Numerical investigation of turbulent gas flow in an over-expanded nozzle flows: hysteresis phenomenon and asymmetrical configuration

R. Benderradji [1,2] *, H. Gouidmi [1] and A. Beghidja [1]

[1] Laboratory of Renewable Energy and Sustainable Development, LREDD
Université des Frères Mentouri- Constantine1, 25000 Algeria
[2] Faculty of Science, Université Med Boudiaf, 28000 M'Sila, Algeria

(reçu le 20 Décembre 2017 - accepté le 25 Décembre 2017)

Abstract - *The flow in an Over-Expanded Nozzle is subjected to shock waves leading to the unsteady separation of the boundary layer. Free detachment may be followed by a restricted detachment. During the expansion regime in propellant nozzles, several physical phenomena are encountered: supersonic jet, jet separation, adverse pressure gradient, shockwave, turbulent boundary layer, highly compressible mixture layer, return flow, large-scale turbulence. These very complex phenomena can considerably affect the performance of the nozzle. This numerical study contributes to the analysis of the character of the detachment as well as the transition from one type of detachment to the other, and the numerical precision of the different models of turbulence that can predict the properties and physical phenomena of the flow in a flat nozzle. The study was conducted on a test case of the ATAC project {Aerodynamics of the Hoses and Backs-bodies}. Girard [1], using a CFD-Fastran calculation code, based on the resolution of the Navier-Stokes equations by the finite volume method. The study highlights the phenomen on hysteresis and the asymmetrical configuration due to the detachment of the boundary layer.*

Résumé - *L'écoulement dans une buse surexploitée est soumis à des ondes de choc conduisant à la séparation instable de la couche limite. Le détachement libre peut être suivi d'un détachement restreint. Au cours du régime d'expansion des tuyères propulsives, plusieurs phénomènes physiques sont rencontrés: jet supersonique, séparation par jet, gradient de pression défavorable, onde de choc, couche limite turbulente, couche de mélange fortement compressible, flux de retour, turbulence à grande échelle. Ces phénomènes très complexes peuvent affecter considérablement les performances de la buse. Cette étude numérique contribue à l'analyse du caractère du détachement ainsi qu'à la transition d'un type de détachement à l'autre, et à la précision numérique des différents modèles de turbulence qui permettent de prédire les propriétés et les phénomènes physiques du flux dans un buse plate. L'étude a été réalisée sur un cas test du projet ATAC {Aerodynamics of Hoses and Backs-bodies}. Girard [1], utilisant un code de calcul CFD-Fastran, basé sur la résolution des équations de Navier-Stokes par la méthode des volumes finis. L'étude met en évidence les phénomènes d'hystérésis et la configuration asymétrique due au détachement de la couche limite.*

Keywords: Nozzle - Over-expanded - Detachment - Sshockwave - In-stationarity.

1. INTRODUCTION

The discipline of aerodynamics helps to define the technical specifications of a launcher for the calculation of performance and for the reduction of unsteady charges which can compromise the structural integrity of the propulsion system and damage the payload. A lot of work has been done on the internal aerodynamics of propulsive nozzles.

This work consisted mainly of drawing nozzles whose evolution of the profile from the neck is very brutal. The ratio of the section being thus more important, the theoretical propulsion is found to increase. In this way, such nozzles make it possible to

* benderradjirazik@gmail.com

maximize the thrust in the flight phases at high altitude, where the pressure is very low. The propellant gases then undergo a strong relaxation. However, when starting the engine on the ground, takeoff and the initial phase of the flight, the latter encounters external pressures higher than the pressure in the outlet section of the nozzle.

The abrupt adaptation of the pressure of the propulsive flow to the external pressure is done by means of a recompression shock which is positioned in the nozzle. The diet is called the over denture diet. In the case of high overdrive, the recompression shock causes the turbulent delaminating of the boundary layer.

This phenomenon due to its asymmetrical and unsteady nature is at the source of lateral loads exerted on the structure of the nozzle which is exerted on the structure of the nozzle.

