CFD Analysis of UAV Flying Wing

Vasile PRISACARIU*

*Corresponding author
"Henri Coandă" Air Force Academy of Braşov, Romania,
Mihai Viteazul, nr. 160, Brasov 500187, Romania
aerosavelli73@yahoo.com

DOI: 10.13111/2066-8201.2016.8.3.6

Received: 27 July 2016/ Accepted: 26 August 2016/ Published: September 2016

***Abstract**: Numerical methods for solving equations describing the evolution of 3D fluid experienced a significant development closely related to the progress of information systems. Today, especially in the field of fluid mechanics, numerical simulations allow the study of gas-thermodynamic confirmed by experimental techniques in wind tunnel conditions and actual flight tests for modeling complex aircraft. The article shows a case of numerical analysis of the lifting surface on the UAV type flying wing.*

***Key Words**: mesh networks, CFD analysis, UAV, flying wing.*

NOTATIONS

F_z	-	lift force	AoA	-	angle of incidence
F_y	-	lateral force	F_x	-	drag force
V_x	-	speed	c	-	chord

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is a field of fluid mechanics that uses numerical methods and algorithms to analyze problems that involve fluid flows. IT systems are used to perform the calculations required to simulate fluid-fluid interaction, fluid-solid with defined surface boundary conditions. The software tools provide high yields of research that improve accuracy and simulation speed in user-defined scenarios. Validation is performed using numerical simulations tests in wind tunnels and then in the virtual and real flight tests [17, 19, 20]. The analysis of CFD comprises three main stages: the first stage is the pre-processing, offering the solver all the information necessary for computing.

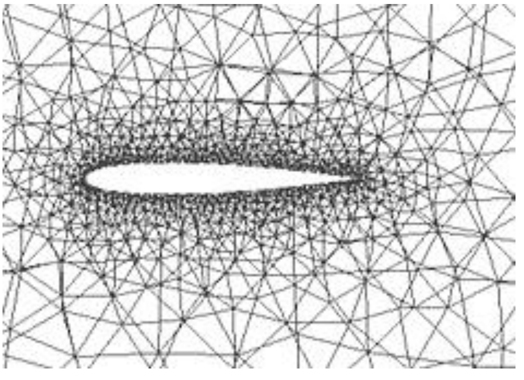
The pre-processing comprises the geometry generation (the calculation of the input parameters: flow conditions, thermal parameters, the generation of the mesh network (Figure 1) and the physical modeling (equations of motion, boundary conditions); the second stage includes the numerical analysis itself or the numerical solutions generation (resolving iterative of the partial differential equations) and the last step is the post-processing analysis, visualization of the solution (numerical results or graphs) and the generation of the final reports that include all the information about CFD analysis (initial parameters, hardware configuration, partial and final results), [1].

CFD modeling is based on solving a series of differential equations (conservation) and completed models for the treatment of turbulence, pressure, cavitations, heat exchange and dispersed phases.

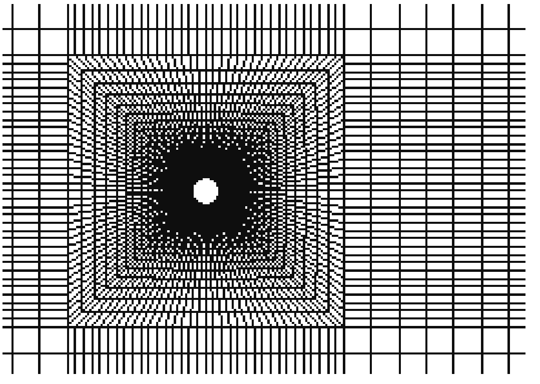
Shaped areas are divided into small cells resulting in mesh networks with more nodes. The equations are written for each node; they are assembled into an overall system of equations, which is then resolved.

The stability of selected mesh is generated numerically and analytically in simple linear problems to ensure discontinuous discrete solutions.

