

Large-Eddy Simulation of Supersonic Ejectors

Using a Newly Developed Wall Model

Doctoral dissertation presented by

Romain DEBROEYER

in partial fulfillment of the requirements for
the degree of Doctor in Engineering Sciences

January 2025

Thesis committee

Prof. Yann BARTOSIEWICZ (co-supervisor, UCLouvain)

Prof. Grégoire WINCKELMANS (co-supervisor, UCLouvain)

Prof. Hadrien RATTEZ (chair, UCLouvain)

Dr. Matthieu DUPONCHEEL (secretary, UCLouvain)

Prof. Stefan HICKEL (TU Delft)

Dr. Koen HILLEWAERT (ULiège)

Remerciements

Tout d’abord, j’aimerais exprimer ma profonde reconnaissance à mes deux promoteurs, Yann Bartosiewicz et Grégoire Winckelmans, pour leur soutien et leurs conseils tout au long de ce travail. Merci à Grégoire pour les longues heures passées à revoir mes papiers et à discuter des détails des belles équations utilisées dans cette thèse. Merci à Yann pour toutes les discussions sur la physique des éjecteurs, les subtilités du “choking” et des flux d’exergie.

J’aimerais également remercier les membres de mon comité d’encadrement, Laurent Bricteux, Matthieu Duponcheel et Koen Hillewaert pour leurs réponses à mes questions pendant ces années. Merci également aux membres de mon jury de thèse pour leur lecture de mon manuscrit et la discussion constructive qui s’en est suivie pendant ma défense privée.

Ensuite, un grand merci aux membres du TFL pour la superbe ambiance de travail qui m’a permis de toujours garder la motivation à venir au bureau, même quand les travaux de recherche n’avançaient plus. Un merci tout particulier à mes collègues assistants, présents et passés, Billu, Gauthier, Anouk, Joauma, PB, Gilles, Xavier et Youri qui m’ont également soutenu pendant les longues heures à imprimer, scanner, surveiller et corriger examens et interros.

Merci à Guillaume, Nicolas, Loric et Quentin pour l’excellent travail sur le design du nouvel éjecteur cylindrique et la campagne de tests expérimentaux et numériques. Vos résultats ont permis de grandement alimenter les discussions et analyses concernant la physique des éjecteurs.

J’aimerais également remercier les personnes qui ont contribué d’un peu plus loin à cette thèse mais sans qui les résultats présentés n’auraient pas pu exister: Thomas

et Michel de Cenaero pour leur aide précieuse dans l'appropriation d'ArgoDG et de GMSH; ainsi que Franck, Victor, Julien et Arnaud du CREDEM et aussi Grégoire et Pascal d'Ateliers François pour l'aide dans la partie expérimentale de cette thèse.

Merci au CECI/CISM et à Cenaero pour la mise à disposition des ressources de calcul, indispensables à l'obtention des nombreux résultats numériques présentés dans cette thèse, ainsi que pour le support technique.

Entreprendre l'aventure d'un doctorat a inévitablement un impact sur la vie dans son ensemble et je n'y serais pas arrivé sans le soutien de ma famille et de mes amis.

Finalement, et sans doute le plus important, merci Camille de m'avoir soutenu, supporté et encouragé pendant ces 5 dernières années (et bien plus avant ça). Une thèse est synonyme de hauts et de bas pour celui qui l'entreprend et cela se répercute inévitablement sur la personne qui partage sa vie. Je te remercie infiniment de m'avoir redonné la motivation dans les moments difficiles, d'avoir été compréhensive lors de mes changements d'humeur fréquents au gré des passages positifs et négatifs de ma thèse, de m'avoir toujours poussé à aller au bout de ce dont j'étais capable. Il est évident que je n'y serais pas arrivé sans toi.

À mon papa...

Contents

1	Introduction	1
1.1	Context	1
1.2	Research objectives	5
1.3	Structure of the manuscript	7
2	Models and methods	9
2.1	Conservation equations	9
2.2	Fluid properties	11
2.3	Numerical methods	11
2.3.1	Large-Eddy Simulations	11
2.3.2	Reynolds and Favre averaging	15
2.3.3	Reynolds-Averaged Navier Stokes simulations	16
3	A wall model for LES of compressible flows based on a new scaling of the law of the wall	19
3.1	Introduction	19
3.2	Wall-resolved simulations of compressible channel flows at $Re_\tau \simeq 1000$	22
3.2.1	Simulation setup	22
3.2.2	Results	23
3.3	Wall modeling strategy	27
3.3.1	Baseline wall model for incompressible flows	28
3.3.2	Scalings for compressible flows	29
3.3.3	A priori error estimation	35
3.4	Wall-modeled simulations of compressible channel flows at $Re_\tau \simeq 1000$	36

3.4.1	Simulation setup	37
3.4.2	Results	40
3.5	Wall-modeled simulations of compressible channel flows at $Re_\tau \simeq 5200$	51
3.5.1	Simulation setup	51
3.5.2	Results	52
3.6	Conclusions and perspectives	56
4	Large-Eddy Simulation of a planar ejector - Total exergy flux analysis	59
4.1	Introduction	59
4.2	Simulation setup	60
4.2.1	Geometry	60
4.2.2	Boundary conditions	60
4.2.3	Mesh	62
4.2.4	Extrusion depth	63
4.3	Results	64
4.3.1	Instantaneous fields	68
4.3.2	Averaged fields	70
4.3.3	Flux of total exergy through the dividing streamlines	79
4.4	Conclusions and perspectives	91
5	Design and characterization of a new cylindrical supersonic ejector	95
5.1	Introduction	95
5.2	Design and numerical setup	96
5.2.1	Geometry	96
5.2.2	Experimental setup	97
5.2.3	RANS simulations setup	100
5.3	Experimental and RANS simulation results	102
5.4	Large-eddy simulation at a critical point	106
5.4.1	Numerical setup	107
5.4.2	Instantaneous results	108
5.4.3	Averaged results	109
5.5	Conclusions and perspectives	120
6	Impact on the ejector performance of the addition of chevrons to the primary nozzle	123
6.1	Introduction	123
6.2	Geometry and numerical setup	125
6.2.1	Chevron geometry	125

6.2.2	Mesh	126
6.2.3	Boundary conditions	126
6.3	Simulation results	127
6.3.1	Instantaneous results	128
6.3.2	Averaged fields	130
6.3.3	One-dimensional analysis	134
6.4	Conclusions and perspectives	143
Conclusions and perspectives		147
Bibliography		153
A Total exergy fluxes in the RANS framework		163
B Derivation of the compound Mach number for a compound flow with general transfers between streams		167

Introduction

1.1 Context

Supersonic ejectors have become an increasingly widespread replacement solution for compressors in industrial applications. They constitute a promising solution to replace classical compressors in refrigeration cycles, allowing the use of low-grade energy sources [7]. Moreover, as they perform the entrainment and compression of a flow passively, i.e. with no moving parts, they are cheaper to manufacture and easier to maintain than a compressor. In transcritical CO₂ refrigeration cycles, two-phase supersonic ejectors are introduced to recover part of the expansion losses at the isenthalpic valve and thus reduce the exergetic losses [62]. A solar air-conditioning system using a supersonic ejector was also proposed [39] and is represented in Figure 1.1. In ultra high bypass ratio engines, their use is studied to improve the efficiency of the engine bleed systems [85]. These devices are also used in propulsion applications to increase the thrust [74].

To achieve the compression and entrainment of a fluid, supersonic ejectors use a high total pressure primary flow, accelerated in a converging-diverging nozzle. This produces a supersonic jet, which entrains the secondary flow at a lower total pressure. Both flows exchange momentum and energy in the mixing chamber. The mixed flow is then brought to an intermediate total pressure in the diffuser, thus achieving the compression of the secondary flow. The schematic of a typical cylindrical ejector is shown in Figure 1.2, with its principal geometric characteristics. Note that in the case of a rectangular ejector, the radii R_* are replaced by the half-heights h_* and the height of the channel is noted $H_* = 2h_*$. The subscript p (respectively s) is used to designate quantities relative to the primary (respectively secondary) flow. The index t is used for

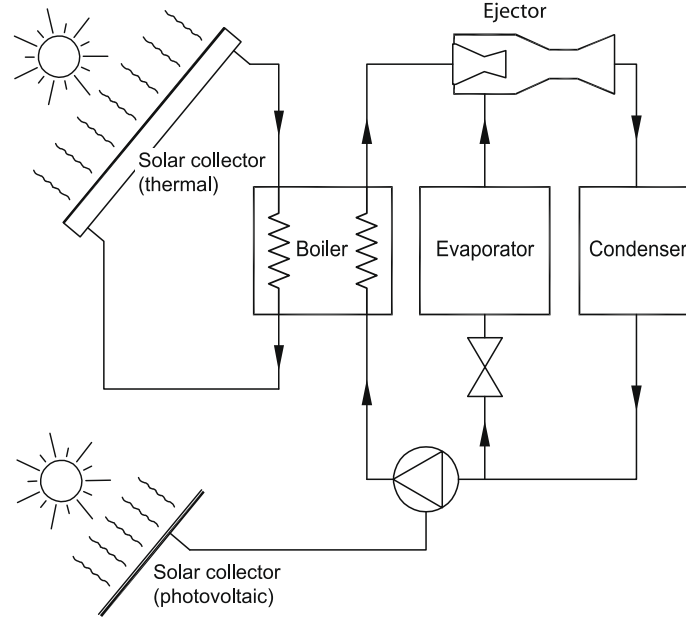


Figure 1.1: A solar air-conditioning system using a supersonic ejector. Reproduced from Hemidi et al. [39].

the primary nozzle throat. The nozzle exit position (NXP) is the distance between the exit of the primary nozzle, noted with a subscript e , and the beginning of the constant area mixing duct, labeled with the subscript m .

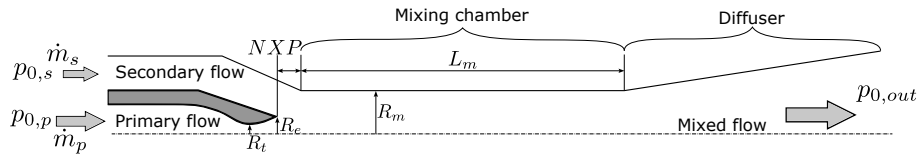


Figure 1.2: Schematic of a typical supersonic ejector.

The operating conditions of a supersonic ejector are represented using its characteristic curve, showing the evolution of the entrainment ratio $ER = \dot{m}_s / \dot{m}_p$ as a function of the compression ratio, i.e. the ratio between the mixed flow total pressure and the secondary flow total pressure, and which exhibits the behavior shown in Figure 1.3.

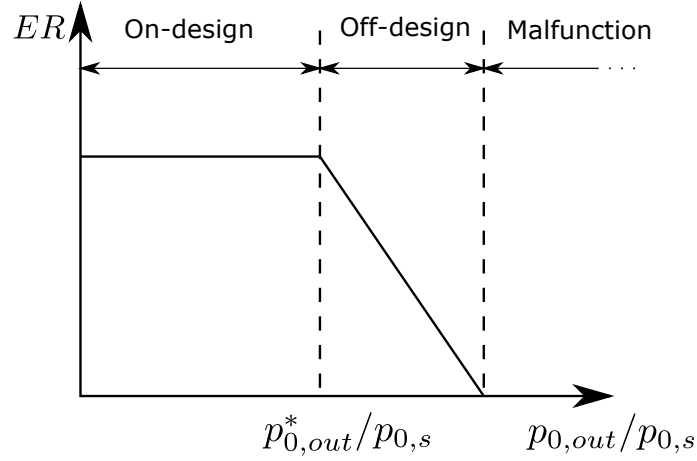


Figure 1.3: Schematic of a typical characteristic curve for a supersonic ejector.

Three main regimes can be identified:

1. The on-design, or double-choked regime where the entrainment ratio is maximum and constant. This regime appears for low compression ratios and extends up to the critical back-pressure $p_{0,out}^*$. In this regime, both flows are said to be choked.
2. From the critical pressure and up to the breakdown pressure, the ejector is said to be in off-design or single-choked regime. In this regime, the entrainment ratio decreases almost linearly until it reaches zero. This corresponds to an unchoked secondary flow situation.
3. The ejector then goes into malfunction or back-flow regime and part of the primary flow exits through the secondary port.

Such a characteristic curve exists for each set of primary and secondary total conditions (pressure and/or temperature).

For most supersonic ejector applications, the objective is to maximize the on-design entrainment ratio and the value of the critical pressure. To do so, several approaches can be employed, depending on the design phase, desired accuracy, and/or available resources.

1. First, thermodynamical or “quasi-1D” models can be used. Such models rely on conservation laws and simplified choking mechanisms to predict the entrainment ratio for a given back pressure, using the main geometric parameters of the ejector, typically, the area ratios. The first model was introduced by Keenan et al. [51] in 1950 and was based on mass, momentum and energy conservation. However, this model did not include any model for the choking of the secondary flow. Fabri and Siestrunck [30] later proposed a model where the on-design regime was achieved by considering that the primary and secondary flows remain independent in the mixing duct, up to a particular location where the expansion of the primary flow would force the secondary flow to reach sonic conditions, hence causing its choking. They however considered the critical pressure to be a parameter rather than a result. The model introduced by Munday and Bagster [64] then also predicted the critical pressure. Many later thermodynamical models used this explanation of the choking of the secondary flow, called the “Fabri-choking”, as a base to compute the entrainment ratio, adding irreversibilities, real gas behavior or the ability to predict the off-design operation mode (see Eames et al. [29], Huang et al. [42], Rogdakis and Alexis [72], Chen et al. [15]).

More recently, Lamberts et al. [57] proposed a new explanation for the limitation of the mass flow-rate in a supersonic ejector. The new model relied on the compound choking theory of Bernstein et al. [6] and was shown to predict more accurately the on-design entrainment ratio. Moreover, it was observed to be more consistent with the observation made in Reynolds Averaged Navier Stokes (RANS) simulations. Metsue et al. [61] adapted this model to real gases and further improved it with the addition of efficiencies to take irreversibilities into account. The constant mixing pressure constraint was also lifted, allowing to predict the whole characteristic curve, even its off-design portion. They also proved that the compound-choking was equivalent to maximizing the mass flow-rate, which constitutes the primary physical definition of a choked flow.

These models usually only require the main area ratios of the ejector, i.e. A_m/A_t and A_e/A_t , to produce a result. However, the longitudinal dimensions also play a role in the overall performance of ejectors and García del Valle et al. [33] proposed a one-dimensional model to take the variations of pressure and area of the primary supersonic jet into account. Van Den Berghe et al. [83] also used a one-dimensional two-stream approach and added the shear between streams and the wall friction effects in their model. This model was then tuned

using a machine-learning algorithm, thus needing data from simulations and experiments. The compound-choking theory was also extended to incorporate and study the separate effects of the shear between the streams and the wall-friction in Van den Berghe et al. [82].

2. Then, other geometrical parameters, such as the mixing duct length, the secondary flow angle, diffuser angle, etc., can be optimized using RANS simulations and the complete characteristic curve can be studied in details. This approach allows to perform a large number of parametric studies and it can also provide much insight into the flow features and physical phenomena at play in an ejector. As such, Lamberts et al. [55] used a database of RANS simulations to analyze the transfers of total exergy in a supersonic ejector. These simulations are also used to train machine learning algorithms [83], allowing to obtain better simplified models. However, these simulations do not capture the turbulent phenomena responsible for the mixing in ejectors; these phenomena are only modeled through the use of some turbulence closure models.
3. Finally, to properly capture the turbulent phenomena occurring in supersonic ejectors, scale-resolving simulations must be performed. Large-Eddy Simulations (LES) were performed by Bouhanguel et al. [10] to study the flow instabilities in ejectors. This study was however only performed to prove the superiority of the LES over RANS to properly capture the flow instabilities observed experimentally and did not dig deeper into the analysis of the flow. More recently, Croquer et al. [19] performed the LES of the supersonic ejector geometry designed by Lamberts [54]. Their results showed a good match with experimental and RANS results regarding the main flow features, such as the primary jet shock structures. Their results however mainly focused on the global diagnostics, such as the entrainment ratio and choking analysis. They did not analyze the mixing phenomena at play in the ejector.

1.2 Research objectives

As shown above, there is a very limited number of high-fidelity scale-resolving simulations of supersonic ejectors performed in the literature, despite their promising potential to fully understand the physics at play in such devices. The main objective of this thesis is thus to answer the following questions :

- What are the tools necessary to perform high-fidelity LES of supersonic ejectors and how can the post-processing tools used to analyze RANS simulation results be transposed to this higher accuracy framework?
- How can LES results complement RANS simulation results to improve our understanding of supersonic ejector physics?

To answer those questions, several steps will be taken. First, as the Reynolds numbers involved in supersonic ejectors, and thus the number of grid-point required to perform wall-resolved simulations are far too high to be affordable, wall modeling is unavoidable. In this regard, the accuracy of the available wall model, which proved to accurately predict the wall shear stress for incompressible flows, must be evaluated for flows where compressibility effects are significant. If necessary, this model should then be improved to reach the desired accuracy for the wall friction prediction of supersonic flows with adiabatic walls, up to a Mach number of around 2.

Then, the post-processing tools using total exergy fluxes developed by Lamberts et al. [55] for RANS simulations should be adapted to the LES framework. This involves the proper averaging of the evolution equation of the total exergy to obtain the exact total exergy fluxes in the simulation.

Finally, the LES of the rectangular ejector designed and built by Lamberts [54] must be performed and compared to the significant amounts of data obtained experimentally and numerically through RANS simulations. This should highlight the limitations of the RANS simulation to evaluate the transfers between streams as the LES allows to compute the average transfer of total exergy, whereas the RANS only allows to compute the transfer of total exergy of the mean flow, thus missing higher-order fluctuation terms. Moreover, since the LES captures many of the turbulent scales, the turbulent fluctuating terms can be directly computed in this framework, whereas they are modeled in the RANS.

Additionally, a new cylindrical supersonic ejector will be designed and built, as this is the preferred geometry in industrial applications. The ejector design and experimental setup will build on previous experience and the used instrumentation will allow a thorough comparison with numerical results. The change to a cylindrical cross-section also has the added benefit of significantly reducing the three-dimensional effects due to the side-walls present in the rectangular cross-section experiment, and which are difficult and expensive to accurately reproduce in the simulations. Overall, this new supersonic ejector geometry should lead to a more accurate comparison

between experiment, RANS and LES results, thus allowing to focus on the physical phenomena responsible for the discrepancies between frameworks.

1.3 Structure of the manuscript

- **Chapter 2**

This chapter presents the equations used to describe the physics of the system as well as the characteristics of the flow solvers used to produce the results presented in all the subsequent chapters.

- **Chapter 3**

To perform a scale-resolving simulation of a full ejector, it is required to model the effect of the walls to circumvent the need to discretize the inner part of the boundary layers, which would significantly increase the computational cost. Chapter 3 presents the new wall model developed to accurately predict the wall-friction that must be imposed at the wall. This model is built on an existing algebraic wall model that was shown to be very accurate for incompressible flows but introduces significant deviations when used for highly compressible flows. This chapter is largely inspired by the following paper:

R. Debroeyer, M. Rasquin, T. Toulorge, Y. Bartosiewicz, G. Winckelmans, A wall model for large-eddy simulation of highly compressible flows based on a new scaling of the law of the wall, Journal of Fluid Mechanics 980 (2024) A9, doi: 10.1017/jfm.2023.1086

- **Chapter 4**

This chapter presents the results of the LES of the rectangular ejector designed and studied numerically through RANS simulations and experimentally by Lamberts [54]. The post-processing tools using exergy fluxes, introduced by Lamberts et al. [55] for RANS simulations are generalized for the framework of LES and used to analyze the mixing of the flows in the ejector. This chapter is largely inspired by the following paper:

R. Debroeyer, T. Toulorge, M. Rasquin, G. Winckelmans, Y. Bartosiewicz, Analysis of Entrainment and Mixing in a Supersonic Air Ejector Using Large-Eddy Simulation, International Journal of Heat and Fluid Flow doi: 10.2139/ssrn.4877056, [Under review]

- **Chapter 5**

Chapter 5 presents the characterization of the new cylindrical ejector designed and built in the laboratory. The results of an experimental campaign and complementary RANS simulations are presented and analyzed. The results of the LES of one critical point are thoroughly analyzed to gain additional insight into the flow phenomena, and to help explain the differences between experimental and RANS simulation results. This chapter is largely inspired by the following paper:

R. Debroeyer, G. Winckelmans, Y. Bartosiewicz, Numerical and experimental characterization of a cylindrical supersonic air ejector, International Journal of Heat and Fluid Flow doi: 10.2139/ssrn.4992378, [Under review]

- **Chapter 6**

This final chapter presents the results of the LES of the same cylindrical ejector as that studied in Chapter 5, with the addition of chevrons at the primary nozzle exit. This geometric modification is proposed to promote mixing at the very beginning of the mixing pipe. The impact of this modification on the performance of the ejector is studied using all the tools developed in the two previous chapters.

Models and methods

This chapter is dedicated to the presentation of the equations used to describe the physics of the system under investigation. Then, the numerical methods used to perform the simulations are briefly reminded.

2.1 Conservation equations

The fundamental conservation equations are those for mass, momentum, and total internal energy. These constitute the compressible Navier-Stokes equations which, written in conservative form and using Einstein's notation with $i, j = 1, 2, 3$, are:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) &= 0, \\ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) &= -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}, \\ \frac{\partial}{\partial t} (\rho e_0) + \frac{\partial}{\partial x_j} (\rho e_0 u_j) &= -\frac{\partial}{\partial x_j} (p u_j) + \frac{\partial}{\partial x_j} (u_i \tau_{ij}) - \frac{\partial q_j}{\partial x_j}, \end{aligned} \quad (2.1)$$

where $e_0 = e + k$ is the total internal energy, with e the internal energy and $k = (u_k u_k)/2$ the kinetic energy; and

$$\tau_{ij} = 2\mu \left(d_{ij} - \frac{1}{3} d_{kk} \delta_{ij} \right) + \mu_v d_{kk} \delta_{ij} = 2\mu d_{ij}^d + \mu_v d_{kk} \delta_{ij} \quad (2.2)$$

$$q_j = -\kappa \frac{\partial T}{\partial x_j} \quad (2.3)$$

are the shear stress tensor and the heat flux density vector, respectively, with μ the shear viscosity, μ_v the bulk viscosity, κ the thermal conductivity, $d_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$

the strain rate tensor (and thus $d_{kk} = \frac{\partial u_k}{\partial x_k}$ is the divergence of the velocity field) and d_{ij}^d its deviatoric part; δ_{ij} being the Kronecker tensor. We will neglect the bulk viscosity μ_v in the present study.

Combining those equations allows to obtain conservation equations that will be useful for the total exergy analysis performed in Section 4.3.3 . Those are the conservation equations for :

- (1) the enthalpy $h = e + \frac{p}{\rho}$:

$$\frac{\partial}{\partial t} (\rho h) + \frac{\partial}{\partial x_j} (\rho h u_j) = \frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} + \tau_{ij} d_{ij} - \frac{\partial q_j}{\partial x_j}, \quad (2.4)$$

$$\frac{\partial}{\partial t} (\rho h) - \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (\rho h u_j + q_j - p u_j) = -p \frac{\partial u_j}{\partial x_j} + \tau_{ij} d_{ij}; \quad (2.5)$$

- (2) the kinetic energy:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = -u_i \frac{\partial p}{\partial x_i} + u_i \frac{\partial \tau_{ij}}{\partial x_j}, \quad (2.6)$$

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j - u_i \tau_{ij} + p u_j) = p \frac{\partial u_j}{\partial x_j} - \tau_{ij} d_{ij}; \quad (2.7)$$

- (3) the entropy (with $T ds = dh - \frac{dp}{\rho}$):

$$\frac{\partial}{\partial t} (\rho s) + \frac{\partial}{\partial x_j} (\rho s u_j) = \frac{\tau_{ij} d_{ij}}{T} - \frac{1}{T} \frac{\partial q_j}{\partial x_j}, \quad (2.8)$$

$$l \frac{\partial}{\partial t} (\rho s) + \frac{\partial}{\partial x_j} \left(\rho s u_j + \frac{q_j}{T} \right) = \underbrace{\frac{2\mu}{T} d_{ij}^d d_{ij}^d}_{>0} + \underbrace{\frac{\kappa}{T^2} \frac{\partial T}{\partial x_j} \frac{\partial T}{\partial x_j}}_{>0}. \quad (2.9)$$

For each of those quantities, the second form contains the time derivative and the flux term associated to this quantity on the left hand side, whereas the source or sink terms are grouped on the right hand side. The conservation equation for the total enthalpy, $h_0 = h + k = e_0 + \frac{p}{\rho}$, is also obtained from summing the first two equations:

$$\frac{\partial}{\partial t} (\rho h_0) + \frac{\partial}{\partial x_j} (\rho h_0 u_j) = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (u_i \tau_{ij}) - \frac{\partial q_j}{\partial x_j}, \quad (2.10)$$

$$\frac{\partial}{\partial t} (\rho h_0) - \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (\rho h_0 u_j + q_j - u_i \tau_{ij}) = 0. \quad (2.11)$$

2.2 Fluid properties

For all simulations, dry air will be considered, and it will be treated as a perfect gas with $R = c_p - c_v = 287.1 \text{ J/(kg K)}$ and $\gamma = c_p/c_v = 1.4$. We then also have that $de = c_v dT$ and $dh = c_p dT$. The Sutherland law is used for the dynamic shear viscosity:

$$\frac{\mu}{\mu_{ref}} = \left(\frac{T}{T_{ref}} \right)^{3/2} \left(\frac{T_{ref} + S}{T + S} \right), \quad (2.12)$$

with $\mu_{ref} = 1.716 \times 10^{-5} \text{ Pa s}$ at $T_{ref} = 273.15 \text{ K}$ and $S = 110.4 \text{ K}$. In the range of temperatures encountered in the present ejector application, i.e. between $T \simeq 140 \text{ K}$ and $T \simeq 320 \text{ K}$, the dynamic viscosity will thus vary from $\mu = 9.64 \times 10^{-6} \text{ Pa s}$ to $\mu = 1.94 \times 10^{-5} \text{ Pa s}$, showing the importance of taking the temperature dependence of the dynamic viscosity into account.

The specific heat at constant pressure, c_p , is considered constant. The thermal conductivity $\kappa(T)$ is determined from the dynamic viscosity $\mu(T)$ and the Prandtl number $Pr = \frac{\mu c_p}{\kappa}$ taken as constant and equal to $Pr = 0.71$. The obtained thermal conductivity values then ranges between $\kappa(T = 140 \text{ K}) = 13.6 \text{ mW/(m K)}$ to $\kappa(T = 320 \text{ K}) = 27.4 \text{ mW/(m K)}$, presenting an error of at most 3 % when compared to the most accurate model for thermophysical properties to date and as provided by Bell et al. [5].

2.3 Numerical methods

2.3.1 Large-Eddy Simulations

The high fidelity scale-resolving simulations considered in this work are performed using the ArgoDG code developed by Cenaero [40], which uses a Discontinuous Galerkin (DG) method to discretize the compressible Navier-Stokes equations (2.1). We here cannot resolve all the physical scales, down to the Kolmogorov scale; yet we use grids that are quite fine, and hence we resolve many of the flow scales. The ArgoDG code is then used here in Large-Eddy Simulation (LES) mode.

The DG method is a specific type of Finite Element (FE) methods where the continuity of the solution is not imposed at the interface between elements. As in classical FE methods, the exact solution $\tilde{w}_m$ is approximated in each element e by $w_{m,e}$, which is obtained as the linear combination of nodal values $W_{m,i,e}$ and the shape

functions $\phi_{i,e}$ which form a base of the test space Φ_e of each element :

$$w_{m,e} = \sum_{i=0}^{N_\phi(p)} W_{m,i,e} \phi_{i,e}, \quad (2.13)$$

where $N_\phi(p)$ represents the number of degrees of freedom at interpolation order p . Since no continuity of the solution is imposed between the elements, the solution over the whole computational domain is the sum of the local solutions for all elements

$$w_m = \sum_e \sum_{i=0}^{N_{\phi,e}(p)} W_{m,i,e} \phi_{i,e}. \quad (2.14)$$

The discretized Navier-Stokes equations can be written in a general form as

$$\frac{\partial w_m}{\partial t} + \frac{\partial}{\partial x_k} C_{m,k} + \frac{\partial}{\partial x_k} D_{m,k} + S_m = 0, \quad (2.15)$$

where $C_{m,k}$, $D_{m,k}$ and S_m are the convective, diffusive and source terms respectively. Equation (2.15) is then solved by using the variational principle, thus finding the function w_m for which the residual of the model equation is orthogonal to each shape function $\phi_{i,e}$, as those form a basis of the interpolation space:

$$\begin{aligned} \sum_e \int_e \phi_{i,e} \frac{\partial w_m}{\partial t} dV + \sum_e \int_e \phi_{i,e} \frac{\partial}{\partial x_k} C_{m,k} dV + \sum_e \int_e \phi_{i,e} \frac{\partial}{\partial x_k} D_{m,k} dV \\ + \sum_e \int_e \phi_{i,e} S_m dV = 0. \end{aligned} \quad (2.16)$$

Integrating the convective and diffusive terms by parts and then applying the divergence theorem generates interface fluxes which do not cancel because of the absence of continuity between elements. To characterize those interface fluxes, the interface average $\{\cdot\}$ and jump $[[\cdot]]$ operators are introduced:

$$\{a\} = \frac{a^+ + a^-}{2} \quad [[a]] = a^- \mathbf{n}^- + a^+ \mathbf{n}^+, \quad (2.17)$$

where $+$ and $-$ indicate lower and upper values with respect to the face normal $\mathbf{n}$. The Jacobian of the diffusive flux with respect to the solution gradients is also defined as the fourth order tensor $D_{m,n}^{k,l}$ such that:

$$D_{m,k} \approx D_{m,n}^{k,l} \frac{\partial \tilde{w}_n}{\partial x_l}. \quad (2.18)$$

The weak formulation of Equation (2.16) then becomes, $\forall \phi_{i,e} \in \Phi_e, \forall m$:

$$\begin{aligned}
& \sum_e \int_e \phi_{i,e} \frac{\partial w_m}{\partial t} dV - \sum_e \int_e \frac{\partial \phi_{i,e}}{\partial x_k} (C_{m,k} + D_{m,k}) dV + \sum_e \int_e \phi_{i,e} S_m dV \\
& + \underbrace{\sum_I \oint_I [[\phi_{i,e}]]_k n_k \mathcal{H}_m(w^-, w^+, \mathbf{n}) dS}_{C_{interface}} + \underbrace{\sum_I \oint_I [[\phi_{i,e}]]_k \left\{ D_{m,n}^{k,l} \frac{\partial w_n}{\partial x_l} \right\} dS}_{D_{interface}} \\
& + \underbrace{\sum_I \oint_I [[w_n]]_k \left\{ D_{m,n}^{k,l} \frac{\partial \phi_{i,e}}{\partial x_l} \right\} dS}_{D_{symmetry}} + \underbrace{\sum_I \sigma \oint_I [[w_m]]_k [[\phi_{i,e}]]_k dS}_{D_{penalty}} = 0.
\end{aligned} \tag{2.19}$$

The three first terms of this equation are similar to other FE formulations and represent the temporal, convective and diffusive fluxes, and source term volume contributions. The terms $C_{interface}$ and $D_{interface}$ essentially correspond to jumps in surface fluxes. The modified Simple Low dissipation Advection Upstream Splitting Method, proposed by Kitamura and Shima [52], is used as an approximate Riemann solver $\mathcal{H}_m$ at interfaces between elements for the convective flux. The remaining terms, $D_{symmetry}$ and $D_{penalty}$, are defined according to the Symmetric Interior Penalty (SIP) method [1], with optimal values for the penalty parameter σ , as obtained by Hillewaert [40] to guarantee stability.

The time integration is implicit and second order, using the 3-points backward difference formula. The non-linear system is solved using a Newton/Generalized minimal residual method preconditioned with block-Jacobi. A reduction in residual of 10^{-4} compared to the value of the residual at the beginning of each time step is imposed in order to obtain a good convergence. The time step is constant and such that the element-wise convective Courant-Friedrichs-Lewy (CFL) number stays close to unity at most.

The solver performs implicit LES meaning that no explicit subgrid scale model is used to account for the effect of the unresolved scales of the flow. The additional dissipation due to these scales is supplied directly by the numerical dissipation caused by the discretization. Figure 2.1 presents the dissipation properties of the DG method obtained by Gassner and Kopriva [34] as a function of the wavenumber $k^* = (\pi p)/h$, where h is the cell size and p is the interpolation polynomial degree. This figure shows that the dissipation errors of the DG method targets the high wavenumbers, which are also plagued with higher dispersion errors. On the other hand, the low and medium well-resolved wavenumbers are left intact. Moreover, this high dissipation

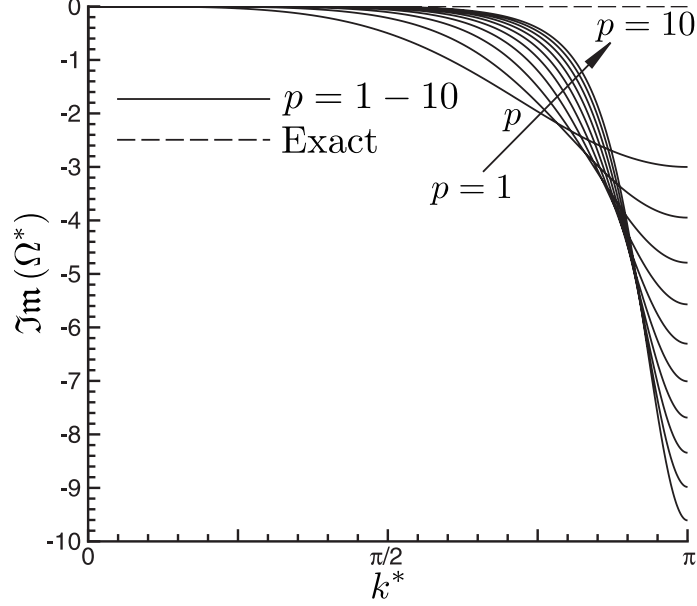


Figure 2.1: Dissipation properties of the DG method for polynomial degrees $p = 1$ to $p = 10$. Reproduced from Gassner and Kopriva [34].

region is stronger and targets higher wavenumbers as the polynomial order increases. These dissipation properties make this method an excellent candidate for implicit LES. The accuracy of this method was validated on benchmark cases such as homogeneous isotropic turbulence and channel flow cases [22]. The simulations presented in this work were carried out using a third-order ($p = 3$) polynomial finite element space over the whole domain, resulting in a fourth-order accurate scheme, which was found to offer a good compromise between accuracy and computational cost for the present application.