Nevertheless, other sources of lateral loads exist. It has been observed in this type of nozzle the presence of an internal shock resulting from the focusing of Mach lines from the neck. This shock will interfere with the overdrive system forming particular shock structures. During this process, the boundary layer peels off and sticks to the wall.

These are turbulent transient phases resulting from priming and defusing for which the charges are very intense during these passages from one configuration to the other. The reasons leading to the re-bonding of the turbulent boundary layer are still poorly understood, with some studies focusing on the influence of internal shock. This is the origin of the boundary layer recollection and associated transient that we will seek to study, analyze and understand in this study.

2. NUMERICAL PROCEDURE

2.1 Optimization of the numerical simulation

The study was conducted on a test case of the ATAC project. The simulated nozzle is a 2D divergent converging nozzle, (figure 1) with a section ratio of 1.7, a Mach number at the output greater than 2, and a divergence half-angle of 10 °.

2.2 CFD code description

The numerical study has been conducted using a finite volume Reynolds-Averaged-Navier-Stokes (RANS) solver (CFD-Fastran) developed by the CFD Re-search Corporation. This code offers two upwind differencing schemes with a variety of higher order limiters to calculate the convective terms in the transport equations. Both explicit and fully implicit time integration schemes are available for steady and unsteady flow simulations.

The code is based on a cell-centered finite volume discretisation. Inviscid fluxes may be computed using Roe's flux difference splitting scheme and Van Leer's flux vector splitting. Both schemes are first order spatially accurate. Flux limiters may however be used to rise up the spatial accuracy.

For Min-mod and Van Leer limiter the accuracy is 2^{nd} order while for Osher Chakravarty limiter the accuracy is up to third order. CFD-Fastran has five turbulence models ($k-\varepsilon$, $k-\omega$, $k-\omega$ SST-Menter, Spalart-Allmaras and Baldwin-Lomax). For this study, the flux vectors are evaluated, at each time step, using Roe's upwind flux difference splitting, with a MINMOD flux limiter in order to achieve a high-order spatial accuracy.

Three turbulence models are used: Baldwin-Lomax, Spalart-Allmaras and $k-\omega$ SST. Time-integration is achieved using a fully implicit scheme. Local time stepping is also used to accelerate convergence to steady state.

2.3 Meshing and boundary conditions

Figure 1 shows the profile of the nozzle used in the 2D calculations. The block-structured mesh is composed of four blocks respectively for calculations on a complete nozzle. Figures 2**a** and 2**b**, show the meshes used for numerical calculations with the associated boundary conditions.

A first zone meshing inside the nozzle (1), it contains the largest number of cells. Zone (2) covers the external field downstream of the nozzle. Zones (3) and (4) located at the top and bottom of the nozzle respectively. The refinement of the mesh is also taken into account near the walls in order to better simulate the boundary.

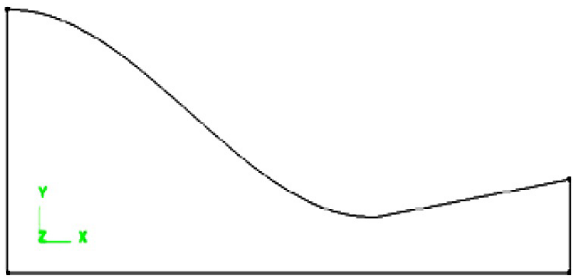


Fig. 1: Profiles of the nozzle used in 2D calculations

Numerical calculations are performed for turbulent and stationary flow. A subsonic input condition is imposed at the inlet of the nozzle where the generating conditions and the direction of the velocity are imposed. The walls of the nozzle and the upstream outer domains are adherent and adiabatic.

The upper and lower boundaries are provided with non-reflective conditions. Finally, a subsonic output condition is imposed on the downstream boundary of the domain. This last condition requires a significant longitudinal extension to allow the jet to become subsonic by diffusion of the momentum by the viscosity.

2.4 Influence of turbulence models

In numerical calculations, the choice of the turbulence model significantly affects the results. Several models were tested: the algebraic model of Baldwin_Lomax, the model with an equation of Spalart_Allmaras and the models with two equations of transport ($k-\varepsilon$ and $k-\omega$).