The most common methods of meshing are finite volume methods, finite element methods (dimensional, three-dimensional, spectral), methods of finite differences and boundary element method [2, 3, and 13].



a. unstructured network



b. structured network

Fig. 1 Types of meshing domain

2. THE CURRENT STAGE ON CFD ANALYSIS

In the 1980s the birth of supercomputer enabled the application of techniques in fluid dynamics research efforts in the area of industrial design. NASA Ames Research Center (Moffett Field, California), has an entire division dedicated to the development and validation of applying CFD techniques.

In aerodynamics, gas dynamics numerical simulations allow the study of the conditions confirmed by experimental techniques in wind tunnels and real flight tests for modeling complex aircraft evolutions.

CFD analysis is widely used in both research agencies and academia in European and international communities [10, 14].

According to [4, 5, 11, 12] we can select a number of specific CFD software tools that are used in aero and hydrodynamics, see examples in Figures 2 and 3:

- *commercial codes:* generators of grids (ICEM-CFD/ANSYS, CFD-Geom/ESI group, Automesh4/NUMECA, Pointwise, ADINA-AUI), solvers (CFD-ACE/ESI group, CFD-FASTRAN/ESI group, CFX/ANSYS, COMSOL Multiphysics, Fluent/ANSYS), data visualization (CFD-VIEW/ESI group, CFX-Post/ANSYS, COMSOL, Hyperview, Tecplot) [8, 11, 18].

- *free/ open source codes:* grid generators (Engrid, NETGEN) solvers (CFD2D, ELMER, OpenFoam, SU2), data visualization (GnuPlot, Paraview) or integrated (XFLR5, Stallion 3D/Hanley Innovation), [15, 16].

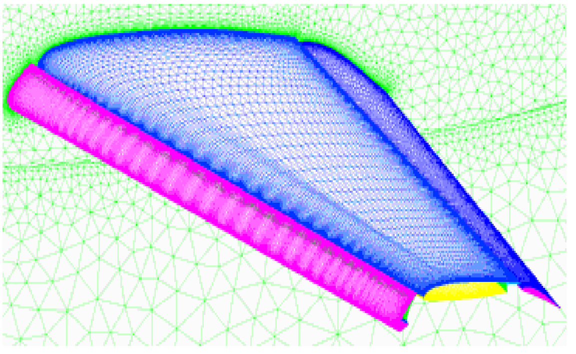


Fig. 2 Meshing wing – Pointwise CFD Mesher [6]

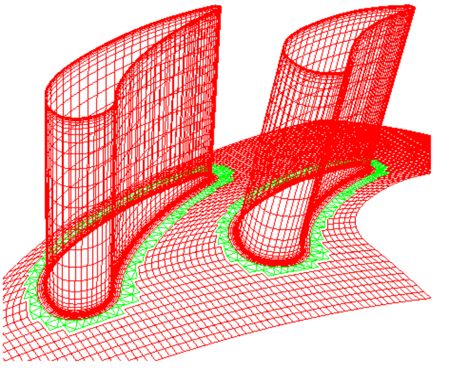


Fig. 3 Meshing turbine blades – DRAGON [7]

For a highly accurate simulation input data must be clearly outlined. An increase in detail creates a corresponding increase response time for the calculations and memory requirements of the IT system.

Simulations can have different system requirements: multicore processors with speeds above 2 GHz extremely high memory, from a few gigabytes (GB) of RAM for a typical simulation up to 128 GB of RAM for more detailed models. On a single processor, a typical simulation may take several hours or even days, and this time can be reduced by running it in parallel on multiple processors (clusters), see Table 1.

Table 1. Features PC

Features	Values					
CPU cores	4	8	16	32	64	128
CFD size (cells)	>2 mil	2...5 mil	4...10 mil	4...20 mil	30...40 mil	70...100 mil
RAM memory	2..3 GB	6...8 GB	10...15 GB	20...30 GB	40...60 GB	100..150 GB

We performed a series of tests with SolidWorks 2013 - Floworks (64-bit) on a computer system equipped with Ivy Bridge i7 processor 2.4-3.4 Ghz processing time results are valid for 6 of the 8 simultaneous tasks of the processor for a 32 GB RAM memory; it also can be observed a percentage increase of RAM with the increase of the grid mesh cells number, see table 2, [9].

