Numerical Simulation of Overexpanded Jets in Internal and External Flows

Y. Perrot and A. Hadjadj
Institut National des Sciences Appliquées, LMFN-CORIA UMR 6614 CNRS
Avenue de l'Université, 76801 Saint Etienne du Rouvray, France
Phone: (+33) 232.959.725; Fax: (+33) 232.959.780, e-mail: perrot,hadjadj@coria.fr

Abstract
A numerical investigation was conducted to assess the ability of the three-dimensional Navier-Stokes solver, N3S-Natur [1], using the k-ω SST turbulence model when computing nozzle-afterbody flows with propulsive jets. Three nozzle configurations were selected as test cases for the computational method: the first is the ONERA TIC nozzle, the second is an axisymmetric boattailed afterbody configuration and the third is a fully 3D transonic nozzle. In most situations, internal and external flow-field regions are modeled. The obtained results are carefully analyzed and compared to the experimental data. A three-dimensional computation was done to make evidence of 3D phenomena which are not negligible. A particular attention was payed to the appearance of a recirculation zone on the afterbody.

1. INTRODUCTION
The aerothermodynamic phenomena over space launch vehicles, rockets, missiles and other supersonic vehicles are a challenging problem for aeronautical applications.

Basically, the physical problem met in this flow configuration is essentially the result of the interaction of two merging flows; one coming from a propulsive jet, and the other caused by the ambient high speed stream (see Figure 1). Several complex purely viscous phenomena, such as boundary layers with adverse pressure gradients, shocks, induced separation, recirculation bubbles, shear layers can occur and may strongly affect the engine's performance. In addition, the heat loading at the base of a vehicle can be considerable, since most of these phenomena occur in hot gases at very high temperature with complex chemical reactions and important heat-transfer mechanisms.

From numerical point of view, the quality of computational results depends strongly on the accuracy of the turbulence model used for CFD and a rather fair prediction of the flow-field can be accompanied by large errors in the calculation of wall properties, affecting mainly transfer coefficients, skin friction and heat flux.

The current investigation assesses the capability of the Navier-Stokes method N3S-Natur, version 1.4.5-DP, using the k-ω SST model to predict complex flows at high speed including afterbody flows. The ONERA TIC nozzle [2, 3] as well as the RTO afterbody [4, 5] and the fully 3D AGARD nozzle [6, 7] (referenced as B.4.2) are retained as test cases, and numerical results are compared with the experimental data. These geometries were chosen because they were proposed by the ATAC, the RTO

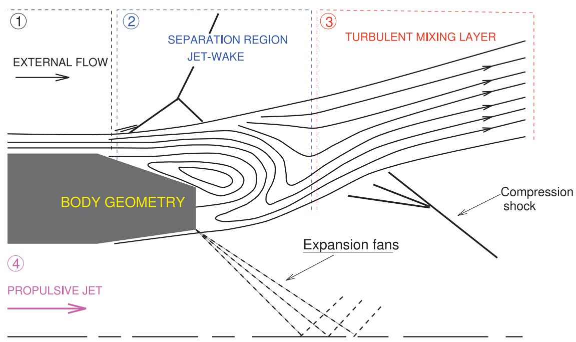

Figure 1. Afterbody nozzle configuration.

and the AGARD working groups on afterbody aerodynamics, respectively as a reliable cases to evaluate CFD codes for the analysis of complex jet interactions. A description of some of the most important flow phenomena occurring on typical nozzles and afterbodies is presented. A three-dimensional computation was done to make evidence of 3D phenomena which are not negligible. It is concluded that the k-ω Shear Stress Tensor turbulence model provides an interesting prediction of the wall pressure distribution particularly for shock/boundary-layer interactions problems.

2. COMPUTATIONAL RESULTS

In this study, we used the N3S-Natur code, version 1.4.5-DP, which is a general-purpose, fully 3D unstructured parallel compressible Navier-Stokes solver that is supported through a joint effort between SNECMA-Moteurs, Renault, EDF and SIMULOG Development Center [1]. Currently, the N3S-Natur code is being used by both researchers and industrial groups for many practical applications (nozzles, turbomachine, jet propulsion, combustion and many other aero- thermodynamics applications). The Roe upwind scheme with second-order accuracy is used to evaluate the explicit part of the governing equations. Several turbulence models and advanced numerical techniques are included in this code.

