NUMERICAL STUDY OF THE COMPRESSIBLE FLOW INSIDE A SUPERSONIC NOZZLE TO 2D

[1]SAID SELLAMI, [2]OMAR KHOLAI

[1,2]Laboratory LITE Department Engineering of Transport University Constantine1, Algeria
E-mail: [1]sellamisaid1@gmail.com, [2]kohlai.omar@gmail.com

Abstract— The object of this study is performed phenomena, which appear in the flow of a two-dimensional convergent-divergent nozzle. The fluid used is a gas, which assumed to be an ideal gas, by using the finite volume method for solving partial differential equations and turbulence model used in this study is the model with two types of transport equations (k-ω) *SST Menter*, is considered the most suitable for this kind of problem. This simulation performed using FLUENT software.

Keywords—nozzle, converging-diverging, turbulence, shock wave, supersonic, compressible flow, finite volume.

I. INTRODUCTION

This work deals with the numerical simulation of compressible and turbulent internal flow using FLUENT software. In this approach, the physical and thermal magnitudes of the flow are calculated. The flow is considered a two-dimensional compressible flow turbulent supersonic nozzle, characterized by the presence of walls requires the consideration of viscosity effects.

The flow that particularly concerns us is a high-speed flow and very high temperature where it produces shock waves that interact with the boundary layers. The existence of shocks in this type of flow produces heavy load losses. These flow phenomena in the nozzles are very harmful to the solid structures of the nozzle. The existence of these phenomena also require a request from the industry and researchers depth analytical and numerical studies of flows in all its forms (in-stationary, compressible external or internal) particularly to better understand the current physical problems, among these research study: R. Haouia and all [1] presented the results of a flow of high temperature gas in an axisymmetric nozzle at hypersonic regime. They used a gas mixture composed of five chemical species (O2, N2, NO, O, N). The in-stationary partial differential equations (Euler equations), which governs this flow is solved with an explicit scheme using digital finite volume method with two kinetic models to Zeldovich (3 and 17 reactions chemical). They used a 150 mesh node along the X axis and 10 node according to the radius Y. They got very interesting results in both cases models mentioned above. A.Nebbache [2] presented the separation phenomenon in an asymmetric nozzle, where the test gas is assumed as an ideal gas. The system of equations governing the flow is solved using the finite volume method fully implicit type predictor-corrector Mac - Cormack. The turbulence model used is (k-ω). This work is based on two configurations: The first configuration consists of (a main nozzle, a box and a secondary nozzle removable collar); the mesh size used is (236 × 200). The second box without configuration and field of study comprises (a main nozzle and noted areas "Jet" and "Wind"). Several meshes (four mesh) were tested to study the independence of the mesh. The results were compared in both configurations. He presented the separation phenomenon in an asymmetric nozzle, where the test gas is assumed as an ideal gas. The system of equations governing the flow is solved using the finite volume method fully implicit type predictor-corrector Mac - Cormack. The turbulence model used is (k-ω). This work is based on two configurations: The first configuration consists of (a main nozzle, a box and a secondary nozzle removable collar); the mesh size used is (236 × 200). The second box without configuration and field of study comprises (a main nozzle and noted areas "Jet" and "Wind"). Several meshes (four mesh) were tested to study the independence of the mesh. The results were compared in both configurations. The same author [3] studied numerically the separation of a turbulent flow in an axisymmetric nozzle truncated where the test gas is perfect supposed nitrogen. The system of equations governing the flow is solved using the finite volume method with a fully implicit scheme predictor-corrector type of Mac-Cormack. The field of digital integration of this study consists of three distinct parts: the nozzle, the jet and the lower field. Three meshes have been used to study the independence of the mesh. E.Mahfoudi and All [4] worked on the physical analysis and numerical simulation of turbulent separated flow in a supersonic nozzle truncated ideal contour, the turbulence is modeled by a statistical approach (FRANS) in generalized coordinates with the use of the model (SST-Menter). The system of equations governing this flow is solved using the finite volume method structured mesh. The time integration is performed by the fully implicit numerical scheme predictor-corrector type of MacCormack. M.Y. Bouzid and R. Dizene [7] Have studied by the two-dimensional numerical simulation of the behavior of compressible flow was highly turbulent through a nozzle of converging-diverging supersonic, with the use of four turbulence models built into the system of Navier Stokes averaged by the statistical method Favre.