Table 2. Processing time

CFD size (cells)	16 mil	8 mil
RAM memory	68%	51%
Time on	122 h	57 h
CFD size (cells)	16 mil	8 mil

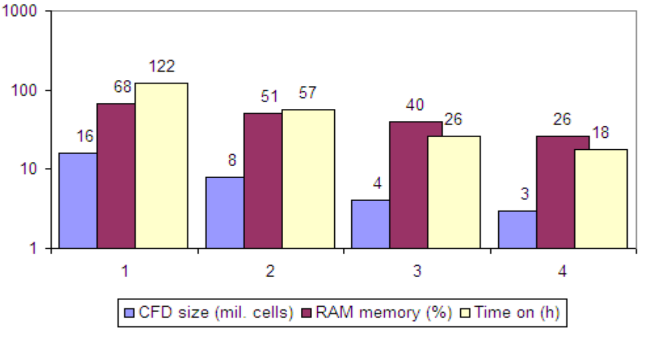


Fig. 4 The graph processing time

3. CFD ANALYSIS

3.1. 2D CFD Analysis of an Airfoil

We propose to review the Clark Y airfoil type (see Figure 5) with the conditions of analysis presented in Table 3.

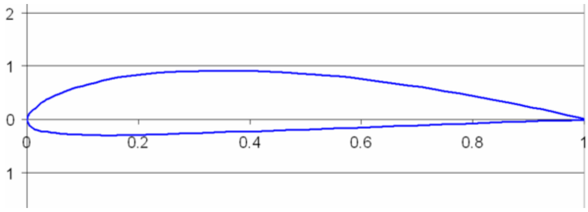
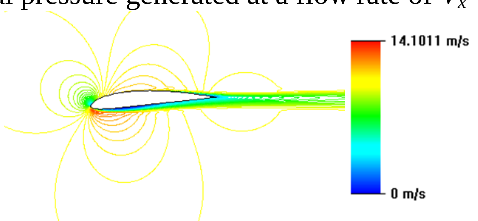


Fig. 5 Clark Y Airfoil

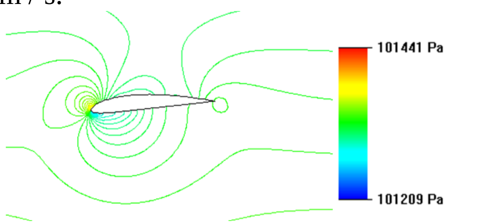
Table 3. Conditions for 2D analysis

Speed	5 … 30 m/s
Mesh 2D	aprox 88000
Turbulence	0,1%
Analisys plan	XZ
Flow type	Laminar-turbulent
Analisys time	3h

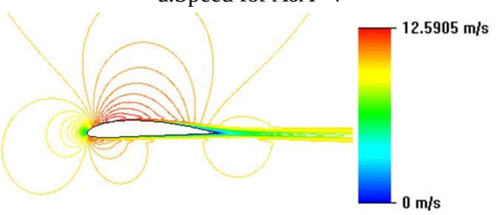
2D CFD analyzes were carried out using solver Floworks (Solidworks) with finite element; the purpose of the CFD analysis is to find the aerodynamic behavior of the profile at different incidence angles (-40, 00, 40). Figure 6 highlights the specific line speed and total pressure generated at a flow rate of V_x = 10 m / s.