The shocks are detected using the shock sensor proposed by Persson and Peraire [66]. A smoothness indicator

$$S_e = \frac{(w_{m,e} - \hat{w}_{m,e}, w_{m,e} - \hat{w}_{m,e})}{(w_{m,e}, w_{m,e})} \quad (2.20)$$

is computed in each element e where the $(\cdot, \cdot)$ operator is the scalar product, $w_{m,e}$ being the computed solution as defined in Equation (2.13) and $\hat{w}_{m,e}$ is a truncated version of

this solution, only containing the terms up to order $p - 1$:

$$\hat{w}_{m,e} = \sum_{i=0}^{N_\phi(p-1)} W_{m,i,e} \phi_{i,e}. \quad (2.21)$$

The amount of viscosity added to stabilize the shocks, ϵ_e , is then taken as constant per element and is computed as

$$\epsilon_e = \begin{cases} 0 & \text{if } s_e < s_0 - \kappa, \\ \frac{\epsilon_0}{2} \left(1 + \sin \left(\frac{\pi(s_e - s_0)}{2\kappa} \right) \right) & \text{if } s_0 - \kappa \leq s_e \leq s_0 + \kappa, \\ \epsilon_0 & \text{if } s_e > s_0 + \kappa, \end{cases} \quad (2.22)$$

where $s_e = \log(S_e)$ and ϵ_0 , s_0 and κ are tuning parameters, chosen empirically. In this study, it was observed that the parameter controlling the width of the transition window κ should be set to 0.5. To properly stabilize the strong shocks occurring in the ejector simulations, the amplitude parameter was set to $\epsilon_0 = 0.35$, whereas the threshold parameter s_0 was found to lie between -3 and -3.5 .

The computed value of ϵ_e is finally scaled to have the proper dimensions for each of the conservation equations and applied as a Laplacian viscosity, i.e. the divergent of $\epsilon_e \nabla w_m$ is added to each equation of the system.

2.3.2 Reynolds and Favre averaging

To produce meaningful results, the flow statistics are averaged in time and in the homogeneous directions. This averaging is noted $(\bar{\cdot})$. With that, we define the Reynolds-averaged density $\bar{\rho}$ and pressure $\bar{p}$. The Favre averaging operator is also defined:

$$\tilde{\phi} = \frac{\overline{\rho \phi}}{\bar{\rho}}. \quad (2.23)$$

The fluctuations associated to the defined time and Favre averaging operators are defined as

$$\phi' = \phi - \bar{\phi} \quad \text{and} \quad \phi'' = \phi - \tilde{\phi}, \quad (2.24)$$

respectively. By definition, $\overline{\phi'} = 0$ and $\widetilde{\phi''} = 0$.

When performing the Reynolds-averaging of the Navier-Stokes equations, the average of the product of three or four quantities can arise. It is then useful to express

those terms as a combination of Favre-averaged quantities and fluctuations:

$$\overline{\rho A B} = \bar{\rho} \tilde{A} \tilde{B} + \overline{\rho A'' B''} \quad (2.25)$$

$$\overline{\rho A B C} = \bar{\rho} \tilde{A} \tilde{B} \tilde{C} + \overline{\rho A'' B'' C} + \overline{\rho A'' C'' B} + \overline{\rho B'' C'' A} + \overline{\rho A'' B'' C''} \quad (2.26)$$

2.3.3 Reynolds-Averaged Navier Stokes simulations

Reynolds-Averaged Navier Stokes (RANS) simulations are also performed to allow the rapid estimation of local and global quantities of the flow for many different cases. Those simulations are performed using the Ansys Fluent (2023 R1) simulation software and which solves the compressible RANS equations :

$$\frac{\partial}{\partial t} (\bar{\rho}) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j) = 0, \quad (2.27)$$

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_i \tilde{u}_j) = - \frac{\partial}{\partial x_i} \left(\bar{p} + \frac{2}{3} \bar{\rho} \tilde{k} \right) + \frac{\partial}{\partial x_j} \left(\hat{\tau}_{ij} + \hat{\tau}_{ij}^t \right), \quad (2.28)$$

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho} \tilde{e}_0) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{h}_0 \tilde{u}_j) = & - \frac{\partial}{\partial x_j} \left(\hat{q}_j + \hat{q}_j^t \right) + \frac{\partial}{\partial x_j} \left(\tilde{u}_i \left(\hat{\tau}_{ij} + \hat{\tau}_{ij}^t \right) \right) \\ & + \frac{\partial}{\partial x_j} \left(\overline{u_i'' \tau_{ij}} - \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} \right), \end{aligned} \quad (2.29)$$

where $\hat{\tau}_{ij}^t = \hat{\sigma}_{ij}^t + \frac{2}{3} \bar{\rho} \tilde{k} \delta_{ij}$ is the turbulent shear stress tensor, with $\hat{\sigma}_{ij}^t = -\overline{\rho u_i'' u_j''}$ the turbulent stress tensor and $\tilde{k} = \overline{u_k'' u_k''}$ the turbulent kinetic energy; and $\hat{q}_j^t = \overline{\rho c_p T'' u_j''}$ is the turbulent heat flux density. In these equations, the notations $\bar{\cdot}$ and $\tilde{\cdot}$ are the time and Favre averaging operators introduced previously and $\hat{\cdot}$ stands for mean variables computed from mean quantities. It is thus not stricto sensu a time average, especially for non-linear quantities, but it is a quantity based on the mean-flow variables. These are defined as

$$\hat{\tau}_{ij} = 2 \mu(\tilde{T}) \hat{d}_{ij}^d, \quad (2.30)$$

$$\hat{q}_j = -\kappa(\tilde{T}) \frac{\partial \tilde{T}}{\partial x_j}, \quad (2.31)$$

where $\hat{d}_{ij}^d$ is the deviatoric part of the strain rate tensor of the mean flow $\hat{d}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)$. The terms $\overline{u_i'' \tau_{ij}}$ and $\frac{1}{2} \overline{\rho u_k'' u_k'' u_j''}$ are neglected in the RANS solver.

The definitions of the turbulent stress tensor $\hat{\sigma}_{ij}^t$ and the turbulent heat flux density $\hat{q}_j^t$ imply exact time averaging and if these were computed directly, the $\bar{\cdot}$ notation

should be used. However, in this case, as their modeling uses definitions similar to their molecular counterparts, namely,

$$\hat{\tau}_{ij}^t = 2 \mu_t \hat{d}_{ij}^d, \quad (2.32)$$

$$\hat{q}_j^t = -\kappa_t \frac{\partial \tilde{T}}{\partial x_j}, \quad (2.33)$$

these quantities are here written using the $\hat{\cdot}$ notation. The turbulent viscosity μ_t is computed using the $k - \omega - SST$ turbulence model, with $\tilde{\omega} = \frac{1}{C_v} \frac{\tilde{\epsilon}}{k}$ with $C_v = 0.090$, $\tilde{\epsilon}$ the turbulent kinetic energy dissipation rate, and $\mu_t = \bar{\rho} \frac{k}{\tilde{\omega}}$. This model was shown by Bartosiewicz et al. [4] to produce the best results when compared to experimental data, with a constant turbulent Prandtl number $Pr_t^{RANS} = \frac{\mu_t c_p}{\kappa_t} = 0.85$.

A wall model for Large-Eddy Simulation of compressible flows based on a new scaling of the law of the wall

This chapter is largely inspired by an article published in the Journal of Fluid Mechanics, entitled "A wall model for Large-Eddy Simulation of highly compressible flows based on a new scaling of the law of the wall" [24], by R. Debroeyer, M. Rasquin, T. Toulorge, Y. Bartosiewicz, and G. Winckelmans.

3.1 Introduction

Scale-resolving simulations of flows at industrial scales and high Reynolds numbers Re can currently not reasonably be performed using wall-resolved Large-Eddy Simulations (wrLES) because of their too high computational cost. According to Choi and Moin [16], the number of grid-points increases as $Re^{13/7}$ for such simulations. Moreover, it was shown by Piomelli and Balaras [67] that, for Reynolds numbers above $\mathcal{O}(10^6)$, 99 % of the grid points in the mesh are used to resolve the inner layer (= near-wall region) of the boundary layer which only represents around 10 % to 15 % of its thickness. The grid-point requirements can thus be drastically reduced by modeling the effect of this inner layer on the flow using a so-called wall model. When such a

model is used, the number of grid points scales linearly with Re , as also shown by Choi and Moin [16].

Wall models for large-eddy simulation (LES) rely on the relative decoupling between the inner part of the boundary layer and the rest of the flow. The flow is assumed to follow a universal profile in the near-wall region of the boundary layer whereas the rest of the flow is assumed to be properly captured by the LES. The LES thus still captures a significant part of the boundary layer and is driven by the problem geometry. Three main types of wall models can be distinguished:

- i Algebraic (or analytical) models where the wall shear stress is directly computed from the LES flow data using the law of the wall for the near-wall region. These models were the first used to perform wall-modeled LES (wmLES) by Deardorff [23] and later by Schumann [73]. Piomelli et al. [68] proposed an improvement of such model by using the velocity a short distance downstream to evaluate the local wall shear stress. In such framework, Reichardt's formula [71] is often used to approximate the law of the wall in a convenient way as it reproduces, in a single formula, the laminar viscous sublayer, the buffer region, and the fully turbulent outer region (with the log-law); hence allowing to maintain accuracy when the y^+ value of the used LES data falls below the log region.
- ii Two-layer models where a separate set of equations is solved within the near-wall part of the boundary layer. The equations solved there are the so-called boundary layer equations, as in Balaras et al. [3], Duprat et al. [28], Kawai and Larsson [50], Piomelli and Balaras [67], Park and Moin [65], and which all require an appropriate modeling of the turbulent viscosity profile. Frère et al. [32] studied the balance of the different terms constituting the boundary layer equations using wrLES data of the flow around an airfoil and observed that, for values of y^+ sufficiently large ($y^+ \geq 30$), the convective and pressure gradient terms essentially compensate each other. This observation allows to further simplify the boundary layer equations to only consider the diffusive terms, which can then be integrated on a 1D mesh.

Other authors [18, 90] use a specifically modified Subgrid-Scale model in the near-wall region (relying thus only on the LES solver but also requiring to discretize this near-wall region). In Davidson and Peng [20], the near-wall layer is modeled using a turbulent viscosity obtained from the $k - \omega$ Reynolds-Averaged Navier-Stokes (RANS) equations. Even though these approaches

solve the same set of equations in the whole domain, they can still be considered as two-layer modeling as the unresolved scales are modeled differently in the near-wall layer and the rest of the domain.

- iii Machine learning and in particular neural networks (NN) were also used by several authors, such as Zhou et al. [93] or Boxho et al. [11]. Such approaches could possibly allow the development of models for non-equilibrium flows; yet they require the training of NN, hence the need for reference data.

Developments were made to generalize the former two approaches to take compressibility effects into account. For instance, Bocquet et al. [9] used a two-layer model in which the compressible boundary layer equations were solved and compared it to a quasi-analytical model based on Reichardt's formula with the temperature, viscosity, and density measured at the wall. This was however proposed for an isothermal wall and the Kader law [47] was used to provide the wall heat flux. Even though the proposed quasi-analytical model and the compressible boundary layer model showed similar performance for low Reynolds number supersonic channel flow test cases, the error on the velocity profile was significant.

A similar quasi-analytical wall model was used by Catchirayer et al. [13] for an isothermal wall. They also proposed a model to compute the wall temperature in the case of an adiabatic wall, using a Crocco-Busemann type relation and the off-wall temperature and velocity data. This wall model was however only tested on the adiabatic wall in quasi-incompressible cases, yielding satisfactory results.

The present work proposes to investigate the possibility of using a quasi-analytical wall model where the compressible flow data are scaled before being fed to the wall model. Such scalings have already been proposed to better collapse the compressible simulation data on the universal incompressible laws, the most commonly used being the Van Driest transformation [84]. However, in the late 1940s, Howarth [41] and Stewartson [75] proposed scalings to deduce the results of compressible flows from equivalent incompressible results. To the authors' knowledge, such scalings have never been investigated to propose a simple wall model for compressible flows.

In this chapter, the results of wrLES of adiabatic compressible turbulent channel flows at friction Reynolds number of $Re_\tau \simeq 1000$ and at two centerline Mach numbers M_c (one subsonic, one supersonic) are first presented, as they will later be used as reference for the evaluation and validation of our proposed wall model for wmLES. Then, the complete wall modeling strategy is explained, including the specific tabulated

non-linear solver of Reichardt's formula, the determination of the wall temperature, and the investigated scalings to take into account compressibility effects. The results of wmLES of the same compressible turbulent channel flow cases are presented and compared to those of the reference wrLES. The consistency of the new wall model at low Mach numbers is also verified by comparing the results of wmLES at $M_c = 0.25$ to those of reference incompressible Direct Numerical Simulation(DNS) data from Lee and Moser [59] at $Re_\tau = 1000$ and 5200. Finally, the proposed wall model is also applied to a turbulent channel flow at $M_c = 1.5$ and $Re_\tau = 5200$.

3.2 Wall-resolved simulations of compressible channel flows at $Re_\tau \simeq 1000$

In order to obtain reference data for the wall model assessment, two wall-resolved LES (wrLES) of compressible and turbulent channel flows have been performed. The target Reynolds number was set to $Re_\tau = 1000$ and two centerline Mach numbers M_c were investigated: (1) a subsonic case at $M_c \simeq 0.76$ and (2) a supersonic case at $M_c \simeq 1.5$.

3.2.1 Simulation setup

Adiabatic walls are considered and the meshes are identical for both cases. The domain size in the three directions is $L_x \times L_y \times L_z = 2\pi h \times 2h \times \pi h$ and the mesh is composed of $N_x \times N_y \times N_z = 64 \times 32 \times 64$ structured hexahedra. Since $p = 3$ elements are used, this represents $256 \times 128 \times 256$ degrees of freedom per equation. The mesh is uniform in the homogeneous directions (x and z) and is stretched in the y direction using

$$\frac{y}{h} = 1 + \frac{\tanh(\alpha(\xi - 1))}{\tanh(\alpha)}, \quad (3.1)$$

where $\xi = \frac{2i}{N_y}$ with $i = 0, \dots, N_y$. The stretching parameter is set to $\alpha = 2.6$. The resolution in wall coordinates for both cases is given in Table 3.1. Note that this table shows the effective resolution, which corresponds to the length of the element divided by the polynomial order, instead of the usual element length, in order to take into account the higher resolution capability of high-order Finite Element methods.

A second simulation was performed in the supersonic regime with a doubling of the elements in the vertical direction to ensure that the proposed mesh allows to have an accurate reference. This mesh preserves the wall resolution but reduces the stretching,

	Δx^+	y_1^+	Δz^+
$M_c \simeq 0.76$	29.1	1.2	14.5
$M_c \simeq 1.51$	20.4	0.8	10.2

Table 3.1: Effective resolution in wall-coordinates for the wall-resolved simulations at $Re_\tau \simeq 1000$.

effectively increasing the resolution in the center of the channel. The results of this simulation are superimposed to those of the base mesh presented previously and are observed to match almost perfectly with the coarser mesh.

The flow statistics are averaged in time over roughly 11 normalized times $t^+ = h/u_\tau$, and in the homogeneous directions. The Favre averaging is then applied to the time-averaged velocity and temperature, defined as

$$\tilde{u} = \frac{\overline{\rho u}}{\bar{\rho}}, \quad \tilde{T} = \frac{\overline{\rho T}}{\bar{\rho}} = \frac{\bar{p}}{R\bar{\rho}}. \quad (3.2)$$

The notation $\tilde{\mu}$ stands for $\mu(\tilde{T})$ and the notation $\bar{\tau}_w$ stands for the time-averaged wall shear stress.

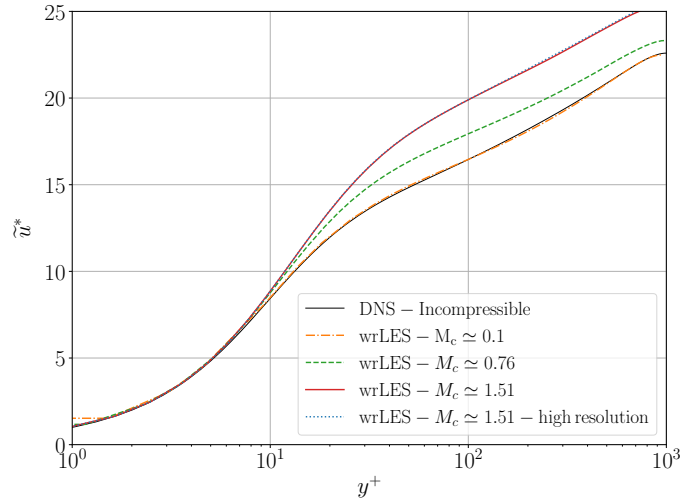
The mean pressure gradient imposed in the x direction is dynamically adapted so as to maintain a constant mass flow rate. To maintain thermal equilibrium, the power produced by the forcing pressure gradient is omitted in the total internal energy equation. This implicit forcing was observed to allow the total temperature of the flow to remain within 1 % of the target value throughout the simulation.

3.2.2 Results

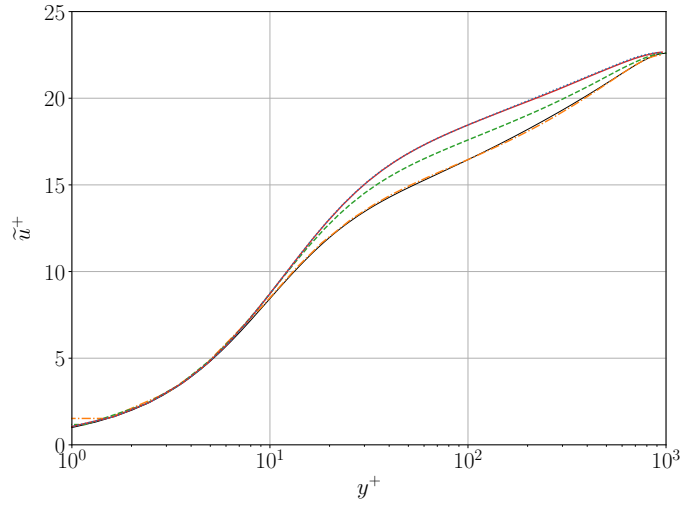
The friction Reynolds number, defined as $Re_\tau = h^+ = \frac{h\sqrt{\bar{\rho}_c \bar{\tau}_w}}{\tilde{\mu}_c}$ for a compressible channel flow, was measured as $Re_\tau = 1001$ in the subsonic case, with a centerline Mach number of $M_c \simeq 0.76$; while it was measured as $Re_\tau = 955$ in the supersonic case, with $M_c \simeq 1.51$.

Figure 3.1 shows the velocity profile for both cases: first using the semi-local normalization for both quantities, i.e.

$$y^+ = \frac{\sqrt{\bar{\rho} \bar{\tau}_w}}{\tilde{\mu}} y, \quad \tilde{u}^* = \sqrt{\frac{\bar{\rho}}{\bar{\tau}_w}} \tilde{u}, \quad (3.3)$$



(a) Using the semi-local normalization for both quantities.



(b) Using the semi-local normalization for y and the Van Driest transformation for $\tilde{u}$.

Figure 3.1: Velocity profiles in wall-coordinates for the cases at $M_c \simeq 0.76$ (dashed line) and $M_c \simeq 1.51$ at normal (thick solid line) and higher resolution (dotted line). The incompressible flow DNS data from Lee and Moser [59] (thin solid line) and the wrLES data from de Wiart et al. [22] at $M_c \simeq 0.10$ (dash-dotted line) are also shown for comparison.

in Figure 3.1(a); and then using the non-dimensionalization recommended by Huang et al. [44] which uses the semi-local normalization for y and the Van Driest transformation [84] for $\tilde{u}$:

$$\tilde{u}^+ = \sqrt{\frac{\bar{\rho}_w}{\bar{\tau}_w}} \int_0^{\tilde{u}} \sqrt{\frac{\bar{\rho}}{\bar{\rho}_w}} d\tilde{u}' , \quad (3.4)$$

in Figure 3.1(b).

This figure shows that the Van Driest transformation allows to reduce the Mach number dependence on the normalized velocity profile, while still not perfectly collapsing on the incompressible DNS, or wrLES data. This is due to the relatively low Reynolds number and the related large contribution of the viscous sub-layer. Several authors have proposed more advanced transformations to better take the effect of the viscous sub-layer into account, see Huang and Coleman [43], Brun et al. [12], Zhang et al. [92], Trettel and Larsson [80], Modesti and Pirozzoli [63].

The velocity profiles are also shown in outer coordinates in Figure 3.2, i.e. non-dimensionalized using the centerline velocity $\tilde{u}_c$ and the channel half-height h . The observed behavior is very similar to that seen in LES of channel flows at lower Reynolds numbers performed by Brun et al. [12] with a steeper (resp. flatter) velocity profile close to the wall (resp. centerline) with increased Mach number.

The velocity fluctuation profiles are shown in Figure 3.3. These fluctuation profiles are non-dimensionalized using the wall shear stress and the local density. Thus:

$$\widetilde{u''_{i,rms}}^+ = \sqrt{\frac{\widetilde{u''_i u''_i}}{\bar{\tau}_w / \bar{\rho}}} \quad \text{where} \quad \widetilde{u''_i u''_i} = \frac{\rho u''_i u''_i}{\bar{\rho}} . \quad (3.5)$$

The evolution of the near-wall streamwise fluctuation peak with increasing Mach number is similar to what is obtained by Modesti and Pirozzoli [63] in DNS and using the same normalization. They indeed observed an increase of the streamwise velocity fluctuation $\widetilde{u''_{rms}}^+$ peak when increasing the Mach number. For the spanwise and vertical profiles, a slight decrease of fluctuation amplitude is observed when increasing the Mach number. Overall, the dependence in Mach number of the velocity fluctuation profiles is not as significant as for the average velocity profile.

Figure 3.4 shows the temperature profile, normalized by the total temperature at the center of the channel,

$$\tilde{T}_{0,c} = \tilde{T}_c \left(1 + \frac{\gamma - 1}{2} \tilde{M}_c^2 \right) , \quad (3.6)$$

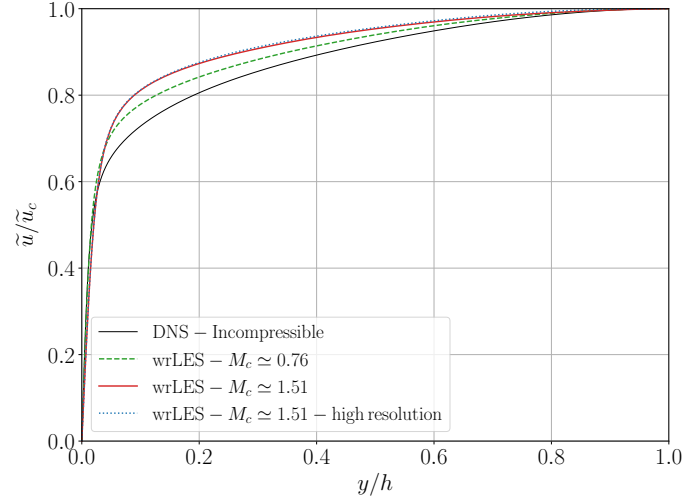


Figure 3.2: Velocity profiles in outer coordinates for the cases at $M_c \approx 0.76$ (dashed line) and $M_c \approx 1.51$ at normal (thick solid line) and higher resolution (dotted line). The incompressible flow DNS data from Lee and Moser [59] are also shown for comparison (thin solid line).

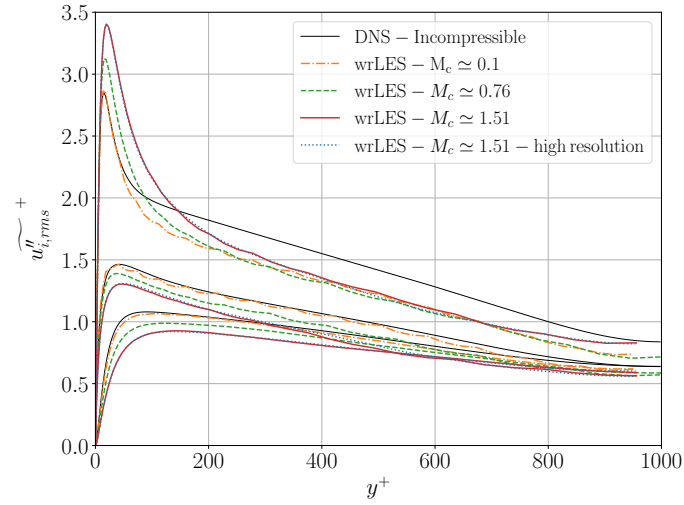


Figure 3.3: Velocity fluctuation profiles in wall-coordinates for the cases at $M_c \approx 0.76$ (dashed line) and $M_c \approx 1.51$ at normal (thick solid line) and higher resolution (dotted line). The incompressible flow DNS data from Lee and Moser [59] and the wrLES data from de Wiart et al. [22] at $M_c \approx 0.10$ (dash-dotted line) are also shown for comparison (thin solid line). Top : $\widetilde{u''_{rms}}^+$, middle : $\widetilde{w''_{rms}}^+$, bottom : $\widetilde{v''_{rms}}^+$.

where $\tilde{T}_c$ is the average temperature at the center of the channel. The normalized wall temperature is seen to decrease when the Mach number is increased, which is consistent with the experimental observations of Van Driest [84] for fluids with Prandtl number lower than unity (see also later the modeling of the profile using a recovery factor with Equation (3.9)).

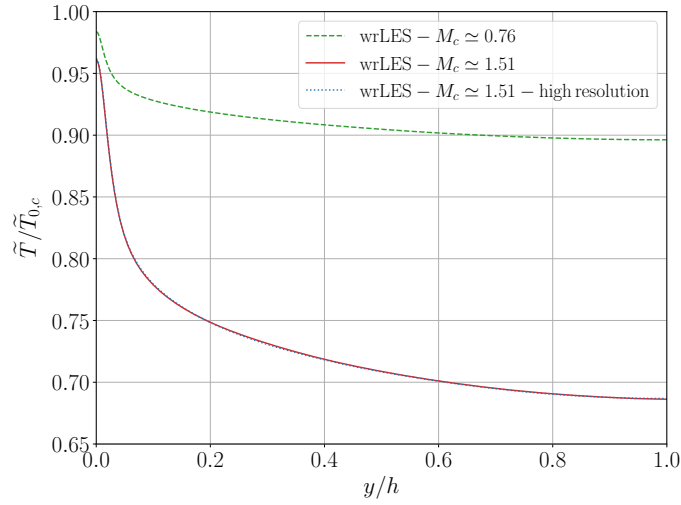
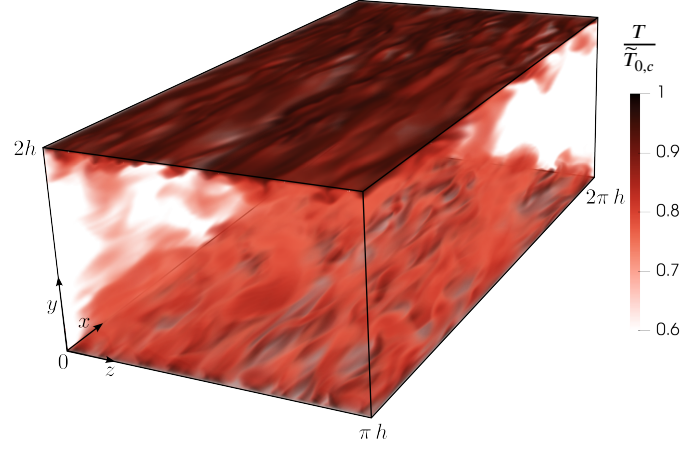


Figure 3.4: Temperature profiles for the cases at $M_c \approx 0.76$ (dashed line) and at $M_c \approx 1.51$ at normal (thick solid line) and higher resolution (dotted line).

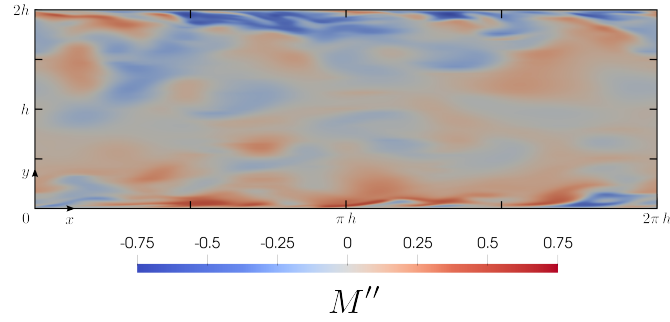
Finally, a volume rendering of the instantaneous normalized temperature field, as well as the instantaneous Mach number fluctuations in a slice of the domain, computed as $M'' = M - \tilde{M}$, are shown in Figure 3.5. We observe that the Mach number fluctuations are significant, with amplitudes up to 0.75.

3.3 Wall modeling strategy

The different wall models used to perform the wmLES of the compressible channel flow cases are presented in this section. First, the base wall model, validated for the wmLES of incompressible flows by Frère et al. [31] is introduced. Then, the three scalings for compressible flows are presented and their implementation is detailed. Since these scalings partly rely on unknown wall quantities, the approach used to estimate these quantities is explained and its accuracy assessed on the two compressible channel flow cases.



(a) Volume rendering of the temperature field.



(b) Mach number fluctuations in a slice.

Figure 3.5: Instantaneous views of the case at $M_c \simeq 1.51$.

3.3.1 Baseline wall model for incompressible flows

The base wall model used in this study relies on the law of the wall for incompressible flows, and fitted using Reichardt's formula :

$$\begin{aligned} \bar{u}_{Reichardt}^+(y^+) &= \frac{1}{\kappa} \log(1 + \kappa y^+) \\ &+ \left(C - \frac{1}{\kappa} \log \kappa \right) \left(1 - \exp\left(-\frac{y^+}{11}\right) - \frac{y^+}{11} \exp\left(-\frac{y^+}{3}\right) \right). \end{aligned} \quad (3.7)$$

We use $\frac{1}{\kappa} = 2.61$ and $C = 4.25$, in accordance with latest calibrations (see also Winckelmans and Duponcheel [89]). This equation can be solved for $\bar{\tau}_w$ given the averaged values of $\bar{u}_1$ measured at the wall model injection location y_1 . Only the component of velocity tangent to the wall is used to compute the wall shear stress. Several authors [73, 67, 36] choose to average the velocity in time and/or space. However, as investigated by Lee et al. [58] and Frère et al. [31], imposing the value of the wall shear stress instantaneously and locally is sufficient to predict the mean velocity profile. This point-wise and instantaneous approach is also used in the present work, as it can be generalized to transient and non-homogeneous flows. The complete wall modeling strategy will thus be presented using local and instantaneous quantities.

To solve Equation (3.7), instead of using a Newton-Raphson method, as done by Bocquet et al. [9], a tabulated approach is used here, as proposed by Maheu et al. [60]. In this version, the value of $\log(y_1^+ u_1^+) = \log\left(\frac{y_1 u_1 \rho}{\mu}\right)$ is computed. The value of y_1^+ is then retrieved from pre-calculated tabulated values of $\log(y^+ u_{Reichardt}^+)$ stored as a function of $\log(y^+)$. The use of the logarithmic version of the relation is based on the observation that the relation between $\log(y^+ u^+)$ and $\log(y^+)$ is almost linear; hence an accurate value of y^+ can be efficiently computed from the look-up table with only a few tabulated entries and using a linear interpolation between nearby entries. The wall shear-stress is finally computed as

$$\tau_w = \frac{1}{\rho} \left(\mu \frac{y_1^+}{y_1} \right)^2. \quad (3.8)$$

It is then projected in the global frame to be aligned with the component of the velocity tangent to the wall and imposed at the wall through a Neumann boundary condition.

This wall model was shown to provide accurate results in the framework of DGM/ILES by Frère et al. [31] for high Reynolds number incompressible flows, and it will serve as the baseline model for the present work. It will also be the foundation of the wall models investigated further.

3.3.2 Scalings for compressible flows

In order to derive a wall model compatible for compressible flows, the flow quantities measured in the simulation must be scaled so as to better follow the law of the wall valid for incompressible flows. The baseline wall model described above is then applied on those scaled quantities to finally obtain the wall shear stress.

Three transformations are studied in this work to achieve the necessary scaling: (1) the Howarth-Stewartson (HS) transformation which maps a compressible boundary layer to an incompressible boundary layer with same mass flow-rate; (2) a modified Van Driest transformation proposed by Brun et al. [12], and (3) a blending of those two scalings.

These transformations rely on the density and temperature variations close to the wall; an estimation of these quantities at the wall is thus necessary. In the following, the quantities related to the transformed flow will be noted with uppercase symbols.

