U.P.B. Sci. Bull., Series D, Vol. 72, Iss. 3, 2010 ISSN 1454-2358

ON THE ACCURACY OF NUMERICAL PREDICTION IN TRANSONIC-SUPERSONIC FLOW ARROUND MISSILES

Cristina MIHAILESCU[1], Teodor Viorel CHELARU[2], Sterian DANAILA[3], Corneliu BERBENTE[4], Costică SAVA[5]

Scopul lucrării este de a valida un model numeric prin comparație cu datele experimentale existente pentru o configurație alungită cu ampenaj canard. Pentru acesta sunt considerate 4 configurații de bază pentru care sunt analizate comparativ rezultatele experimentale şi teoretice. Modelarea numerică s-a facut utilizând programul FLUENT (v6.1.12). Pentru comparație s-au folosit rezultatele experimentale obtinute în tunelul aerodinamic. In final se vor prezenta o serie de concluzii privind acuratețea metodei dezvoltate.

The aim of the paper is to validate a numerical model using a comparison with available experimental data for a slender body configuration with canard fins. Four basic configurations are considered, for which the theoretical results are analyzed in comparison with experimental ones. The numerical model was developed using the commercial code FLUENT (v6.1.12). For comparison experimental results obtained in aerodynamic wind tunnel were used. Finally, some conclusions related to the accuracy of this method are presented.

Key words: canard-controlled missile, aerodynamic characteristics

1. Introduction

Over the past decade, static Computational Fluid Dynamics (CFD) simulations over increasingly complex vehicles have become commonplace. Many aerodynamic prediction codes are used to provide values for aerodynamic coefficients. But how well these coefficients reflect reality is a problem of accuracy. Nowadays, this problem is presented in many papers, being part of code validations [1], error estimations [2], mesh refinement [3], [4] and others.

Obtaining the aerodynamic coefficients for missiles is a problem that

[1] Drd. Eng., Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest, Romania
[2] Reader., Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest, Romania
[3] Prof., Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest, Romania
[4] Prof., Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest, Romania, e-mail: berbente@yahoo.com
[5] Drd. Eng., Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest, Romania

involves high responsibilities from researchers in fluid flow domain. To obtain these aerodynamic data, many models could be built and tested in a wind tunnel, with different positions related to the flow. Such tests are expensive and require also an expensive model execution. Computational Fluid Dynamics (CFD) is an alternative to wind tunnel testing, but the necessary requirement of using such a numerical model is that the results that yield must be a realistic simulation of a fluid in motion. That is why it is necessary to compare numerical results with experimental ones.

Canard control is also quite commonly used, especially on short-range missiles. The primary advantage of canard control is better manoeuvrability at low angles of attack, but canards tend to become ineffective at high angles of attack due to flow separation that causes the surfaces to stall. Since canards are ahead of the center of gravity, they cause a destabilizing effect and require large fixed tails to keep the missile stable.

For the case of a slender body configuration with fins and canards, studies about the influence of canard deflection on aerodynamic characteristics for missile are necessary. This is due to air deflection in presence of canards, a phenomenon that can produce commands inversion, especially for the case of asymmetrical deflection of canards for roll control.

The aim of this study is to make a comparison between experimental and numerical data of aerodynamic coefficients for a guided missile and to evaluate the accuracy of numerical prediction in transonic-supersonic flow around missiles with canard fins.

Similar studies involving canard-controlled missiles were performed by James DeSpirito et al. [5], [6], [7] at Army Research Laboratory between 2000 and 2004, with good agreement between numerical and experimental data.

2. Mathematical model

The three-dimensional Reynolds-Averaged Navier-Stokes equations are:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho u_i\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right)\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_{lj}}{\partial x_l}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{2}$$

where the Reynolds stresses can be express in terms of mean velocity gradients using the Boussinesq approach:

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij} \tag{3}$$

and:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

For turbulence modelling, a modified form of the k-ε two-equation turbulence model proposed by Shih et al. [8], called Realizable k-ε model was used. It differs from the standard k-ε model because it contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate.

