Experimental results and numerical simulations for transonic flow over the ONERA M4R model

Andreea BOBONEA*[,1], Mihai Leonida NICULESCU[1], Mihai Victor PRICOP[1], Adrian CHELARU[1], Florin MUNTEANU[2], Marius Gabriel COJOCARU[1]

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

DOI: 10.13111/2066-8201.2013.5.2.1

Abstract: *This paper presents a comparison between experimental results of transonic flow over the ONERA M4R calibration model obtained in the INCAS Trisonic wind tunnel and the numerical results. The first purpose, emphasized in this paper is to compare and validate the computational fluid dynamics (CFD) techniques for internal transonic flows and to try to find the most suitable numerical methodology for these flows in both accuracy and computational resources. The second purpose is to develop a general method in experimental data correction and flight Reynolds extrapolation, using numerical simulations for both global and local pressure coefficients, as a replacement for the classical vortex lattices based method. That will be developed in a future paper. Besides the computational work, the periodic wind tunnel calibration is required as a quality insurance operation and a numerical model is developed such that future hardware modifications to be included and their impact to be properly considered.*

Key Words: *transonic internal flow, CFD, Trisonic wind tunnel*

1. INTRODUCTION

A wind tunnel performance is reflected in the degree of uniformity of flow parameters in the experimental section (size and direction of the velocity, Mach number), the accuracy of parameters measurements (which depends on the instrumentation used), and also the size of interaction effects between the measured object and the measuring instrument. These performance deviations are determined from free flight results within an infinite atmosphere or, more practically, with the help of computational fluid dynamic simulations. For a wind tunnel to be accepted from the viewpoint of quality and performance testing, tests of calibration should be conducted and regular calibration reports on the entire system should be published. When constructive and functional changes are made to the configuration; these calibrations are compulsory. The objective of the test program mentioned in this paper was to determine the aerodynamic force and moment coefficients of the ONERA M4R model equipped with an internal 6-component strain gauge balance, in transonic flow.

2. EXPERIMENTAL SETUP

Due to the necessity of calibration of the Trisonic wind tunnel experimental chambers, it was decided to design and build a wind tunnel calibration model, using the geometry of ONERA models established in 1969 by the French agency (Fig.1, Tab.1). Taking into account the dimensions of the Trisonic wind tunnel, an intermediate scale between Onera M3 and Onera

M5 named “Onera M4R” [1], [2], was chosen, having a wing span of 635mm. The geometry of the first model produced had some deviations and it was necessary for the model wing to be rebuilt [3], [4], this being the final version used in this study.