Estimation of thermodynamic quantities at the wall

In order to find an estimation of the wall temperature, Walz's relation [88] is used:

$$T = T_w + (T_r - T_w) \left(\frac{u}{u_e} \right) + (T_e - T_r) \left(\frac{u}{u_e} \right)^2, \quad (3.9)$$

where

$$T_r = T_e \left(1 + r \frac{\gamma - 1}{2} M_e^2 \right) \quad (3.10)$$

is the recovery temperature and the index e denotes the edge of the boundary layer. In this study, the recovery factor r is taken as $r \simeq Pr^{1/3}$, as proposed by Van Driest [84] for a turbulent boundary layer. As this study only deals with adiabatic walls, the recovery to wall temperature ratio is unitary and Equation (3.9) can be simplified to :

$$T = T_w - r \frac{\gamma - 1}{2} \frac{u^2}{\gamma R}. \quad (3.11)$$

Introducing the quantities measured in the flow at the location where the wall model is injected, noted with subscript "1", Equation (3.11) becomes :

$$T_w = T_1 \left(1 + r \frac{\gamma - 1}{2} M_1^2 \right), \quad (3.12)$$

thus providing an estimation of the wall temperature using only the local velocity and temperature at y_1 .

We also note that combining Equation (3.12) with Equation (3.11) taken at any location in the boundary layer gives:

$$T = T_1 \left(1 + r \frac{\gamma - 1}{2} M_1^2 \right) - r \frac{\gamma - 1}{2} \frac{u^2}{\gamma R}, \quad (3.13)$$

which makes it possible to compute the temperature at any location in the boundary layer from the velocity data. Using Equation (3.13) with the velocity data from the wrLES of the compressible turbulent channel flows and comparing the result to the actual temperature data of the wrLES, as done in Figure 3.6, shows that this simplified model accurately approximates the real temperature profile in the compressible cases. The injection location of the wall model will here be taken at $y/h \simeq 0.14$; which is a typical first-cell height for wmLES of a turbulent channel flow. The error on the T_w prediction is then 0.05 % and 0.11 % for the subsonic and supersonic cases, respectively.

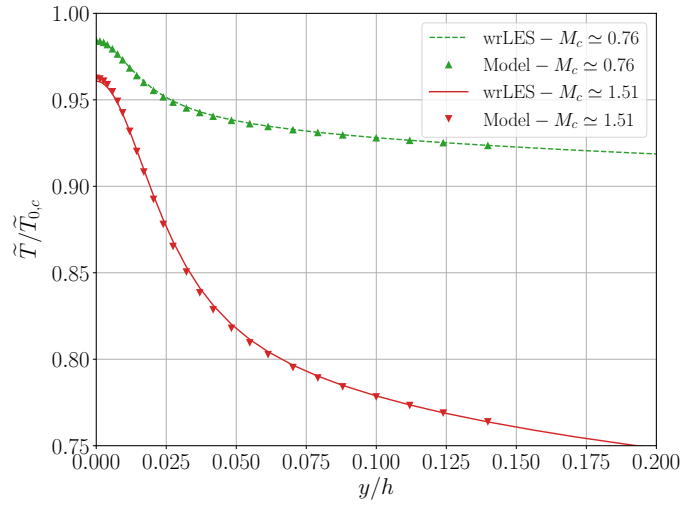


Figure 3.6: Comparison between the temperature profile measured in the wrLES (lines) and the modeled temperature profile (symbols) for the cases $M_c \simeq 0.76$ and $M_c \simeq 1.51$.

Finally, to fully characterize the thermodynamical state at the wall, the pressure is considered constant across the first element. Thus,

$$p_w = p_1, \quad (3.14)$$

$$\Rightarrow \rho_w = \frac{p_1}{RT_w}. \quad (3.15)$$

Howarth-Stewartson (HS) transformation

This transformation, proposed by Howarth [41] and Stewartson [75] in the late 1940's, can be used to map the compressible boundary layer equations to an equivalent

incompressible form. It uses a virtual incompressible streamfunction that carries the same mass flow-rate as the real compressible flow, down to the wall. This transformation requires the definition of a reference state to which the initial compressible flow is mapped; in our case, an incompressible flow with constant density ρ_w and constant viscosity $\mu_w = \mu(T_w)$.

The wall-normal coordinate of the scaled flow is then defined as:

$$Y_1 = \int_0^{y_1} \left(\frac{\rho}{\rho_w} \right) dy. \quad (3.16)$$

The compressible streamfunction ψ is introduced so that $\rho u = \rho_w \frac{\partial \psi}{\partial y}$ and $\rho v = -\rho_w \frac{\partial \psi}{\partial x}$. To ensure conservation of mass-flow rate between physical and transformed flow, this compressible streamfunction must have the same value as its scaled incompressible counterpart $\hat{\psi}(X, Y)$. Thus,

$$\rho_w \frac{\partial}{\partial y} \psi(x, y) = \rho_w \frac{\partial}{\partial Y} \hat{\psi}(X, Y) \frac{dY}{dy}, \quad (3.17)$$

$$\rho u = \rho U, \quad (3.18)$$

which thus results in a unitary scaling for the velocity:

$$U_1 = u_1. \quad (3.19)$$

The baseline model, described in Section 3.3.1 is then applied on the transformed Y_1 and U_1 , along with the fluid properties of the transformed flow (ρ_w and μ_w), providing the wall shear-stress τ_w . However, the integral in Equation (3.16) cannot be computed explicitly as the density profile is not known below the first grid point. An iterative approach, using the Crocco-Busemann relation and the velocity profile given by the Reichardt formula could be implemented. However, this method was found in *a priori* testing to yield a value for Y_1 only around 0.5 % different from that provided when using a simple explicit trapezoidal integration method; the latter was thus implemented to perform the y -coordinate scaling.

Modified Van Driest (VD) scaling

The Van Driest transformation [84] was obtained by developing the turbulent boundary layer equations for compressible flows and obtaining the compressible version of Prandtl's wall differential equation, i.e. :

$$\frac{\partial \bar{u}}{\partial y} = \frac{1}{\kappa} \sqrt{\frac{\tau_w}{\rho}} \frac{1}{y}, \quad (3.20)$$

which is identical to the incompressible version with the exception that the density is variable in this case. It was used in previous sections to scale the velocity so as to better fit the near wall incompressible flow velocity profile in wall coordinates. Contrary to the HS transformation, no scaling of the vertical coordinate was used. The ratio of the compressible wall differential equation and its incompressible form, assuming a constant density equal to the wall density, was used to obtain the scaling of the velocity derivative:

$$\frac{dU}{du} = \sqrt{\frac{\rho}{\rho_w}}. \quad (3.21)$$

Brun et al. [12] proposed an improved scaling that also acts on the derivative of the y coordinate. The complete transformation is:

$$Y = \int_0^y \frac{\mu_w}{\mu} dy, \quad (3.22)$$

$$U = \int_0^u \frac{y}{Y} \frac{\mu_w}{\mu} \sqrt{\frac{\rho}{\rho_w}} du. \quad (3.23)$$

It was pointed out by Brun et al. [12] that applying this transformation on the velocity profile decreases the spread of the value of the C coefficient in the log region for different flow configurations, yet does not cancel it entirely.

In the present work, the y -coordinate transformation is approximated, at the first grid-point, using

$$Y_1 = \frac{\mu_w}{\mu_1} y_1, \quad (3.24)$$

as this yields better results when used in the complete wall model than using a trapezoidal rule. Equation (3.23) is then simplified to the expression of the classical Van Driest transformation and can be integrated analytically, recalling that the pressure is uniform near the wall and that the temperature follows Walz's relation [88]:

$$U_1 = \int_0^{u_1} \sqrt{\frac{T_w}{T}} du, \quad (3.25)$$

$$= \int_0^{u_1} \left(1 - \frac{r}{2 c_p T_w} u^2 \right)^{-1/2} du, \quad (3.26)$$

$$= \sqrt{\frac{2 c_p T_w}{r}} \arcsin \left(\sqrt{\frac{r}{2 c_p T_w}} u_1 \right). \quad (3.27)$$

A difference of at most 1 % is observed on the input value of the baseline wall model (i.e. $\frac{U_1 Y_1 \rho_w}{\mu_w}$) between the result provided by the numerical integration of Equation (3.22) and Equation (3.23) using the wrLES data, and the value obtained using Equation (3.24) and Equation (3.27).

Present scaling

The third proposed scaling can be seen as a blending of the two previous ones. As for the modified Van Driest scaling, it relies on a scaling of the vertical coordinate, but is based on the ratio of the semi-local scaling used in Figure 3.1 and the incompressible normalization of the vertical coordinate. The velocity transformation, on the other hand is obtained by ensuring the conservation of mass flow rate between the real compressible flow and the transformed incompressible flow, similarly to the HS scaling. However, the y -coordinate transformation now contains a factor to take the viscosity variation into account:

$$Y_1 = \int_0^{y_1} \frac{\mu_w}{\sqrt{\rho_w}} \frac{\sqrt{\rho}}{\mu} dy, \quad (3.28)$$

$$U_1 = \sqrt{\frac{\rho_1}{\rho_w} \frac{\mu_1}{\mu_w}} u_1. \quad (3.29)$$

The $\frac{\mu_w}{\sqrt{\rho_w}} \frac{\sqrt{\rho}}{\mu}$ scaling factor for y is inspired by the recommendation of Trettel and Larsson [80].

As was the case when using the HS transformation, the integral for Y_1 is evaluated using a trapezoidal rule, thus taking the average of the argument of the integral estimated at the wall and measured at y_1 , providing a simple explicit scaling. When comparing the numerical integration of Equation (3.28) using wrLES data to the trapezoidal rule, the error done on Y_1 is lower than 2 % for both Mach numbers as long as $y_1/h \lesssim 0.15 - 0.20$, which is a necessary condition to use such wall model solely based on the law of the wall.

Other scalings exist to collapse compressible flow results to equivalent incompressible flow results, such as those proposed by Volpiani et al. [87], Trettel and Larsson [80] and Griffin et al. [37]. These more recent scalings proved to collapse more accurately the compressible flow data onto incompressible results in the post-processing phase. However, the latter two rely on using derivatives of the time-averaged flow quantities in the wall normal direction. Using such derivatives, evaluated locally and instantaneously in the simulation, would be prone to large errors, and also in the under-resolved first off-wall elements; hence introducing large deviations in the model. The

scaling proposed by Volpiani et al. [87] is also similar to the modified Van Driest transformation Brun et al. [12] but the integral present in the velocity scaling cannot be evaluated analytically; which then also introduces large errors when evaluated using the trapezoidal rule. The *a priori* error was nevertheless computed for this scaling as well, and is shown in Figure 3.8.

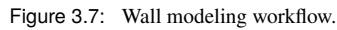
3.3.3 A priori error estimation

Based on the results of the wrLES at $Re_\tau \simeq 1000$ and at the two centerline Mach numbers, the error on the wall shear stress prediction can be evaluated *a priori*. To do so, the wall modeling workflow is followed, taking as input the averaged flow quantities all along the wrLES profiles. The complete workflow, including the baseline wall model, is summarized in Figure 3.7. Note that, in the case where no scaling is used, the center dotted box is by-passed and thus $Y_1 = y_1$ and $U_1 = u_1$.

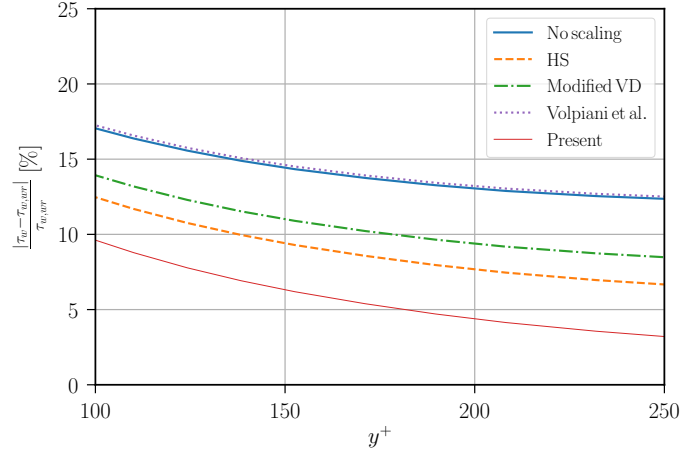
Applying this methodology to the wrLES data, we obtain the results shown in Figure 3.8 for the error on τ_w when using the different wall models compared to the actual wrLES wall shear-stress. This figure shows the range between $y^+ = 100$ and 250 although the present wall model should not be used above $\simeq 200$ for such wmLES at $Re_\tau \simeq 1000$, as the contribution of the complement function that adds to the law of the wall becomes important.

This figure clearly demonstrates how the scaling procedure improves the results concerning the estimation of the wall shear stress that will be further applied via the wall model. In both cases, the use of a scaling decreases the error on τ_w . At $y^+ = 150$, which would be a reasonable first cell height for wmLES at $Re_\tau \simeq 1000$, the error when going from $M_c \simeq 0.76$ to $M_c \simeq 1.51$ goes from 14.3 % to 38.5 % when no scaling is applied, and from 12.4 % to 30 % when the VD scaling is used. At the same location, the error only increases from 9.3 % to 17 % when the HS scaling is used, and from 6.2 % to 6.5 % when the present scaling is used. The present scaling thus performs best. We also note that its error remains at the same level when increasing the Mach number.

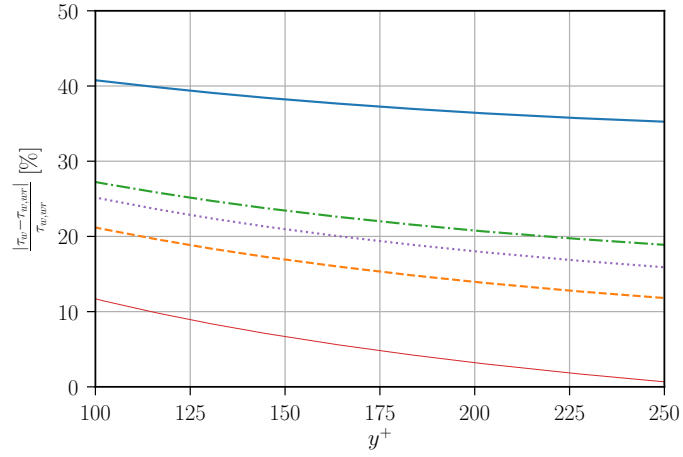
Figure 3.9 shows the scaled velocity profiles, using each of the presented scalings. As for Figure 3.8, those profiles are only shown up to the location where a wall model could be applied ($y^+ \simeq 250$ for this Reynolds number), thus removing the wake region, where no collapse of the velocity profiles is expected. This figure confirms the conclusions based on the error on τ_w presented in Figure 3.8 and shows that the velocity profile scaled using the hybrid scaling produces a velocity profile that collapses better onto the Reichardt formula.


$$\text{Re}_\tau \simeq 1000$$

Mach numbers.



(a) $M_c \approx 0.76$

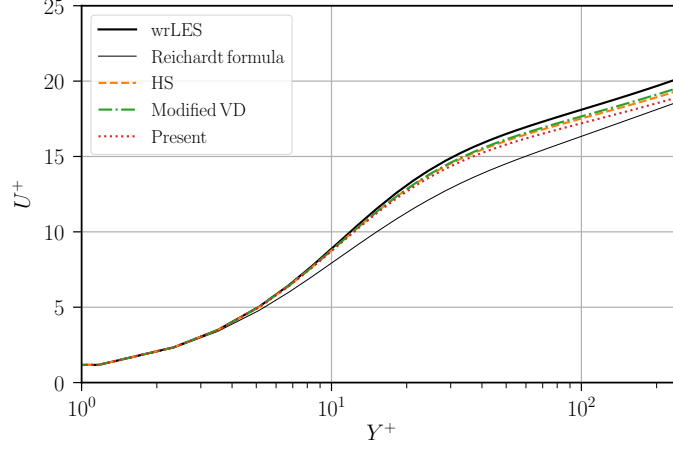


(b) $M_c \approx 1.51$

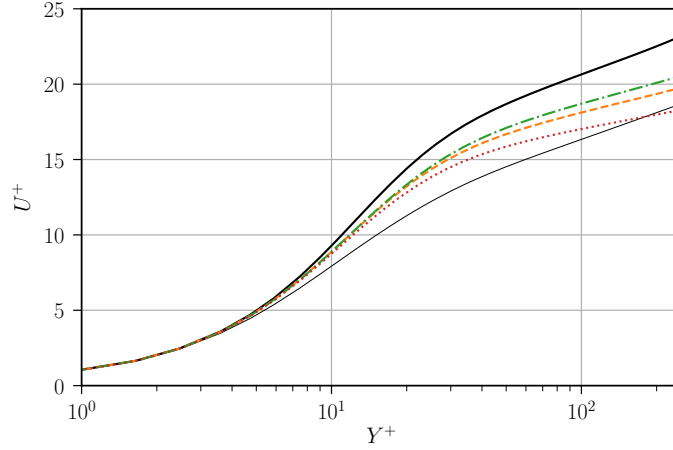
Figure 3.8: Error on the wall shear-stress in % when using the wall models on the wrLES data compared to the wrLES averaged wall shear-stress.

3.4.1 Simulation setup

The computational domain size for the wall-modeled simulations is the same as that for the wall-resolved simulations. The order of the interpolation polynomials is set to 3 in the bulk of the domain and is increased to 4 in the first wall-adjacent elements as



(a) $M_c \approx 0.76$



(b) $M_c \approx 1.51$

Figure 3.9: Normalized scaled velocity profiles. Zoom on the region where a wall model could be applied.

this was observed to give the best results in the framework of wall-modeled ILES/DGM by Frère et al. [31] using the same code. For the wall-modeled simulations, the number of elements in the three directions is $N_x \times N_y \times N_z = 26 \times (11 p_3 + 2 p_4) \times 13$. The same stretching function is used (see Equation (3.1)) and the stretching parameter is set to $\alpha = 0.4$.

Still following the recommendations of Frère et al. [31], the computation of the wall shear stress through the wall model is based on the flow quantities measured at the bottom of the second wall-adjacent element. The mesh is shown in Figure 3.10 with the locations at which the wall model is evaluated shown using a star. There is thus one DG element (with five degrees of freedom) between the wall and the location (bottom of the second DG element) where the flow quantities are measured to evaluate the wall shear stress, allowing to have an accurate measurement of the flow quantities in the LES, as was also shown by [31]; and as was also recommended by Kawai and Larsson [50]. The resolution in wall-coordinates for the wmLES, rounded to 5, is given in Table 3.2. Note that in this case, the resolution in the x and z directions corresponds to the effective resolution, i.e. the size of the element divided by the polynomial order, whereas y_1^+ is the height of the first element.

	Δx^+	y_1^+	Δz^+
$M_c \simeq 0.25$	60	140	60
$M_c \simeq 0.76$	60	135	60
$M_c \simeq 1.51$	50	115	50

Table 3.2: Resolution in wall-coordinates for the wall-modeled simulations at $Re_\tau \simeq 1000$, rounded to 5. The resolution in the x and z directions correspond to the effective resolution, i.e. the size of the element divided by the polynomial order, whereas y_1^+ is the height of the first element.

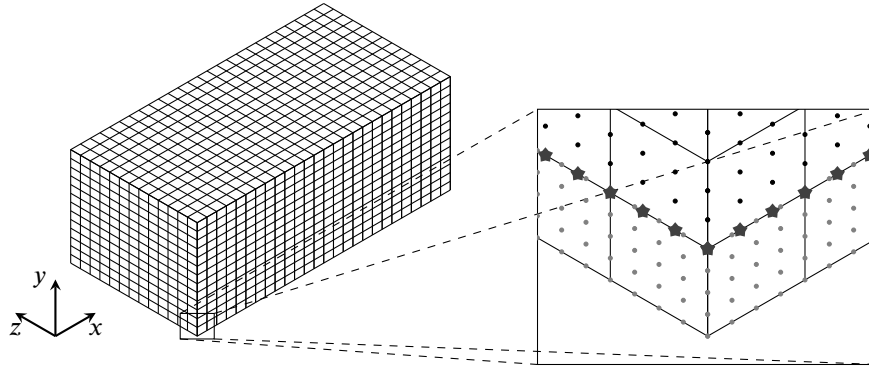


Figure 3.10: Mesh used for the wall-modeled simulations. On the right part of the figure, the degrees of freedom are shown in gray in the buffer element ($p = 4$) and in black in the bulk elements ($p = 3$). The stars represent the degrees of freedom where the quantities are measured to compute the wall shear stress using the wall model.

Contrary to the wrLES, these wall-modeled simulations are driven using a constant imposed streamwise pressure gradient. The use of this forcing source term was preferred as it was observed to result in a faster statistical convergence in the case of the wall-modeled simulations. This forcing source term is computed using the incompressible DNS results to reach $Re_\tau = 1000$ for the case at $M_c \simeq 0.25$. For the higher Mach number cases, it is taken as the time-averaged pressure gradient of the wrLES. The wall shear stress of each wmLES will thus be essentially equal to that of the corresponding wrLES, and the error will be evaluated on the mass flow rate.

3.4.2 Results

The velocity profiles in wall coordinates are shown in Figure 3.11 for the quasi-incompressible simulations and are compared to the incompressible DNS results of Lee and Moser [59] at the same Reynolds number. Moreover, the quasi-incompressible wall-resolved ILES data obtained using DGM/SIP at $Re_\tau = 950$ and $M_c \simeq 0.10$ and presented by de Wiart et al. [22] are also shown for a full comparison over the whole range of resolutions of scale-resolving simulations. The wmLES profiles of the

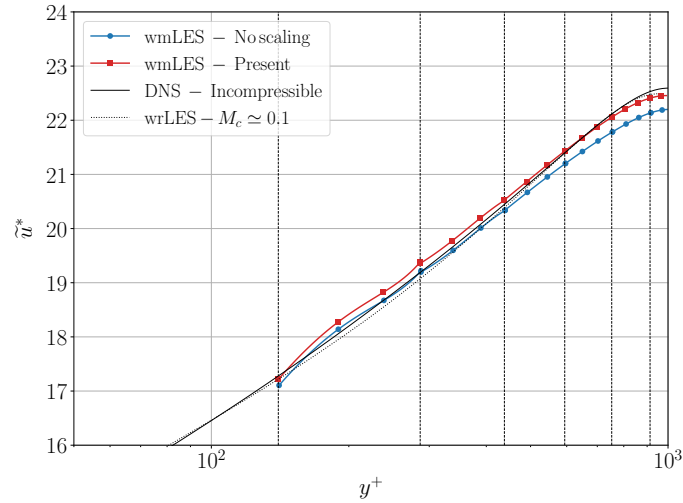


Figure 3.11: Velocity profiles in wall-coordinates for the channel flows at low Mach numbers: reference incompressible flow DNS (Lee and Moser [59], solid black), wrLES at $M_c \simeq 0.10$ (de Wiart et al. [22], dotted black), wmLES at $M_c \simeq 0.25$ without scaling (blue) and with present scaling (red). The vertical dashed black lines show the element boundaries (for readability reasons, those are only shown for the profile obtained using the present scaling).

compressible channel flows are shown in Figure 3.12 and are compared to the wrLES profiles presented in Section 3.2.

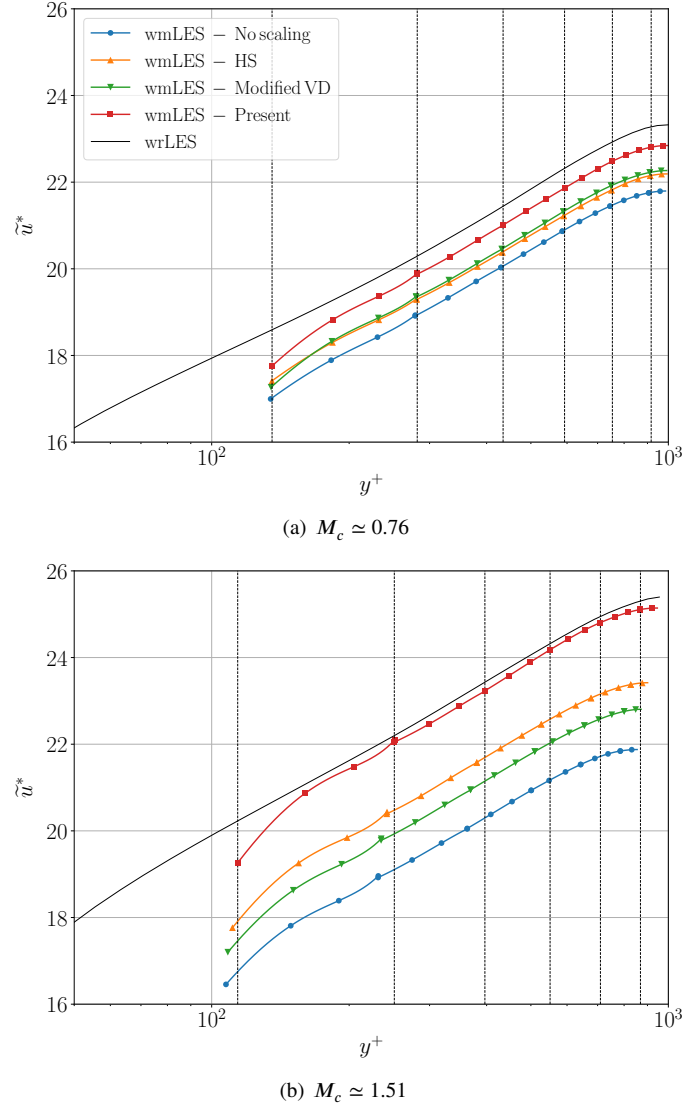


Figure 3.12: Velocity profiles in wall-coordinates for the compressible channel flows. The vertical dashed black lines show the element boundaries (for readability reasons, those are only shown for the profile obtained using the present scaling).

The semi-local normalization, defined in Equation (3.3), is used for the wall-distance and velocity in Figures 3.11 and 3.12 because the solution computed in the first wall-adjacent element being nonphysical, the integral present in the classical Van Driest transformation (see Equation (3.4)) cannot be performed with sufficient precision.

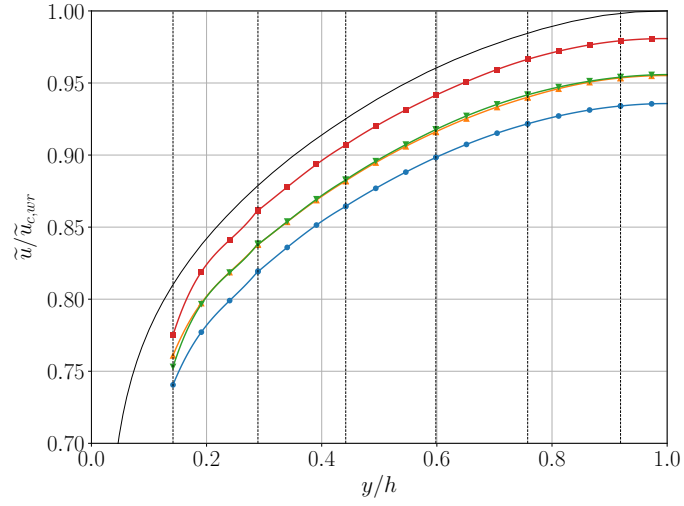
Computation element boundaries are shown by vertical dashed lines. There are no degrees of freedom on the channel centerline as it cuts the center $p3$ element in two. The degrees of freedom in the first off-wall element will be omitted in all the following graphs as they do not bring any meaningful information. The first degree of freedom shown in those graphs is thus where the flow quantities are evaluated for the calculation of the wall shear stress.

As expected, the proposed wall model yields virtually identical results to the baseline incompressible wall model at low Mach numbers, see Figure 3.11. The obtained velocity profile is also found to be very close to that of the reference incompressible flow DNS.

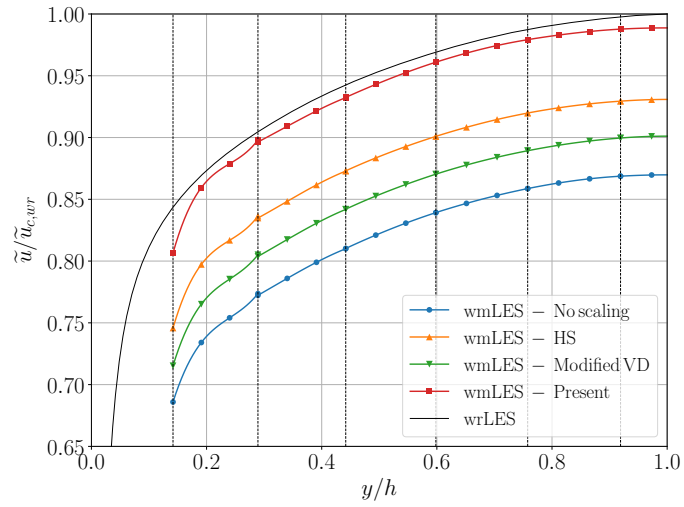
It can be seen in Figure 3.12 that the error on the velocity profile follows the trends obtained in the *a priori* error analysis (Section 3.3.3). Indeed, the velocity profiles obtained with the unscaled wall model exhibit the largest error in both cases, its magnitude increasing with Mach number. The Howarth-Stewartson and modified Van Driest scalings yield very similar results at subsonic speeds but the accuracy of the former decreases less than that of the latter when going to the supersonic case. Lastly, the presently proposed scaling produces the most accurate velocity profile in both cases, with a significant error reduction for increasing Mach number.

The same velocity profiles are shown in outer coordinates in Figure 3.13. In this figure, the velocity is normalized by the centerline velocity of the wrLES $u_{c,wr}$ as this better highlights the error on the velocity profile for the different scalings. This figure shows the same trends as what was observed in wall-coordinates, the unscaled wall model underevaluating the centerline velocity by 6% in the subsonic case and by 14% in the supersonic case; whereas the proposed scaling maintains this error below 2 % in both cases. The error when using the other two scalings lies between these boundaries. A similar behavior can be observed on the temperature profile, in Figure 3.14.

The density profiles are shown in Figure 3.15. At the lower Mach number, the three scaled wall models, as well as the baseline wall model, produce essentially the same results, accurately retrieving the density variation of the wrLES. When the Mach number is increased to $M_c \simeq 1.51$, a more significant difference is observed between

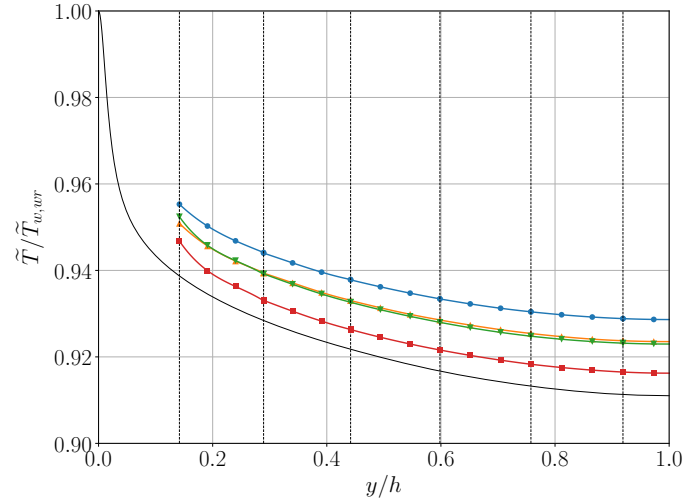


(a) $M_c \simeq 0.76$

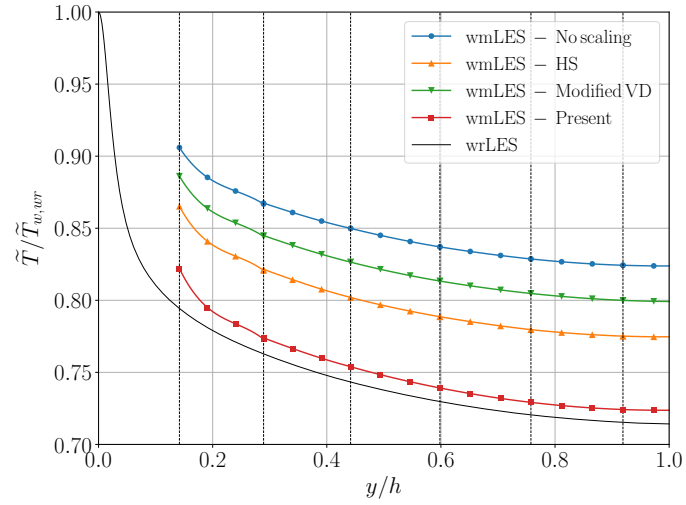


(b) $M_c \simeq 1.51$

Figure 3.13: Velocity profiles in outer coordinates for the compressible channel flows. The vertical dashed black lines show the element boundaries.



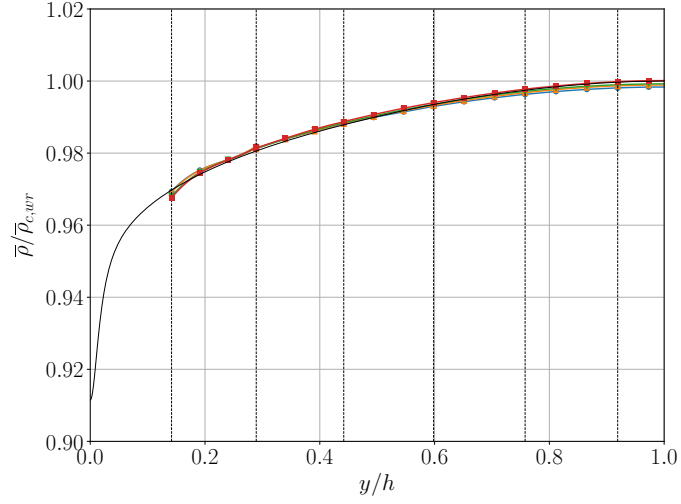
(a) $M_c \simeq 0.76$



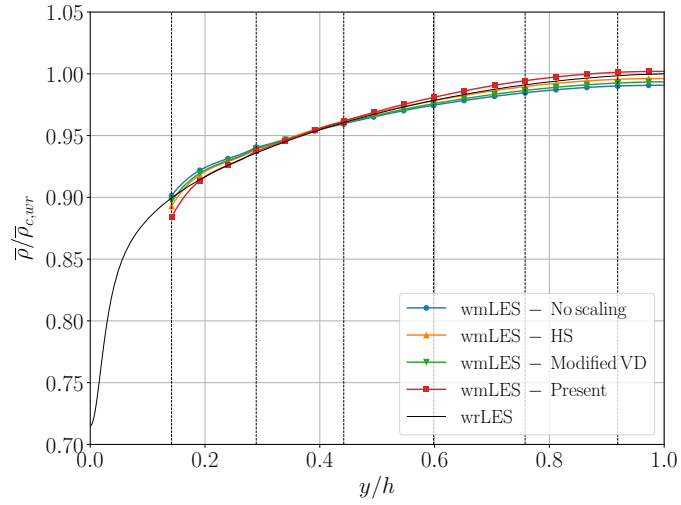
(b) $M_c \simeq 1.51$

Figure 3.14: Temperature profiles for the compressible channel flows.

these models, and the Howarth-Stewartson and present scalings provide the best overall results.