In this study, all solutions were developed using a cell vertex unstructured meshes with a maximum grid clustering in regions of high gradients (boundary layers and shocks). The grid was stretched in both axial and radial directions to enable matching the fine grid clustering near the body to the coarse grid in the regions far away from it. Finer grid clustering was also used in the axial direction to capture the birth and the early growth of the shear layer as well as the shock cells structure of the jet. Approximately, six hundred thousand points are used in all the computational domain. The computations reported here were performed on a parallel computer (IBM P4 1600) using 10 processors. Turbulent computations are involved in this study using two turbulence models (the k-$\in$ model including a wall function and the k-ω SST-Menter model) to qualify the turbulent flow-field. An implicit solver was used to achieve steady state numerical solution.

The boundary conditions used for this study include inflow, outflow, free stream, solid walls and geometrical symmetry. Reservoir conditions (total temperature and total pressure) are fixed as inflow conditions. Two outflow boundary conditions are needed: constant pressure for subsonic flows are extrapolated for supersonic flows. On solid walls, no-slip boundary conditions are specified and axisymmetric conditions (for the first two cases) are imposed on the centerline of the jet.

2.1 The ONERA TIC nozzle

The first case considered in this paper is an overexpanded subscale rocket-nozzle having a Truncated Ideal Contour (TIC) studied experimentally at ONERA Chalais Meudon Center [2]. This nozzle has no internal shock and produces a nearly uniform flow at the exit. For highly overexpanded regime, when the exit pressure is much lower than the ambient pressure, a separated flow occurs inside the nozzle and continues as a free jet. An open recirculation zone exists which matches the ambient free-stream to the upstream supersonic jet. The numerical simulation reveals the existence of a small recirculation bubble trapped between the wall (at the nozzle lip) and the main recirculation nozzle. This small vortex may play an important role in the amplification of the flow oscillations especially at the end effect regime. The central part of the jet is characterized by a strong shock waves interaction leading to a Mach disk configuration with a triple point configuration and a slip line discontinuity.

A typical flow separation is shown in Figure 2 for a pressure ratio p_c/p_a=40.

The corresponding wall pressure distribution is shown in Figure 3 where the computational results are compared with the experimental data. Good agreement is obtained with the SST model.

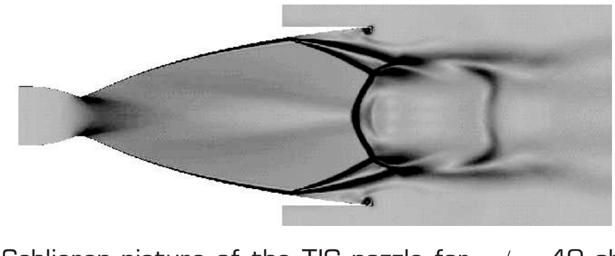

Figure 2. Numerical Schlieren picture of the TIC nozzle for p_c/p_a=40 showing a free shock separation pattern.

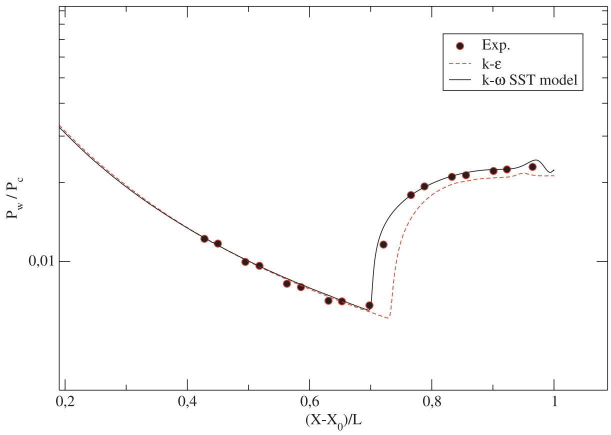

Figure 3. Normalized wall pressure distribution.