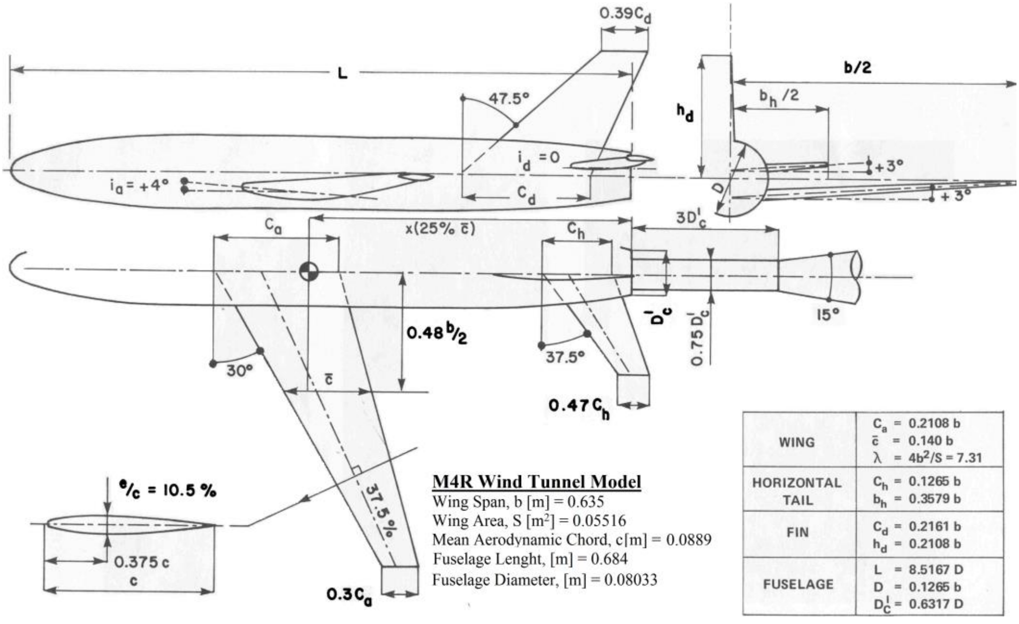

Fig. 1 ONERA M4R wind tunnel model geometry

The TASK balance is fixed in a stable position within the model layout so that the center of balance corresponds to 25% of CMA on the horizontal axis of the fuselage, which is the reference center for reducing the aerodynamic forces and moments.

Fig. 2 ONERA M4R wind tunnel model in the transonic experimental chamber

The intended calibration test matrix includes a number of runs at different Mach numbers: M = 0.25; 0.5; 0.7; 0.84 and 0.9, corresponding to the Reynolds number values $Re=1.5\cdot 10^6$, $2\cdot 10^6$, $2.5\cdot 10^6$, $4\cdot 10^6$ (calculated for the aerodynamic mean chord of the model wing). From the completed tests, run #7612 at Mach number M=0.7, $Re=2\cdot 10^6$ was chosen as a reference because the flow is transonic and fully turbulent.

3. GOVERNING EQUATIONS

For a three-dimensional stationary Cartesian coordinate system, the unsteady Reynolds-averaged Navier-Stokes equations using the Favre averaging (a mass-weighted averaging) could be written in the conservative form as [5-6].

$$\frac{\partial Q}{\partial t}+\frac{\partial\left(F_x-G_x\right)}{\partial x}+\frac{\partial\left(F_y-G_y\right)}{\partial y}+\frac{\partial\left(F_z-G_z\right)}{\partial z}=S \tag{1}$$

where

$$Q=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho E\end{bmatrix}\quad F_x=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uw\\ \rho uH\end{bmatrix}\quad F_y=\begin{bmatrix}\rho v\\ \rho vu\\ \rho v^2+p\\ \rho vw\\ \rho vH\end{bmatrix}\quad F_z=\begin{bmatrix}\rho w\\ \rho wu\\ \rho wv\\ \rho w^2+p\\ \rho vH\end{bmatrix} \tag{2}$$

If one assumes that the fluid is Newtonian and the thermal boundary layer is neglected, the diffusive flux G may be written as

$$G_x=\begin{bmatrix}0\\ \tau_{xx}^{tot}\\ \tau_{xy}^{tot}\\ \tau_{xz}^{tot}\\ u\tau_{xx}^{tot}+v\tau_{xy}^{tot}+w\tau_{xz}^{tot}+\alpha\frac{\partial T}{\partial x}\end{bmatrix}\quad G_y=\begin{bmatrix}0\\ \tau_{xy}^{tot}\\ \tau_{yy}^{tot}\\ \tau_{yz}^{tot}\\ u\tau_{xy}^{tot}+v\tau_{yy}^{tot}+w\tau_{yz}^{tot}+\alpha\frac{\partial T}{\partial y}\end{bmatrix}$$
$$G_z=\begin{bmatrix}0\\ \tau_{xz}^{tot}\\ \tau_{yz}^{tot}\\ \tau_{zz}^{tot}\\ u\tau_{xz}^{tot}+v\tau_{yz}^{tot}+w\tau_{zz}^{tot}+\alpha\frac{\partial T}{\partial z}\end{bmatrix} \tag{3}$$