(a) $M_c \simeq 0.76$

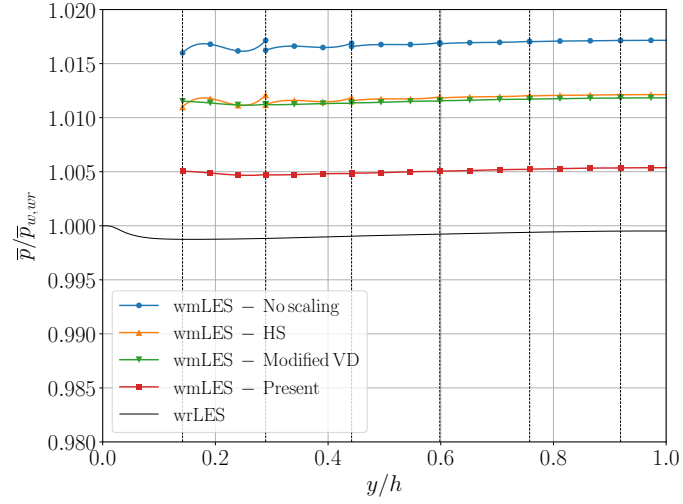


(b) $M_c \simeq 1.51$

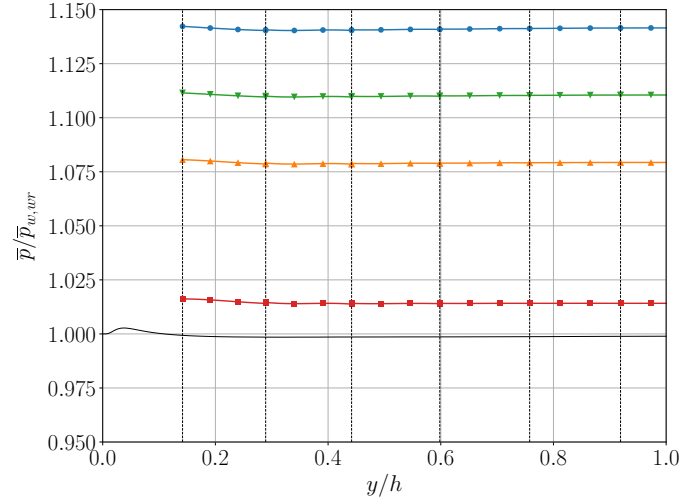
Figure 3.15: Density profiles for the compressible channel flows.

Finally, Figure 3.16 shows the pressure profile in the channel for both Mach numbers. This shows that the pressure is indeed almost constant in the whole span of the channel, except for the small variations observed in the near-wall region. These

variations however only amount to less than 0.5 %. This figure again shows the superiority of the present wall model compared to the other wall models as the error on the pressure compared to the wrLES profiles is largely decreased as well.



(a) $M_c \approx 0.76$



(b) $M_c \approx 1.51$

Figure 3.16: Pressure profiles for the compressible channel flows.

The differences in velocity and density profiles yield differences in the computed mass flow rate for each of the scalings and the error on this global quantity compared to the reference simulation (DNS for the quasi-incompressible case at $M_c \simeq 0.25$ and wrLES for the cases at $M_c \simeq 0.76$ and $M_c \simeq 1.51$) is summarized in Table 3.3 for all studied cases. Two diagnostic quantities are computed: (i) the mass flow rate in the channel, including the under-resolved buffer element, noted Q_m and (ii) the integral of momentum in the resolved part of the channel in the wmLES (i.e. excluding the region covered by the under-resolved buffer element) and noted Q_m^{res} . The use of this second diagnostic makes it possible to better appreciate how the unphysical data of the buffer element impact the evaluation of the total mass flow rate.

		No scaling	HS	Modified VD	Present
Error on Q_m [%]	$M_c \simeq 0.25$	1.33	/	/	0.67
	$M_c \simeq 0.76$	6.90	4.86	4.78	2.19
	$M_c \simeq 1.51$	14.48	7.71	11.03	1.14
Error on Q_m^{res} [%]	$M_c \simeq 0.25$	1.07	/	/	0.43
	$M_c \simeq 0.76$	6.60	4.60	4.36	1.82
	$M_c \simeq 1.51$	14.22	7.55	10.82	1.03

Table 3.3: Error on the total mass flow-rate Q_m and on the integral of momentum in the resolved part of the wmLES Q_m^{res} , at $M_c \simeq 0.25$, $M_c \simeq 0.76$ and $M_c \simeq 1.51$.

As expected from the profiles previously shown, the scaled and unscaled wall models have a similar accuracy in terms of mass flow rate at low Mach number. For the higher Mach numbers, the present scaling provides the best results while all the scaled wall models perform better than the unscaled wall model. For all the considered wall models, the error on the second diagnostic is slightly lower than the error on the total mass flow rate but this is not significant. The increase of centerline Mach number from $M_c \simeq 0.76$ to $M_c \simeq 1.51$ has the effect of increasing the error on the mass flow rate by more than a factor two for the unscaled wall model and the wall model scaled using the modified Van Driest scaling. For the Howarth-Stewartson scaling, a slightly lower, yet very high 50 % increase in error is observed. For the present scaling, the evolution of the error on mass flow-rate is non-monotonous: it first increases when increasing the Mach number from $M_c \simeq 0.25$ to $M_c \simeq 0.76$; then, a slight decrease in error is observed when going to the supersonic range. This decrease is however marginal. When using the integral of momentum on the resolved part of the channel, it is even less significant. Concerning the increased error for the wmLES at $M_c \simeq 0.76$ with respect to the wmLES at $M_c \simeq 0.25$, it can also be partially attributed to the

fact that the reference used in the low subsonic case is an incompressible flow DNS, whereas that used for the high subsonic case is our compressible flow wrLES.

Globally, the proposed scaling clearly provides improved performances on the prediction of the local quantities (velocity and temperature profiles) and of the global mass flow rate, both for high subsonic flows and for supersonic flows (at least up the Mach number of 1.5 so far investigated).

The velocity fluctuation profiles are shown in Figures 3.17 and 3.18. For all cases, the variations in results depending on the used scaling are much less significant than those obtained for the mean velocity profiles. For the low subsonic case in Figure 3.17, both wmLES retrieve well the reference DNS profiles for the vertical and spanwise directions. In the streamwise direction, the fluctuation amplitude is underestimated in the wmLES. However, the obtained profile compares well with that of a wrLES at similar Re_τ and $M_c \simeq 0.10$. The underestimation of the fluctuation amplitude is thus inherent to the LES framework.

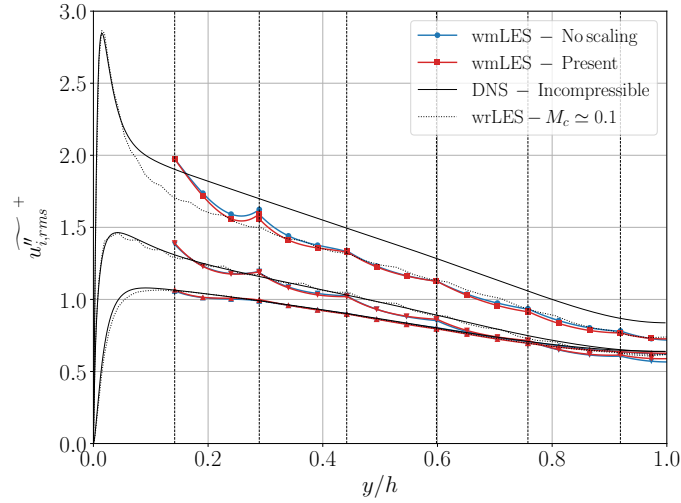
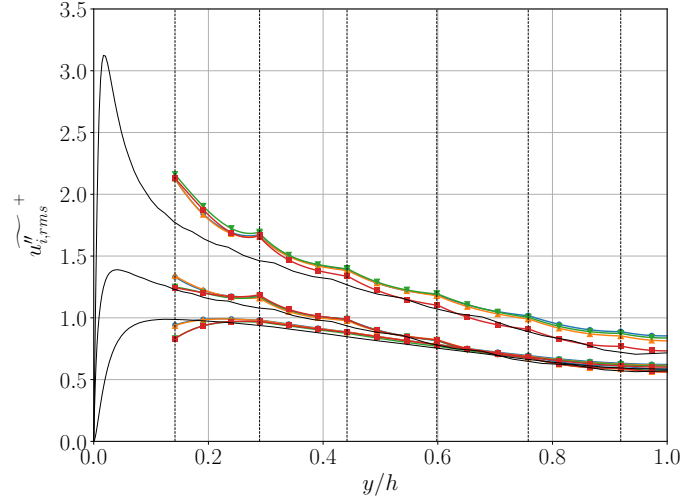
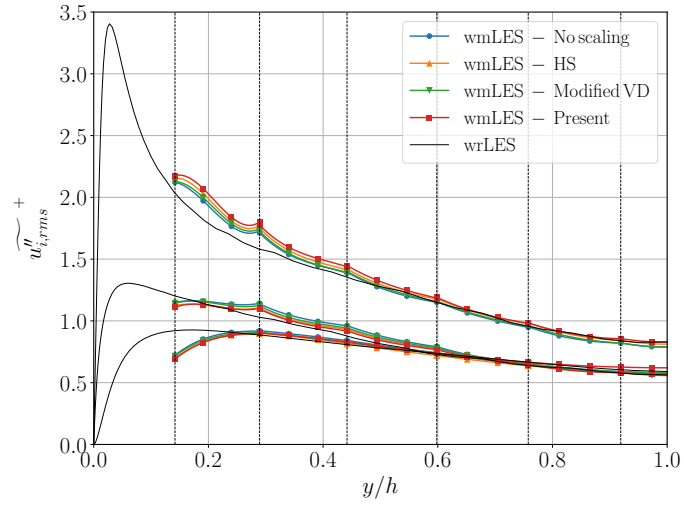


Figure 3.17: Fluctuation profiles for the channel flows at low Mach numbers: reference incompressible flow DNS (Lee and Moser [59], solid black), wrLES at $M_c \simeq 0.10$ (de Wiart et al. [22], dotted black), wmLES at $M_c \simeq 0.25$ without scaling (blue) and with present scaling (red).

For the high subsonic and supersonic cases in Figure 3.18, the same accuracy is obtained in the transverse directions. In the longitudinal direction the present scaling provides slightly better results in the region close to the channel center in the subsonic



(a) $M_c \simeq 0.76$



(b) $M_c \simeq 1.51$

Figure 3.18: Fluctuation profiles for the compressible channel flows.

case, and results similar to those of the other models in the lower part. In the supersonic case, the wmLES profiles are similar and they follow well the wrLES profile.

Finally, the Reynolds stresses are shown in Figure 3.19, showing that all the wall models predict very similar stresses in the whole channel. Moreover, the wmLES profiles compare well with the reference simulations, with a slight loss of accuracy

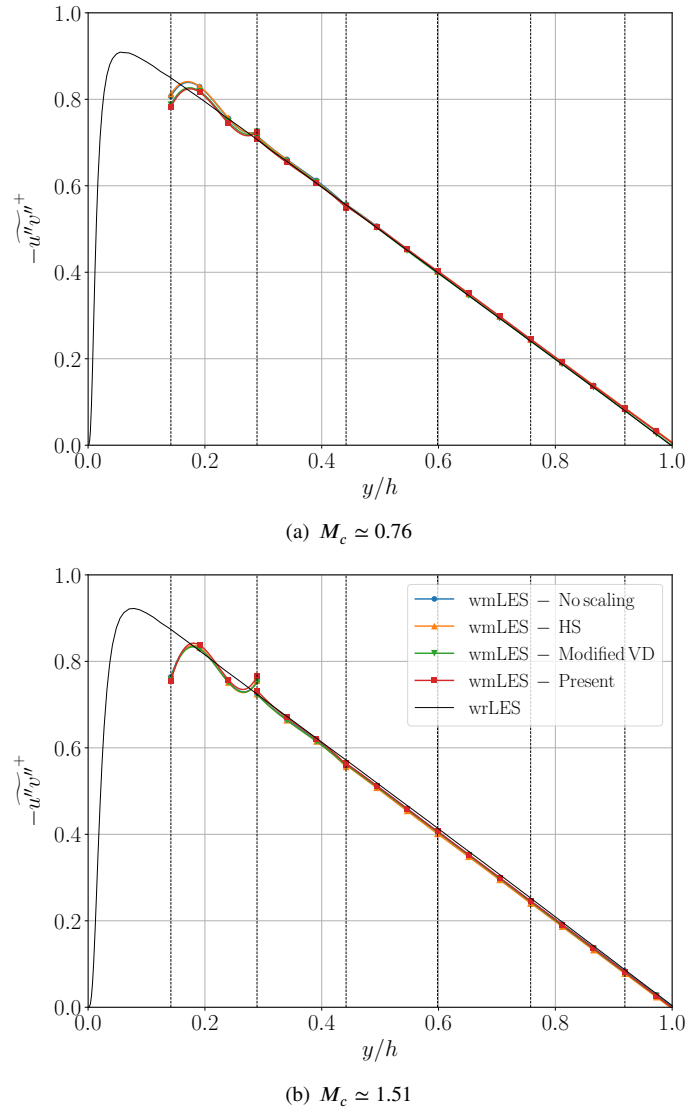


Figure 3.19: Reynolds stress profiles for the compressible channel flows.

in the supersonic case. However, in the first resolved element, large oscillations are observed. This is due to the significant under-resolution used in that region and was already observed in the framework of wall-modeled LES using DGM [21].

3.5 Wall-modeled simulations of compressible channel flows at $Re_\tau \simeq 5200$

In order to investigate the behavior of the proposed wall model in more practical cases where the inner and outer regions of the boundary layer are well separated, two cases at $Re_\tau \simeq 5200$ are studied with the wmLES. In a first case, an adiabatic channel flow with $M_c \simeq 0.24$ is computed. The results can thus be compared to the incompressible DNS database of Lee and Moser [59] at the same Reynolds number. The second case is a supersonic channel flow with centerline Mach number $M_c \simeq 1.46$. No reference data is available for this Mach number at such high Reynolds number; the computational cost of a wrLES would be too high and is beyond the scope of this study. The results obtained for this case are thus analyzed in a more qualitative manner.

3.5.1 Simulation setup

The simulation setup is similar to that of the wmLES at $Re_\tau \simeq 1000$ (see Section 3.4.1): the domain dimensions, the number of elements, and their interpolation orders are the same. Maintaining the number of elements constant when increasing the Reynolds number by a factor of five might seem in contradiction with the requirements of Choi and Moin [16], which recommend that the number of grid points should increase linearly with the Reynolds number. However, the proposed mesh resolution gives a theoretical wall model injection location of $y_1^+ \simeq 800$ or $y_1/h = 0.154$, which lies at the end of the log-layer (as also measured by Winckelmans and Duponcheel [89]), thus within the range of validity of the law of the wall. Moreover, this mesh resolution is the same as that used by Frère et al. [31] for a channel flow at the same Reynolds number (and even for a case at $Re_\tau = 50\,000$), and was shown to give good results in the frame of an ILES/DGM. The only change in the mesh compared to that used for the channel flows at $Re_\tau \simeq 1000$ concerns the stretching in the vertical direction which has been removed here, hence exactly matching the mesh from Frère et al. [31] with the same interpolation orders.

The other solver parameters are the same as previously. The Reynolds number is enforced by imposing a constant pressure gradient in the streamwise direction. The friction Reynolds number obtained for the low Mach number simulation ($M_c \simeq 0.24$)

is $Re_\tau = 5178$ whereas the supersonic channel reached $M_c \simeq 1.46$ and $Re_\tau = 5166$. The wall-resolution is $\Delta y_1^+ = 790$ and 700 for the subsonic and supersonic cases, respectively.

3.5.2 Results

Low subsonic channel flow

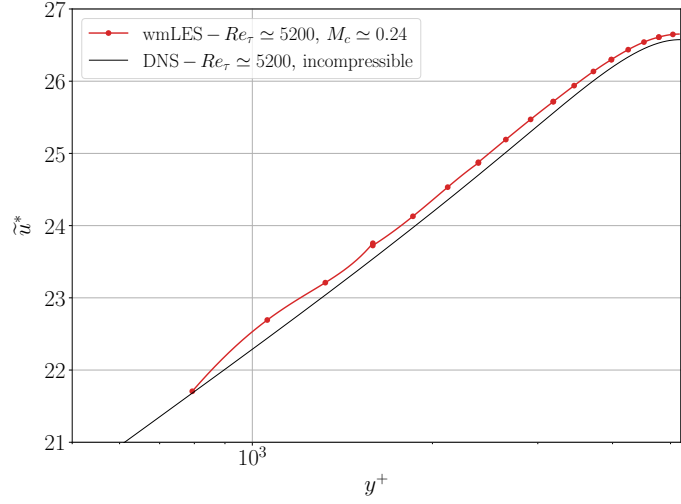
The obtained mean velocity profile is shown in Figure 3.20 in both wall and outer coordinates. A slight over-prediction of the velocity over the whole channel span is observed. This does not exceed 1 %, which also corresponds to the actual density variation across the channel. Thus, the observed deviation can be attributed to the rather small compressibility effects when increasing the Mach number, resulting in an upwards shift in the velocity profile; as was also observed in Figure 3.1.

Concerning the fluctuations, shown in Figure 3.21, the wmLES underestimates the longitudinal velocity fluctuations compared to the DNS, as usual for LES simulations. The fluctuation profiles in the y and z directions are well represented by the wmLES.

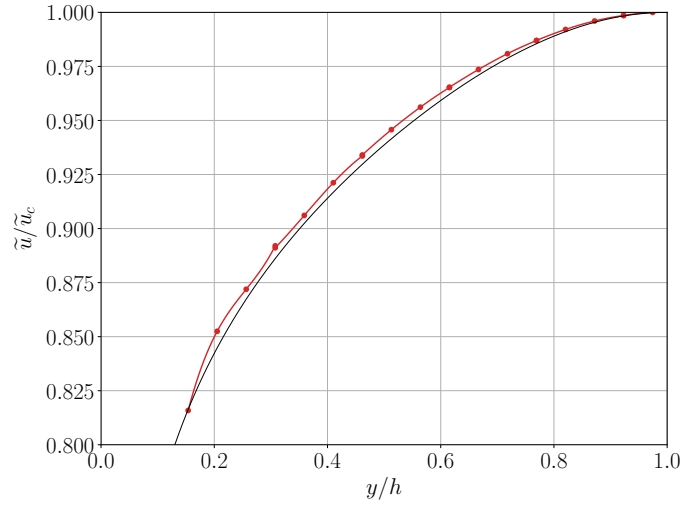
Supersonic channel flow

The obtained mean velocity profile for the supersonic channel at high Reynolds number is shown in Figure 3.22 in both wall and outer coordinates. Given the lack of reference to assess the validity of the results for this case, the wrLES results at $Re_\tau \simeq 1000$ and $M_c \simeq 1.51$, as well as the incompressible flow DNS data from Lee and Moser [59] at $Re_\tau \simeq 5200$, are used as a basis for comparison.

Figure 3.22(a) shows a significant upward shift of the velocity profile in wall coordinates when compared to the incompressible flow at the same Reynolds number. This shift is physical and is essentially due to the high Mach number; it is indeed seen to be very similar to that observed at $Re_\tau \simeq 1000$ in Figure 3.1(a). In outer coordinates, the previously observed lower slope near the center of the channel with an increase of the Mach number is also physical; it is clearly visible when comparing to the incompressible flow profile. The profile at $Re_\tau \simeq 5200$ and $Re_\tau \simeq 1000$ almost overlap. This very low dependency on the Reynolds number for the bulk velocity profile can also be observed in the DNS results of Yao and Hussain [91] for cooled channel flows at the same Mach number and at increasing Reynolds numbers ($Re_\tau = 145 - 1266$). The above elements allow to have some confidence in the quality of the velocity profiles obtained in our wmLES using the present scaling for supersonic flows at high Reynolds number.



(a) In wall-coordinates.



(b) In outer coordinates.

Figure 3.20: Velocity profiles for the channel flow at low Mach number: wmLES at $M_c \simeq 0.24$ with present scaling (red) and incompressible flow DNS (Lee and Moser [59], solid black).

The obtained fluctuation profiles are shown in Figure 3.23. It was observed (Figure 3.3) for wrLES that the spanwise and vertical fluctuations were not strongly

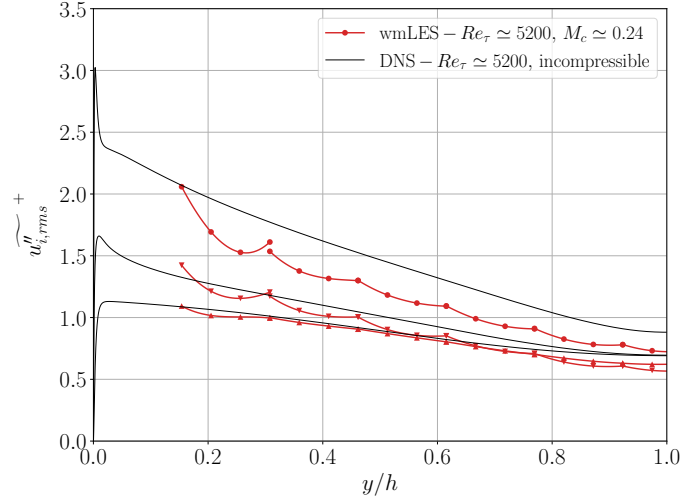
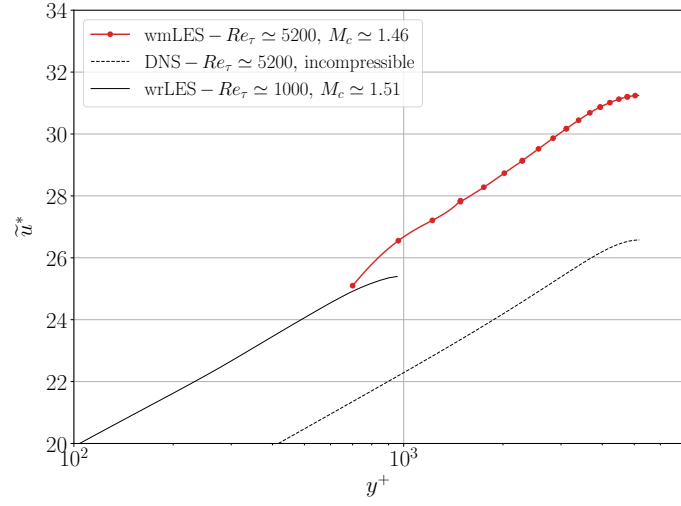


Figure 3.21: Velocity fluctuation profiles for the channel flow at low Mach number and high Reynolds number: wmLES at $M_c \simeq 0.24$ with present scaling (red) and incompressible flow DNS (Lee and Moser [59], solid black).

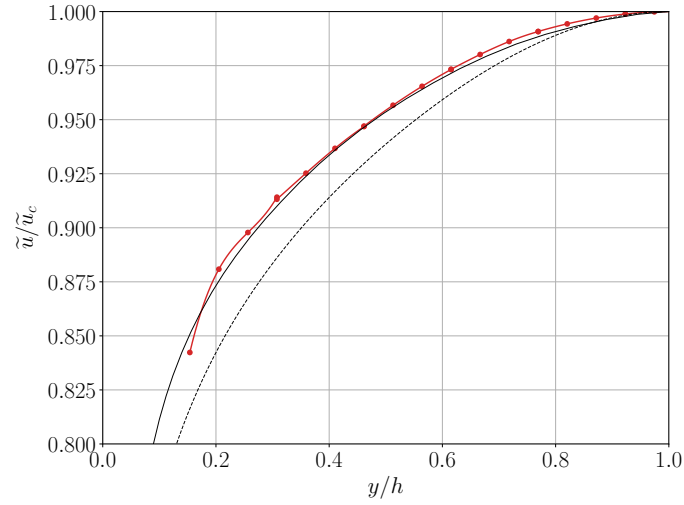
dependent on the Mach number. The deviation observed between the incompressible flow and the supersonic channel flows at similar Reynolds numbers should thus exhibit a similar behavior; it is indeed the case with the usual slight underestimation by the wmLES (attributed to the intrinsic under-prediction of the fluctuations by a LES compared to a DNS). This shift was also observed in the wrLES results. Moreover, the velocity fluctuations in these transverse directions for the wmLES are slightly increased compared to the wrLES at similar Mach number and $Re_\tau \simeq 1000$. This evolution with increasing Reynolds number is also observed in reference incompressible flow DNS data (see for instance Lee and Moser [59]). The above observations strengthen our confidence in the spanwise and vertical velocity fluctuations obtained by our wmLES.

In the streamwise direction, the obtained wmLES profile is similar to that of the wrLES profile at similar Mach number and $Re_\tau \simeq 1000$. As expected, they both stand above the profile for incompressible flow.

Finally, Figure 3.24 shows the obtained density and temperature profiles for this wmLES. These profiles can only be compared to those obtained in the wrLES at similar Mach number and $Re_\tau \simeq 1000$. As no values are computed at the wall, the profiles were here non-dimensionalized using the channel centerline values. This figure shows, for both quantities, a slightly flatter profile for the wmLES compared to



(a) In wall-coordinates.



(b) In outer coordinates.

Figure 3.22: Velocity profile for the wmLES of supersonic channel flow at high Reynolds number using the present scaling (red). The profiles of the wrLES at similar Mach number and $Re_\tau \simeq 1000$ (solid black) and of the incompressible flow DNS at $Re_\tau \simeq 5200$ (Lee and Moser [59], dotted black) are provided for comparison.

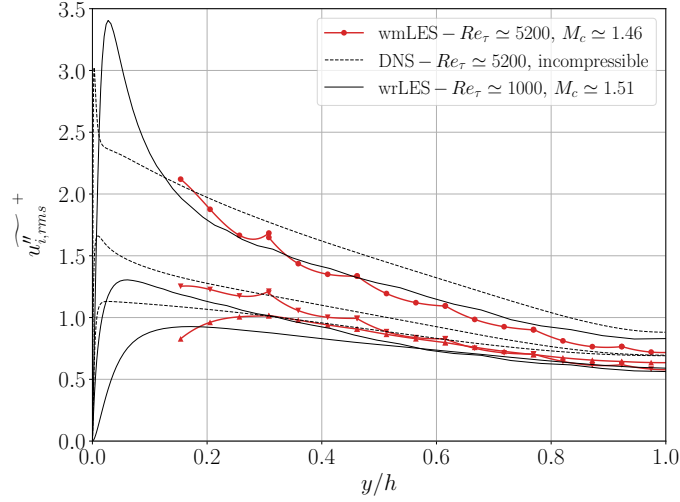
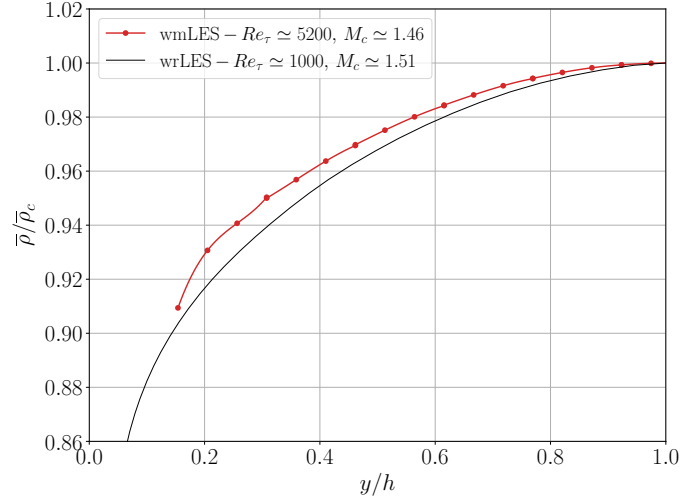


Figure 3.23: Velocity fluctuation profiles for the wmLES of supersonic channel flow at high Reynolds number using the present scaling (red). The profiles of the wrLES at similar Mach number and $Re_\tau \simeq 1000$ (solid black) and of the incompressible flow DNS at $Re_\tau \simeq 5200$ (Lee and Moser [59], dotted black) are provided for comparison.

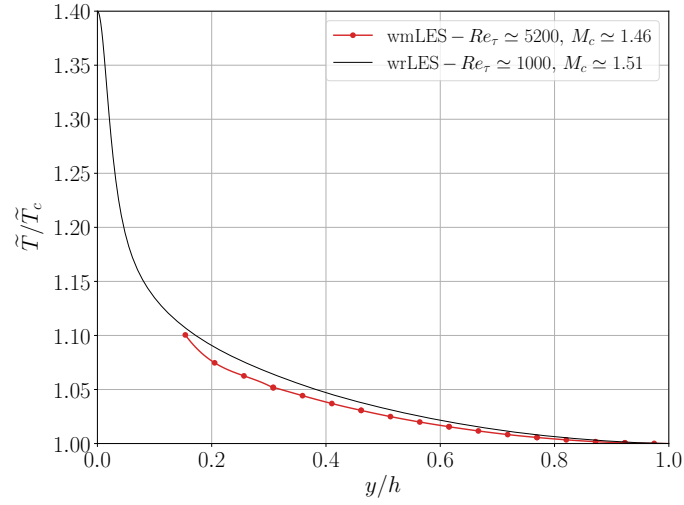
that of the wrLES at a lower Reynolds number. This is expected when increasing the Reynolds number as the near-wall variations will be larger; giving some confidence in the temperature and density profiles obtained in our wmLES.

3.6 Conclusions and perspectives

In this chapter, different scalings that are used to map the mean profiles of compressible flow into their equivalent incompressible flow were recalled. They were simplified with the objective to design a simple quasi-analytical wall model for wall-modeled large eddy simulation (wmLES) of highly compressible flows, and still using the Reichardt formula for the law of the wall in incompressible flows. Then, an additional scaling was proposed, originating from the combination of the existing ones. The accuracy of the obtained wall models was evaluated on three cases of turbulent channel flow at $Re_\tau \simeq 1000$: a low-Mach number subsonic case (thus directly comparable to an incompressible flow), a high-Mach number subsonic case, and finally a supersonic case. For each case, a wall-resolved LES (wrLES) was also performed to serve as reference.



(a) Density



(b) Temperature

Figure 3.24: Density and temperature profiles for the wmLES of supersonic channel flow at high Reynolds number using the present scaling (red). The profiles of the wrLES at similar Mach number and $Re_\tau \approx 1000$ are provided for comparison.

The comparison of the wmLES with their wrLES counterpart showed that the proposed hybrid wall model accurately replicates the wrLES profiles: in terms of

velocity as well as temperature and density. For the velocity fluctuations, no significant differences were observed between the wall models and they all satisfactorily reproduced the wrLES profiles. The error on the mass flow rate in the channel was used as global metric to compare the wall models and it was shown that the model based on the proposed new scaling achieved a consistent error of about 2 % for both the high-subsonic and the supersonic cases; whereas the other scalings produced an increase in the error (by a factor of 1.5 to 2). As expected, the unscaled wall model yielded consistently the worst results. In the low-subsonic regime, using the proposed new scaled version also led to smaller deviations from the reference wrLES data than when using the unscaled version.

The proposed wall model was then further investigated on a channel flow at high Reynolds number of $Re_\tau \simeq 5200$: for the low-subsonic and the supersonic regimes. The low-subsonic wmLES correctly predicts the reference incompressible DNS data, with a deviation on the mass flow-rate of only 0.3 % (also attributed in part to the small compressibility effects). For the supersonic case, the velocity profile was shown to be aligned with its lower Reynolds number counterpart at identical Mach number, which corresponds to the expectations. In terms of fluctuations, both cases gave expected results (in comparison to wrLES at lower Reynolds number and to incompressible DNS at similar Reynolds number). However, the lack of a reference for a supersonic channel at high Reynolds number makes it difficult to provide here a more quantitative analysis of the wmLES performance.

The wall model developed here is used to perform wmLES of supersonic ejectors where the Mach numbers and the Reynolds numbers are comparable to those investigated here. The results of these simulations are presented in the next chapters and will complement the results and observations of previous experimental studies and RANS simulations [55, 57].

The accuracy of the wall model for flows at Mach numbers higher than $M_c \simeq 1.5$ should also be further evaluated. Furthermore, this study only focused on a scaling valid for flows with adiabatic walls. Its extension to flows with other boundary conditions will also have to be investigated.

Large-Eddy Simulation of a planar ejector - Total exergy flux analysis

This chapter is largely inspired by an article submitted to the International Journal of Heat and Fluid Flow, entitled "Analysis of entrainment and mixing in a supersonic air ejector using Large-Eddy Simulation"[26], by R. Debroeyer, T. Toulorge, M. Rasquin, G. Winckelmans, and Y. Bartosiewicz.