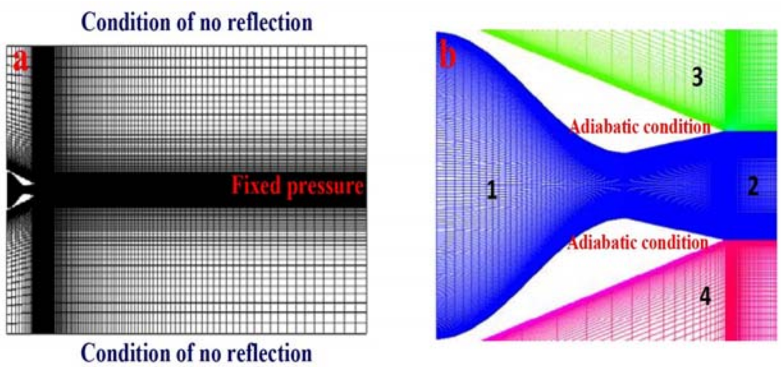


Fig. 2: Structured mesh of the nozzle

Figure 3 shows the influence of turbulence models on the distribution of wall pressure along the nozzle divergence.

Numerical calculations are carried out at $NPR = 6$. We note that all the models used, for example, the case of the model $k-\varepsilon$. The model of Spalart Allmaras, Balwin_Lomax and $k-\omega$ reproduce the area of separation appropriately compared to the experiment. Given these results, the k-ω model will be used in subsequent 2D calculations.

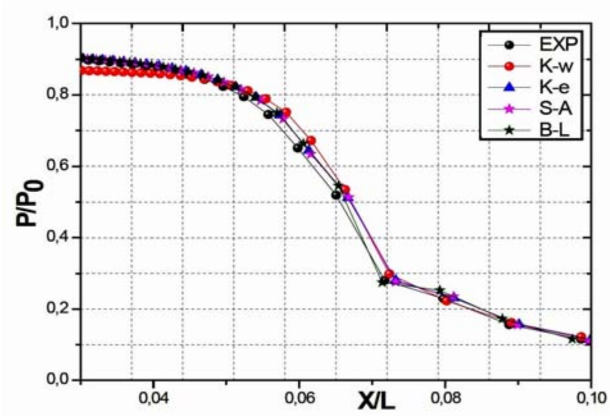
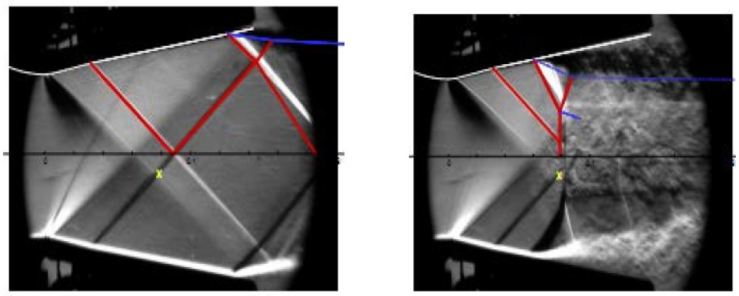


Fig. 3: Influence of the turbulence model on the distribution of pressure at $NPR = 6$

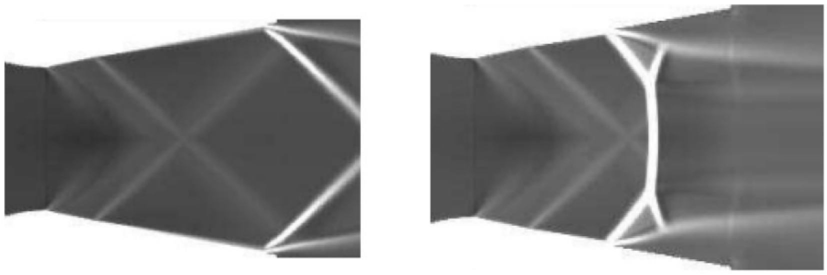
2.5 Validation of the results

Figure 4 shows the Mach number iso-contours. It shows a case of validation of our calculation with the experimental results, analytical Girard [1], and numerical, Sellam [2]. It can be seen that the three configurations are almost the same. The shock-like or relaxation wave structures are visible in the downstream flow, and the structures observed numerically are similar to those of the experiment.