According to the Boussinesq hypothesis, the shear stresses τ^{tot} may be written as

$$\tau_{xx}^{tot}=\frac{2}{3}\left(\mu+\mu_t\right)\left(2\frac{\partial u}{\partial x}-\frac{\partial v}{\partial y}-\frac{\partial w}{\partial z}\right)\quad \tau_{yy}^{tot}=\frac{2}{3}\left(\mu+\mu_t\right)\left(2\frac{\partial v}{\partial y}-\frac{\partial u}{\partial x}-\frac{\partial w}{\partial z}\right)$$
$$\tau_{zz}^{tot}=\frac{2}{3}\left(\mu+\mu_t\right)\left(2\frac{\partial w}{\partial z}-\frac{\partial v}{\partial y}-\frac{\partial u}{\partial x}\right)\quad \tau_{xy}^{tot}=\left(\mu+\mu_t\right)\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) \tag{4}$$
$$\tau_{xz}^{tot}=\left(\mu+\mu_t\right)\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\quad \tau_{yz}^{tot}=\left(\mu+\mu_t\right)\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)$$

The Sutherland's formula could be used to determine the dynamic viscosity μ as a function of temperature, while the eddy viscosity μ_t is computed using a turbulence model.

For gases, the external force f_e due to the gravitational acceleration is very small, therefore it can be neglected. Moreover, we can assume that the thermal conductivity is the single heat source, therefore the source term S becomes null.

$$S = 0 \tag{5}$$

The pressure is obtained from the equation of state of ideal gas

$$p = \rho R T \tag{6}$$

Furthermore, we could assume that air is a perfect gas; therefore

$$e = \frac{R}{\gamma - 1} T \quad h = \frac{\gamma R}{\gamma - 1} T \tag{7}$$

4. NUMERICAL SIMULATION

Adequate CAD models for the supersonic Wind tunnel test section and ONERA M4R calibration model have been produced for all included incidence angles in test matrix. A parametric CAD model for the sting kinematics has been prepared, enabling automatic vertical positioning, as a function of incidence.

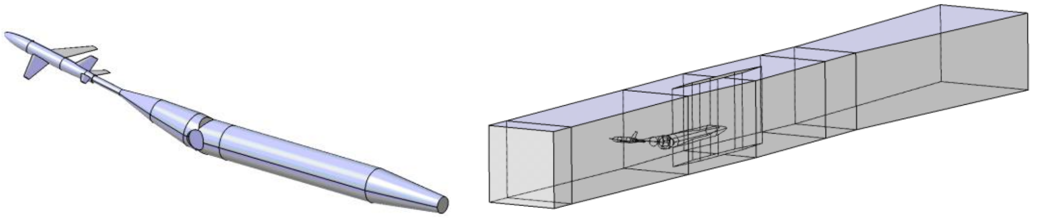

Fig. 3 Parametric CAD of sting and M4R left, full CAD model right

The numerical simulations of the three-dimensional turbulent flow were carried on ONERA M4R model using Ansys Fluent 14.5 [7]. The internal flow modeling requires distinct CAD geometries and meshes, for each incidence. In order to avoid an excessive effort, relatively coarse meshes have been built, at around 4.5 million cells as shown in Fig. 4, keeping in mind that external flow has to be also computed in the near future, to answer to the second purpose of this work. Also the parametric meshing (scripting) will be considered in the next phase, to minimize meshing effort. The imposed mesh coarseness constrains the wall distance y^+ to a value of about 30, suitable for the adopted turbulence model.

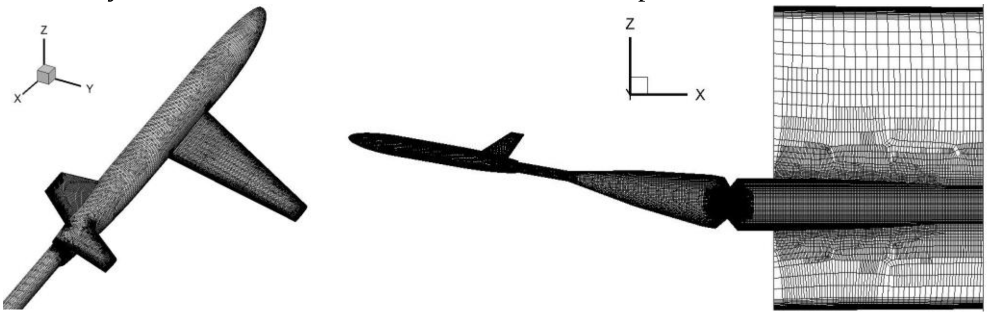

Fig. 4 Detail of the mesh around ONERA M4R model, AoA= 10^o

Because the realizable k-ε model [8] is a high-Reynolds turbulence model, we used the non-equilibrium wall functions [9] that are suitable in complex flows with separations and reattachments. In order to significantly decrease the computational time, the implicit formulation has been adopted. To take into account the physical flow properties, the convective fluxes are discretized with the Roe scheme, which is a Godunov-type scheme [10-14].