4.1 Introduction

In this chapter, a high-fidelity LES of a rectangular supersonic ejector is performed, using periodic boundary conditions in the extruded spanwise direction. First, the simulation setup is presented, the extrusion depth and spatial resolution are then analyzed. The evolution of the boundary layers in the primary nozzle and secondary flow entrance, as well as the shear layers, are examined. The principal averaged fields are also presented and compared to experimental and RANS simulation data. The post-processing tools using exergy fluxes, inspired from the work of Lamberts et al. [55] for RANS simulations and adapted to the LES framework are presented. The local total exergy fluxes are then compared between the RANS and the LES frameworks, with a focus on the turbulent shear stress and turbulent heat flux; which are obtained in LES but only modeled in RANS. This total exergy flux analysis also leads to a new criterion to evaluate the degree of mixing in a supersonic ejector. Finally, the mixing of the flows in a supersonic ejector is shown to be insufficient to optimize the mixing

duct length as the balance between shear stress between streams and wall-friction has a much larger impact on the performance of such a device.

4.2 Simulation setup

4.2.1 Geometry

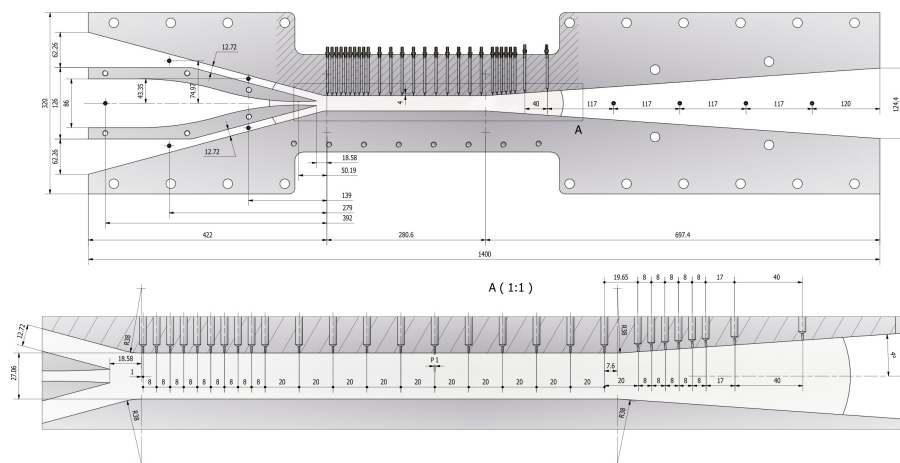
The ejector geometry considered in this study was designed by Lamberts [54] and is shown with its dimensions in Figure 4.1. The main geometric characteristics of this ejector are summarized in Table 4.1. This geometry is extruded in the spanwise direction. Transparent side-walls are fitted on the experimental setup in order to allow flow visualization using shadowgraphy. Pressure sensors are also placed along the top wall of the ejector, and spaced as shown in Figure 4.1(a).

Parameter		Value
Area ratio	H_m/H_t	4.51
Primary nozzle area ratio	H_e/H_t	1.33
Nozzle exit position	NXP/h_m	1.37
Mixing duct length to half-height ratio	L_m/h_m	20.74

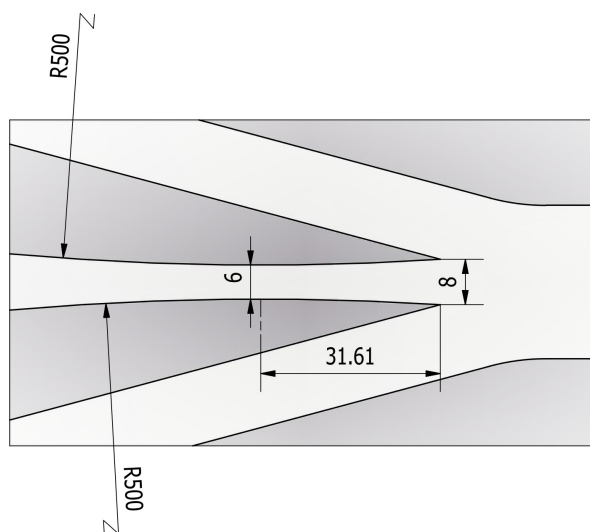
Table 4.1: Main geometric characteristics of the ejector.

4.2.2 Boundary conditions

The primary nozzle inlet is fed with a turbulent channel flow obtained using a co-simulation of a channel flow, thus supplying a time-dependent velocity field at the inlet. This is done in order to obtain the correct profile of turbulence at the primary nozzle exit, thus better capturing the interaction between the primary and secondary flows. Volume source terms, computed using a proportional-integral controller, are used in the periodic channel co-simulation to ensure the values of the total conditions $p_{0,p}$ and $T_{0,p}$. At the secondary flow inlets, the total pressure $p_{0,s}$ is imposed, along with the total temperature $T_{0,s}$. No turbulence is injected at the secondary inlets as those are connected to a plenum in the experiment. Thus, the flow does not have sufficient distance to transition to a fully turbulent flow. In this study, the ratio $T_{0,s}/T_{0,p}$ was taken as unity. The static back-pressure p_{out} is imposed at the ejector exit. The operating point simulated in this study is such that $p_{0,p}/p_{0,s} = 5.133$, $p_{out}/p_{0,s} = 1.232$ and it lies in the on-design regime as the critical pressure was measured experimentally at $p_{out}/p_{0,s} \simeq 1.75$.



(a) Full ejector



(b) Zoom around the primary nozzle exit

Figure 4.1: Dimensional drawing of the geometry, reproduced from Lamberts [54]. All dimensions are in mm.

The walls are assumed adiabatic and the algebraic wall model proposed by Debroeyer et al. [24] is used to impose the wall shear stress τ_w . We thus perform a wall-modeled LES (wmLES). The wall shear stress is computed using the flow quanti-

ties measured at the bottom of the second wall-adjacent element and the interpolation order is increased to $p = 4$ in the first wall-adjacent element, following the recommendation from Frère et al. [31]. This model was shown to accurately predict the velocity profile in adiabatic compressible channel flows with a centerline Mach number up to $M_c \simeq 1.5$ and at friction Reynolds number of $Re_\tau \simeq 5200$. Those values are of the same order of magnitude as those reached in the present ejector application.

4.2.3 Mesh

The mesh is generated using the open-source Gmsh mesh generator [35] and is composed of first-order structured hexahedra in the whole domain, except in the first quarter of the mixing duct, where it is refined around the dividing streamlines in order to better capture the evolution of the shear layers. The mesh in a slice of the domain is shown in Figure 4.2 where only the elements are shown. For third-order elements, as used to compute the solution in this study, there are ten degrees of freedom in triangles and sixteen degrees of freedom in quadrangles. A sponge layer measuring 20 % of the diffuser's length is added at the ejector's end in order to absorb the fluctuations and provide a clean exit condition to the highly fluctuating flow. A damping of the solution is added in the equations in this region and this term is proportional to the difference between the instantaneous solution and its moving time average. Moreover, to achieve the best damping possible, the interpolation order of the elements in the sponge layer progressively decreases from three to zero. This region of the domain will be omitted for all the results presented here. The two-dimensional mesh is extruded in the spanwise direction. The mesh contains a total of 687 thousand elements, resulting in 45 million degrees of freedom and is partitioned using Metis [49] in 888 partitions to reach the desired number of elements per processor.

The near wall resolution obtained with this mesh, normalized by the viscous length $\mu/\sqrt{\rho\tau_w}$, lies between $\Delta x^+ \times y_1^+ \times \Delta z^+ = 80 \times 450 \times 15$ and $120 \times 700 \times 30$ in the mixing duct. whereas for the streamwise and spanwise directions Δx^+ and Δz^+ are the effective resolutions (i.e., the length and depth of the near wall element are each divided by the interpolation order). Since the friction Reynolds number Re_τ is $\mathcal{O}(10^4)$ in the mixing duct, the value of y_1^+ lies in the range where the flow can be assumed to be universal and not yet significantly impacted by the outer flow region, which thus respects the requirements to properly apply the wall model developed in Chapter 3. In the streamwise direction, Δx^+ is low enough to accurately capture the streamwise evolution of the flow, as this value is typical of high quality wmLES. Finally, the spanwise resolution is high compared to the streamwise resolution, but this is imposed

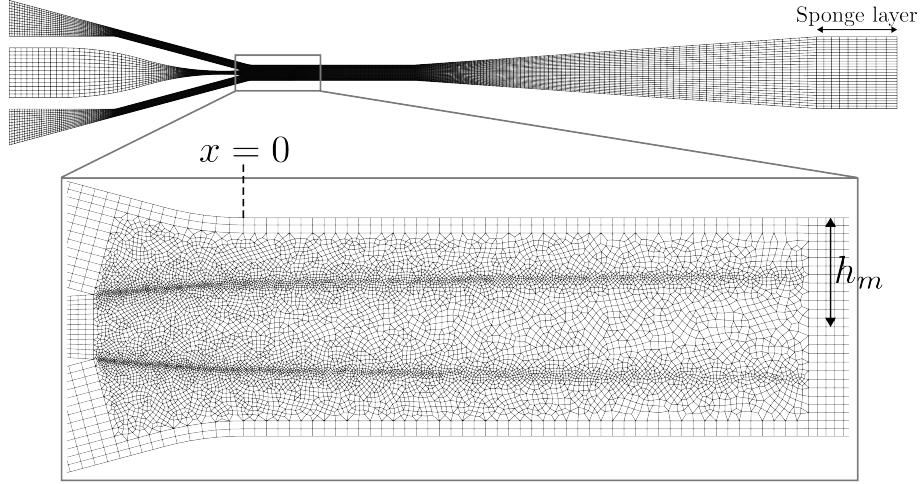


Figure 4.2: Elements of the mesh in a slice of the domain, and zoom in the region near the nozzle exit and with the shear layers.

by the accuracy needed to capture the 3-D dynamics in the shear layers and where isotropic elements are thus preferred. Indeed, since the 2-D mesh was extruded in the spanwise direction, Δz is uniform over the entire mesh. Finally, and most importantly, the x and y resolutions are taken much finer in the regions near the nozzle exit and within the shear layers than in the region near the walls, see the zoom in Figure 4.2, with an effective resolution of at least $\theta/6$ where

$$\theta = \int_{y_1}^{y_2} \frac{\rho u - (\rho u)_1}{(\rho u)_2 - (\rho u)_1} \left(1 - \frac{u - u_1}{u_2 - u_1} \right) dy, \quad (4.1)$$

where y_1 is the centerline of the ejector and y_2 is a point located within the secondary flow potential core. This is achievable thanks to the combination of structured and unstructured elements.

4.2.4 Extrusion depth

The presented geometry is extruded in the spanwise direction and a periodic boundary condition is used. However, in order to approximate an infinite domain, the two-point correlation of the spanwise velocity fluctuations (recall that the mean

spanwise velocity is zero), defined as

$$f_{ww}(x, y, \Delta z) = \frac{\overline{w(x, y, z + \Delta z)w(x, y, z)}}{\overline{w(x, y, z)w(x, y, z)}}, \quad (4.2)$$

should fall close to zero when $|\Delta z|$ approaches $L_z/2$. Because of the periodicity, we obtain that $f_{ww}(x, y, -\Delta z) = f_{ww}(x, y, \Delta z)$.

Moreover, it is considered that an extrusion depth of around eight times the integral length scale,

$$\mathcal{L}(x, y) = \int_0^{L_z/2} f_{ww}(x, y, z) dz \quad (4.3)$$

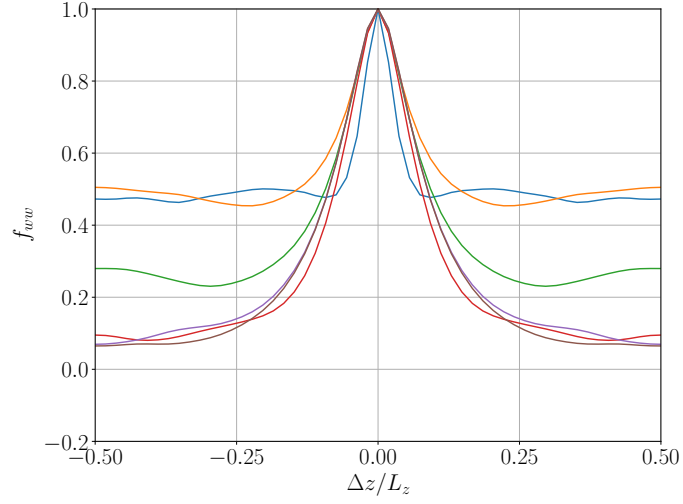
should be used [69].

In this study, two extrusion depths were considered : $L_z/h_m = 0.887$ (case I) and $L_z/h_m = 1.478$ (case II). Note that the spanwise depth of the experimental setup with side walls was $L_z/h_m = 3.695$.

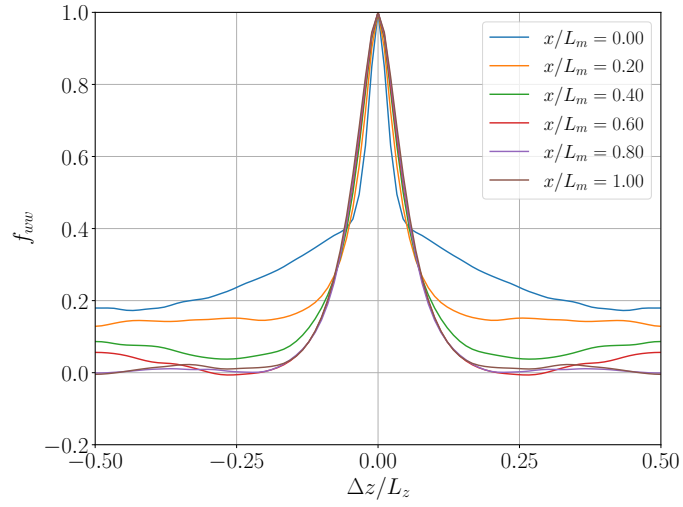
The two-point correlation functions, as well as the integral length scale were computed along the top and bottom dividing streamlines (those being obtained using the time-averaged mean flow) for both cases, and are shown in Figures 4.3, 4.4, and 4.5. It is seen that the smallest extrusion depth of case I does not allow for the two-point correlation function to fall close to zero within the half span of the ejector. Moreover, the integral length scale is at best, for $x/L_m \geq 0.6$, equal to the recommended $1/8$ of the extrusion depth, and is significantly larger than that for more than half of the mixing duct length. On the other hand, the two-point correlation function of case II is close to zero from the beginning of the mixing duct, and the region of the flow where the integral length scale exceeds $1/8$ of the extrusion depth is limited; and it does not exceed 20 %. This extrusion depth will be thus considered satisfactory to properly capture the spectrum of the turbulent flow in the shear layers. In the rest of this study, the results are only presented for case II. For this case, the simulation of one characteristic time $t_c = \frac{L_m}{\bar{u}_e} \simeq 0.6$ ms, $\bar{u}_e$ being the cross-section averaged flow velocity at the primary nozzle exit and corresponding to a Mach number of around $\overline{M}_e = 1.7$, takes around 2 days on 1792 AMD EPYC 7763 cores on the “Lucia” supercomputer (two cores per partition and 16 partitions for the channel flow co-simulation).

4.3 Results

This section presents the results of the LES with the largest extrusion depth. Both instantaneous and averaged fields are analyzed. The averaging operation is performed



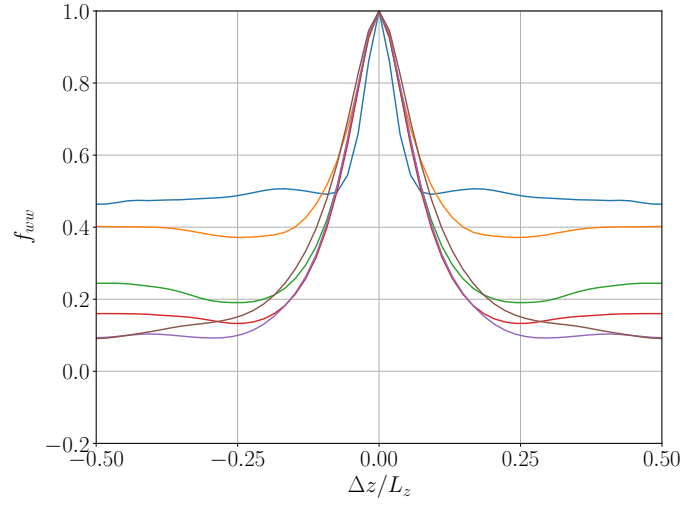
(a) Case I



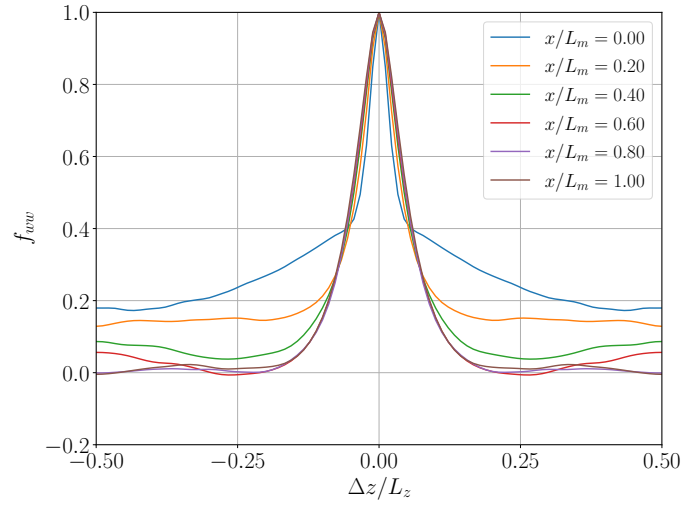
(b) Case II

Figure 4.3: Two-point correlation function at different streamwise locations along the top dividing streamline in the mixing chamber.

in the spanwise direction and in time, over around $4.5 t_c$. The constraint on the CFL number yields a time step of $\Delta t/t_c \simeq 4 \times 10^{-4}$.



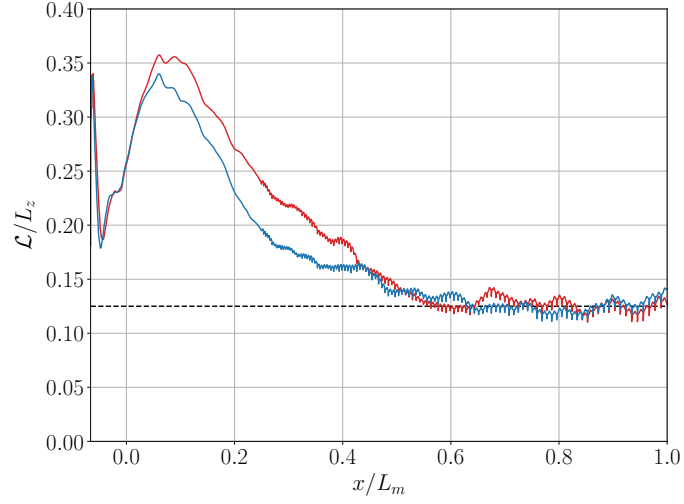
(a) Case I



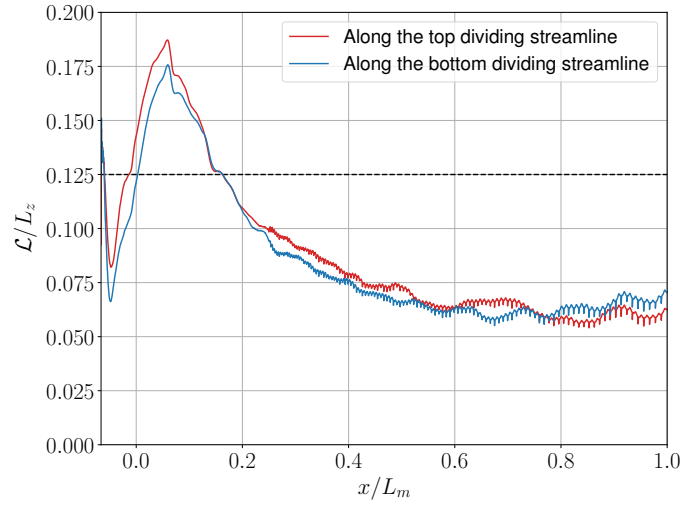
(b) Case II

Figure 4.4: Two-point correlation function at different streamwise locations along the bottom dividing streamline in the mixing chamber.

The LES averaged results are compared to those of a RANS simulation and to the experimental results obtained by Lamberts [54]. The present RANS simulation



(a) Case I



(b) Case II

Figure 4.5: Ratio between the integral length scale $\mathcal{L}$ and the extrusion depth L_z at different streamwise locations along the top and bottom dividing streamlines in the mixing chamber.

results were also verified to match those of Lamberts et al. [55], using a similar setup.

The experimental results include mass flow-rate and pressure measurements, as well as flow visualization, using a shadowgraphy setup.

The entrainment ratio computed in the LES is $ER = 0.532$ with a peak-to-peak variation of at most 1.3 % over the averaging period. Since the flow quantities are averaged in the spanwise direction, the mass flow-rates are computed per unit span and are thus obtained in $[\text{kg}/(\text{s m})]$. This entrainment ratio is 0.5 % higher than that obtained in the RANS simulation ($ER^{RANS} = 0.529$), which thus slightly worsens the agreement with the experimental entrainment ratio of $ER^{EXP} = 0.486$ obtained by Lamberts [54]; this is however not significant. The overestimation of the entrainment ratio by both numerical frameworks is attributed to the presence of sidewalls in the experiment, which create additional friction and secondary flows, not present in the 2D or spanwise periodic simulations.

4.3.1 Instantaneous fields

The temperature field in a slice of the ejector is presented in Figure 4.6. The field visualizations are obtained using the high-order solution visualization plugin from Paraview, developed by Rasquin et al. [70]. This figure shows the main structures of the flow: the two expansion fans emanating from the primary nozzle lips, due to the under-expanded nature of the primary flow; the growth of the primary supersonic jet with the surrounding turbulent shear layers responsible for the entrainment; and finally the secondary shock train in the diffuser. The flow downstream of the shock is asymmetrical relatively to the ejector centerline. This is a well-known effect, largely studied in classical converging-diverging nozzles (see for example Verma and Manisankar [86], Telega et al. [76]). This flow asymmetry was also observed in supersonic ejectors by Croquer et al. [19].

The density gradient magnitude in a cross-plane of the ejector is presented in Figure 4.7. This figure even better shows the expansion fans at the primary nozzle exit and the subsequent reflections caused by their interaction with the shear layers. These shear layers are observed to present Kelvin-Helmholtz instabilities from a very small distance downstream of the NXP. They then grow very quickly by vortex pairing and they merge at around $x \simeq 0.5 L_m$. The secondary shock in the diffuser is also seen to be captured within one element.

This figure also shows the state of the boundary layers coming in the mixing duct. More specifically, the boundary layers of the primary flow appear to be laminar whereas those of the secondary flow appear to be turbulent. To better evaluate the

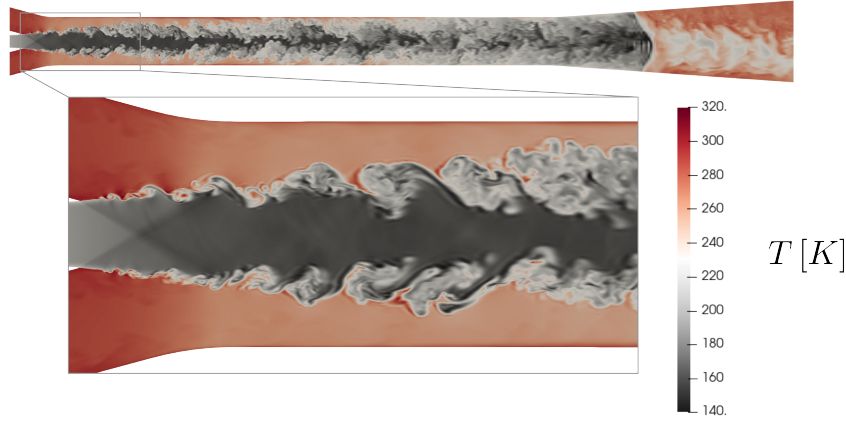


Figure 4.6: Temperature field in a cross-plane, and zoom in the part close to the primary nozzle exit.

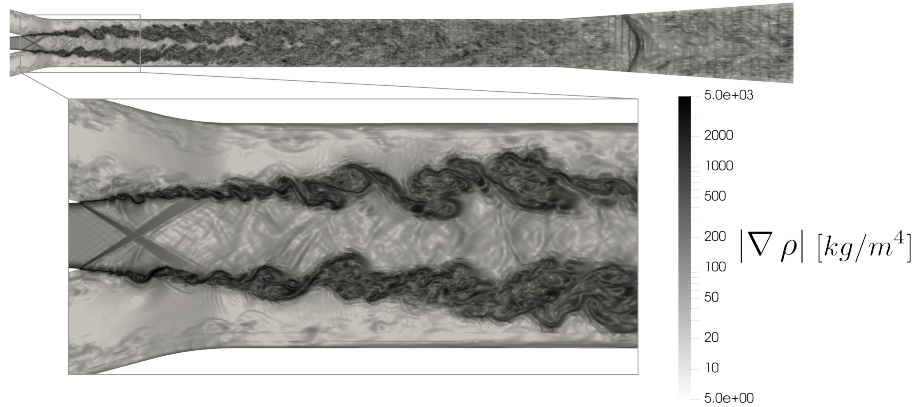


Figure 4.7: Magnitude of the density gradient field in a cross-plane, and zoom in the part close to the primary nozzle exist.

state of the boundary layers, Figure 4.8(a) shows the instantaneous wall shear stress on the top wall of the top secondary flow entrance channel and on the top wall of the primary nozzle. The streaks in the wall shear stress present at the inlet of the primary nozzle (from the co-simulation of a fully developed turbulent channel flow used at inlet) and characteristic of turbulent boundary layers disappear within the converging section. This is due to the very high acceleration in the primary nozzle

(up to $M \simeq 1.7$) causing the relaminarization of the boundary layers. This effect was previously studied by Bader et al. [2] and Debroeyer et al. [25] for example, and shows that it is not essential to feed the primary nozzle with a turbulent inflow, as the boundary layers will relaminarize in these conditions.

On the other hand, no turbulent inflow condition is imposed at the inlets of the secondary flows. Indeed, the entrained flow is a result of the LES, not an input. The entrained flow that enters the simulation is not yet turbulent, and the wall shear stress is thus devoid of streaks. However, along the very long inlet channel (around 18 times its half-height), the boundary layers naturally become turbulent. In the end part of this channel, the boundary layers are seen to be fully turbulent. This is confirmed in Figure 4.8(b) where the three-dimensional structures at the end of the secondary flow entrance are clearly seen to emanate from the wall. We also see that the wall-attached structures disappear in the primary nozzle when close to the exit, confirming that the boundary layers have completely relaminarized.

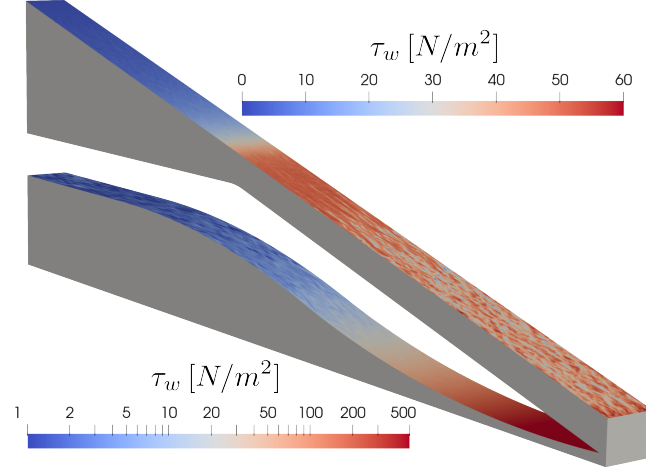
Figure 4.9 shows the Q-criterion colored by the spanwise vorticity at the beginning of the mixing duct in the top shear layer. This clearly shows the appearance of three-dimensional instabilities at the very beginning of this shear layer, which then rapidly transitions to turbulence, producing a fully turbulent shear layer in the majority of the mixing duct.

From Figures 4.8 and 4.9, it can be concluded that, contrary to what was proposed by Croquer et al. [19], the rapid transition of the shear layers to fully turbulent shear layers is not conditional to the injection of turbulence in the primary nozzle.

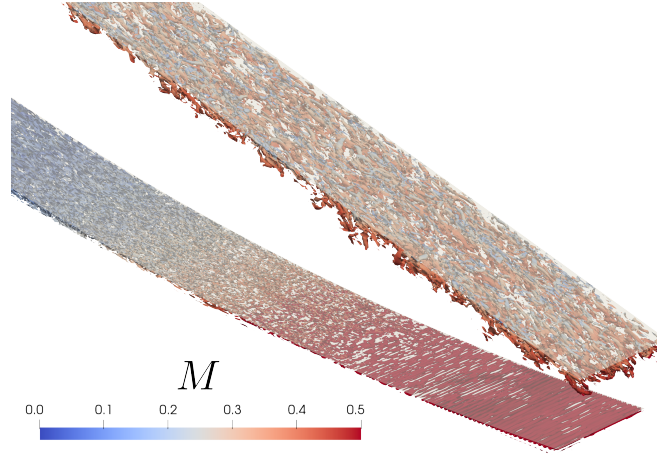
4.3.2 Averaged fields

The Mach number field of the averaged flow in the mixing duct is shown in Figure 4.10. This highlights again the expansion fans at the primary nozzle exit. The dividing streamlines are shown in black on the Mach number field. The position of those dividing streamlines is computed by performing a Runge-Kutta 4 integration on the time and space averaged momentum field. The figure also shows the sonic lines, i.e. where $\overline{M} = 1$ in red, highlighting how the secondary stream progressively becomes supersonic along the mixing duct.

This Mach number field is then averaged for each individual stream in order to obtain an equivalent one dimensional view of both flows as well as the compound one. Each Mach number is obtained by averaging the Mach number across that said fluid



(a) Wall shear stress on the top walls of the top secondary flow entrance and of the primary nozzle. For improved readability, the color mapping on the primary nozzle wall is logarithmic.



(b) Iso-contour of the Q-criterion along the top walls of the top secondary flow entrance and of the primary nozzle colored by the Mach number, and zoomed in on the end of the primary and secondary flow ducts.

Figure 4.8: Boundary layers in the primary and secondary flow inlets.

region using

$$\overline{M}_n = \frac{1}{H_n} \int_{H_n} \overline{M} dy, \quad (4.4)$$

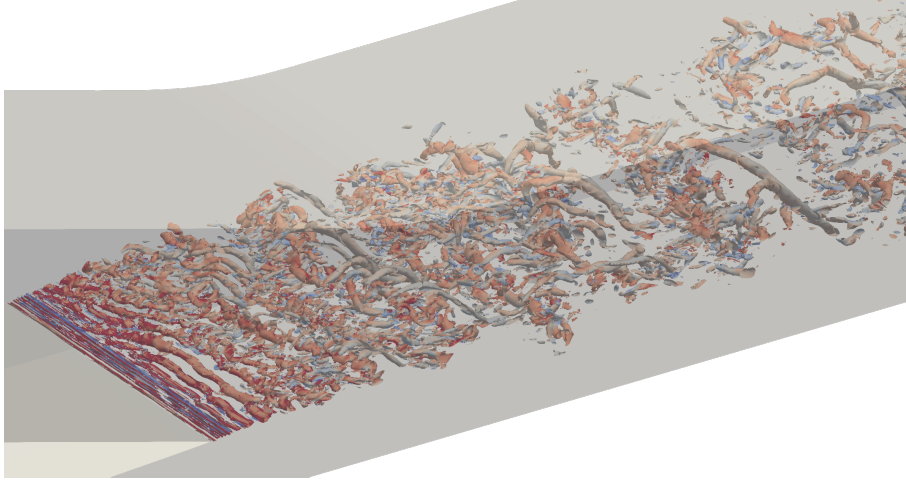


Figure 4.9: Q-criterion in the top shear layer at the beginning of the mixing duct, colored by the spanwise vorticity component.

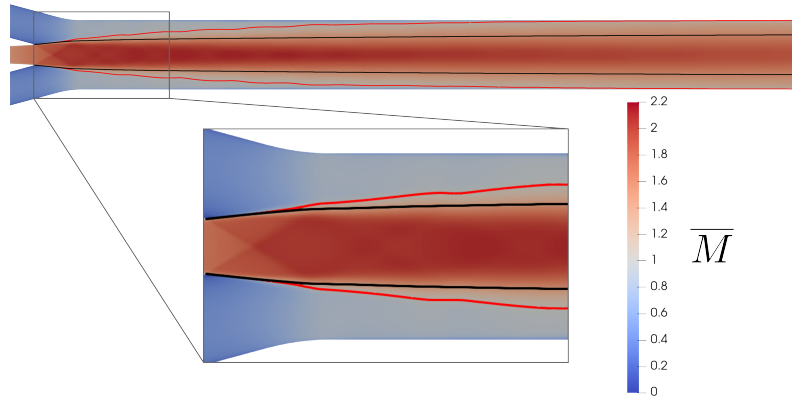


Figure 4.10: Mach number field of the averaged flow in the mixing duct. The dividing streamlines are shown in black and the sonic lines, i.e. where $\overline{M} = 1$ are shown in red.

where H_n is the flow section considered and is delimited by the dividing streamlines (and by the mixing duct wall for each secondary flow). The results are presented in Figure 4.11. For the secondary flows, the results are well superimposed on each other all along the mixing duct. This is “as expected”, and also opposed to what was reported by Croquer et al. [19], where a significant difference in averaged Mach numbers was

observed between the top and bottom secondary flows, and from the very beginning of the mixing duct. Here, the only observed asymmetry is that in the diffuser, downstream of the exit shock (recall Figure 4.6); and it does not affect the flow in the mixing duct.

An equivalent, or a compound, isentropic Mach number $\overline{M}_{eq}^{is}$ is derived from the compound choking theory [57]. In the case of a perfect gas, it is defined as

$$\overline{M}_{eq}^{is} = \left[\frac{A}{\sum_n \frac{A_n}{M_n^2}} \right]^{1/2}. \quad (4.5)$$

where $A = \sum_n A_n$ is the total flow cross-section and A_n is replaced by H_n for the case of a two-dimensional ejector. This quantity has also been added to Figure 4.11. Similarly to classical nozzle flows, the flow is compound-choked when the equivalent Mach number reaches unity and is compound-supersonic (respectively compound-subsonic) when $\overline{M}_{eq}^{is} > 1$ (respectively $\overline{M}_{eq}^{is} < 1$).