2.2 The axisymmetric boattailed RTO nozzle

The second test-case concerns the S8Ch single-flux axisymmetric boattailed nozzle [4, 5], for which detailed measurements are available. The square sectional length is 120 mm. The uniform supersonic external flow of Mach number near 2 is obtained via a plane cross-section nozzle. The model tested reproduces a generic shape of a missile afterbody, equipped with a truncated conical boat-tail of 25° and 34.5 mm length. The external diameter is D = 30 mm. The thickness of the incident external boundary-layer, measured by laser velocimetry, is 6 mm, with an external free stream speed U_∞ of 507 $m.s^{-1}$, which corresponds to a Mach number M1 of 1.94. The model is equipped with a conical nozzle (exit angle: 10°) and feeded with compressed air. The Mach number M_j at the outlet of the nozzle on the axis of symmetry is 1.75, for an outlet diameter of 14.9 mm. The numerical Schlieren picture (see Figure 4) shows the major aerodynamics phenomena acting in the wind-tunnel. We first notice the appearance of a λ shape shock in the region of the flow separation on the boat-tail. This shock wave reflects on the wind-tunnel wall and interacts with the jet mixing layer. The barrel shock, appearing at the nozzle-tail, interacts on the centerline of the jet and forms a Mach reflection, with a visible Mach disk located at approximately $X/D \approx 2{:}5$. The turbulent mixing layer, which forms the isobar line of the under-expanded jet, grows progressively until a certain distance where the containment effects of the wind-tunnel seem to be important. At the outlet of the nozzle, the flow in the potential core of the jet is accelerated to a Mach number of approximately 6:35.

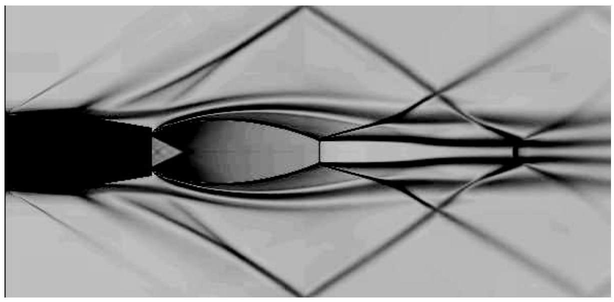

Figure 4. Numerical Schlieren picture of the RTO afterbody flow-field.

Figure 5 shows the streamlines that bound the separation region just outside the nozzle. Good qualitative agreement between computational results (using the SST-Menter model) and experimental data is obtained. Additional comparison between computation and experiment shows that the SST-Menter model is able to predict the major features of the flow-field (separation point location (see Figure 6), structure of the jet). Particularly, the shock/boundary-layer interaction region which is a crucial issue for wall loads is well predicted by the k-ω SST model, whereas the k-$\in$ model fails to reproduce this flow accurately.

Experiment

Computation

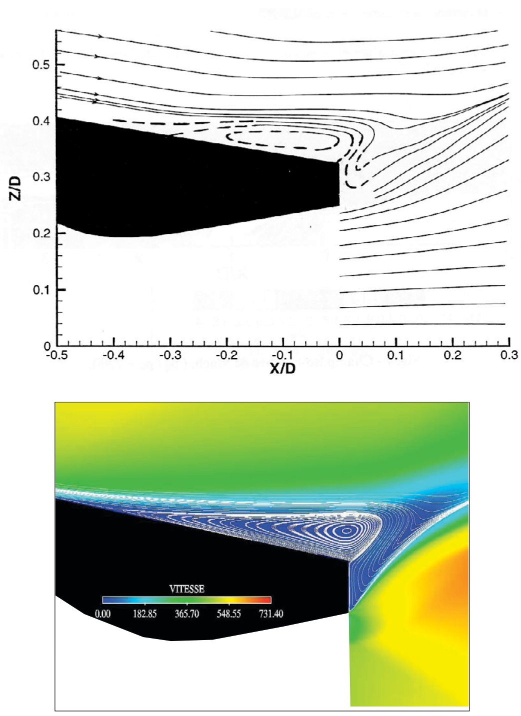

Figure 5. Streamlines patterns showing the boundary layer separation at the base of the body.

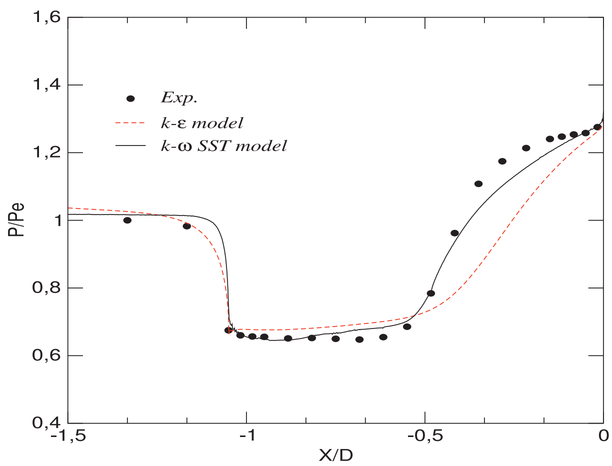

Figure 6. Normalized afterbody surface pressure.