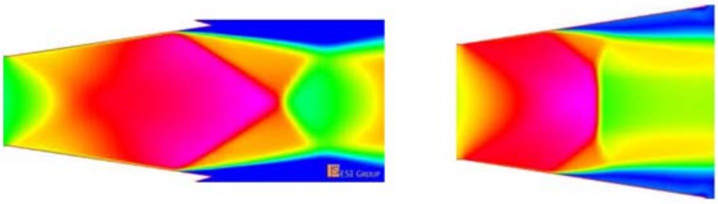
Also shown in figure 5; is a comparison of the upper wall pressure distribution, between the experiment, and the numerical simulation for the overpressure regime, $NPR = 3.80$. We note a good agreement between the calculation results and the experimental values, and that they have the same distribution except that the pressure plateau after separation point is a little apart.



Theoretical Girard [1] and experimental Sellam [2]



Numerical calculation Sellam [2]



Present CFD-Fastran calculation

Fig. 4: Comparison of theoretical, experimental, and numerical structures

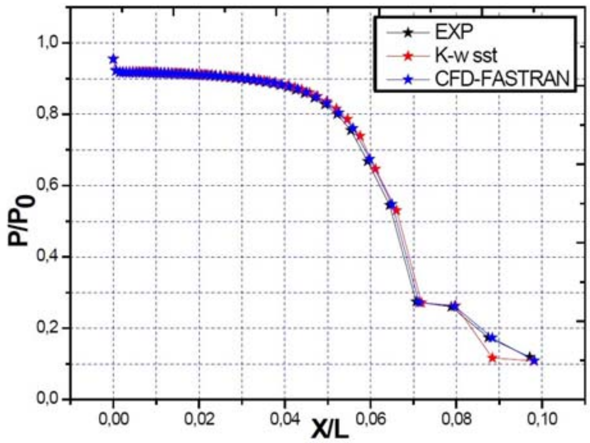


Fig. 5: Distribution of the parietal pressure of the upper walls
$\text{NPR} = 3.80$. Comparison of our results with Sellam [**2**]

4. RESULTS AND DISCUSSION

4.1 Asymmetric flow configuration and hysteresis phenomenon

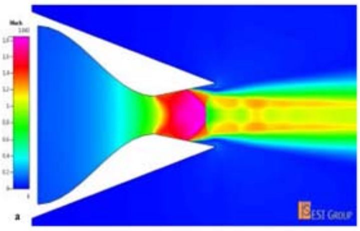
To reproduce the hysteresis phenomenon in the supersonic nozzle, we use the NPR expansion rate (the ratio between the nozzle generating pressure and the ambient pressure), defined by the P_{i0} / P_a ratio to parameterize the calculations. This ratio between the pressure generating the nozzle and the ambient pressure is at the origin of the formation of a recompression shock. The major disadvantage of the existence of this shock is that it can go inside the nozzle, favoring the delaminating of the boundary layer.

In some cases, the boundary layer takes off without re-sticking, and in other cases, the delaminating is followed by a re-bonding of the downstream boundary layer inducing the creation of a recirculation bulb.

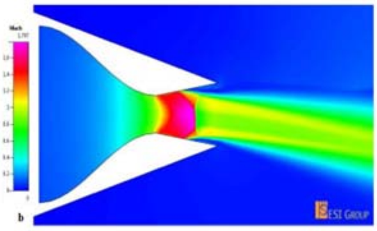
These two configurations are respectively named: Free Shock Separation (FSS) and Restored Shock Separation (RSS). The in stationary of this interaction is the source of significant oscillating mechanical loads and intense variation of heat flux on the wall of the nozzle. This phenomenon has been demonstrated experimentally, then numerically for the first time, by Sellam [1].

Several NPR's, varying from $NPR = 1.80$ to $NPR = 5.42$, were studied by 2D RANS numerical simulation carried out on a complete nozzle, in order to observe the different configurations (asymmetric and symmetrical flaking, hysteresis phenomenon).