Figure 4.11 shows that $\overline{M}_{eq}^{is}$ is significantly higher than the Mach number $\overline{M}_2$ of the secondary flows. Moreover, it is seen to reach unity slightly before the geometrical throat of the ejector, located at $x/L_m = 0$, where the secondary flow is still largely subsonic. The ejector is then compound-choked well upstream the location where the

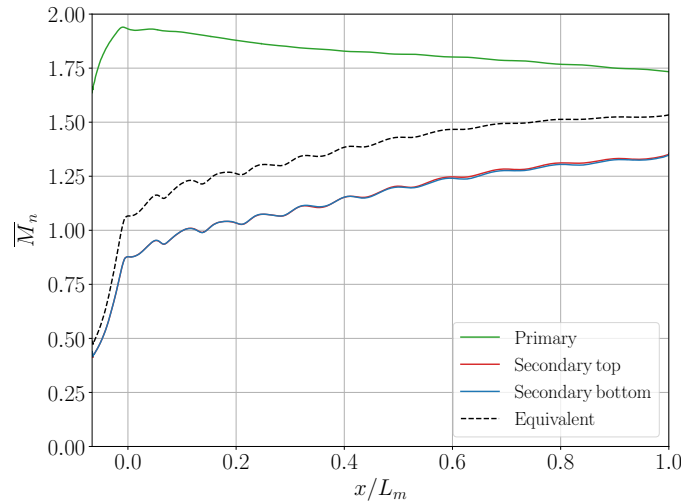
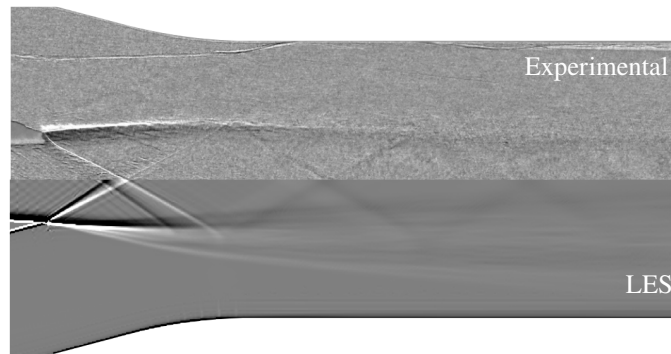


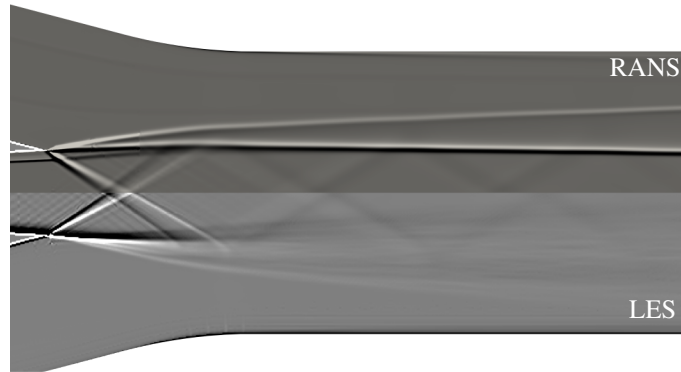
Figure 4.11: One-dimensional averaged Mach numbers in the mixing duct. The equivalent isentropic Mach number $\overline{M}_{eq}^{is}$ is also shown.

secondary Mach number becomes unity. This shift of the choking section, upstream of the geometrical throat, is attributed to the shear between the flows overwhelming the wall-friction as shown by Van den Berghe et al. [82].

Figure 4.12(a) shows the comparison between an experimental shadowgraphy image and a numerical one obtained using the Laplacian of the time and spanwise-



(a) Comparison between the experimental shadowgraph image (upper-half) and that computed using the LES results (lower-half).



(b) Comparison between the shadowgraphy image computed using the LES (upper-half) and the RANS (lower-half) results.

Figure 4.12: Shadowgraphy image in the region near the exit of the primary nozzle.

averaged density field. This shows an excellent agreement between the LES and the experimental results regarding the position of the primary expansion fans generated by the primary supersonic jet, as well as the secondary expansion and compression waves. It must be noted that those reflected waves are not visible in instantaneous cross-section views such as those previously shown, but they appear when a time averaging is performed; the waves reflection pattern also agrees well with the flow visualizations. Moreover, the experimental shadowgraphy image shows a contraction of the primary jet after its initial expansion. This is also observed in the image obtained numerically, where this contraction is seen to correspond to the inner edge of the shear layer.

In addition, the comparison between numerical shadowgraphy images obtained using the RANS and the LES results is shown in Figure 4.12(b). This shows that the position of the first expansion wave matches almost perfectly between both numerical frameworks. However, after the first interaction of the reflected wave with the shear layer, a shift in the position of the expansion waves appears. This is caused by the differences in position of the shear layer, which further amplifies the shift between numerical results at each reflection.

To study the impact of characteristics of the secondary flow on the behavior of the beginning of the shear layer, Figure 4.13 compares the normalized average velocity profile at the exit of the secondary entrance pipe obtained using RANS and LES. This corresponds to the velocity profile along a line emanating from the (upper) NXP and perpendicular to the secondary flow main axis. This figure shows that, despite the significant differences in numerical frameworks, both velocity profiles are very similar, which suggests that the differences in shear layer behavior are primarily due to the scale-resolving aspect of the LES, combined with the appropriate mesh.

The wall pressure obtained in the LES is compared to that obtained in the RANS simulation in Figure 4.14. We observe a significant variation of the wall pressure in the mixing duct, which opposes the early simplified model assumptions of a constant pressure mixing. We note that this assumption was removed from the model proposed by Metsue et al. [61], which allowed to accurately predict the critical pressure and the off-design portion of the characteristic curve, without any additional calibration coefficients.

Figure 4.14 also shows an over-prediction of the wall pressure by the RANS simulation. This is attributed to a slightly higher expansion of the primary flow in the LES, resulting in a smaller cross-section area available for the secondary flow. This effect is further amplified by a more rapid shear layer growth. This then forces

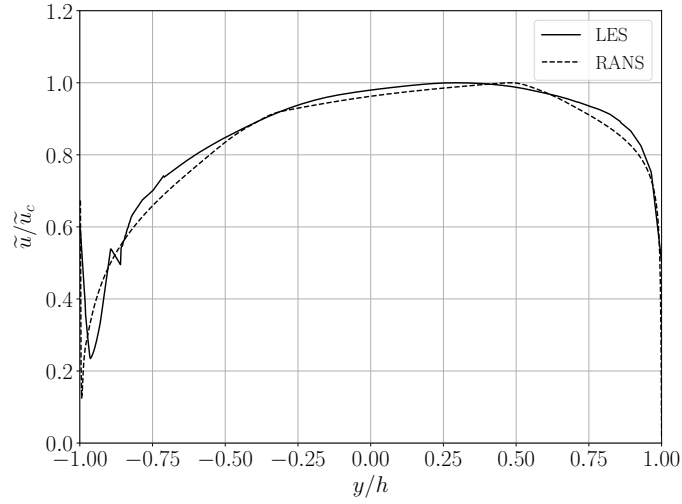


Figure 4.13: Normalized average velocity profile at the secondary entrance exit. Comparison between the LES (solid line) and the RANS (dashed line) simulation results. $y/h = -1$ corresponds to the NXP, whereas $y/h = 1$ is the upper wall of the ejector.

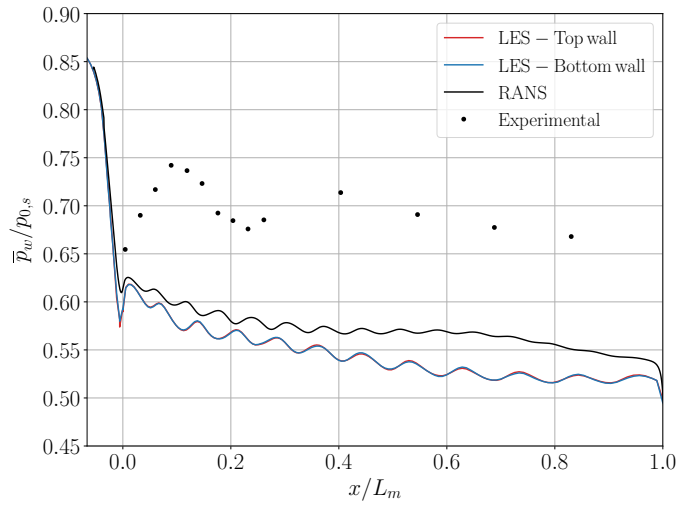


Figure 4.14: Wall pressure in the mixing chamber. Comparison between the LES and the RANS simulation results.

the secondary flow to accelerate more in the LES, leading to a lower pressure. A similar effect was observed by Croquer et al. [19] when comparing the wall-pressure

obtained in RANS simulations and in LES. We recall that the turbulent entrainment and mixing is solely modeled in RANS (through the closure model for the effective turbulent viscosity), whereas the turbulent structures responsible for the entrainment and mixing are well resolved in the LES.

We also note that both numerical frameworks largely underestimate the results of the experiment with side walls of Lamberts [54], the LES being slightly worse than the RANS. Lamberts [54] however partially solved this discrepancy by performing a three-dimensional RANS simulation that took into account the side walls of the experiment. Given the very high computational cost and complexity of performing such a simulation in the LES framework, the effect of side walls was not investigated further in the present study.

Figure 4.15 shows the evolution of the averaged temperature in the primary and secondary flows. This shows the expected decrease of the secondary flow temperature as it accelerates, whereas the primary flow temperature increases, after an initial decrease caused by the expansion fans at the primary nozzle exit.

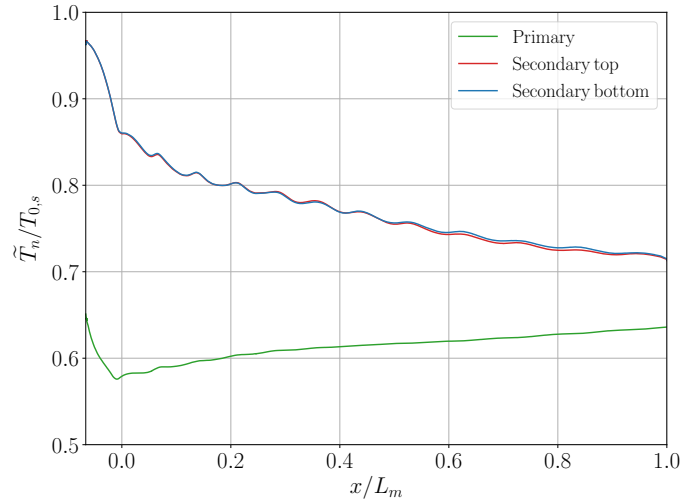


Figure 4.15: One-dimensional averaged temperature in the mixing duct.

Finally, Figure 4.16 shows the evolution of the averaged total exergy in each stream. As a reminder, total exergy is defined as the maximum mechanical work that can be

extracted from a system and is expressed as

$$\xi_0 = (h + k) - h_{ref} - T_{ref} (s - s_{ref}), \quad (4.6)$$

where $h + k$ is the total enthalpy h_0 (with h the enthalpy and k the kinetic energy), s is the entropy (with $T ds = dh - dp/\rho$), and the subscript ref represents an arbitrary reference state at rest (i.e. at total conditions). In this application, the reference state is chosen to be the total conditions of the secondary flow at its inlet: $p_{ref} = p_{0,s}$ and $T_{ref} = T_{0,s}$; and thus $h_{ref} = c_p T_{0,s}$ and $\xi_{0,s} = 0$. The total exergy of the primary flow at its inlet is noted $\xi_{0,p}$.

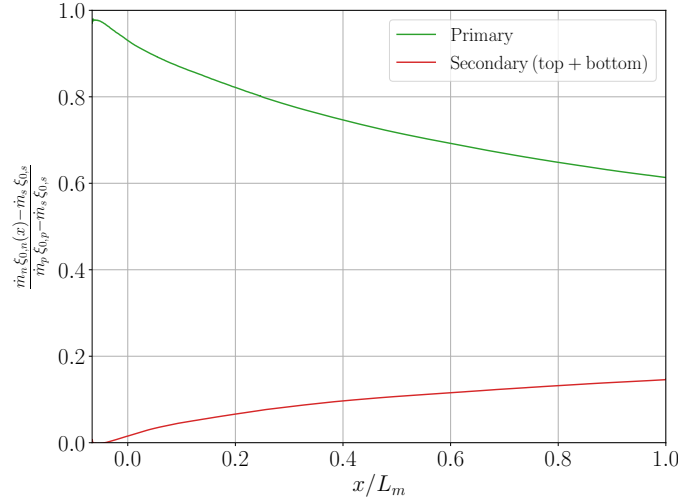


Figure 4.16: Evolution of the normalized total exergy in the mixing duct.

As the primary and secondary mass flow-rates in the ejector are different, this figure presents the average convective flux of total exergy for both streams

$$\dot{m} \xi_0(x) = \int_H \overline{\rho u \xi_0} dy \quad [\text{W/m}] \quad (4.7)$$

compared to the secondary flow inlet and normalized by $\dot{m}_p \xi_{0,p} - \dot{m}_s \xi_{0,s}$. In the present case, out of the 40 % total exergy lost by the primary flow, only 15 % was transferred to the secondary flow, meaning that 25 % was lost by irreversibilities.

4.3.3 Flux of total exergy through the dividing streamlines

In order to study the exchanges between both streams along the mixing chamber, it was proposed by Lamberts et al. [55] to use total exergy tubes as these allow to visualize how the useful mechanical work flows in the ejector. A total exergy tube is defined such that no flux goes through its boundary. The exergy flux leaving the tube thus only differs from the exergy flux entering the tube because of the internal sink terms coming from irreversibilities. The particular interest in using the total exergy to perform this analysis is that the sink terms are by definition always negative, such that within a total exergy tube, the transported total exergy can only decrease. The evolution equation for the total exergy must thus be established to properly split the flux terms and the sink terms.

From the definition of Equation (4.6), the differential of the total exergy is

$$d\xi_0 = d(h + k) - T_{ref} ds. \quad (4.8)$$

Using the conservation equations for the enthalpy, the kinetic energy and the entropy reminded in Chapter 2, the conservation equation for the total exergy is obtained as

$$\frac{\partial}{\partial t} (\rho \xi_0) + \frac{\partial}{\partial x_j} (\rho \xi_0 u_j) = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (u_i \tau_{ij}) - \frac{T_{ref}}{T} \tau_{ij} d_{ij} - \left(1 - \frac{T_{ref}}{T}\right) \frac{\partial q_j}{\partial x_j}. \quad (4.9)$$

To properly analyze the steady-state mixing in the ejector, this equation must be averaged in time and in the homogeneous spanwise direction, which leads to

$$\frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{\xi}_0 \tilde{u}_j + \overline{\rho \xi_0'' u_j''} - \overline{u_i \tau_{ij}} + \bar{q}_j - T_{ref} \frac{1}{T} \bar{q}_j \right] = -T_{ref} \left[\overline{\frac{1}{T} \tau_{ij} d_{ij}} - \overline{\frac{1}{T^2} q_j \frac{\partial T}{\partial x_j}} \right], \quad (4.10)$$

where the averaged total exergy, in the case of a perfect gas, is

$$\tilde{\xi}_0 = c_p \left(\tilde{T} - T_{ref} \right) + \frac{\tilde{u}_k \tilde{u}_k}{2} + \frac{\overline{u_k'' u_k''}}{2} - T_{ref} (\tilde{s} - s_{ref}), \quad (4.11)$$

and $\xi_0'' = \xi_0 - \tilde{\xi}_0$ by definition. The total exergy-velocity correlation arising from this averaging can be decomposed in its components:

$$\overline{\rho \xi_0'' u_j''} = \overline{\rho c_p T'' u_j''} + \overline{\tilde{u}_i \rho u_i'' u_j''} + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} - T_{ref} \overline{\rho s'' u_j''}. \quad (4.12)$$

Entropy being non-linear with respect to the other state variables, the entropy-velocity correlation cannot be further simplified. The final averaged conservation equation can

then be written:

$$\begin{aligned} \frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{\xi}_0 \tilde{u}_j + \overline{\rho c_p T'' u_j''} + \tilde{u}_i \overline{\rho u_i'' u_j''} + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} - T_{ref} \overline{\rho s'' u_j''} \right. \\ \left. - \overline{u_i \tau_{ij}} + \bar{q}_j - T_{ref} \overline{\frac{1}{T} q_j} \right] = -T_{ref} \left[\overline{\frac{1}{T} \tau_{ij} d_{ij}} - \overline{\frac{1}{T^2} q_j \frac{\partial T}{\partial x_j}} \right]. \end{aligned} \quad (4.13)$$

The terms on the right-hand side are sink terms and are linked to the entropy production

a) due to the viscous dissipation

$$\frac{\Phi}{T} = \frac{1}{T} \tau_{ij} d_{ij} = \frac{1}{T} \left(2\mu d_{ij}^d d_{ij}^d + \mu_v (d_{kk})^2 \right) \geq 0, \quad (4.14)$$

b) due to the heat conduction

$$\frac{\Phi_\Theta}{T^2} = -\frac{1}{T^2} q_j \frac{\partial T}{\partial x_j} = \frac{k}{T^2} \frac{\partial T}{\partial x_j} \frac{\partial T}{\partial x_j} \geq 0. \quad (4.15)$$

The terms on the left-hand side provide the expression for the total exergy flux, noted $\bar{F}_{\xi_0,j}$:

$$\begin{aligned} \bar{F}_{\xi_0,j} = \bar{\rho} \tilde{\xi}_0 \tilde{u}_j + \overline{\rho c_p T'' u_j''} + \tilde{u}_i \overline{\rho u_i'' u_j''} + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} - T_{ref} \overline{\rho s'' u_j''} \\ - \overline{u_i \tau_{ij}} + \bar{q}_j - T_{ref} \overline{\frac{1}{T} q_j}. \end{aligned} \quad (4.16)$$

We also recall that $\bar{\tau}_{ij}^t = -\overline{\rho u_i'' u_j''}$ and $\bar{q}_j^t = \overline{\rho c_p T'' u_j''}$ are defined as being the turbulent shear stresses and heat fluxes, respectively. The averaged equation can thus also be written as

$$\begin{aligned} \frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{\xi}_0 \tilde{u}_j - \left(\overline{u_i \tau_{ij}} + \tilde{u}_i \bar{\tau}_{ij}^t \right) + \left(\bar{q}_j + \bar{q}_j^t - T_{ref} \overline{\frac{1}{T} q_j} \right) + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} \right. \\ \left. - T_{ref} \overline{\rho s'' u_j''} \right] = -T_{ref} \left[\overline{\frac{1}{T} \tau_{ij} d_{ij}} - \overline{\frac{1}{T^2} q_j \frac{\partial T}{\partial x_j}} \right]. \end{aligned} \quad (4.17)$$

The transfer of total exergy between both streams can be quantified by computing the local flux of total exergy through the dividing streamline. In what follows, the flux across the dividing streamline will be noted:

$$F_{\xi_0} = \bar{F}_{\xi_0,j} n_j, \quad (4.18)$$

where n_j is the j -th component of the unitary normal vector to the dividing streamline, pointing towards the secondary flow. A positive flux will thus represent a transfer from the primary flow to the secondary flow. A schematic representation of the flux of total exergy through the dividing streamline is shown in Figure 4.17.

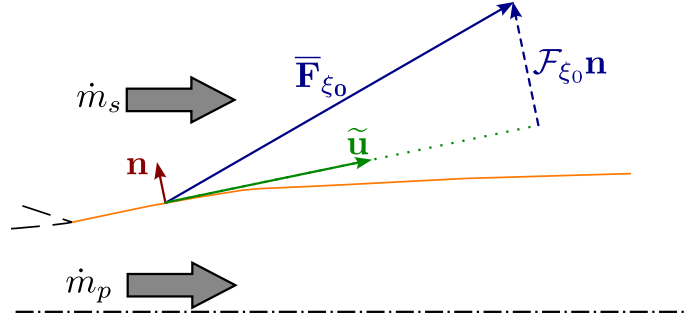


Figure 4.17: Schematic representation of the total exergy flux through the dividing streamline at a local position.

By definition of the dividing streamline, the convective component of the total exergy flux is zero (since $\tilde{u}_j n_j = 0$). The total exergy flux can be split into three basic components : the flux of

(1) enthalpy:

$$\mathcal{F}_h = \left[\bar{q}_j + \bar{q}_j^t \right] n_j , \quad (4.19)$$

(2) kinetic energy:

$$\mathcal{F}_k = \left[- \left(\overline{u_i \tau_{ij}} + \tilde{u}_i \bar{\tau}_{ij}^t \right) + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} \right] n_j , \quad (4.20)$$

(3) entropy:

$$\mathcal{F}_s = -T_{ref} \left[\frac{1}{T} \bar{q}_j + \overline{\rho s'' u_j''} \right] n_j . \quad (4.21)$$

The flux of total exergy is then obtained as $\mathcal{F}_{\xi_0} = \mathcal{F}_h + \mathcal{F}_k + \mathcal{F}_s$. The flux of total exergy due to the flux of total enthalpy (flux of enthalpy and flux of kinetic energy) can also be considered, by defining $\mathcal{F}_{h_0} = \mathcal{F}_h + \mathcal{F}_k$. It is also convenient to define the flux of total exergy only due to thermal effects, i.e. $\mathcal{F}_q = \mathcal{F}_h + \mathcal{F}_s$.

It should be noted that two of the fluxes, namely the enthalpy and kinetic energy fluxes, are independent from the *arbitrarily* chosen reference state, whereas the entropy flux is not. Thus, to properly analyze the total exergy fluxes within an ejector

independently of the choice of T_{ref} , these two contributions should be treated separately: on one hand, the enthalpy and kinetic energy fluxes, which constitute the total enthalpy flux, and on the other hand the entropy flux. However, this splitting of the fluxes maintains the monotonicity of the source terms within the corresponding tube, which was one of the particular interests of using total exergy as quantity of interest. Indeed, no source or sink term is present in the total enthalpy equation (see Equation (2.11)), whereas the positive source terms of the entropy equation (Equation (2.9)) are the same as the sink terms of the total exergy equation, after being multiplied by the factor $-T_{ref}$. The choice of the reference state is dictated by the application of the supersonic ejector and the entropy flux is scaled accordingly to then be added to the total enthalpy flux, thus yielding the total exergy flux. In this case, as the ejector is studied as a standalone device, the reference state is chosen as the total conditions of the secondary flow so that it enters the ejector with a zero total exergy.

In the following, a non-dimensional form of each flux will be reported in the figures, defined as:

$$f_* = \frac{H_m}{\dot{m}_p \xi_{0,p}} F_* . \quad (4.22)$$

All total exergy flux components are compared to those obtained in the RANS simulation, and with the corresponding expressions, developed in Appendix A, in Figure 4.18. The RANS simulation results were also observed to match those obtained by Lamberts et al. [55] using a similar setup. We observed large discrepancies between the results from both frameworks. This is especially true for the first half of the mixing duct, where the peak values captured by the LES at the very beginning of the mixing duct have significantly higher amplitudes and spread over a larger axial area than those observed in the RANS. Moreover, a second bump in the kinetic energy flux is observed around $x/L_m \simeq 0.2$, and that is not captured in RANS. In the second half of the mixing duct, all components of the total exergy flux show a good agreement between LES and RANS.

To better understand the large differences observed between both frameworks, the turbulent shear stress and turbulent heat flux most responsible for exergy transfer are shown at several streamwise positions, for the lower half of the mixing duct, in Figure 4.19. For the sake of clarity, these streamwise locations are also shown in the instantaneous density field in Figure 4.20. These turbulent quantities are normalized similarly to the total exergy flux, i.e. using $\frac{\dot{m}_p \xi_{0,p}}{\bar{u}_e H_m}$ for the turbulent shear stress and $\frac{\dot{m}_p \xi_{0,p}}{H_m}$ for the turbulent heat flux. This normalization is consistent with the normalization of the local fluxes defined in Equation (4.22) and used in Figure 4.18. The LES results

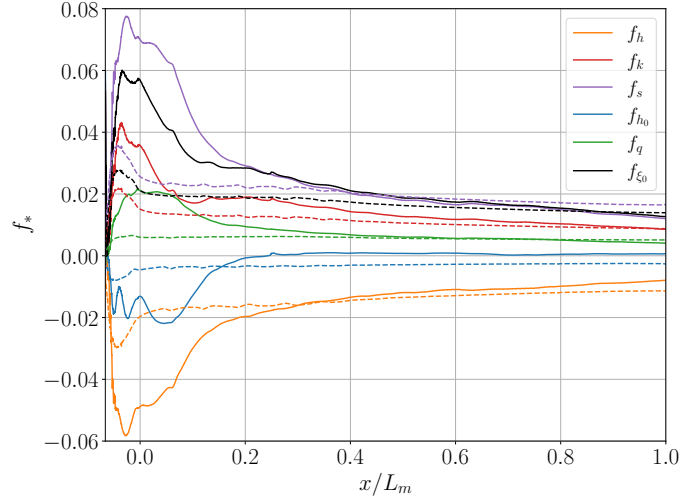


Figure 4.18: Non-dimensional fluxes through the dividing streamlines computed from the LES (solid). The fluxes computed from the RANS simulation (dash) are also provided for comparison.

presented in Figure 4.19 are not shown below the location where the wall shear stress was evaluated using the wall model.

These figures highlight the significant differences between the RANS simulation results where the turbulent shear stress and heat fluxes are modeled and the LES results where they are captured. Even though the profiles predicted by the RANS simulation follow the trend of those obtained in the LES, their amplitude is largely underestimated up to $x/L_m \simeq 0.4$, and for both the turbulent shear stress and the turbulent heat flux. Moreover, the profiles spanning from $x/L_m \simeq 0.2$ up to $x/L_m \simeq 0.6$ exhibit a much larger spread in the LES compared to those in the RANS, where the region of high turbulence intensity is much more localized. This is attributed to a lower shear layer growth-rate predicted in the RANS simulation. However, because of the confinement due to the wall of the mixing duct, the shear layer thickness saturates in the LES between $x/L_m \simeq 0.4$ and $x/L_m \simeq 0.6$. The difference in shear layer thickness is thus reduced past that location; from $x/L_m \simeq 0.8$, both frameworks produce similar profiles.

These figures also allow to interpret the differences observed in the local total exergy fluxes through the dividing streamlines introduced in Figure 4.18. Indeed, the values of the turbulent shear stress and heat flux at the location of the dividing streamline, represented by the markers in Figure 4.19 are much higher in the LES compared to

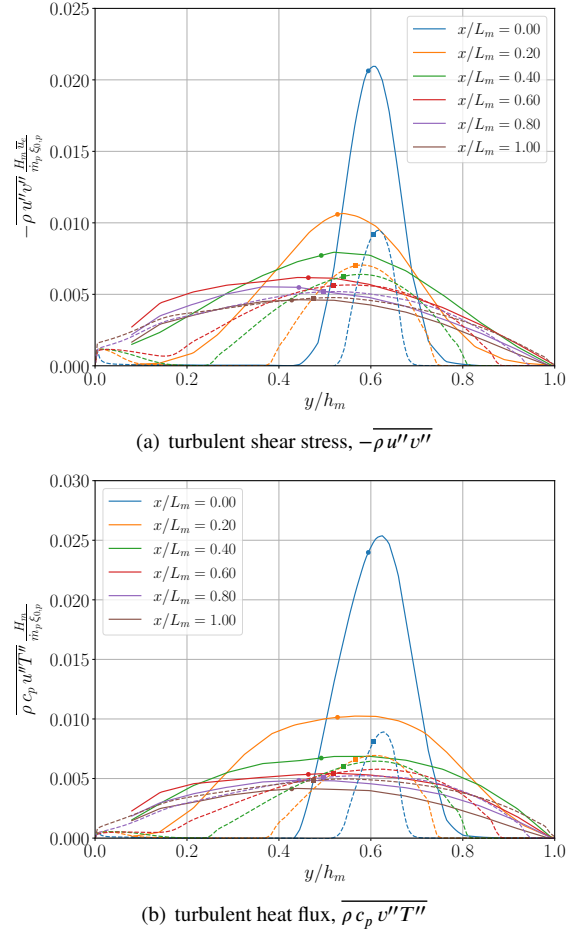


Figure 4.19: Non-dimensional turbulent shear stress and turbulent heat flux in the mixing duct for LES (solid) and RANS (dash) along the lines shown in Figure 4.20. $y/h_m = 0$ corresponds to the bottom wall and $y/h_m = 1$ to its centerline. The markers indicate the location of the dividing streamline computed in the LES (circles) and the RANS simulation (squares).

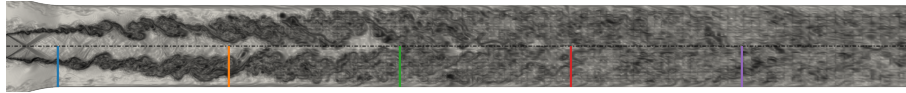


Figure 4.20: Instantaneous density gradient field in the mixing duct, with lines indicating the locations where the turbulent shear stress and the turbulent heat flux are shown in Figure 4.19.

those obtained in the RANS simulation, up to around $x/L_m \simeq 0.8$. Note that the position of the dividing streamline is also slightly different in the RANS compared to the LES from $x/L_m = 0$, the primary jet being always slightly more expanded in the LES. Comparing those values to the total exergy fluxes through the streamlines shows a good match (once a factor of two is applied to take into account both dividing streamlines), showing the importance of those terms when evaluating the total exergy fluxes; hence also the need for properly capturing them using either scale-resolving LES or properly modeling them (using some improved RANS modeling).

In the RANS simulation, a constant Prandtl number $Pr_t^{RANS} = 0.85$ was used. To assess the validity of this hypothesis, the local turbulent Prandtl number $Pr_t = \frac{\mu_t c_p}{\kappa_t}$ is computed from the turbulent shear stress and heat flux obtained in the LES. The turbulent viscosity μ_t and thermal conductivity κ_t are obtained using their usual definition:

$$\mu_t = \frac{-\overline{\rho u'' v''}}{\frac{\partial \tilde{u}}{\partial y}}, \quad (4.23)$$

$$\kappa_t = \frac{\overline{\rho c_p u'' T''}}{-\frac{\partial \tilde{T}}{\partial y}}. \quad (4.24)$$

The derivatives used in the denominator were smoothed using a moving averaging with block-size equal to $0.05h_m$ to remove the spurious oscillations caused by the discontinuities between elements, inherent to the DG method. The result is shown in Figure 4.21. This figure shows that the value of $Pr_t \simeq 0.85$ is a reasonable estimation of the turbulent Prandtl number away from the shear layer. This value is however too high in the shear layer (i.e. where the turbulent intensity is very high) where it drops consistently to 0.6; it even drops to 0.4 at $x/L_m = 0.2$. Those observations are in line with the conclusions of Chambers et al. [14] who observed a reduction of turbulent Prandtl number in a thermal mixing layer. Further studies on more accurate modeling of Pr_t for RANS simulations should be carried out, as well as on the impact such improved modeling might have on the fraction of total exergy fluxes due to thermal effects. Another RANS simulation, using this time $Pr_t^{RANS} = 0.6$ could be also conducted. However this would not help to correct the primary issue concerning the incorrect prediction by the RANS approach of the turbulent viscosity μ_t as defined by Equation (4.23) and illustrated by the discrepancies observed for the shear stress profiles in Figure 4.19.

The three components of the total exergy flux given in Equations (4.19) to (4.21) are also shown in Figure 4.18 for both the LES and the RANS simulations. This

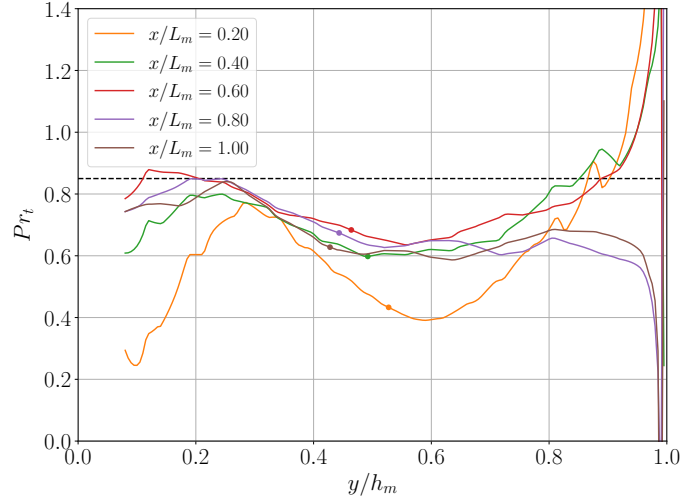


Figure 4.21: Local turbulent Prandtl number, Pr_t , computed in the LES along the lines shown in Figure 4.20. The markers (circle) indicate the location of the dividing streamline obtained in the LES. The constant value used in the RANS simulation is also shown for comparison (dash).

figure also illustrates the total enthalpy component that is independent of the reference state. This decomposition gives additional insight into the mixing process. Indeed, in this case, the enthalpy contribution is directed towards the primary flow, which is consistent with its much lower temperature due to the supersonic expansion. This negative transfer (from the view point of the secondary stream) is counterbalanced by the kinetic energy component and by the entropy contribution which is the largest in magnitude and directed towards the secondary flow. The entropy of the primary flow at inlet being lower than that of the secondary flow because of its higher total pressure and the evolution of total exergy being opposite to that of entropy, this is expected. This last flux is less intuitive to analyze. However, linearizing the expression of the entropy flux given in Equation (4.21), this flux is seen to contain a term closely related to the total enthalpy flux, only being multiplied by the ratio T_{ref}/T . When this ratio is increased, the entropy flux is increased as well, meaning that to produce the largest exchange of entropy between the flows, this exchange should occur at the lowest temperature possible, compared to the reference state. A similar analysis can be performed regarding the pressure ratio p_{ref}/p . This highlights the importance of producing a supersonic jet, effectively reducing both temperature and pressure in the mixing chamber.