II. GOVERNING EQUATIONS

For flow compressible, viscous and perfect supposed, the fundamental equations of flow can be given by the following laws: the conservation of mass (1), the conservation of momentum (2) and the conservation of energy (3). Their formulation in the Cartesian coordinate system, [12] and [13].

$$\begin{cases} \dfrac{\partial \rho}{\partial t} + \dfrac{\partial(\rho u_j)}{\partial x_j} = 0 & (1) \\ \dfrac{\partial(\rho u_j)}{\partial t} + \dfrac{\partial(\rho u_i u_j)}{\partial x_j} + \dfrac{\partial p}{\partial x_i} - \dfrac{\partial \tau_{ij}}{\partial x_j} = 0 & (2) \\ \dfrac{\partial(\rho E)}{\partial t} + \dfrac{\partial(\rho E u_j)}{\partial x_j} + \dfrac{\partial q_j}{\partial x_j} + \dfrac{\partial p u_j}{\partial x_j} - \dfrac{\partial u_i \tau_{ij}}{\partial x_j} = 0 & (3) \end{cases}$$

Where:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right); q_j = -\lambda \frac{\partial T}{\partial x_j}; Pr = \frac{\mu c_p}{k} = \gamma \frac{\mu c_v}{k}$$

The total energy is expressed as follows:

$$E = e + \frac{u^2}{2} \quad (4)$$

Gas is calorically perfect, then e by unit mass is:

$$e = c_v T \quad (5)$$

The flow pressure is obtained by the equation of state:

$$p = \rho r T \quad (6)$$

Where: $r = \frac{R}{M}$

The flow temperature is calculated from the energy equation (5).

With ρ the density, p is the static pressure, u and v Cartesian velocity components, E the total energy per unit mass and e internal energy per unit mass.

III. TURBULENCE MODEL

In this study, we used the model with two transport equations (k-ω) of Menter SST with correction (Shear Stress Transport), given its effectiveness and his popularity in the numerical studies calculation of compressible flows [4] and [10].

The model (k-ω) of Mount consists of a combination of the model (k-ω) of the Wilcox and (k-ε) of the Launder-Sharma [10].

The k-ω Shear Stress Transport (SST) turbulence model of Menter, merges the k-ε model of Wilcox, with a high Reynolds number k-ε model (transformed into the k-ω formulation). The SST model seeks to combine the positive features of both models. Therefore, the k-ω approach is employed in the sublayer of the boundary layer. The reason is that the k-ω model needs no damping function. This leads, for similar accuracy, to signifi-cantly higher numerical stability in comparison to the k-e model. Furthermore, the k-ω model is also utilised in logarithmic part of the boundary layer, where it is superior to the K-e approach in adverse pressure flows and in compressible flows. On the other hand, the K-ε model is employed in the wake region of the boundary layer because the k-ω model is strongly sensitive to the freestream value of ω. The k-ε approach is also used in free shear layers since it represents a fair compromise in accuracy for wakes, jets, and mixing layers, [13].

The equations to be solved are the equation of the turbulent kinetic energy k, the specific dissipation rate ε and the freestream value of ω.

Turbulent Viscosity is expressed by:

$$\upsilon_t = C_\mu \frac{k}{\omega} = \rho . c_\mu . \frac{k^2}{\varepsilon}$$

Transport equations are illustrated by the following equations:

$$\frac{\partial(\bar{\rho} k)}{\partial t} + \frac{\partial(\bar{\rho} u_j k)}{\partial x_j} = \bar{\rho} P - \bar{\rho}\omega k + \frac{\partial}{\partial x_j}\left[\left(\bar{\mu} + \frac{\bar{\mu}_t}{\sigma}\right)\frac{\partial k}{\partial x_j}\right]$$

$$\frac{\partial(\bar{\rho}\omega)}{\partial t} + \frac{\partial(\bar{\rho} u_j \omega)}{\partial x_j} = C_{\omega 1}\frac{\bar{\rho} P \omega}{k} - C_{\omega 2}\bar{\rho}\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\bar{\mu} + \frac{\bar{\mu}_t}{\sigma_\varepsilon}\right)\frac{\partial \omega}{\partial x_j}\right]$$