The transport equations for k and ε for Realizable k-ε model are:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon-Y_M \tag{5}$$

and

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_j}(\rho \varepsilon u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\rho C_1 S\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon} \tag{6}$$

where

$$C_1=\max\left[0.43,\frac{\eta}{\eta+5}\right],\ \eta=S\frac{k}{\varepsilon},\ S=\sqrt{2S_{ij}S_{ij}}\ ,\ S_{ij}=\frac{1}{2}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right) \tag{7}$$

For this model, C_μ is no longer a constant. Instead, it is computed from:

$$C_\mu=\frac{1}{A_0+A_s\dfrac{kU^*}{\varepsilon}} \tag{8}$$

where A_0 and A_s are constants.

The term G_k represents the production of turbulence energy and it is defined as:

$$G_k=\frac{1}{2}\mu_t\left(\frac{\partial u_j}{x_i}+\frac{\partial u_i}{x_j}\right)^2 \tag{9}$$

The term Y_M represents the "dilatation dissipation" term for high Mach number flows. This term is modelled according to Sarkar [9], as:

$$Y_M=2\rho\varepsilon\frac{k}{\gamma RT} \tag{10}$$

The Realizable k-ε Equation (1÷6) cannot be applied to wall-bounded flows directly. For the wall-bounded flows, we must specify the boundary conditions appropriate to a solid boundary for velocity, turbulence kinetic energy k and dissipation rate ε. The wall function is used to avoid the unphysical behaviour of the k and ε equations inside the viscous sub layer region and to match the outer flow to the wall. The no-slip condition: $u,v,w=0$ is used on the wall.

This model was validated for many flow types including strong streamline curvature, vortices, and rotation. For all these cases, the Realizable k-ε model showed substantial improvements over the standard k-ε model.

3. Numerical solution

All computations were performed for steady-state case using the commercial CFD code FLUENT (v6.1.12).

The geometry and unstructured meshes were generated using the preprocessor GAMBIT. The number of tetrahedral cells for the resulted meshes is presented in Table 1. Because of the necessity to analyze the influence of differential canard deflection, it was impossible to find a symmetry plan for the domain flow. That is why the analyses are performed in three-dimensional space. In generation of meshes, the boundary layer mesh spacing was used near the missile body and fin surfaces.

The domain was large enough, it's extension on Ox axis was between -5L before missile and 10L after it, where L is the total length of the missile. On Oy and Oz axis the distance from missile to the outer boundary was about 6L.

The imposed boundary conditions were:

- For missile's solid surfaces, wall boundary condition was used;
- A far-field pressure boundary condition at the downstream, upstream, and outer boundary with imposed values for free stream direction, speed and pressure was used;
- The ideal gas hypothesis for air and Sutherland law for viscosity was used.

4. The influence of canard geometry

In this study the aerodynamic coefficients for a guided missile will be evaluate considering different canard deflections. The configuration has four canard fins, which can be deflected in a range of $-15^0 \div 15^0$, to realize the guidance of missile, and six fins. The missile configuration is presented in Fig. 1.

The analyses were performed at Mach number 1.11 for a range of incidences $-4^0 \div +11^0$ for four cases with different configurations presented in Table 2.

The experimental results used for comparison have been obtained only for cases B, C and D in the wind tunnel [10] for a range of attack angle of $-4^0 \div 15^0$.

The normal force (Cz,), axial force (Cx,), and pitching moment (Cm,) coefficients are presented in a missile's axis system.

The results obtained in previous conditions have been graphically represented for a comparison with experimental results. In this way, in Fig. 2, 3, 4 and 5 the force coefficients and the moments coefficients respectively are represented for every case.