2.3 The 3D AGARD nozzle

The third test-case is the 3D single-flux axisymmetric boat-tailed nozzle proposed by AGARD working group for CFD validations.

The nozzle is divided into two parts: (1) an internal part which features a circular feed/plenum chamber, transitioning to a throat with a rectangular cross section, and a two-dimensional supersonic expansion passage, (2) an external skin with a square inlet section. The two parts join together at a rectangular outlet section of the nozzle (see Figure 7).

The challenge from the computational point of view is the accurate resolution of the complex shock structure, resulting in the need for highly accurate discretisation techniques and fine meshes to capture the multiple shock reflections within the jet and the nozzle boundary layer separation.

As a first step towards accurate solution with reliable turbulence model, we have used a coarse grid (with approximately 180,000 points mostly concentrated near the afterbody wall to solve the boundary layer) and a simplified version of the k-$\in$ using wall functions to describe the inner region of the

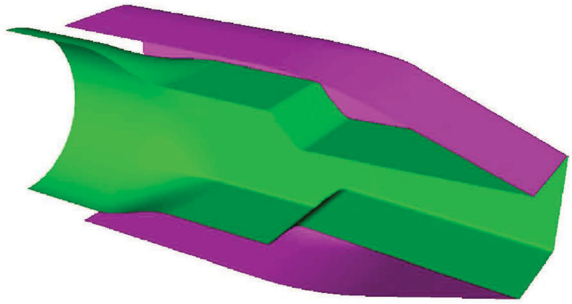

Figure 7. 3D view of the nozzle geometry.

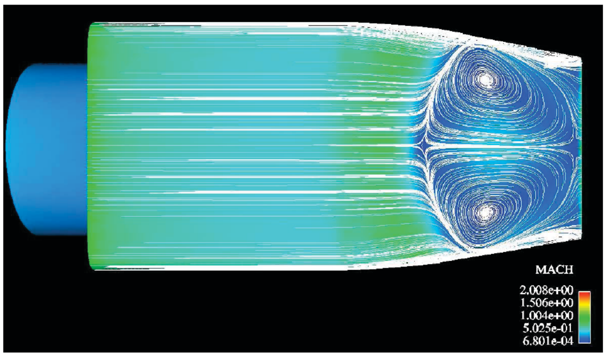

Figure 8. 3D view of the flow-field issue from the nozzle. Surface streamlines with two recirculation zones.

boundary layer. Due to the double symmetric (both left/right and top/bottom) of the geometry, only the quarter of the domain is computed. A graphical illustration of the wall streamlines on the body surface are presented in Figure 8. The computation shows the separation region downstream of the standing shock characterized by two counterrotative recirculation bubbles. A Schlieren visualization on the two symmetry planes of the computational domain are presented in Figure 9. Most flow features (shocks, mixing layer and supersonic jet plume) are clearly visible. Figure 10 shows the 3D organization of the outer flow in term of vorticity surface where two main separation surfaces exit. These separations roll up to constitute two horseshoe vortices escaping downstream. In addition, the simulation reveals the existence of various large scale motions whose properties play an essential role when important choices need to be made to adjust the design of these aeronautical systems.

3. CONCLUSION

In this study, we presented the computational results of a plume-induced flow separation on an axisymmetric boattailed afterbody as well as a 3D transonic nozzle. Two turbulence models were investigated: k-ω SST-Menter and k-$\in$ with a wall function. The obtained results show that the boundary-layer separation in the nozzle as well as on the boat-tail is correctly predicted with the SST-Menter model. In particular, this model gives the nearest detachment position to the experiment. The wall static pressure on the afterbody is in good agreement with the experiment. The mean velocity field and turbulence quantities in the jet flow are analyzed and compared to the measurements. Mean velocities and turbulent kinetic energy profiles near the outlet of the nozzle are in good agreement with experimental data (results not shown). Currently, a new version of the SST-Menter is under development. This investigation is motivated by the desire to increase the robustness of the model and to make it less Δy_1^+ dependent (where Δy_1^+ is a smallest normalized distance to the wall). This improvement is very attractive for industrial applications where the grid is not necessary cartesian near walls.