The numerical simulations have been performed within the following inlet pressure conditions: total pressure = 174200 Pa, total temperature = 281.2 K, turbulence intensity = 0.2 %, turbulent viscosity ratio = 0.2, angle of attack (AoA) = from -4º to +12º, considering the values from the experimental report. Because the flow is subsonic to inlet of computational domain, the velocity on this frontier is extrapolated and it is M=0.7, matching the experimental value.

The reference values for the computation of the forces and moment coefficients are: reference surface aria = 0.05516 m^2, reference length = 0.0889 m, density = 1.715kg/m^3, velocity = 222.7 m/s.

5. RESULTS

A relevant aspect for the numerical simulation is the verification and validation with experimental data. The comparison between experimental and numerical data for steady simulations is plotted in Fig. 4. From the numerical perspective it is a challenge with respect to the turbulence modeling, shock wave/boundary layer interaction and the modeling requirements for ONERA M4R wind tunnel model. Generally, the numerical results obtained with second order Roe method and realizable k-ε turbulence model are in good agreement with the experimental ones as shown in Fig. 5; therefore, the numerical methodology for this test case is validated. The zero lift drag difference is to be expected, considering the mesh coarseness.

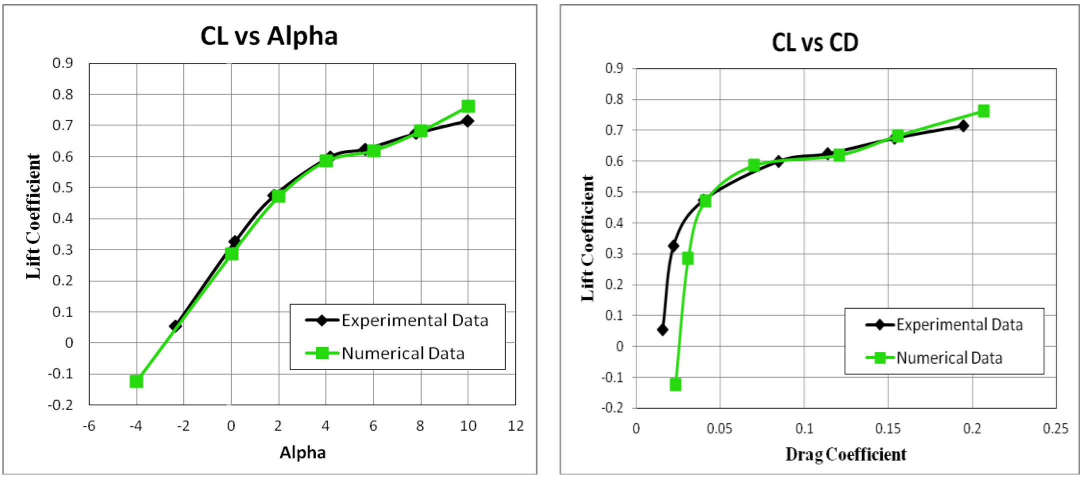

Fig. 5 Aerodynamic performance of ONERA M4R model

The output of numerical flow solutions is not only in the global coefficients increments, but also in the local values, as pressure distributions, thus providing a good insight of the solid wall interference.

For this reason, Fig. 6 shows only numerical results for pressure distributions near the wing root, at mid-span wing and near the wing tip at moderate angle of attack and near stall. One clearly sees that the pressure distributions for pressure side for moderate angle of attack and near stall are close but significant differences appear for the suction side.