For case A (Fig. 2) the accuracy of numerical results is good and predict in good conditions the increase of lift coefficient with the incidence angle. In the same manner, the numerical results for the moment coefficient C_m predict in a reasonable way the influence of incidence. For drag coefficient the accuracy is rather poor, but the conclusion is that numerical results for drag force coefficients are higher than the experimental data. The same results referring to the drag coefficient are obtained for all three cases A, B and C.

For cases B and C (Fig.3 and Fig.4) the accuracy is a little poorer than for case A both for lift and moment coefficients, but the slope of the diagrams is kept.

In the absence of experimental data, for case D represented in Fig.5, only numerical results are available and they predict lift, drag and moment coefficients for the case of configuration without canard fins. We appreciate that results for this case have the same accuracy like in previous cases.

Referring to influence of canard deflection angle, a major influence on increase of the canard deflection angle is the increase of axial force coefficient Cx. The lift coefficient Cz is not influenced when the deflection is asymmetric, but, when the canards are deflected symmetrically in current flow (Case C, Fig.8) an increase of Cz can be observed. The influence of asymmetric deflection angle of canards (case A and B) is quite small comparative to the case of symmetric deflection angle (case C) related to moment coefficient Cm. All these comparative graphs are represented in Fig. 6, 7 and 8.

Table 1

Cells number for meshes

Case	Number of cells
A	625401
B	743569
C	541532
D	640174

Table 2

Configurations

Case	Deflection of horizontal canard
A	$+/-2.5^0$
B	$+/-5^0$
C	$+10^0$
D	No canard fins