It then appears figure 6, that under a critical NPR, the flow in the nozzle becomes asymmetrical with respect to the axis of the nozzle. This observation is in agreement with experiments. This asymmetric configuration has also found also for other geometries, for example, Lawrence [2], Bourgoing [3], Reijasse [4], Pilinski [5], and Shimshi [6].



a- Symmetric configuration for $NPR = 2.60$



b- Asymmetric configuration for $NPR = 2.15$

Fig. 6: Visualization numerical flow configuration at two different NPRs

In order to investigate whether the critical NPR has the same value as the increasing and/or decreasing NPR, the simulation was initialized for $NPR = 5.42$, and the boost ratio was progressively decreased to $NPR = 1.8$, convergence was reached at each step for a steady stationary solution, from the converged initial field for the previous NPR.

Multi-stage numerical simulation results are shown in figure 7. It clear that the flow is initially symmetrical. At $NPR = 1.9$ (critical NPR case) the flow becomes asymmetric and the asymmetry remains as the NPR further decreases. As a result the ratio of relaxation has been increased gradually, we notice that the flow suddenly becomes symmetric and the asymmetry still remains when the NPR increases much higher than the critical NPR (case where increasing), which shows that it is the flow is initialized with an asymmetric configuration the NPR must be increased much higher in order to recover the symmetry, and the critical NPR is slightly different from that obtained in the increasing case.

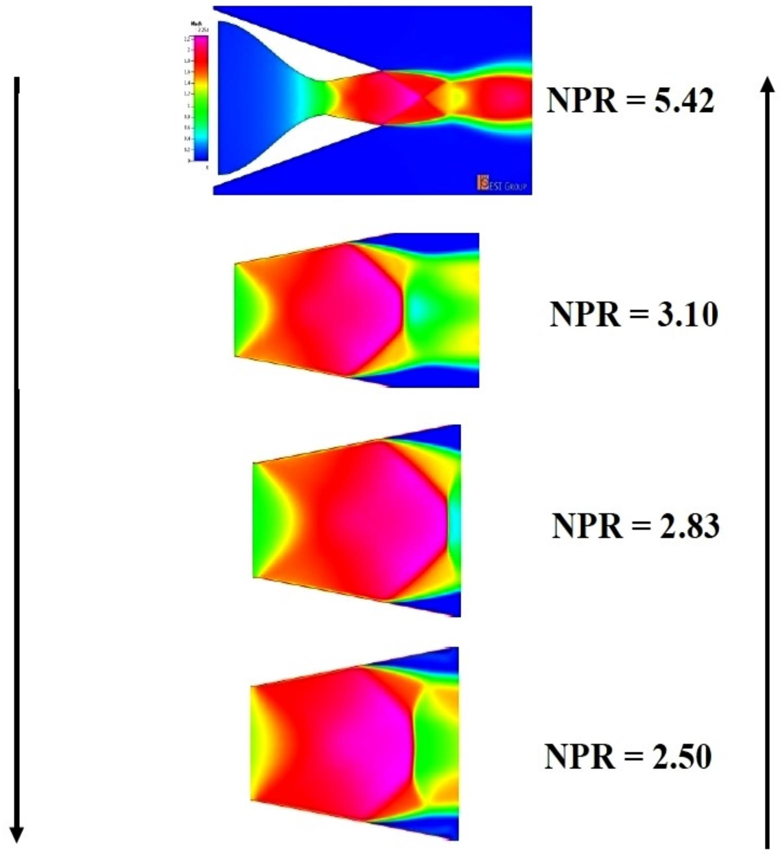
On the other hand, when the value of the NPR decreases again, the case where $NPR = 1.9$ the asymmetry begins to occur for which below this value the symmetrical configuration no longer exists, and when the value of the NPR increased, the case where $NPR = 2.50$, the flow recover the symmetry with respect to the axis of symmetry of the nozzle and beyond this value, the configuration remains symmetrical.