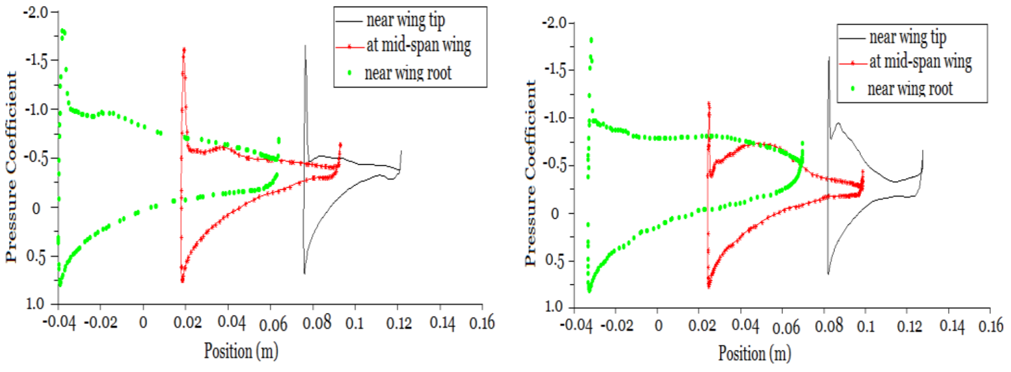

Fig. 6 Pressure coefficient distributions at α = 4º (left) and α = 10º (right)

6. CONCLUSIONS

Generally, the numerical results obtained with second order Roe method are in good agreement with the experimental ones as shown in Fig. 5; therefore this numerical approach is validated for the test case of ONERA M4R wind tunnel model. More useful than the absolute values, are increments between wind tunnel constrained and free-stream model, as a general tool for solid wall interference treatment. Numerical simulation derived increments are the subject of near future work, accompanied by more efficient, parametric meshing. Two different physical models shall be applied: compressible non viscous (Euler equations) and viscous (RANS). Further test cases are necessary to validate the CFD.

REFERENCES

[1] C. Adam et al., Elaborarea temei de proiectare pentru macheta etalon M4 si dispozitivul de prindere în suflerie, raport I.M.F.D.Z., cod N - 5243, nov. 1990.

[2] L. Costian et al., Proiectare machetă ONERA - M4 R, faza I, proiect I.M.F.D.Z., cod N - 5244, dec. 1990

[3] C. Adam, L. Costian, Proiect machetă de calibrare ONERA M4R-M, raport INCAS cod A-3216/2, decembrie 2004.

[4] C. Adam et al., Execuție machetă de calibrare ONERA M4R-M, raport INCAS cod A-3367, iunie 2005.

[5] S. Dănăilă and C. Berbente, *Metode Numerice în Dinamica Fluidelor*, Editura Academiei Române, 2003.

[6] C. Hirsch, *Numerical Computation of Internal and External Flow, Volume 2: Computational Methods for Inviscid and Viscous Flows*, John Wiley and Sons, New York, 1990.

[7] ***ANSYS, Inc., ANSYS FLUENT Theory Guide, Release 14.0, November 2011. Available at www.ansys.com.

[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 and Fluids*, Vol. **24**, issue 3, pp. 227-238, ISSN: 0045-7930, 1995.

[9] S.-E. Kim and D. Choudhury, *A near-wall treatment using wall functions sensitized to pressure gradient*, in ASME FED Vol. **217**, Separated and Complex Flows, ASME, 1995.

[10] B. v. Leer, Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method, *Journal of Computational Physics*, Vol. **32**, Issue 1, pp. 101-136, ISSN: 0021-9991, 1979.

[11] P. L. Roe, The use of Riemann problem in finite difference schemes, *Lecture Notes in Physics*, Vol. **141**, pp. 354-359, 1980.

[12] P. L. Roe, Approximate Riemann solvers, parameter vectors, and difference schemes, *Journal of Computational Physics*, Vol. **43**, Issue 2, pp. 357-372, ISSN: 0021-9991, 1981.

[13] P. L. Roe, Characteristic-based schemes for the Euler equations, *Annual Review of Fluid Mechanics*, Vol. **18**, pp. 337-365, 1986.

[14] G.-C. Zha, Y. Shen and B. Wang, An improved low diffusion E-CUSP upwind scheme, *Journal of Computers & Fluids*, Vol. **48**, pp. 214-220, ISSN: 0045-7930, 2011.