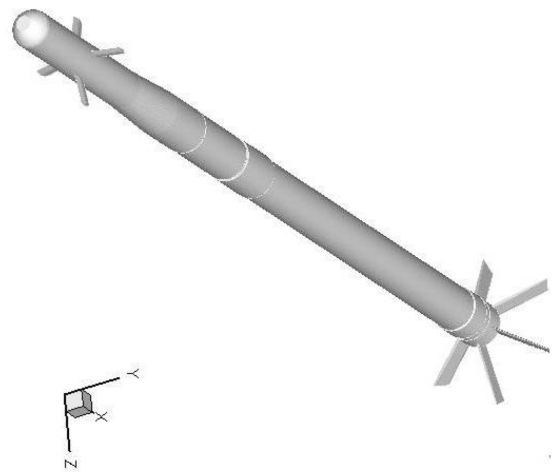

Fig. 1 Missile's configuration

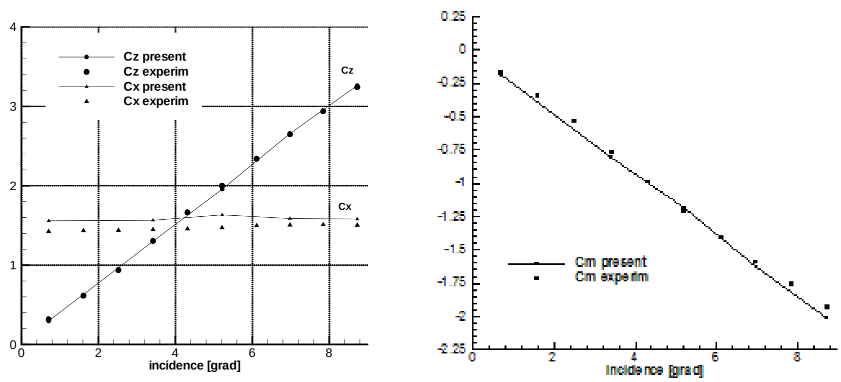

Fig. 2 Forces and moments coefficients for case A

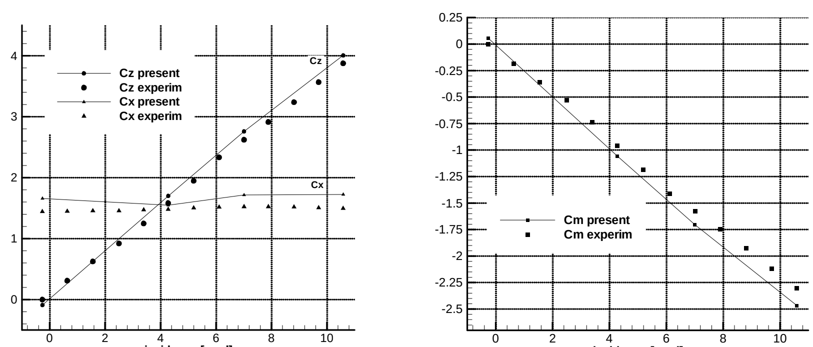

Fig. 3 Forces and moments coefficients for case B

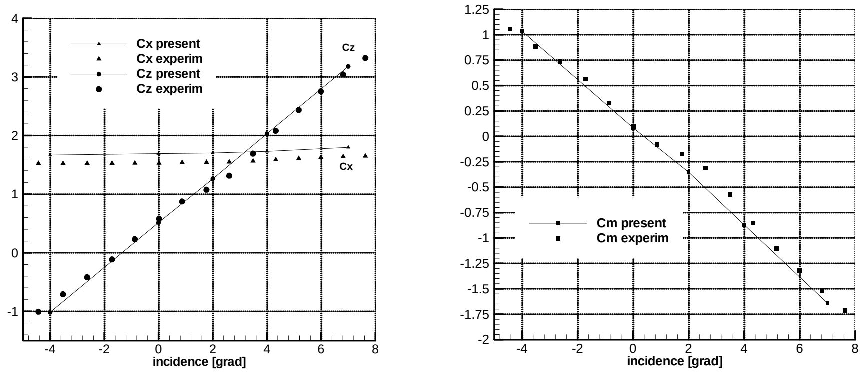

Fig. 4 Forces and moments coefficients for case C

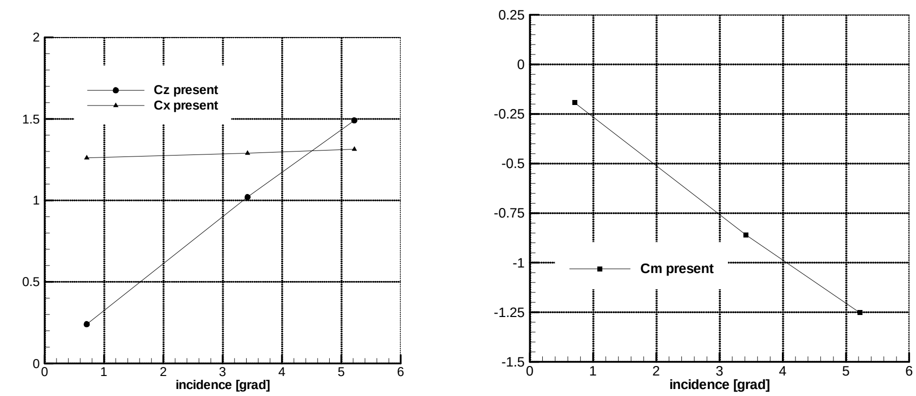

Fig. 5 Forces and moments coefficients for case D

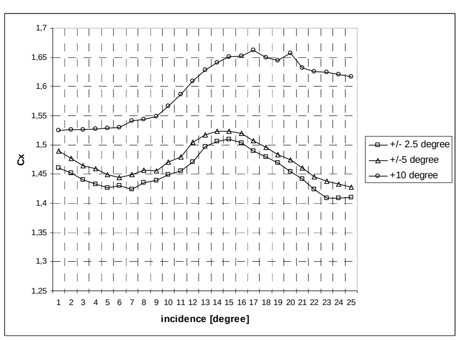

Fig. 6 Influence of canard deflection angle on Cx

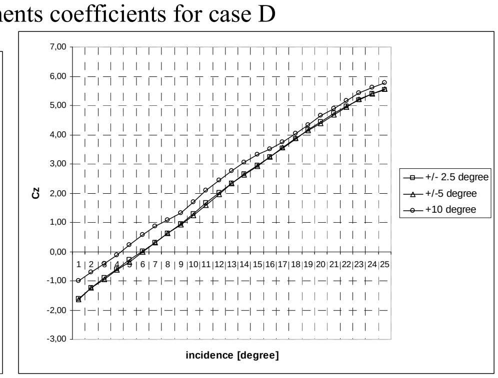

Fig. 7 Influence of canard deflection angle on Cz

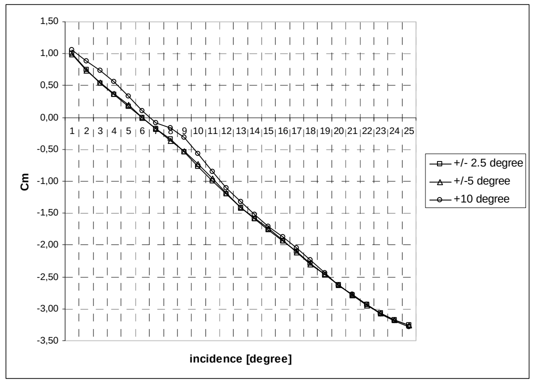

Fig. 8 Influence of canard deflection angle on Cm

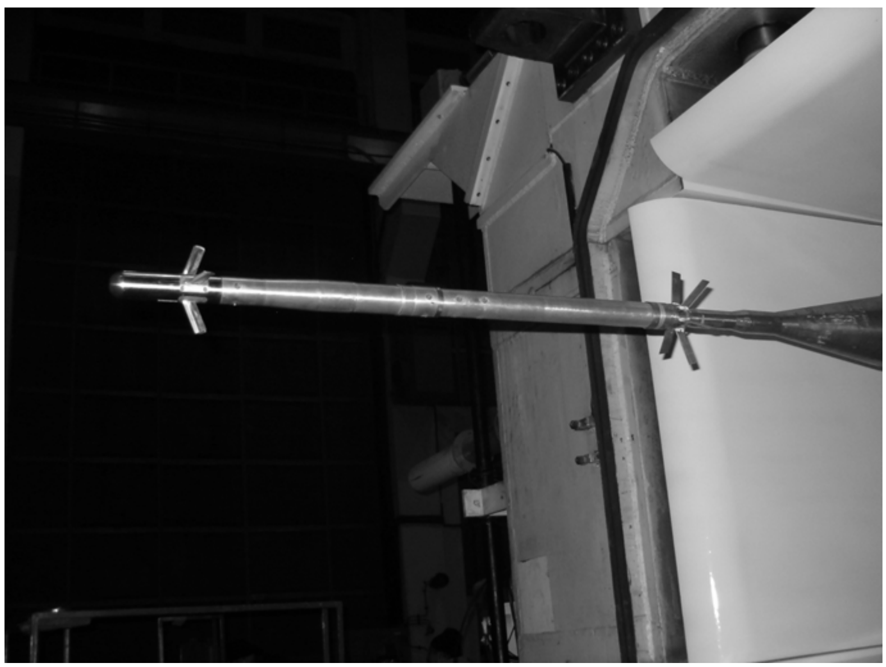

Fig. 9 Missile model during wind tunnel tests

5. Conclusions

Using the FLUENT postprocessor, the viscous and pressure forces were integrated along the missile body and fin surfaces to calculate the aerodynamic coefficients. The pitching moment is expressed about the nose of the missile. The reference area is the cross-sectional area of the missile body, and the reference length is the diameter of the missile. The calculated coefficients are compared to wind tunnel measurements performed at National Institute for Aerospace Research "Elie Carafoli" Bucharest [10].

A major influence of increasing canard deflection angle is the increase of axial force coefficient. The lift coefficient is not influenced when the deflection is asymmetric, but when the canards are deflected symmetrically in current flow (case C), an increasing of Cz can be observed. The influence of asymmetric deflection angle of canards (case A and B) is quite small comparative to the case of symmetric deflection angle (case C) related to moment coefficient Cm. All these comparative lots are represented in Fig. 6, 7 and 8.