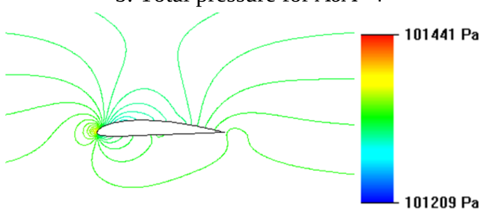
a.Speed for *AoA* -4^0



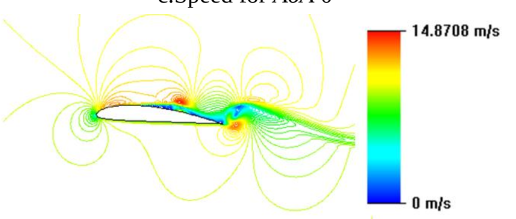
b. Total pressure for *AoA* -4^0



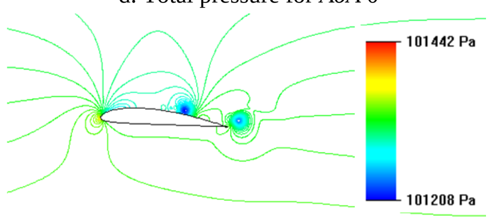
c.Speed for *AoA* 0^0



d. Total pressure for *AoA* 0^0



e.Speed for *AoA* 4^0



f. Total pressure for *AoA* 4^0

Fig. 6 CFD analysis for Clark Y airfoil

Figures 6a, 6c and 6e show the evolution and constant speed lines. Figures 6b, 6d and 6f present the isobars developments at the three angles of incidence considered.

3.2. 3D CFD Analysis of Flying Wing Type

We propose to review the flying wing of Figure 4 with the characteristic data shown in Table 4. The simulation was based on the use of mesh networks (more than 6 million) with maximum density and efficient exploitation of the available computing resources (CPU 2.4GHz RAM 32 GHz). The objective of the CFD analysis is to find the aerodynamic behavior of the flying wing-type carrying capacity surface (see Figure 7 and features shown in Table 4) at zero *AoA*.

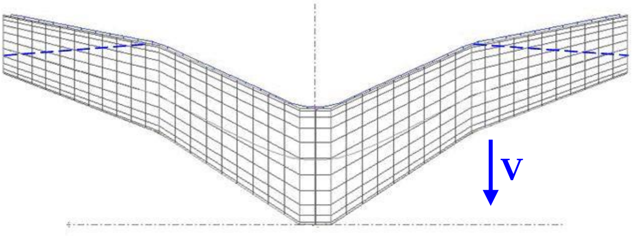


Fig. 7 Flying wing

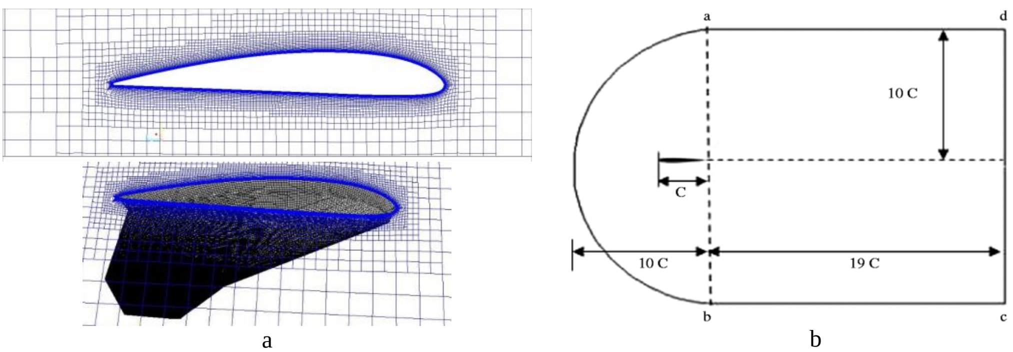
Table 4. Geometric and masic features

Span	2000 mm
Airfoil	Clark Y
Swept angle for c/4	30^0
Surface	0,7 m^2
Aspect ratio	6
Mass	1,2 kg

3D numerical simulations were carried out using Solidworks/Floworks solver by finite element method to investigate the aerodynamic behavior of the carrying capacity surface. The geometry used in CFD simulation has been reproduced for similar models at scale 1:1. For the study of the flow a structured grid was generated and used, connected to a number of parameters according to Table 5, see Figure 8a and the wing position rates shown in Figure 8b [2, 3]. Assessing the flying qualities of the flying wing consisted of 6 phases of analysis from 5÷30 m/s. totaling nearly 180 hours of processing. The CFD analysis revealed a number of relevant results such as: lift - F_z and drag -F_x (Table 5), downstream speed on lifting surface (Figure 9a), turbulence intensity (Figure 9b) and profile lines in the extreme zones (Figure 9d).