Figure 4.18 also shows that, from about $x/L_m \simeq 0.2$, the kinetic energy and the enthalpy components of the total exergy flux compensate each other. The total enthalpy flux thus falls to zero. As shown in Figure 4.22, this also causes the total enthalpy, or equivalently the total temperature to be essentially constant in each stream from $x/L_m \simeq 0.2$, only varying by at most 0.5 %. Thus, the only contribution to the total exergy flux is its entropy component. In this section of the mixing duct, this component has a slight decreasing trend yet remaining significant up to the end of the mixing chamber. This also shows the importance of the entropy flux in the total exergy transfer within a supersonic ejector as it is the only non-zero component in the majority of the mixing pipe and has a beneficial impact on the total exergy transfer. Figure 4.18 also shows that the RANS does not allow to correctly capture this compensation of the kinetic energy and enthalpy fluxes, which is due to the imperfect modeling of the turbulent shear stresses and heat flux density, as shown previously.

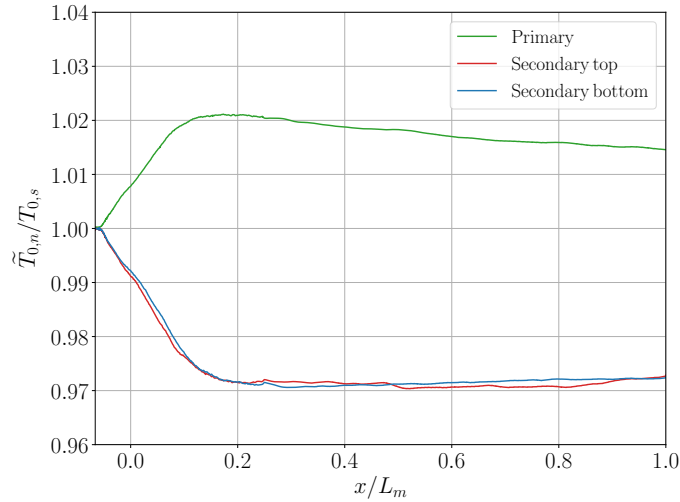


Figure 4.22: Evolution of the total temperature in the mixing duct.

Alternatively, the decomposition of the total exergy flux proposed by Lamberts et al. [55] and using the kinetic energy flux f_k and that of total exergy due to thermal effects f_q , also shown in Figure 4.18 allows to bring a more physical explanation to the purely mathematical observations made previously concerning the entropy flux. Indeed, the base concept behind total exergy is to bring all types of energy to the level of useful mechanical work by including their associated entropic corrections. In this regard, enthalpy being thermal energy, it must be derated by its associated entropic

contribution, which is linked to the temperature at which the transfer is performed, to retrieve its useful mechanical energy content. This corrected enthalpy flux component is in fact f_q . Conversely, kinetic energy being already purely mechanical energy, it does not require an entropic correction. It is thus more valuable exergetically compared to enthalpy. This explains why, even though enthalpy and kinetic energy fluxes balance each other after a short distance in the mixing duct, the net exergetic effect is still positive for the secondary flow after that location since the enthalpic contribution must be reduced by its entropic correction (and significantly so in this case, yielding a positive f_q flux towards the secondary stream), thus directing the net total exergy flux in the same direction as the kinetic energy component.

The zero total enthalpy flux could be used as a criterion to indicate that the streams are mixed. This is confirmed by the profile of the turbulent shear stress shown in Figure 4.19(a) at $x/L_m = 0.2$ which has non-zero values in the whole channel, only reaching zero near the wall, indicating that the upper-bound of the shear layer has reached the mixing duct wall from this position. Moreover, the average velocity profiles shown in Figure 4.23 also indicate that, from $x/L_m = 0.2$, the constant velocity plateau of the secondary flow has already disappeared; from that point, there is no obvious distinction between the primary and secondary streams. This conclusion is also in line with the empirical correlation for mixing length of Karthick et al. [48] which gives a theoretical

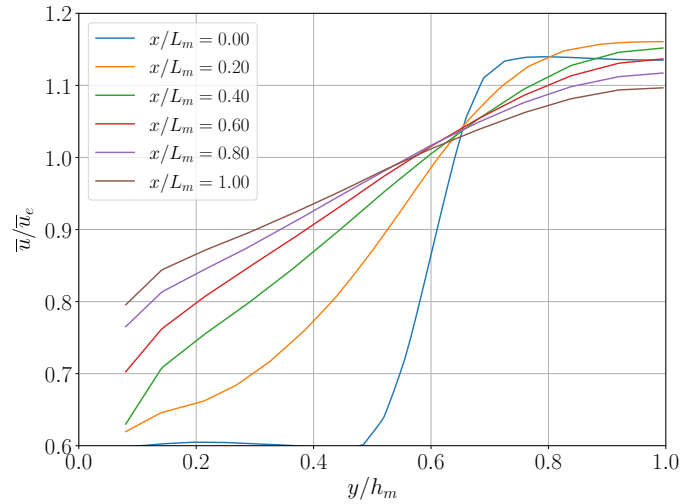


Figure 4.23: Average velocity profiles along the lines shown in Figure 4.20.

mixing length of $0.44 L_m$ for the present case. This is still larger than the value of $0.2 L_m$ proposed here.

Figure 4.24 shows the evolution of the cross-section averaged total pressure along the mixing duct for each stream separately and the compound flow. This shows that there is a consistent increase of the secondary total pressure and a related decrease of the primary total pressure. The total pressure averaged over the whole mixing duct exhibits a more complex evolution. Indeed, the total pressure is observed to be almost constant from $x/L_m = 0$ to $x/L_m \simeq 0.2$, followed by a slow linear decrease further downstream. This trend is consistent with the observation made previously of fully mixed flows from $x/L_m \simeq 0.2$. Moreover, this shows that the mixing occurs at essentially constant total pressure; rather than at constant static pressure as often assumed in the literature.

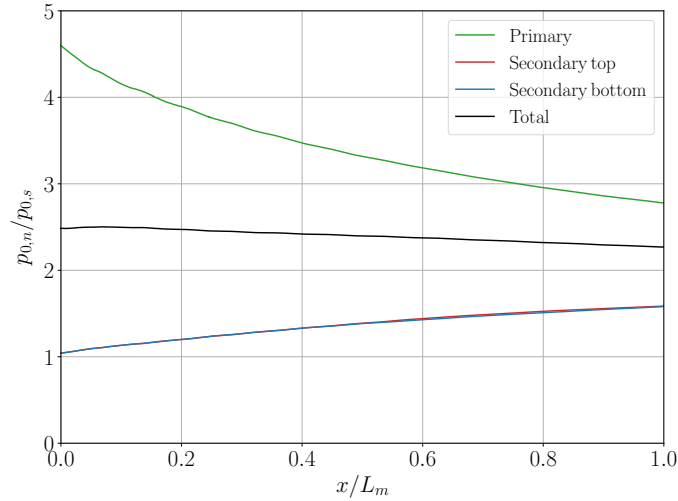


Figure 4.24: Evolution of the total pressure along the mixing duct.

Yet, it is important to realize that the mixing of the flows as defined here is not sufficient for the ejector to work optimally. Indeed, by obtaining the characteristic curve using RANS simulations of the same supersonic ejector, but where the mixing duct length was shortened to a quarter of the current design length $L_{m,design}$ (corresponding to a cut of the actual mixing duct at $x/L_m = 0.25$), it was observed that the critical pressure drastically decreased by almost 20 %, while maintaining the on-design entrainment ratio. This loss in performance is explained by the fact that, even though

the flows are mixed at $x/L_m \simeq 0.2$ according to the criterion previously defined (the upper-bound of the shear layer fills the entire channel section), a significant turbulent shear stress level still remains in the regions surrounding the dividing streamlines, as illustrated in Figure 4.25. At $x/L_m = 0.2$, the beneficial shear stress between the flows is still about 7 times larger than the detrimental wall shear stress. The additional mixing duct length after $x/L_m = 0.2$ hence allows the secondary flow to increase its total pressure, thus increasing the critical back-pressure representing the maximum back-pressure the secondary flow can achieve at this flow rate fixed by the compound choking process upstream. Beyond that pressure, the only solution is to decrease the flow rate and then to unchoke the secondary stream.

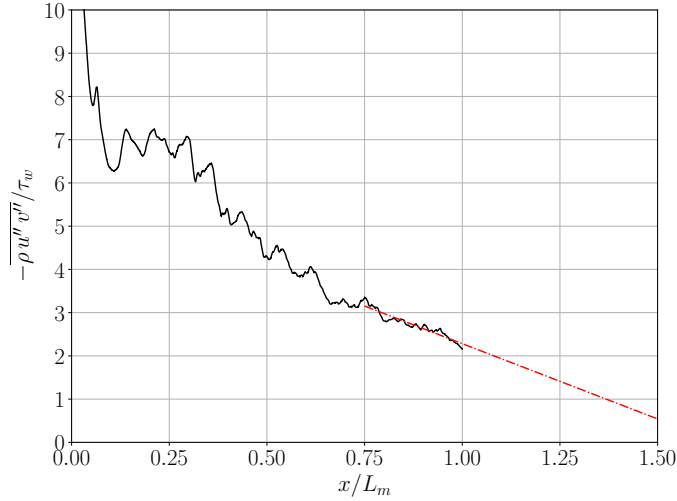


Figure 4.25: Evolution of the turbulent shear stress along the dividing streamlines (top + bottom) normalized by the total wall shear stress (top + bottom) in the mixing duct.

To further confirm the impact of the shear stress to wall-friction ratio, the characteristic curve of an ejector where the mixing duct length was increased to $L_m/L_{m,design} = 1.3$ (i.e., the approximate location where both effects are balanced) was also computed using RANS; it indeed gave a slight increase of the critical pressure, by 1.5 %, and still without change in the on-design entrainment ratio, which is an improvement. This clearly demonstrates that the optimal mixing duct length for an ejector corresponds to the extent along which the driving shear forces dominate the resistive wall-friction forces. Beyond this point, resistive wall-friction forces become

predominant, reducing the ejector performance. The current studied ejector is shown to be not so far from that optimum.

4.4 Conclusions and perspectives

In this study, a high-fidelity LES of a supersonic air ejector was performed. The two-dimensional geometry was extruded over a depth that allowed to properly capture the turbulent content of the shear layers emanating from the interaction between the primary supersonic jet and the secondary subsonic entrained flow. These shear layers were shown to rapidly transition to fully turbulent shear layers. This transition was observed to be independent of the turbulence injected at the inlet of the primary nozzle in the form of a periodic channel co-simulation, as the boundary layers were shown to relaminarize before the nozzle exit, due to the high and continuous acceleration within the nozzle. Moreover, the boundary layers in the secondary flow inlets were shown to naturally transition to turbulent boundary layers, due to the long constant velocity secondary inlet channel.

A shadowgraphy image obtained numerically from the LES showed a good agreement in terms of expansion/shock waves reflection pattern, and shock/shear layer interaction with the experimental shadowgraphy image obtained by Lamberts [54]. The entrainment ratio was shown to match that obtained in a RANS simulation. However, the wall pressure in the mixing duct was observed to be lower for the LES compared to RANS, due to the slightly higher expansion of the primary flow and the faster shear layer growth in the LES.

The local total exergy flux analysis showed how the RANS simulation which uses a constant turbulent Prandtl number (i.e. $Pr_t = 0.85$ in this case) fails to accurately capture the exergy transfer due to turbulence. The Pr_t profile computed from the post-processing of the LES results was indeed found to be non uniform. Although a value of 0.85 appears adequate on the edges of the highly turbulent region between the two flows, the value obtained close to the dividing streamline was closer to $\simeq 0.6$. Further studies on more accurate modeling of Pr_t for RANS simulations of similar flows should be carried out, as well as on the impact such improved modeling might have on the total exergy transfers due to thermal effects. Such improvement would however still not correct the main failures of such models relying on an isotropic description of the turbulent viscosity and a Reynolds analogy to model the turbulent heat fluxes. Alternatively, an additional equation for the turbulent heat flux could be

used in the RANS model to provide a more independent modeling of the turbulent shear stresses and turbulent heat flux.

The LES results also allowed to propose a new criterion to assess the degree of mixing of the flows, based on the flux of total enthalpy through the dividing streamlines. It was indeed observed to fall to zero at $x/L_m \simeq 0.2$, where the velocity profile no longer allowed to clearly separate the primary and secondary flows. This also corresponds to the location where the upper bound of the shear layer comes into contact with the mixing duct wall. Moreover, the one-dimensional averaged total pressure profile is observed to transition from a plateau to a slowly linearly decreasing trend at the same location.

This criterion of mixing of both streams was however shown to be insufficient to choose the correct length of the mixing duct and thus to have an optimal supersonic ejector design. Indeed, even though the flows are considered mixed, a significant turbulent shear stress level still remains in the regions surrounding the dividing streamlines (about 7 times the wall shear stress). It was found that this optimal length would be the necessary length to reach a balance between the driving shear stress at the dividing streamline and the resistive wall shear stress.

In the future, the LES framework could be used to investigate potentially more unsteady operating points; such as in the off-design region of the characteristic curve, or close to the critical point. Indeed, it was observed in this chapter that, for the studied operating point, the significant differences in total exergy fluxes between the RANS and the LES was not reflected in the global performance parameters as both numerical frameworks predict virtually identical entrainment ratios, which is rather unexpected. However, for operating points well in the on-design regime (as the one studied here), Lamberts et al. [55] showed that the total exergy tubes cross the dividing streamline well into the mixing duct (and even in the diffuser in some cases), which corresponds to the region where RANS and LES total exergy fluxes exhibit similar behaviors. This could thus explain why the local differences observed in the beginning of the mixing duct do not translate into differences in the global performance parameters of the ejector. However, for near-critical or off-design operating points, the crossing of the dividing streamline by the total exergy tube was found to move upstreams, reaching the very beginning of the mixing duct, i.e. where large differences in total exergy fluxes are observed between numerical frameworks, which could then result in larger differences in entrainment ratio predictions.

The LES methodology could also be used to study the transition from on- to off-design operations and the physical process that triggers the unchoking of the secondary

stream depending on the fact that it becomes or not supersonic beyond the choking section). The post-processing tools using the streamlines and total exergy fluxes should then be augmented to also account for the unsteadiness of the flow. It must however be recognized that the current computational cost of high-fidelity LES remains a major hurdle. This could be overcome with a GPU accelerated version of the code. More operating points could then be simulated, producing more data to understand the physics behind the mixing of streams in supersonic ejectors, and also to train simplified models.

Finally, this chapter showed that the LES framework did not improve the RANS predictions in terms of entrainment ratio and wall-pressure profile compared to the experiment. In fact, it even slightly worsens it, despite the higher fidelity of the method. The cause of this difference should be studied through the extrusion of the domain to the full experimental setup and the addition of the sidewalls present in the experiment to include the three-dimensional effects in the LES as those showed to significantly impact the wall-pressure profile [54]. However, this poses significant challenges for the wall-modeling due to the presence of sharp corners. The next chapter thus presents the extension of the present LES methodology to cylindrical ejectors, which is the generic geometry for industrial applications and which are less prone to spurious three-dimensional effects.

Design and characterization of a new cylindrical supersonic ejector

This chapter is largely inspired by an article submitted to the International Journal of Heat and Fluid Flow, entitled "Numerical and experimental characterization of a cylindrical supersonic air ejector"[27], by R. Debroeyer, G. Winckelmans, and Y. Bartosiewicz.

5.1 Introduction

Over recent years, the understanding of the internal flow phenomena taking place in supersonic ejectors has significantly increased thanks to many numerical investigations using essentially RANS simulations [56, 57] but also a very limited number of Large-Eddy Simulations [19, 26]. However, the number of experimental studies where the instrumentation allows to study the ejector further than its characteristic curve is rather limited. This provides insightful information about the performance of the device but it does not allow to explain how they are impacted by local flow features and why they might differ from numerical simulations. The most detailed experimental results were obtained by Lamberts et al. [56, 57], featuring 48 pressure taps along the top wall of a rectangular ejector as well as a visualization setup. In these studies, the Schlieren images and the characteristic curves obtained showed a quite good correspondence between the experimental and the RANS simulation results, but large discrepancies in the wall pressure profile were observed. This was largely attributed to three dimensional

effects (due to the presence of the side walls) and a three-dimensional RANS simulation was shown to slightly reduce the differences in wall pressure [54].

The present study aims at producing experimental results with a similar level of detail but using a cylindrical ejector; which is also clearly the preferred geometry since it is that used in industrial applications. This should largely reduce the un-captured three-dimensional effects plaguing the rectangular geometries. Those new experimental results are accompanied by axisymmetric RANS simulations for comparison. Moreover, a high-fidelity scale resolving simulation using LES with a fine grid of an on-design case near the critical point is also carried out. As LES is resolving many of the turbulent scales, and without time-averaging, it is much more accurate than RANS. Hence, it can be used to gain additional insight into the flow and help explain the differences between the RANS and the experimental results.

First, the design of the ejector will be explained, with a focus on the optimization of the mixing pipe length. Then, the full experimental setup will be presented and the numerical setup used to perform the RANS simulations will be laid out. The experimental results will subsequently be compared to their numerical counterparts. Finally, the results of the LES will be presented and used to explain the differences observed between the RANS and the experimental data.

5.2 Design and numerical setup

5.2.1 Geometry

The supersonic air ejector considered in this study is designed to have characteristics, in terms of area ratios and primary mass flow-rate, similar to those of the rectangular ejector designed by Lamberts [54] and extensively studied in subsequent studies [19, 26]. These criteria constrain the values of primary nozzle throat and exit radii. Then, the first improvement over the rectangular design consists in reducing the secondary entrance length in order to decrease the head losses in this part of the ejector. This inlet is also given a convergent geometry to accelerate the secondary flow more gradually. Furthermore, the diffuser consists of a custom made diffuser with an opening angle of 7° to minimize the total pressure losses; followed by an off-the-shelf diffuser with opening angle 19° . The transition between the diffuser angles is done smoothly and at a sufficiently large distance of the mixing duct to have the majority of the important flow features located inside the custom diffuser. This dual diffuser is used to reduce the overall length of the ejector and for manufacturing reasons.

Another major improvement over the design of Lamberts [54] concerns the mixing pipe length. In the earlier design, the mixing duct length was chosen according to best practice guidelines of the literature and was shown to be close to optimal in the previous chapter. For the present ejector, the mixing pipe length was optimized using RANS simulations by obtaining the characteristic curves of ejectors differing only by the mixing duct length. Figure 5.1 shows the characteristic curves for two inlet total pressure ratios in the targeted range. This shows that the maximum critical pressure is obtained for a mixing pipe length of $L_m = 500$ mm or $L_m/R_m \simeq 23.8$ for both total pressure ratios (whereas the mixing duct length of the rectangular geometry was $L_m/h_m = 20.7$ where h_m is the duct half-height). The final design with its main dimensions is shown in Figure 5.2.

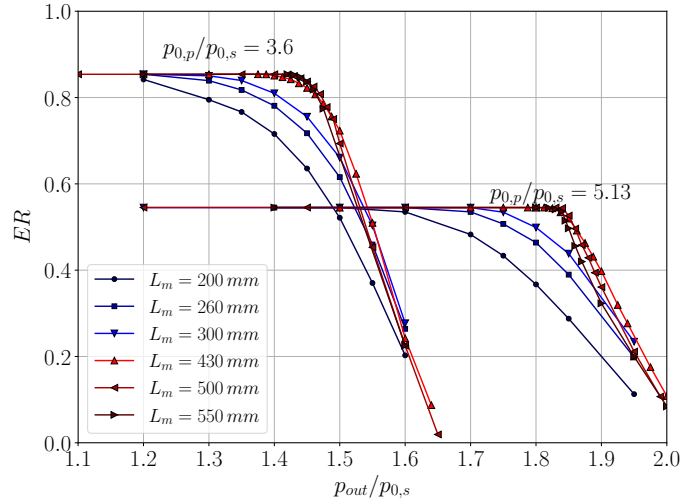


Figure 5.1: Characteristic curves for two inlet total pressure ratios and different mixing pipe lengths, as obtained using RANS simulations.

5.2.2 Experimental setup

The ejector is equipped with mass flow rate measurements at both inlet ports, as well as at the outlet of the ejector for redundancy. Seventy five pressure taps are placed along the mixing pipe and diffuser, connected to multiplexers, which in turn are connected to a relative pressure sensor. The spacing between those pressure taps allows to highlight different flow features, such as the primary jet shock and expansion waves, as well as the secondary shock train. These pressure taps indeed allow to reach

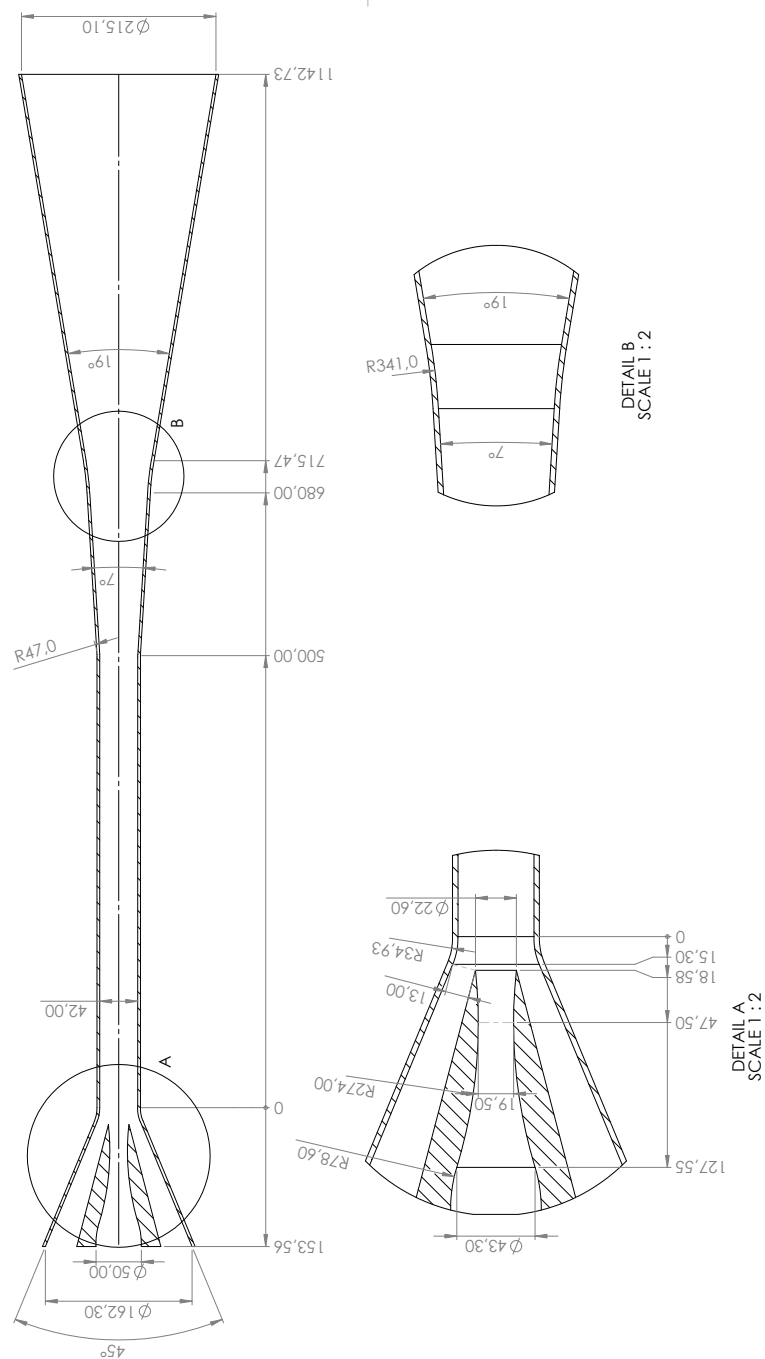


Figure 5.2: Dimensional drawing of the final design of the ejector. All dimensions are in millimeters.

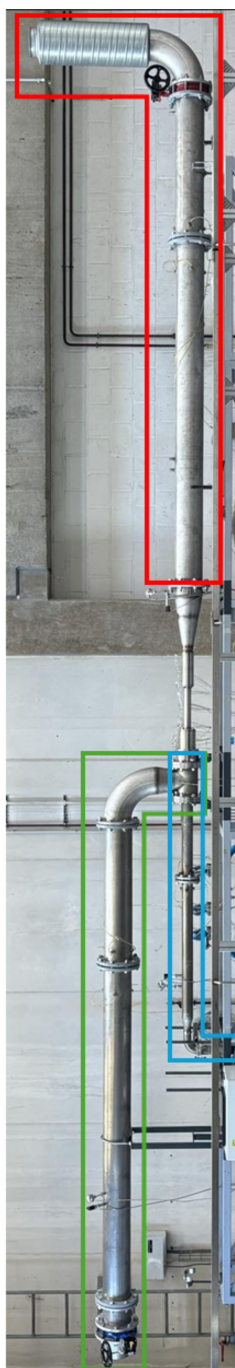


Figure 5.3: Picture of the full experimental setup.

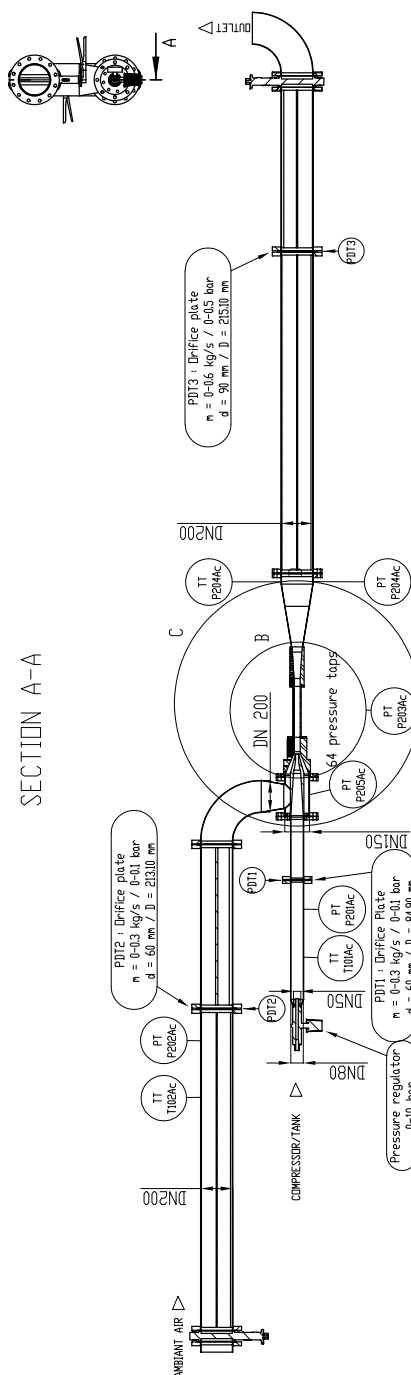


Figure 5.4: P&ID of the experimental setup.

a resolution of $\Delta x/L_m = 0.02$ in most of the mixing pipe and down to $\Delta x/L_m = 0.016$ in the region around the NXP and the end of the mixing duct.

The full experimental setup is shown in Figure 5.3. The long pipes surrounding the ejector, highlighted by colored boxes are not part of the actual ejector but are required to measure the corresponding mass flow-rates, following the norm ISO 5167 [45, 46]. The primary inlet is shown in blue whereas the secondary pipe is shown in green and the outlet pipe is highlighted in red. The full Piping & Instrumentation Diagram (P&ID) is presented in Figure 5.4, with a zoom around the test section provided in Figure 5.5. As shown in this figure, some of the pressure ports are closed (noted C in the P&ID) due to obstruction in the experimental setup, leaving sixty four ports available out of the initially planned seventy five pressure measurement points.

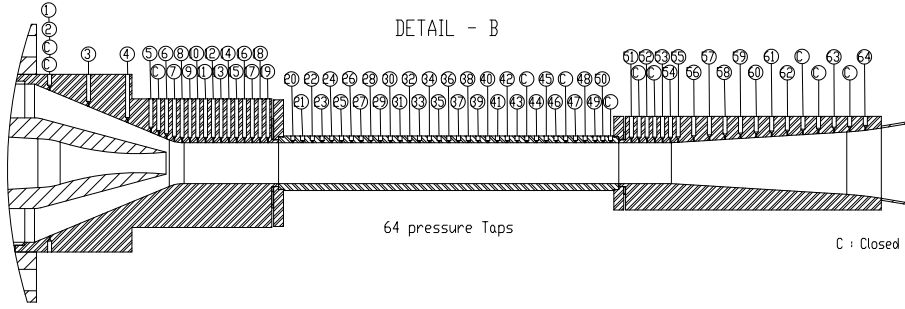


Figure 5.5: P&ID - zoom on the test section

5.2.3 RANS simulations setup

All axisymmetric RANS simulations are performed using the Ansys Fluent (2023 R1) simulation software, and with a $k - \omega - SST$ turbulent viscosity closure model, as this was shown by Bartosiewicz et al. [4] to best compare to experimental results. A turbulent Prandtl number $Pr_t = 0.85$ is used in all simulations. The mesh is constituted of structured quadrangles in essentially the whole domain, with only 0.1 % triangles near the nozzle lip. The mesh is such that the wall is properly resolved in the whole domain, with y_1^+ not exceeding 1.5 in the mixing duct and 5 in the whole domain. There are 500 000 elements. The details of this mesh are shown in Figure 5.6.

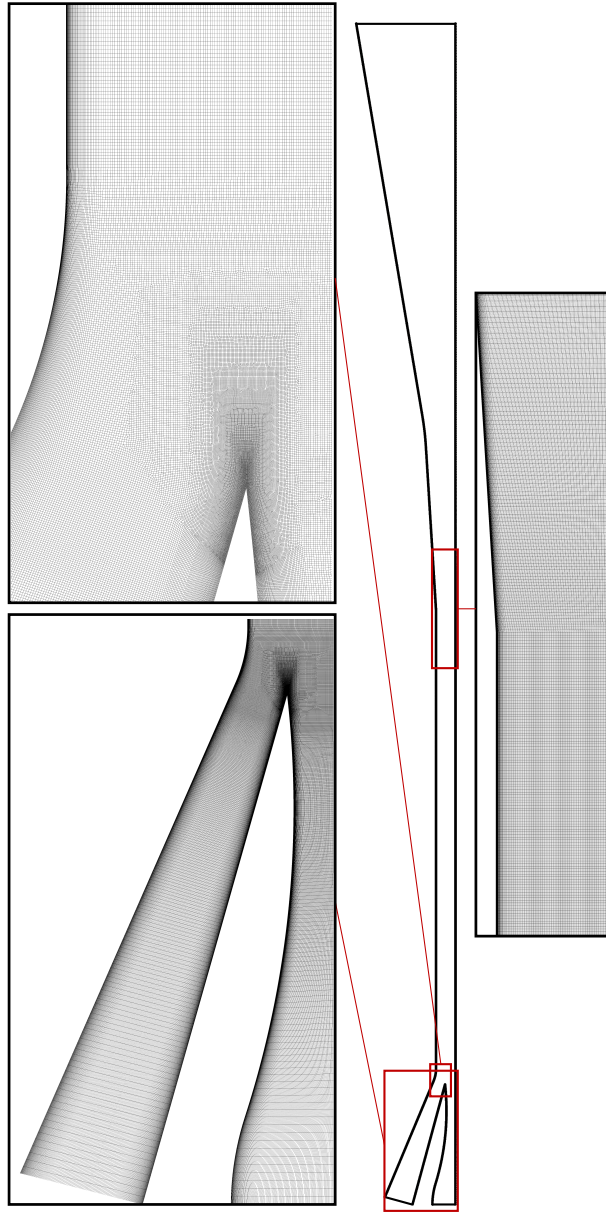


Figure 5.6: Details of the mesh used for the RANS simulations.

5.3 Experimental and RANS simulation results

The characteristic curves obtained experimentally and numerically with RANS simulations are presented in Figure 5.7. The statistics of the experimental study are obtained by averaging the measurements over the duration of the test, as well as across multiple occurrences of each of the operating points. Alongside the mean of the measurements, the uncertainty of each measured quantity is displayed through error bars. Those are obtained by combining the standard deviation across the different experimental runs to the uncertainty coming from the sensors. The uncertainty is computed for a confidence interval of 95 % and following the method exposed in the Guide to the expression of uncertainty in measurements [8].