With constant: $C_\mu = 0.09, C_{\omega 1} = 0.55$

Where:

$$k = \frac{u^2}{\sqrt{c_\mu}}, \varepsilon = \frac{k^{3/2}}{l}, \omega = \frac{\varepsilon}{C_\mu . k}, l = c_\mu^{-3/4} . l_m$$

Transport equations are illustrated by the following

IV. DOMAIN OF CALCULATION

The geometry of the nozzle and the mesh is created with the software "Gambit" 2.2.30. Several methods allow the creation of this geometry, or we rely on predefined geometries, or you just need to enter the coordinates of the different points (x, y) in 2D, create boundaries and finally create the surface. The system studied is a nozzle composed of a converging, a col and a divergent Figure 1, covered by a viscous flow, compressible, turbulent, laminar and instationary. The convergent and divergent are connected to the col by arcs of a circle for ensure continuity of the nozzle profile. Note that this geometry was already being used by Haouia R. et all. [1]

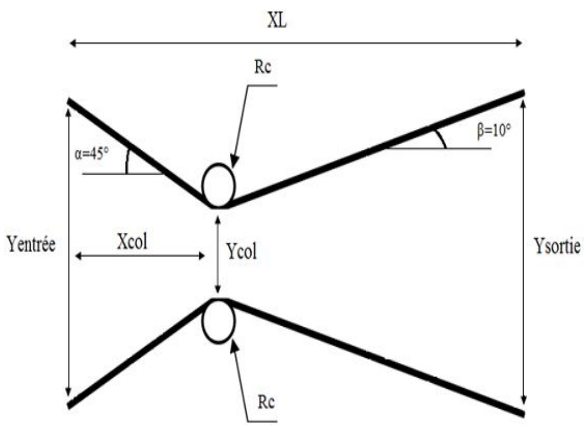

Fig.1. Detail geometry of the nozzle

With *Rc* the radius of the arcs of a circle, *Ycol* the radius of the col of the nozzle, *Yentrée* the inlet radius of the nozzle, *Ysortie* the radius of the nozzle outlet, *Xcol* the abscissa of the geometry col of the nozzle and the *XL* length of the nozzle. Dimensions are in meters.

V. BOUNDARY CONDITIONS

We considered a data of the experience to air flow in the nozzle of wind tunnel with a shock destiny for study to
hypersonic flows around space vehicles produced by R. Haouia and all [1] with conditions (at the inlet is the nozzle) the pressure is 100 (bar) and temperature is 6000 (K), the air flows through the nozzle, accelerates along the nozzle and ejects high speed vilocity air with conditions in the output is the diverging the nozzle: the temperature decreased to the value is 500 (K) and the Mach number is greater than 1.

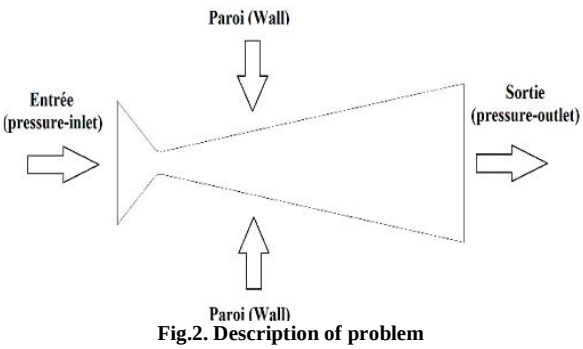

Fig.2. Description of problem

VI. MESH

The choice of mesh plays a very important role in the convergence of calculations including to the col it should be high precision. The mesh that we used in this study consists of 300 nodes along the axis (x) and 100 nodes depending on the radius (y) (Figure 3).

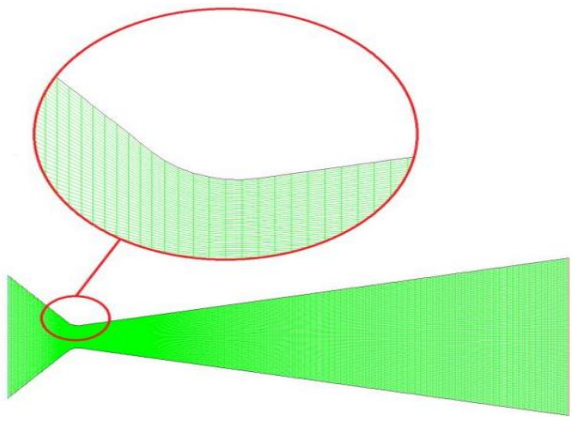

Fig.3. Uniform Mesh (300x100)

VII. VALIDATION OF RESULTS

The validation of the results of our work is very important because it allows us to verify and compared it our results obtained with numerical simulation by other experimental results, theoretical and numerical. Our work has been completed with the numerical results of R.Haouia and all [1]. They obtained profiles of Mach number and temperature in a supersonic nozzle, as shown in Figures 4 and 5 are acceptable and good agreement with our results.