Table 5 Initial parameters

Fluid elements	7,18 mil	Speed	5...30 m/s
Iterations	456	Turbulence intensity	1 %
Density	1,25 kg/m^3	AoA	0^o



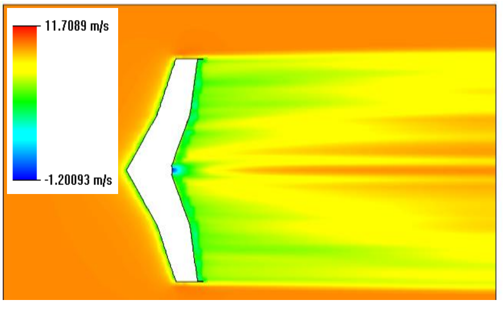
a b

Fig. 8 Lifting surface – a. Meshing structured network, b.ideal levels for analisys

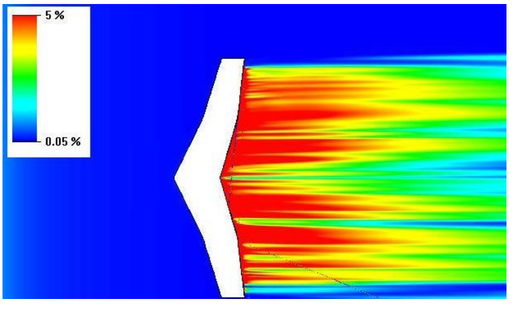
Lateral force F_y values (see Table 6, with values near zero) reveals simulation steps that are within the range of accepted trust. Figure 9b shows the turbulence in the trailing edge of the carrying capacity surface.

Table 6. Normal forces values

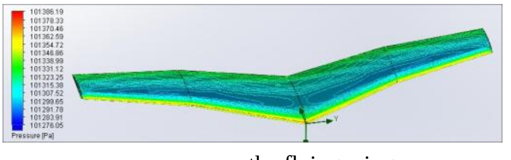
Speed (m/s)	F_z (N)	F_x (N)	F_y(N)
5	1,12	0,09	$5{,}63 \times 10^{-5}$
10	4,33	0,36	$1{,}30 \times 10^{-3}$
15	9,96	0,80	$3{,}01 \times 10^{-3}$
20	18,05	1,38	$6{,}74 \times 10^{-5}$
25	29,26	2,18	$3{,}11 \times 10^{-3}$
30	45,55	3,11	1.01×10^{-3}



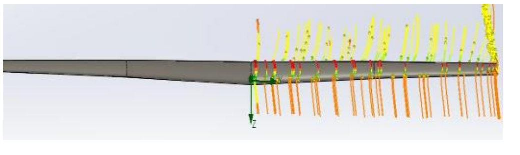
a. speed V_x (m/s)



b. turbulence (%)



c. pressure on the flying wing



d. flow lines

Fig. 9 CFD analisys of the flying wing

The analysis of flow parameters can be assessed graphically by selecting the contours of the flying wing on the same graph for any comparative analysis. The total pressure variation is observed on the selected edges, see Figures 10 and 11.