Between these two values the existence of a double zone of NPR where the two configurations are possible testifies (association of a phenomenon of hysteresis). Note also that for rates of significant relaxation, $\mathrm{NPR}=5.42$, and $\mathrm{NPR}=3.80$, the reflection of the shock of relaxation is done regularly. The lowering of the relaxation rate results in Mach reflection, as well as the rise of the detachment point.

In all these cases, the separation is free and the interaction between the internal shock reflected and the shock of relaxation is highlighted. In the case where $\mathrm{NPR}=2.02$ it is the incident internal shock that intersects the Mach disk, behavior consistent with the experience.

The separation point is located in the same way very upstream. We also see in our case that a free detachment of the boundary layer induces the deformation of the Mach disk towards the lower wall, the wall on which the gluing occurs. Naturally, this tendency is reflected in the shape of the pressure curves which have on the lower wall a point of inflection corresponding to the recirculation bubble.

Calculating at lower NPRs of 1.8 does not change anything. The results are not presented for the low expansion rates. To conclude this part, the study clearly shows that the symmetrical nature of the flow is very dependent on the variation of the NPR in the nozzle, which is of great industrial interest compared to the structural integrity of the nozzle when launching rockets and/or spaceships.



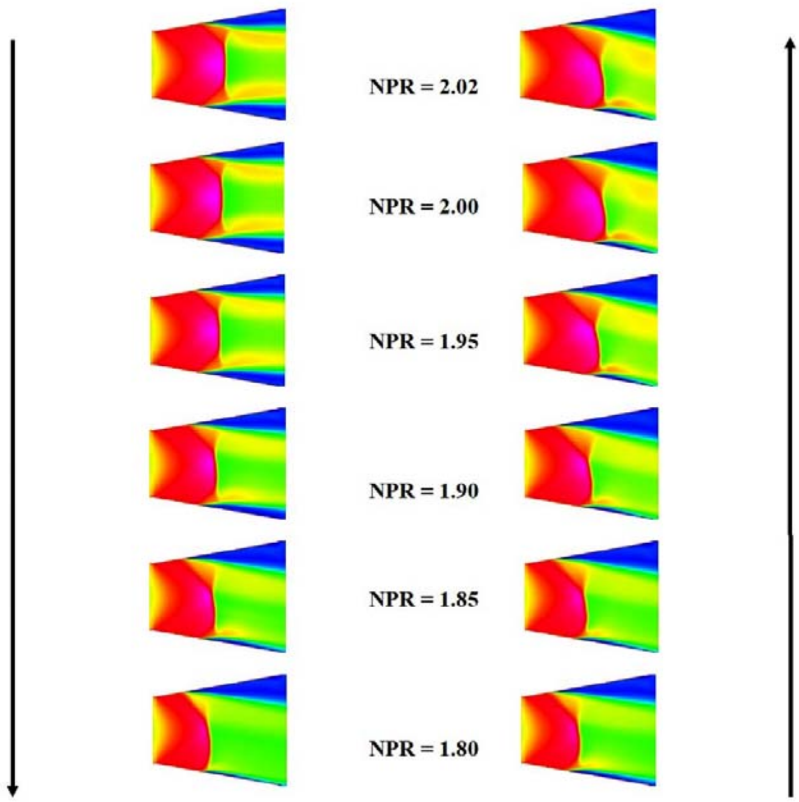


Fig. 7: Hysteresis phenomenon induced by NPR expansion ratio variation

4.2 Influence of flow on pressure relationships

The upper and lower parietal pressure ratios are shown respectively in figure 8A; {the increasing case of the relaxation number}, and figure 8B; {the decreasing case of the expansion number}. They show a difference in the position of the separation point which is more important for $NPR < 3.80$. This gap decreases with the decline in the relaxation rate. The detachment occurs further downstream in the numerical simulations.