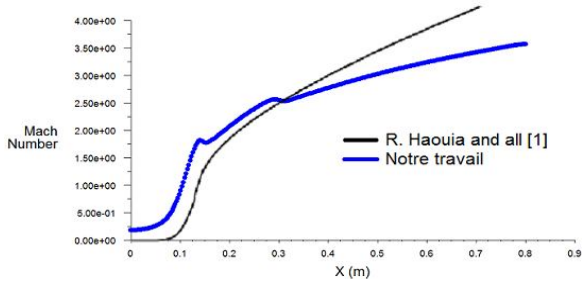

Fig.4. Number match Profiles comparison between our work and that of R.Haouia and all [1]

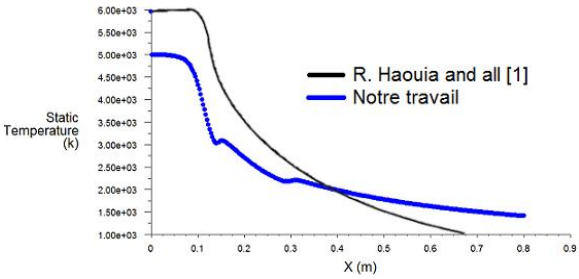

Fig.5. Temperature profiles comparison between our work and that of R.Haouia and all [1]

Considering the supersonic regime in a convergent-divergent nozzle, the solutions presented were obtained after over 300,000 iterations, the calculation time takes several hours.

Figures 6 and 7 show the left iso densities and right its evolution curve. As is noted in these Figures that the density profile takes two different paths, the first path is from the inlet of the nozzle up to its col, the density in this portion is almost constant at its maximum value. The second path that begins just after the col of the nozzle the density thus undergoes a small sharp increase then continues its decrease until the exit of the nozzle because the flow is compressible *(density does not remain the even during the movement).*

The pressure drop inside the nozzle schematically in Figures 8 and 9. Qualitatively this profile follows almost the same shape of density profile.

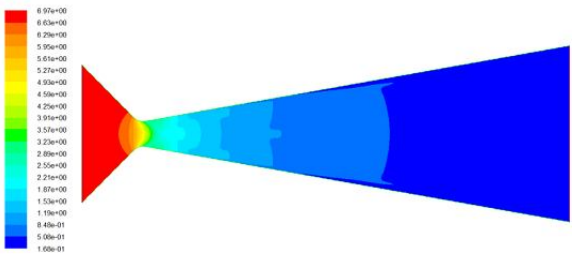

Fig.6. Contours of Density (kg / m^3)

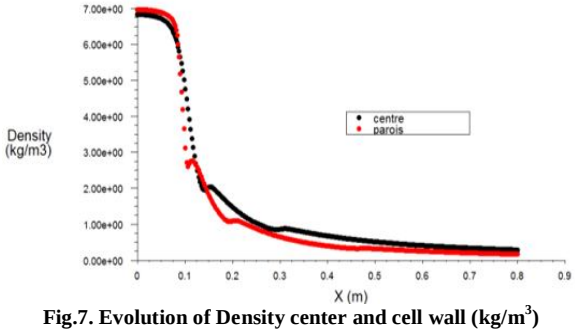

Fig.7. Evolution of Density center and cell wall (kg/m^3)

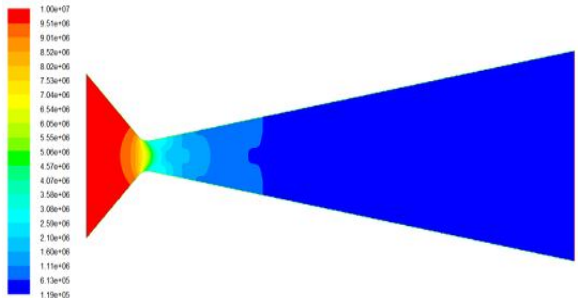

Fig.8. Contours of Static Pressure (pa)