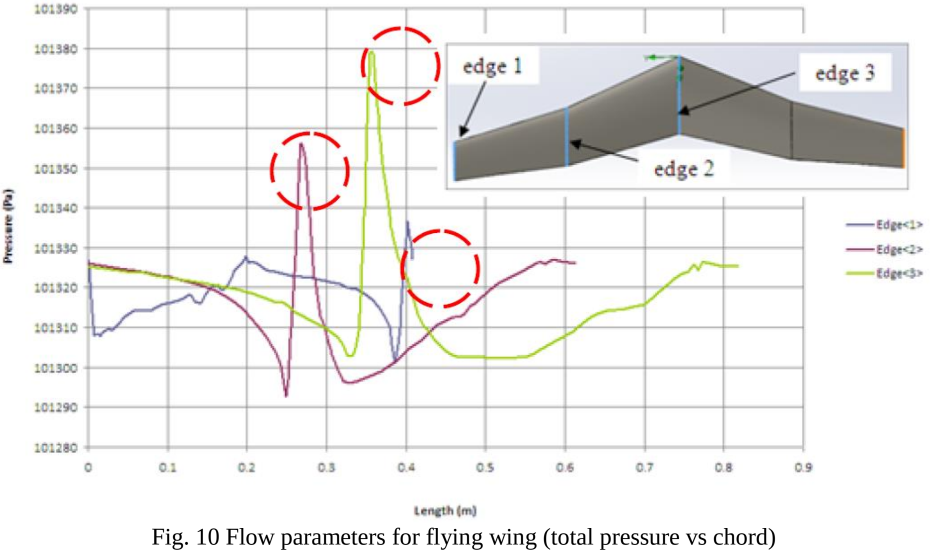


Fig. 10 Flow parameters for flying wing (total pressure vs chord)

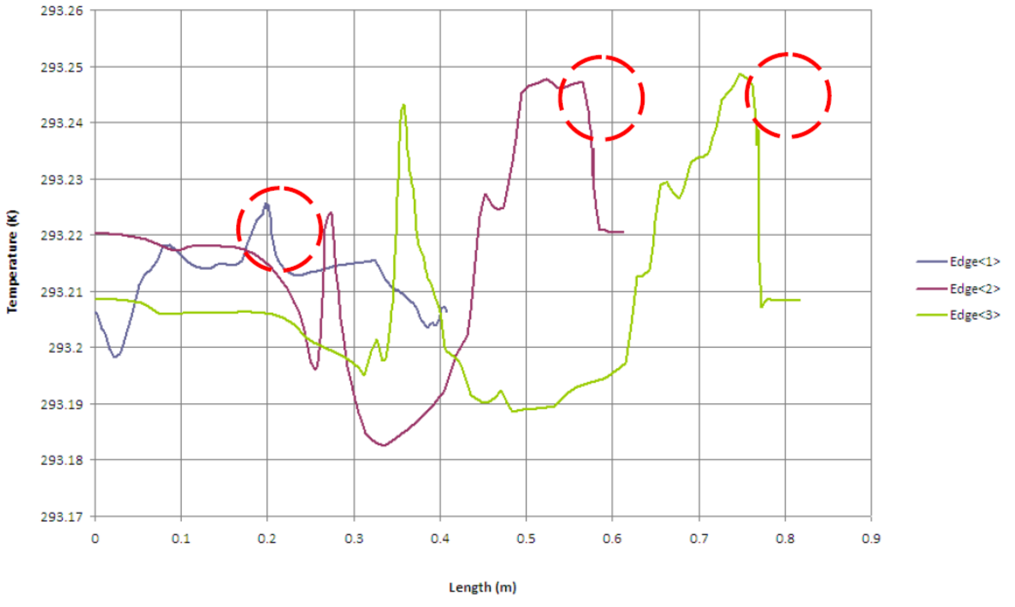


Fig. 11 Flow parameters for flying wing (temperature vs chord)

4. CONCLUSIONS

The results of the simulation of flow around the carrying capacity surface for the flying type wing is affected by two main factors, as follows: the first is the nature and quality (density) of the mesh network and the second is the quality of turbulent flow model used. A comparison between the numerical and experimental (wind tunnel and flight tests) results based on distribution of the velocity vector and distribution of the Reynolds tensor is required. Although the obvious influence of the viscous / inviscid model is observed in the final results we can use both in future CFD analysis.

The 2D and 3D CFD analysis plays an important role; it contributes to minimize the time of design, production and testing processes. The CFD analysis provides early results with minimum possible multiple interventions and resources that can make the difference between an expensive model that does not confirm the expected results and an optimized model.