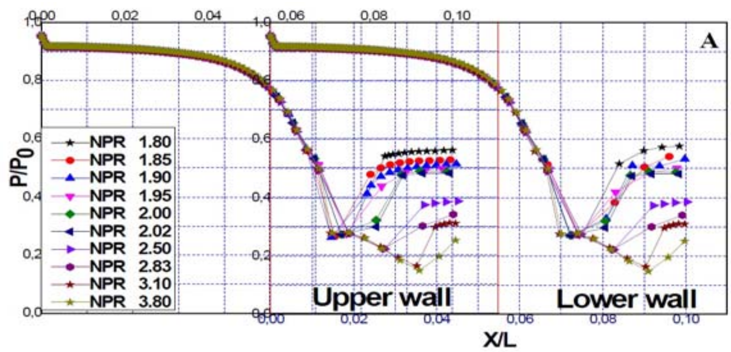


Fig. 8A: Comparison of pressure ratios (NPR increasing)

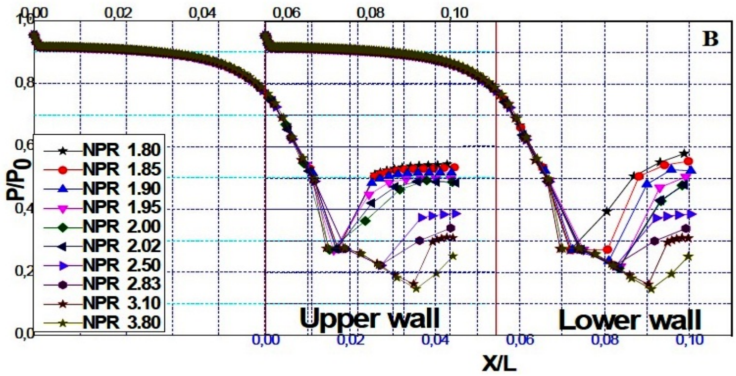


Fig. 8B: Comparison of pressure ratios (NPR decreasing)

The pressure plateau after detachment is very well reproduced, as well as the pressure jump due to the internal shock. Different factors can explain the pressure ratio deviations and the position of the separation point in the 2D numerical calculation:

• Choice or parameterization of the turbulence model.

• Near-wall mesh.

• Incorrect evaluation of the displacement thickness of the boundary layer that develops on both walls of the nozzle.

5. CONCLUSION

The existence of a hysteresis phenomenon associated with the shock wave/boundary layer interaction. This type of interaction, it is present in many fields of aeronautics such as the internal aerodynamic (air intakes, scramjet), external (presence of shock of extrados in transonic, Over-Expanded Nozzle flows).

The presence of this interaction in these mechanical systems is a real practical problem because it is at the origin of strong unsteady stresses, which can lead to structural fatigue and their destruction. It can also lead to instabilities in the operation of the engines (pumping of compressors, instability of combustion in scramjet ...).

REFERENCES

[1] S. Girard, '*Etude des Interférences de Choc dans les Tuyères sur détendues à Choc Interne*', Thèse de Doctorat, Université Pierre et Marie Curie, ONERA-Meudon, 2009.

[2] M. Sellam, G. Fournier and A. Chpoun, '*Numerical Investigation of Over Expanded Nozzle Flows*', Shock Waves, Vol. 24, N°1, pp. 33 - 39, 2014.

[3] R.A. Lawrence, '*Symmetrical and Unsymmetrical Flow Separation in Supersonic Nozzles*', Research Report, N°67-1, Southern Methodist University, 1967.

[4] A. Bourgoing and P. Reijasse, '*Experimental Analysis of Unsteady Separated Flows in a Supersonic Planar Nozzle*', Shock Waves, Vol. 14, N°4, pp. 251 - 258, 2005.

[5] P. Reijasse, B. Corbel and D. Soulevant, '*Unsteadiness and Asymmetry of Shock-Induced Separation in a Planar Two-Dimensional Nozzle, A Flow Description*', AIAA Paper, N°99-3694, 1999.

[6] C. Pilinski and A. Nebbache, '*Unsteady Separated Two-Throat Nozzle Flows. Flow*', Turbulence and Combustion, Vol. 71, N°1-4, pp. 247 - 259, 2003.

[7] H. Li, A. Chpoun and G. Ben-Dor, '*Analytical and Experimental Investigations of the Reflection Shock Wave in Steady Flows*', Journal of Fluid Mechanics, Vol. 390, pp. 25 - 43, 1999.