Secondly, a technical conclusion resulted from this study was that the deflection on the tails is negligible if we use small asymmetrical canard deflection and small attack angle. In this case, we can control the roll of the missile using only canard asymmetrical deflection, without other auxiliary systems.

The results were validated by comparing the computed aerodynamic coefficients for the missile against wind tunnel measurement data. The best agreement between numerical and experimental results is obtained for case A.

We can see from Figures 1-3 a good accuracy for the lift coefficient. We can not say the same thing about the drag coefficient, because in this case the accuracy is rather poor, but the conclusion is that numerical force coefficients are higher than the experimental ones. Anyway, they predicted the same increase with incidence angle as the showed experimental data.

The maximum difference between the calculated and measured normal force coefficient was 7%, while the maximum difference between the calculated and measured axial force coefficient was 14%.

To increase the accuracy of numerical results, a finer mesh is recommended, but this imposes an increase of computer memory and of computing time. In a similar study performed at Army Research Laboratory, computers with 48 and even 64 processors [6] have been used.

The final conclusion of this work is that viscous CFD analysis offers an accurate method for calculating the flow field and aerodynamic coefficients for missiles.

B I B L I O G R A P H Y

[1] *Thomas J. Sooy and Rebecca Z. Schmidt*. Aerodynamic Predictions, Comparisons, and Validations Using Missile DATCOM (97) and Aeroprediction 98 (AP98) Technology Service Corporation, Silver Spring, Maryland 20910 -Journal of Spacecraft and Rockets Vol. 42, No. 2, March–April 2005

[2] *Marian Nemec* Error Estimation and Adaptive Mesh Refinement for Problems with Complicated Geometries MIT ACDL Seminar May 9, 2008
[3] *Marian Nemec Michael J. Aftosmis*, Adjoint Error Estimation and Adaptive Refinement for Embedded-Boundary Cartesian Meshes, 18th AIAA Computational Fluid Dynamics Conference, Miami, FL, June 25–28, 2007.
[4] *Scott M. Murman, Michael J. Aftosmis* Cartesian-Grid Simulations of a Canard-Controlled Missile with a Spinning Tail AIAA-2003-3670, 21st AIAA Applied Aerodynamics Conference, Orlando, Florida, June 23-26, 2003
[5] *James DeSpirito, Harris L. Edge, Paul Weinacht, Jubaraj Sahu and Surya Dinavahi* Computational Fluid Dynamic (CFD) Analysis of a Generic Missile with Grid Fins Army Research Laboratory – Report No. ARL-TR-2318, September 2000
[6] *James DeSpirito, Milton E. Vaughn, Jr., W. David Washington* Numerical Investigations of Aerodynamics of Canard-Controlled Missile Using Planar and Grid Tail Fins, Part I: Supersonic Flow -Army Research Laboratory – Report No. ARL-TR-2848, September 2002
[7] *James DeSpirito, Milton E. Vaughn, Jr., W. David Washington* Numerical Investigations of Aerodynamics of Canard-Controlled Missile Using Planar and Grid Tail Fins, Part II: Subsonic and Transonic Flow -Army Research Laboratory – Report No. ARL-TR-3162 , March 2004
[8] *T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu.* A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids*,* 24(3):227-238, 1995.
[9] *S. Sarkar and L. Balakrishnan*. Application of a Reynolds-Stress Turbulence Model to the Compressible Shear Layer. ICASE Report 90-18, NASA CR 182002, 1990.
[10] ***, Experimental determination of aerodynamic characteristics, National Institute for Aerospace Research "Elie Carafoli" – code C-7074 , 2007.
[11] *B. E. Launder and D. B. Spalding*. The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering*,* 3:269-289, 1974.
[12] *D. C. Wilcox*. Turbulence Modelling for CFD. DCW Industries, Inc., La Canada, California, 1998.