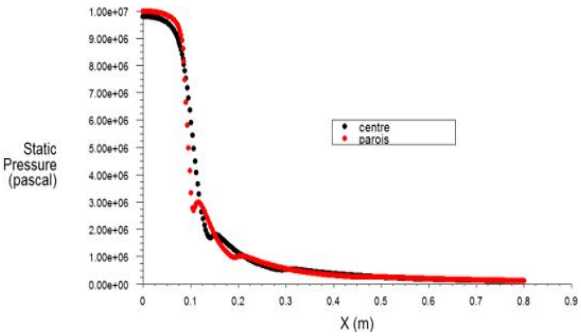

Fig.9. Evolution of Static Pressure center and cell wall (pa)

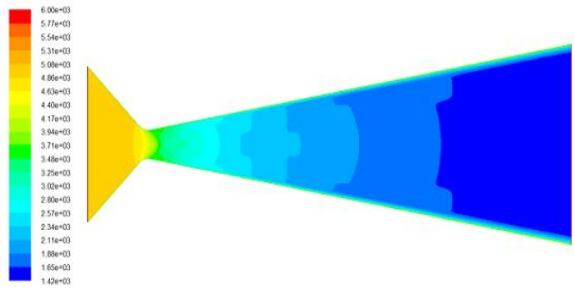

Fig.10. Contours of Temperature (K)

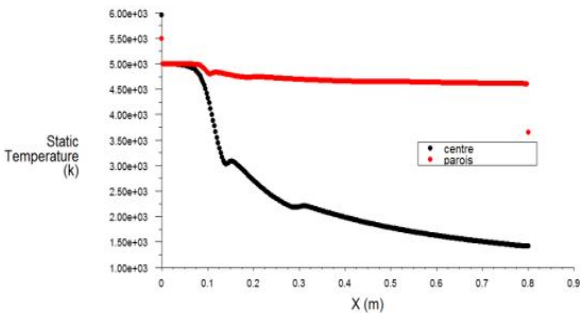

Fig.11. Evolution of Temperature center and cell wall (K)

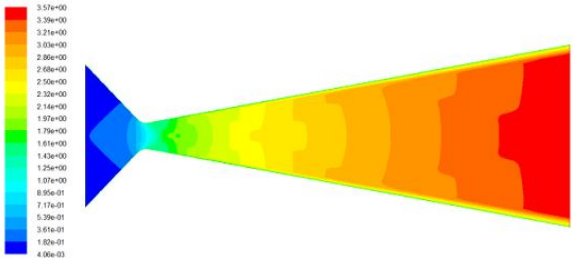

Fig.12. Contours of Mach Number

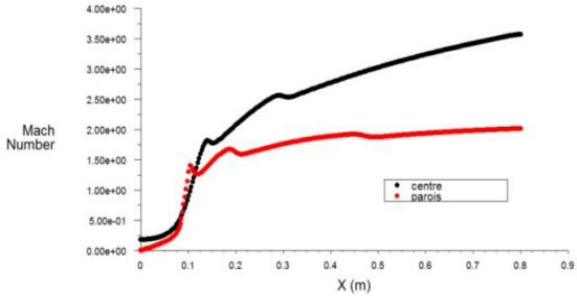

Fig.13. Evolution of Mach Number center and cell wall

The temperature distribution and its evolution respectively in Figures 10 and 11, it was seen that the temperature within the nozzle undergoes three main phases of decline, the first phase is stable in the form of a straight line, it is at the level of converging up to the col of the nozzle, the second phase of decline is a sharp decrease in temperature at the col and the last phase is in the divergent, it continues to decrease until the exit of the nozzle with small sharp increase in the input of the divergent (x = 0.175m) and the other in the middle of the nozzle (x = 0.45m), on the other hand the proportion steady parietal distribution along the nozzle because friction between the flow and the walls.

Iso-values and the evolution of the Mach number in (central and parietal) of the nozzle shown in Figures 12 and 13 respectively, we observe that the regime at the inlet of the nozzle remains almost steady or invariable up to the first contact of the abscissa of point of tangency at the level of convergent and we can say that the regime is subsonic because the value of the Mach number _ is strictly less than one, then it follows by a sharp increase in the vicinity of col of the nozzle exactly between the first point of tangency at the level of converging and the first point of tangency in the divergent in this zone the regime becomes transonic saw his Mach number, as there was a small sharp decrease in the center nozzle, at the inlet to the zone of the divergent (x = 0.15m), the value of Mach number on the inside of the nozzle continues to increase up to the outlet of the nozzle where it reaches a maximum value equal to 6, in this zone the regime is hypersonic.. The same observations for the evolution of Mach number except parietal disturbance in the vicinity of the col due to friction between the fluid and the wall, and the convergent-divergent profile of the nozzle accelerates

the gas to a subsonic velocity to supersonic velocity (hypersonic).