ACKNOWLEDGMENTS

This research is supported by the French aeronautical and aerospace industry (SNECMA-Moteurs). Computational facilities were provided by CRIHAN (Centre de Ressources Informatiques de HAute Normandie, Rouen).

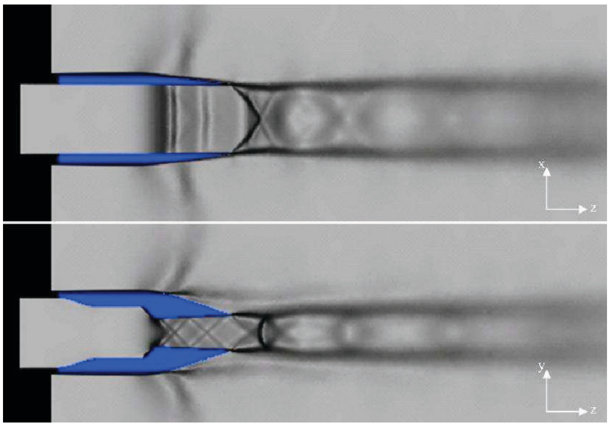

Figure 9. 3D view of the flow-field issue from the nozzle. Numerical Schlieren (top: horizontal symmetry, bottom: vertical symmetry).

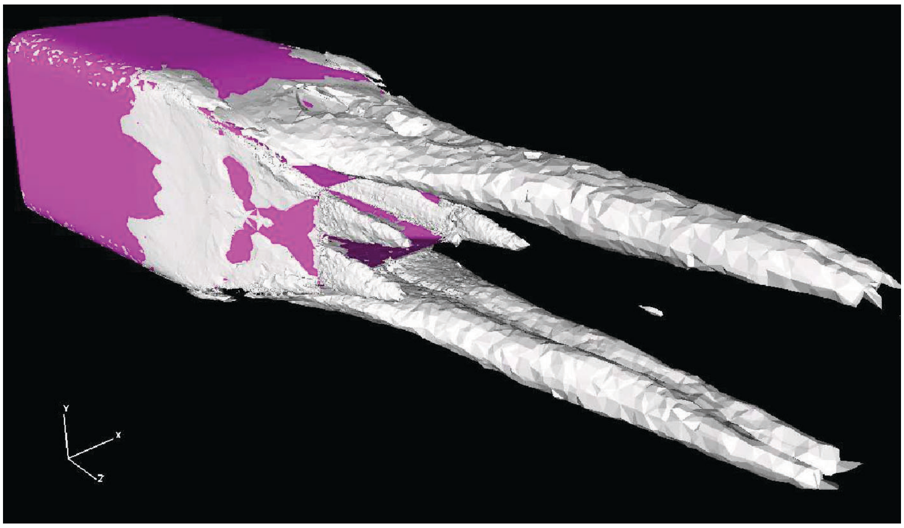

Figure 10. Iso-vorticity field.

REFERENCES

[1] SIMULOG, N3S-Natur 1.4.5-DP, logiciel 3D Navier-Stokes, 2003.

[2] Ph. Reijasse, L. Morzenski, D. Blacodon, J. Birkemeyer, Flow separation experimental analysis in overexpanded subscale rocket-nozzles. *37th AIAA/ASME/SEA/ASEE Joint Propuslion Conference and Exhibit*, AIAA-Paper 2001-3556, 2001.

[3] A. Gross, C. Weiland, *Investigation of shock patterns and separation behavior of several subscale nozzles*. AIAA-Paper 2000-3293, 2000.

[4] Ph. Reijasse, B. Corbel, *Décollement de l'écoulement externe induit par l'éclatement du jet propulsif sur un rétreint d'arrière-corps de missile*. 34ème Colloque d'Aérodynamique Appliquée de l'AAAF, Marseille, France, 1998.

[5] P. Servel, Ph. Reijasse, R. Benay, B. Corbel, *Etudes fondamentales sur les aspects aérodynamiques et thermiques des écoulements à l'arrière-corps des missiles*. ONERA, Technical Note, 1998.

[6] AGARD. Working Group on Aerodynamics of 3-D Aircraft Afterbodies. AGARD Advisory Report, No. AR-318, 1995.

[7] S. Moreau, T. Mauffret, *Numerical simulations of afterbody flowfileds with two-equation turbulence models*. AIAA 96-0570, 1996.