis calculus technology.

The time of execution for the geometric model is substantially reduced because the new parameterization products are friendly, have a great number of characteristics and are designed to obtain rapidity in operation.

The precision of these analyses are guaranteed by the accuracy of parameterized modeling with Pro/ ENGINEER and by the utilization of ANSYS products in solving this type of analysis.

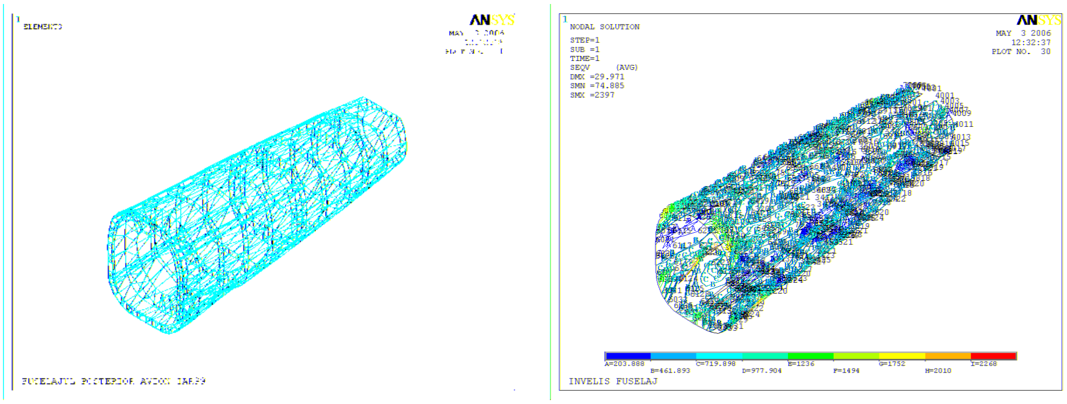


Fig. 5 Teaditional method

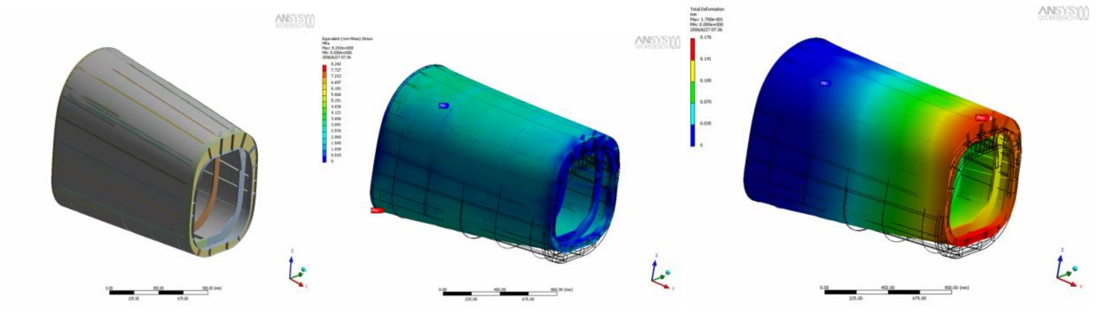


Fig. 6 Parametrized models method

3. CONCLUSIONS

Using the results obtained by direct analysis (the calculated safety margins from Table 1) it can be concluded that the regulatory computing requirements are satisfied, meaning that the positive safety margins (MS) are obtained, namely MS> 0.

The analysis carried out shows the following findings:

1) The parametric model (analysis 2) is much bigger (the direct model has about 2000 nodes and elements as compared to 20,000 nodes and elements for the parametric model). Because of this the analysis using the parametric model takes longer than the direct model analysis.

2) Displacements in analysis 2 are somewhat larger than those in analysis 1, but as expected tensions are slightly lower. A more exact calculation leads to lower tensions. Precisely calculated displacement gradient leads to lower tensions.

3) Following the conclusion 2 it results that safety margins calculated with a less accurate analysis are conservative.

4) As Analysis 2 performs a modeling closer to the cutting geometry it allows a more accurate assessment of tensions in these regions.

5) Changes that may occur during the development of a project can be entered a little easier and more precisely on a parametric model.

4. REFERENCES

[1] *** ANSYS User's Manual for Revision 5.0 – Volume I, II, III, IV.

[2] *** AvP-970- Design Requirements for Service Aircraft, vol. **I**. Airplane Design Requirements, 1975.

[3] *** MIL-F-8785 (ASG), Military Specification-Flying Qualities of Piloted Airplanes, 1969.

[4] *** C-2205/ 06.1999, *Model idealizare structura pentru estimarea duratei de viata a structurii avionului IAR-99,* INCAS Paper.

[5] J. Wright, J. Cooper, *Introduction to Aircraft Aeroelasticity and Loads***,** John Wiley & Sons, Ltd., 2007.

[6] D. P. Raymer, *Aircraft Design***.** *A conceptual Approach***.** AIAA Education Series, 1992.

[7] D. Lozici-Branzei, D. Baran, S. Tataru, Flutter Analysis of the IAR-99 SOIM aircraft, *INCAS Bulletin*, volume **5**, Issue 2, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, DOI: 10.13111/2066-8201.2013.5.2.5, pp. 33-43, 2013.

[8.] D. Dinca Baran, Mathematical Models Used in Studying the Chaotic Vibration of Buckled Beams, *Mechanics Research Communications*, Vol. **21**, Issue 2, March-April, pp. 189-196, 1994.

[9]. D. Baran, M. Stere, S. Tataru, D. Lozici-Branzei, N. Apostolescu, R. Bisca, B. Caloian, Design&Analysis Tools and Techniques for Aerospace Structures in a 3D Environment, *INCAS Bulletin*, Volume **1**, no. 1, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, DOI: 10.13111/2066-8201.2009.1.1.17, pp. 93-95, 2009.

[10]. S. Tataru, D. Lozici-Branzei, D. Baran, M. Stere, Predictive Assessment of an Aircraft Structure Behaviour, *INCAS Bulletin*, Volume **1**, no 2, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, DOI: 10.13111/2066-8201.2009.1.2.18, pp. 129-135, 2009.

[11] D. Lozici-Brinzei, S. Tataru, R. Bisca, *IAR-99 Ground Vibration Tests and Dynamics, Finite Element Model* -AFASES 2011.

[12] D. Lozici-Brinzei, R. Bisca, S. Tataru, Loads in the Design of Flight Vehicles, *INCAS Bulletin*, Volume **2**, no 3, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, DOI: 10.13111/2066-8201.2010.2.3.6, pp. 51-59, 2010.

[13] D. Baran, D. Lozici-Brinzei, S. Tataru, Dynamic study of the virtual prototype of the IAR-99 SOIM Aircraft, *INCAS Bulletin,* Volume **5,** Issue 3, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, DOI: 10.13111/2066-8201.2013.5.3.1, pp. 3-11, 2013.