Finally Figures 14 and 15 shows the behavior of the turbulent viscosity inside the nozzle and the evolution center and parietal respectively, resulting in the vast majority of the nozzle a weak turbulent viscosity, exceptionally just before the col and any long parietal of the nozzle there is a significant viscosity level.

CONCLUSION

In recent years, remarkable progress has been made in the domain of propulsion in general and aerospace propulsion in particularly, but all this progress and the studies of these physical phenomena at high velocity and very high temperature in nozzles often misunderstood, this applies especially several theoretical, numerical and even experimental studies to understand the maximum of these flow phenomena in nozzles used. These progress; numerical study of flow inside the nozzles that can address the problems of physical phenomena without losing a lot of money and make a lot of the time, so we can say that the numerical simulation of turbulent compressible flows is a better means to better understand the physical phenomena.

This numerical simulation of flow dynamics through a nozzle, using the computer code (FLUENT) by the use the type of turbulence model (k-ω) at high vilocety and temperature, which we provides an approach to the behavior of compressible flow in this nozzle and the effects of the geometry of the latter and their parameters on the flow behavior in 2D.

REFERENCES

[1] R. Haouia, A. Gahmousse, and D. Zeitoun, “Ecoulement hors d’équilibre chimique et vibrationnel dans une tuyèrehypersonique axisymétrique,” papier, accepté le 19 octobre 2000, 2001 Editions scientifiques et médicales Elsevier SAS.

[2] A. Nebbache, “Modélisation d’écoulement en tuyère plane et bidimensionnelle,” 19ème Congrès Français de Mécanique, Marseille, 24-28 août 2009.

[3] A.Nebbache, “Aérodynamique d’un écoulement en tuyère idéale tronquée, ” 18ème Congrès Français de Mécanique, pp. 787-795, Grenoble, 27-31 août 2007.

[4] E.Mahfoudi, A.Gahmousse, and K.Talbi, “Etude numérique de l’écoulement compressible turbulent dans une tuyère supersonique,” Revue des énergies renouvelables Vol. 16 N°2 (2013) 285-296, 2013.

[5] E.Mahfoudi, A.Gahmousse, A.Harizi, K.Talbi, and A.Hadjadj, “Simulation Numérique De L’écoulement Compressible Supersonique Application aux Tuyères Propulsives à Combustible liquide Hydrogène, ” Revue des énergies renouvelables Vol. 15 N°3 (2012) 365-372, 2012.

[6] E.Mahfoudi, “Contribution à l’étude des profils de tuyères en écoulements supersoniques visqueux par la méthode des volumes finis,” Thèse de doctorat en sciences de l’université Constantine 1, 2014.

[7] M.Y. Bouzid, and R. Dizene, “Modélisation des écoulements dans les tuyères étude comparative de modèles de turbulence,” 17ème Congrès Français de Mécanique, Troyes, Septembre2005.

[8] A. Hadjadj, “Analyse physique et simulation numérique des écoulements compressibles, applications aux tuyères de propulseur,” Thèse de doctorat, Université de Rouen, 1997

[9] S. Dubos, “Simulation des grandes échelles d'écoulements turbulents supersoniques,” institut national des sciences appliquées de Rouen, 20 septembre 2005.

[10] Y.PERROT, “Etude, mise au point et validation de modèles de turbulence compressible,” institut national des sciences appliquées de Rouen, 19 décembre 2006

[11] L.Thierry, “Etude numérique d’un écoulement gazeux dans une tuyère convergente divergente - technique TVD, ” Rapport de stage de 4ème année,” institut national des sciences appliquées de Rouen, juin-octobre 1993.

[12] JOHN D. ANDERSON, JR, “Computational fluid dynamics,” McGraw-Hill, New York, Edition 1995.

[13] J. Blazek, “Computational Fluid Dynamics: Principles and Applications, ” Second Edition, ELSEVIER 2005

★★★