ACKNOWLEDGMENT

The National Authority for Scientific Research, Romania supported this work – CNCS-UEFISCDI: PN-II-PT-PCCA-2013-4-1349, MASIM project "*Multi Agent Aerial System with Mobile Ground Control Station for Information Management*". The current article benefited from the documentary support of "Transilvania" University of Braşov and "Henri Coandă" Air Force Academy of Braşov, Romania.

REFERENCES

[1] F. Haas, M. Pauritsch, M. Knabl, *Computational fluid dynamics (CFD) for the optimization of products and processes*, The international conference of the Carpathian euro-region specialists in industrial systems 7th edition, 2008.

[2] S. Dănăilă, C. Berbente, *Metode numerice în dinamica fluidelor,* Editura Academiei Române, 2003, ISBN 973-27-0958-8, 362p.

[3] * * * *ANSYS FLUENT User's Guide 14*, 2011, 2948p.

[4] http://www.cfd-online.com/Wiki/Codes, accessed at 12.06.2016.

[5] http://www.taygeta.com/CFD/CFD_codes_p.html, accessed at 14.06.2016.

[6] http://www.pointwise.com/pr/pr-08f-v1512.shtml, accessed at 19.06.2016.

[7] http://www.taitech.com/DragonGrid.htm, accessed at 23.02.2016.

[8] * * * *CFD-ACE+ (GUI V2003.0.124, SOLVER V2003.0.23), Release Notes, 2003*, 34 p., available at www.esi-cfd.com/esi-users/relnotes/ace.V2003.pdf .

[9] * * * Dassault Systemes, *SolidWorks 2008 SP4 Tutorials*, 2008.

[10] G. B. Cosentino, *Computational Fluid Dynamics Analysis Success Stories of X-plane Design to Flight Test*, NASA/TM-2008-214636, 2008, 21p.

[11] D. W. Pepper, X. Wang, *Benchmarking COMSOL Multiphysics 3.5a –CFD problems, COMSOL* Conference Boston 2009, available at:
http://www.ewp.rpi.edu /hartford/~ernesto/F2010/EP1/Materials4Students/Xie/Pepper2009.pdf.

[12] http://www.hanleyinnovations.com/stallion3d.html, accessed at 01.05.2016.

[13] S. Kirkup, *The boundary element method in acoustics (BEM)*, 1998, ISBN 0-953-4031-06, 161p.

[14] * * * *NASA Advanced Supercomputing (NAS) Division*, 2p, available at https://www.nas.nasa.gov/assets/pdf /NAS_Applied_MS_Winter2015.pdf.

[15]* * * *Guidelines for XFLR5 v6.03,* 2011, 71p., available at http://sourceforge.net/projects /xflr5/files/.

[16] V. Prisacariu, A. Luchian, *The aerodynamic analysis of high lift devices*, International conference of scientific paper AFASES 2014, SSN: 2247- 3173, ISSN-L: 2247 – 3173, p 83-89.

[17] V. Prisacariu, I. Cîrciu, S. Pop, *Instruments for the evaluation of the aerodynamic performance of wind tunnels*, International conference "Scientific Research and Education in Air Force" AFASES 2015, ISSN 2247-3173.

[18] http://www.numeca.com/en/products , accessed at 04.05.2016.

[19] M. V. Pricop, C. Stoica, C. Nae şi alţii, Wind Tunnel Testing of Passive High-Lift Systems, *INCAS BULLETIN*, Volume **4**, Issue 1, pp. 67 – 75, ISSN 2066-8201, DOI: 10.13111/2066-8201.2012.4.1.8, 2012.

[20] F. Pomoja, T. S. Mănescu, C. Iacob-Mare, S. Avram, *Rezultate ale simulării computerizate a curgerii prin turbine Cross-Flow tip HYE 10a,* A XIII-a Conferinţă Multidisciplinară cu praticipare internaţională" Profesorul Dorin Pavel – fondatorul hidroenergetici româneşti" Sebeş 2013, p 303-392.