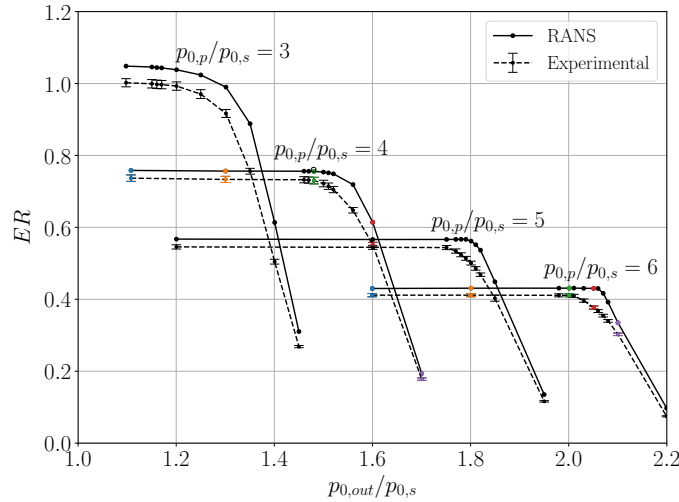
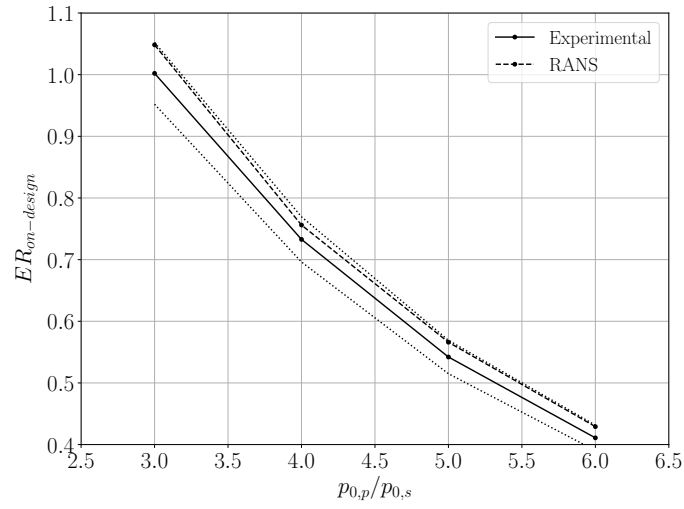
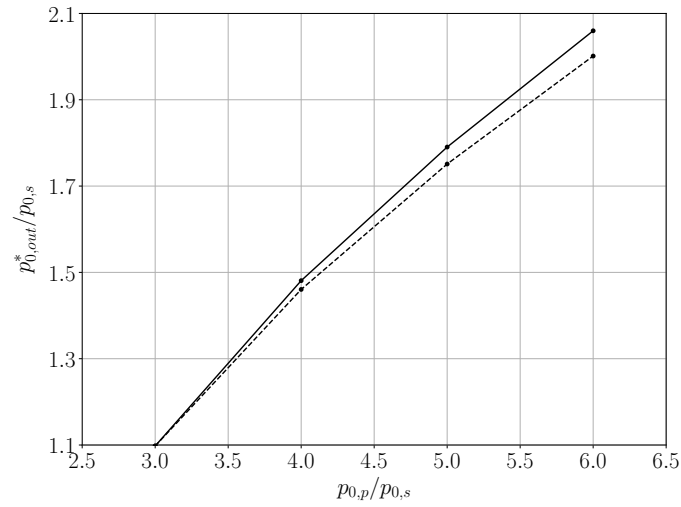


Figure 5.7: Characteristic curves of the ejector. Comparison between experimental and RANS simulation results. The colored markers indicate the cases used to analyze the wall-pressure profiles in Figure 5.9 for each characteristic curve. The entrainment ratio obtained with the LES presented in Section 5.4 is shown with the open black marker on the $p_{0,p}/p_{0,s} = 4$ characteristic curve.

Figure 5.7 shows that the RANS predictions of the performance of the ejector are quite accurate, yet with a systematic over-prediction of the on-design entrainment ratio contained within 5 % compared to the experiment, as shown in Figure 5.8(a). The error on the entrainment ratio is attributed solely to the difference in the secondary mass flow-rate; indeed, the difference in primary mass flow-rate does not exceed 0.5 %. We note that such an over-prediction of the on-design entrainment ratio when using RANS simulations, as compared to experiments, was already observed in the literature



(a) On-design entrainment ratio. The dotted lines represent the $\pm 5\%$ deviation limit from the experimental data.



(b) Critical pressure.

Figure 5.8: Evolution of the main performance characteristics of the ejector. Comparison between experimental and RANS simulation results.

[17, 54, 61]. It is likely linked to experimental imperfections that are not taken into account in the simulation. Those are however limited as the obtained deviation is lower than in previous studies.

Moreover, as evidenced on Figure 5.8(b), the RANS simulations also allow to predict the critical pressure of the ejector. The critical pressure is obtained as the largest back pressure for which the deviation on entrainment ratio compared to the on-design entrainment ratio is below 0.1 %. The critical pressure thus still represents an operating point in the on-design regime. The 0.1 % limit was chosen arbitrarily and was observed to give the most consistent results. Over the whole total pressure ratio range considered here, the error is observed to increase with total pressure ratio but is maintained below 3 %. It should be noted that, as for the on-design entrainment ratio, the RANS simulations always over-estimate the critical pressure. The gradual degradation of the critical pressure prediction can be attributed to the transition from on-design to off-design operation. This transition becomes gradually sharper with increasing total pressure ratio in the RANS, pushing the critical pressure further to the right, whereas it stays consistently smooth in the experiments. The reason behind this difference in behavior remains currently an open question. Despite those deviations in the on-design performance of the ejector, around the critical point and at the transition towards the off-design regime, there is no major discrepancy in the off-design portion of the characteristic curve.

Figure 5.9 shows the evolution of the pressure along the ejector wall for five operating points, for characteristic curves: $p_{0,p}/p_{0,s} = 4$ and $p_{0,p}/p_{0,s} = 6$. The selected operating points span the full back pressure range, with one being the experimental and/or numerical critical point for each total pressure ratio. This figure shows that the numerical simulations tend to underestimate the actual wall pressure profiles for the on-design operation points in the mixing duct. The correct pressure profile is then gradually retrieved in the diffuser to match the outlet pressure. The pressure profiles obtained for the off-design points are significantly better predicted than those obtained for the on-design portion. The largest deviation is observed for a total pressure ratio of 6 and a back-pressure $p_{0,out}/p_{0,s} = 2.05$. This can however be directly explained by the corresponding characteristic curve in Figure 5.7, where this back-pressure is seen to be in the on-design regime for the RANS simulation, whereas it is in the off-design regime for the experiment.

Figure 5.9 also shows a perfect collapse of the on-design curves, up to the location where the flow un-chokes. This is the case for the numerical results and was already observed by Lamberts et al. [55]. This is also the case for the experimental results,

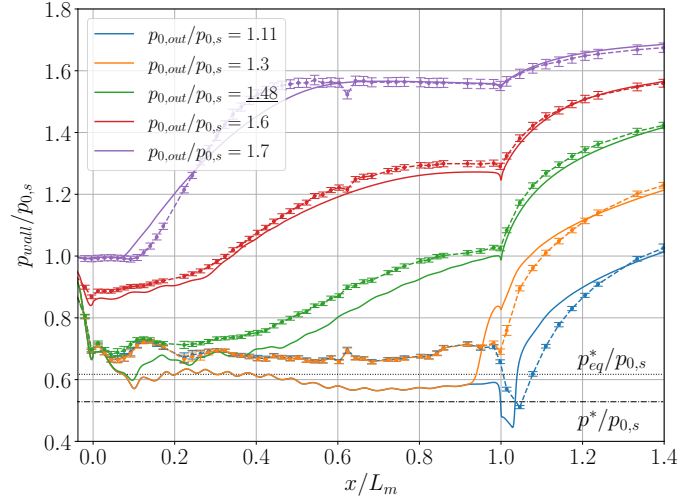
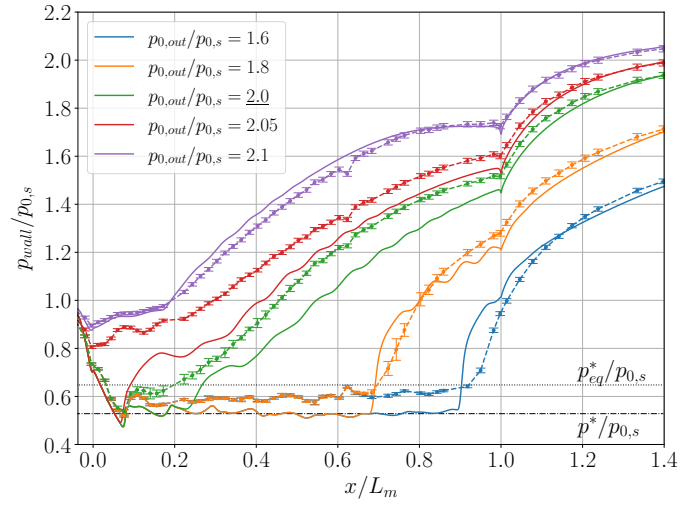
(a) $p_{0,p}/p_{0,s} = 4$ (b) $p_{0,p}/p_{0,s} = 6$

Figure 5.9: Evolution of the wall pressure in the ejector. Comparison between experimental and RANS simulation results for five operating points, and for two of the total pressure ratios considered. The profiles corresponding to the critical conditions obtained experimentally are underlined in the legend.

where the pressure profiles almost perfectly match. This illustrates the high accuracy and repeatability of the experiments. Moreover, the location where those on-design curves separate is quite well predicted by the simulation despite the consistent underestimation of the on-design wall pressure.

The theoretical pressure p_{eq}^* at which the flow should choke, as obtained from the thermodynamic model of Metsue et al. [61], and based on the compound choking theory for isentropic flows, is indicated with a dotted line. This is higher than the sonic pressure of the secondary flow p^* , indicated by a dash-dotted line. This shows that, although in the on-design regime, the experimental wall pressure does not always reach this value, contrary to the RANS simulation results. The pressure obtained experimentally indeed crosses the p_{eq}^* limit for a higher inlet total pressure ratio, but not for $p_{0,p}/p_{0,s} = 4$. This phenomenon was already observed by Lamberts [54] but no further explanations were given. This means however that the actual choking pressure is higher than that predicted by the best theory so far for the choking of a supersonic ejector.

Since the choking pressure increases as the choking model becomes more refined, i.e. from the so-called “Fabri-choking” (which assumes that it is necessary for the secondary flow to become at least sonic to reach the on-design regime), to the “compound-choking” theory, a potential explanation for this yet increased choking pressure could lie in the incompleteness of the model used for the choking. An additional indicator that the error on wall-pressure profiles might be caused by the choking mechanism lies in the fact that the wall-pressure measurements of the off-design operating points, i.e. where the flow is not choked, are more accurately predicted by the RANS.

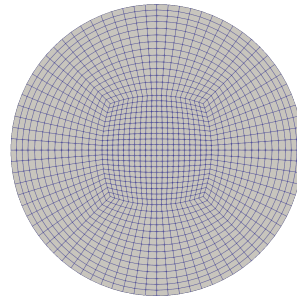
5.4 Large-eddy simulation at a critical point

It was shown in the previous section that the RANS simulations allow to predict fairly well global performance metrics such as the entrainment ratio along the on- and off-design regimes, as well as the critical pressure. However, the prediction of local quantities such as the wall pressure profile still deviates significantly from the experimental results. In order to gain additional insight into the flow physics and to help explain these differences, the LES corresponding to a critical condition for $p_{0,p}/p_{0,s} = 4$ is conducted. The flow statistics are accumulated over five successive windows of around $4.5 t_c$ where $t_c = L_m/\bar{u}_e$ is a characteristic time of the ejector and $\bar{u}_e$ is the cross-section averaged flow velocity at the primary nozzle outlet. These multiple

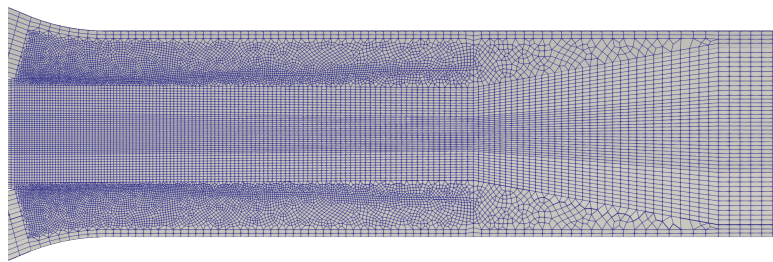
averaging windows will be used to better compare the LES and experimental results using confidence intervals in both cases. Each of the averaging windows represents about 5 ms of physical time, whereas the experimental setup allows averages over several minutes. However, each of those 5 ms averaging windows, representing around 20 000 timesteps, takes around 12 days to simulate on 1536 AMD EPYC 7763 cores on the “Lucia” supercomputer.

5.4.1 Numerical setup

The numerical setup for this LES is similar to that used in Chapter 4 in terms of numerical methods and tools. However, the mesh is here constituted of second order elements and is not extruded to avoid narrow prisms close to the symmetry axis. Instead, the mesh is constructed by connecting streamwise normal sections as shown in Figure 5.10(a). The mesh is thus still essentially structured and composed of



(a) Streamwise-normal section in the region with structured mesh.



(b) Zoom on a slice in the region with unstructured mesh.

Figure 5.10: Details of the mesh used for the LES. Note that only the elements are shown in this figure. As third-order interpolation polynomials are used, each element contains four degrees of freedom per spatial dimension.

hexahedra, except in the first quarter of the mixing pipe where it is unstructured and refined around the computed dividing streamline. Figure 5.10(b) shows the details of the mesh in the unstructured region. A total of 1.1 million elements are used. As a Discontinuous Galerkin Method with third order interpolation polynomials is used, resulting in a fourth-order accurate scheme, this amounts to a little over 70 million degrees of freedom per conservation equation.

The stagnation conditions are imposed at both inlet ports. The inlet total temperatures are identical: $T_{0,p} = T_{0,s} = 300$ K. The inlet total pressure ratio is set to $p_{0,p}/p_{0,s} = 4$. Contrary to what was done in the previous chapter, no turbulent inflow was supplied to the primary nozzle inlet as the strong and continuous acceleration in the primary nozzle causes the relaminarization of the boundary layer. The walls are assumed adiabatic and the wall model developed in Chapter 3 is used to compute the wall shear stress. Finally, the outlet static pressure is set to $p_{out}/p_{0,s} = 1.48$ to match the critical condition according to the RANS simulation and experimental results.

The entrainment ratio is measured to be $ER^{LES} = 0.761$ with a standard deviation of 0.0014. This is in very good agreement with the RANS results presented in Figure 5.7. Indeed, the entrainment ratio obtained in the RANS simulation is $ER^{RANS} = 0.762$ for the same operating conditions. For comparison, in the experiment, the entrainment ratio is $ER^{exp} = 0.73 \pm 0.009$.

5.4.2 Instantaneous results

The instantaneous temperature field in a slice of the domain is shown in Figure 5.11. The field visualizations are obtained using the high-order solution visualization plugin from Paraview, developed by Rasquin et al. [70]. This figure shows the evolution of the primary supersonic jet as it interacts with the secondary flow, generating oblique

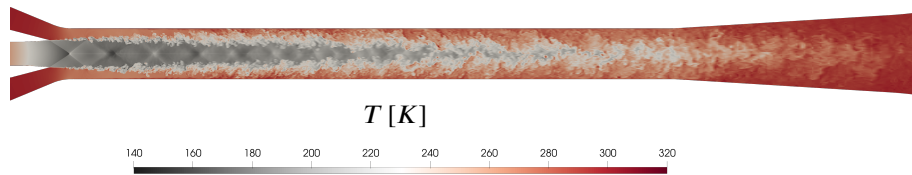


Figure 5.11: Instantaneous temperature field in a slice of the domain.

shock trains and turbulent shear layers. The low temperature region caused by the supersonic jet is seen to reach the end of the mixing pipe.

To further study the evolution of the shear layers at the beginning of the mixing pipe, Figure 5.12 shows the instantaneous fields of the temperature and the longitudinal density gradient in the region near the primary nozzle exit. The slice normal to the flow direction is placed at $x/L_m = 0.10$. These fields show the oblique shock at the primary nozzle exit and the very early transition of the shear layers to turbulence. The density gradient field also shows acoustic waves emanating from the interaction between the oblique shocks and the shear layer.

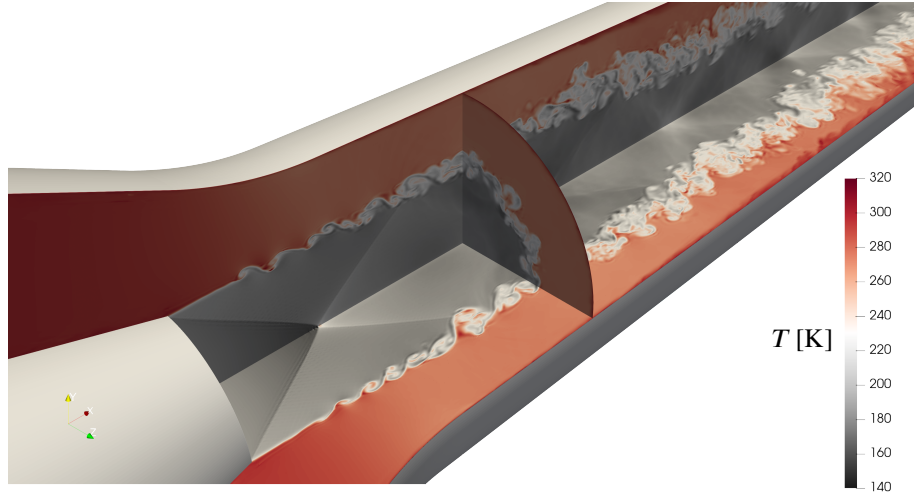
The turbulent nature of the shear layers is further confirmed by the iso-contour of the Q-criterion shown in Figure 5.13, and also colored by the azimuthal component of the vorticity. This clearly shows how the coherent structures quickly become fully three-dimensional and transition to turbulence.

5.4.3 Averaged results

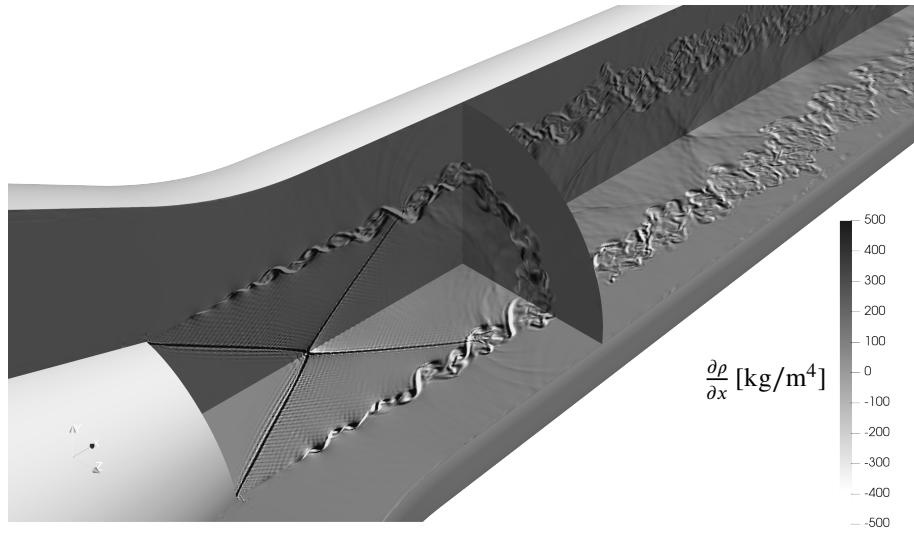
To properly compare RANS simulations, experimental data, and LES results, the time-averaged LES fields are also averaged in the azimuthal direction. Note however that all fields are initially post-processed in three dimensions to detect any flow asymmetries and are then averaged in the circumferential direction. In the following graphs, an image of the temporal and azimuthal spread of the three-dimensional data is included by means of the standard deviation σ of the considered quantity shown as a clear colored band placed at $\pm\sigma$ around the mean.

The dividing streamlines are shown in Figure 5.14 for the LES and the RANS simulation. The dividing streamline is computed in LES by generating streamlines emanating from N_θ equidistant points around the nozzle exit disk, and integrated using a Runge-Kutta 4 scheme on the momentum field. To avoid too large deviations of the streamline from their original azimuthal position, which could lead to streamline intersection further downstreams, each individual streamline is integrated over a series of short streamwise intervals. At the end of each interval, the obtained points are then re-interpolated azimuthally on a uniform grid. The process is then repeated up to the end of the mixing pipe.

The three-dimensional variability of the dividing streamline in the LES is observed to increase with the streamwise coordinate. This is expected as small deviations at the beginning of the mixing pipe will be amplified during the integration of the dividing streamlines. This is however very limited. Moreover, the dividing streamline in LES is



(a) Temperature



(b) Longitudinal density gradient

Figure 5.12: Instantaneous fields in the region near the primary nozzle exit.

observed to match that of the RANS up to $x/L_m \simeq 0.5$. After that location, the oscillations of the dividing streamline caused by the successive compression and expansion

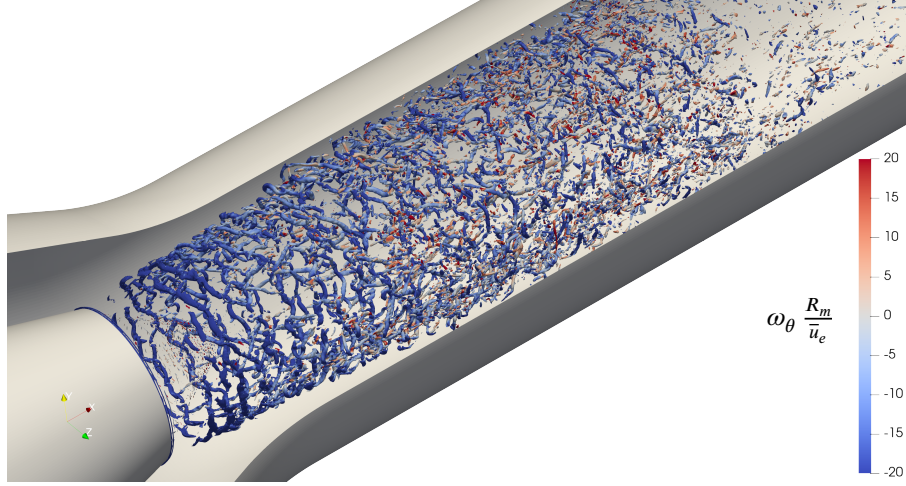


Figure 5.13: Iso-contour of the Q-criterion, colored by the azimuthal vorticity, in the region near the primary nozzle exit.

waves in the primary supersonic jet tend to diverge between both frameworks. The dividing streamline obtained in RANS does however not exceed the standard deviation measured in the LES data.

To further investigate the differences in the dividing streamline, Figure 5.15 presents the time and space averaged fields of Mach number and streamwise density gradient for the LES, and compared to those of the RANS simulation. This confirms the difference in shock structure after a short distance in the mixing pipe. This is caused by a difference in the shear layer growth between the two frameworks, causing the interaction of the shocks and shear layers to differ, starting from the second shock reflection. The difference in shear layers is clearly visible in Figure 5.15(b) as it is laminar in the LES for a short distance before transitioning to turbulence and subsequently growing very rapidly; whereas the RANS predicts a slowly growing turbulent shear layer. This changes the shock angles and thus the position of the streamline. We stress that this LES is of high fidelity, as many of the small scales are captured, thanks to the fine grid size used and accuracy of the DG method. The physics of turbulent entrainment and mixing associated with the shear layers are hence well captured. In RANS, only the averaged behavior is obtained, and here using an “effective turbulent viscosity” closure model. Such model is typically calibrated for boundary layers, and hence it is not necessarily accurate for shear layers. We hence favor the LES results over the

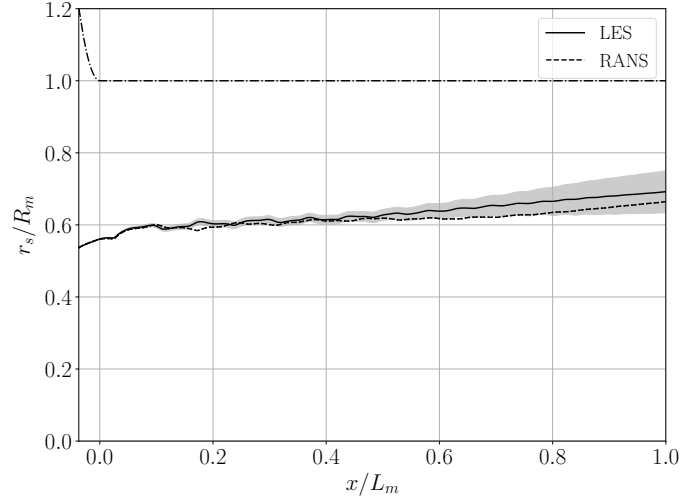
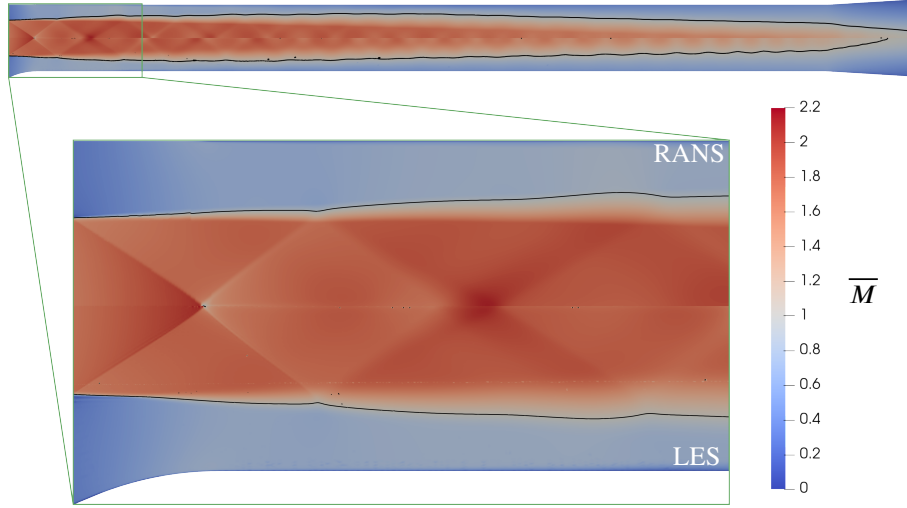


Figure 5.14: Location of the dividing streamline in the mixing pipe. Comparison between RANS (dashed line) and LES (solid line). The shaded area represents the standard deviation of the dividing streamline position along the azimuthal direction. The dash-dotted line represents the position of the ejector wall.

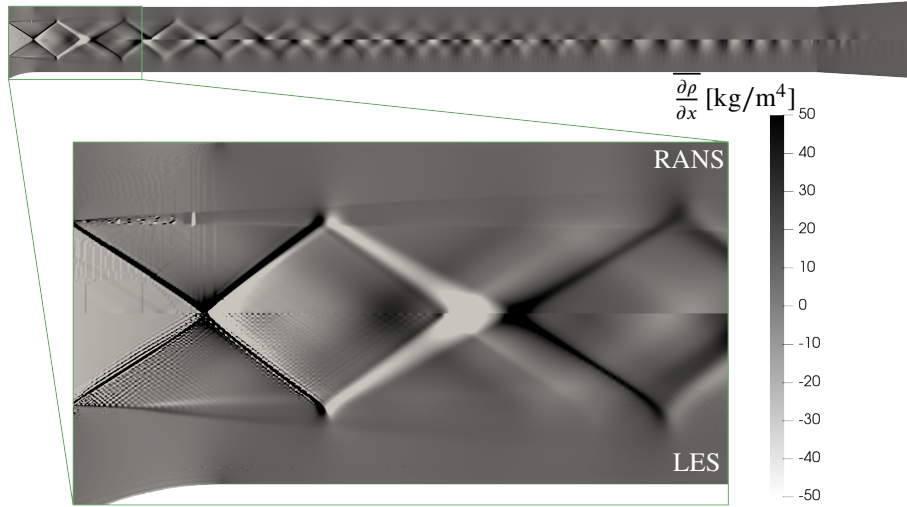
RANS results. It is also interesting to highlight that the present LES captures shocks up to the very end of the mixing pipe, thanks to the very low numerical dissipation of the DG method.

The pressure profiles obtained in RANS simulations were shown in Figure 5.9 to consistently underestimate the experimental pressure measurements for all the cases considered. The wall-pressure profile obtained in the LES, averaged in time and in the circumferential direction, is shown in Figure 5.16. It is compared to the corresponding experimental and RANS profiles. We see that the wall pressure predicted by the LES is much more accurate than that predicted by the RANS up to $x/L_m \simeq 0.2$. Indeed, the LES does not produce the low pressure region observed in RANS which even reaches the compound choking critical pressure p_{eq}^* , symbolized with the dotted line in Figure 5.16. However, between $x/L_m \simeq 0.2$ and $x/L_m \simeq 0.4$, the LES overestimates the increase in pressure and thus does not predict the sudden change in gradient of the experimental pressure profile at around $x/L_m \simeq 0.4$. This leads to a global overestimation of the pressure in the LES for $0.2 < x/L_m \leq 0.8$; whereas the RANS underestimates the pressure all along the mixing pipe.

As shown in Figure 5.17, even in the less accurate region of the mixing pipe, the error on the wall pressure for the LES only slightly exceeds 10 % and is virtually always



(a) Mach number. The sonic lines with $\overline{M} = 1$ are shown in black.



(b) Streamwise density gradient. The impurities present in both LES and RANS are due to post-processing inaccuracies on computing the gradient.

Figure 5.15: Average fields for the RANS (top) and LES (bottom). Complete mixing duct with part of the diffuser, and zoom on the region near the primary nozzle exit.

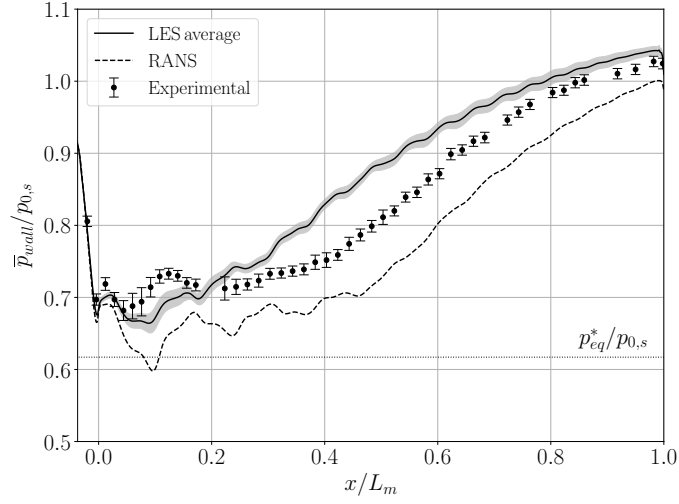


Figure 5.16: Evolution of the wall-pressure along the mixing pipe for $p_{0,p}/p_{0,s} = 4$ and $p_{out}/p_{0,s} = 1.48$. Comparison between experimental (markers), LES (solid line) and RANS (dashed line) simulations. The shaded area around the LES profile indicates the standard deviation of the wall-pressure profile. The dotted line represents the theoretical isentropic choking pressure p_{eq}^* .

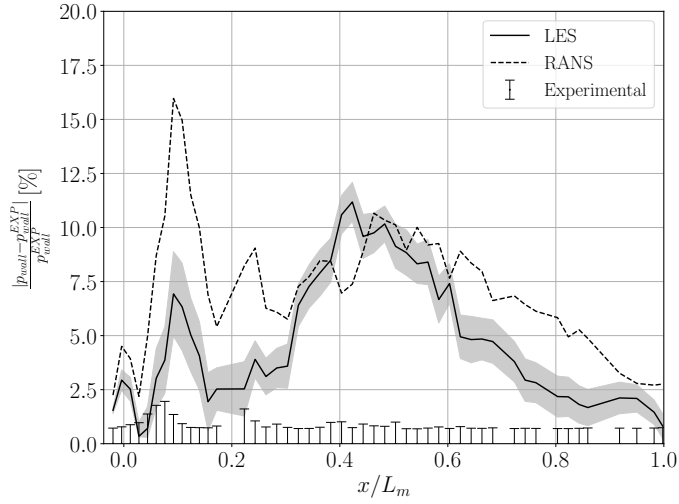


Figure 5.17: Evolution of the error on the wall-pressure prediction along the mixing pipe for $p_{0,p}/p_{0,s} = 4$ and $p_{out}/p_{0,s} = 1.48$. Comparison between LES (solid line) and RANS (dashed line) results. The shaded area around the LES profile indicates the standard deviation of the wall-pressure profile. The error bars on the experimental measurements are also shown for reference.

lower than that given by the RANS simulation. This figure shows how challenging it is to capture the flow phenomena in the mixing pipe, even with a fine grid size LES of the full cylindrical ejector. However, even though the overlap between LES and experimental results is not perfect, results are globally better with the LES, especially during the transition between the expanding flow and its recompression occurring at the inlet of the mixing pipe.

To explain those differences, an equivalent one dimensional view of the flow is obtained by averaging the Mach number within each stream. This is again done by averaging the Mach number in each of the slices obtained from the N_θ points used to generate the dividing streamline. The azimuthal averaging is performed afterwards.

The average Mach number for each stream, and its standard deviation, is shown in Figure 5.18 for LES and compared to RANS. The LES gives lower Mach numbers than the RANS, except for secondary flow where both frameworks agree from $x/L_m \simeq 0.8$. This figure also displays the equivalent, or compound, isentropic Mach number which is obtained from the compound choking theory used by Lamberts et al. [57] and defined by Equation (4.5). Although the primary Mach number still differs from $x/L_m \simeq 0.8$, the compound values also agree as the difference on the Mach number is compensated by the difference in cross-section area of each stream, the LES predicting a higher area for the primary stream.

The expression for $\overline{M}_{eq}^{is}$ is based on the compound flow indicator β introduced by Bernstein et al. [6] and it should reach unity at the location where the flow is choked, when assuming an inviscid situation without transfer at the streams' boundaries (i.e., isentropic streams, hence the notation "is"). It can be seen on Figure 5.18 that it does not exactly reach unity, even though the ejector is known to be choked from the measured mass flow-rate. As this criterion is only valid for isentropic flows, it is expected that the transfers between the streams will impact that compound Mach number. In the RANS simulation, this limit is clearly reached.

As observed in the total exergy transfer analysis performed by Lamberts et al. [55] in RANS and in the previous chapter in LES, the largest transfers occur at the very beginning of the interaction between the streams, and certainly even before the hypothetical choking location. It is thus important to include those transfers, as well as the wall-friction, in the choking analysis of the flow. An extension of the compound choking analysis was performed by Van den Berghe et al. [82] including the shear between the streams (driving effect for the secondary stream) and at the wall (resistive effect for the secondary stream), in the case without heat transfer between the streams. However, the thermal effects are also significant in the present case. It can be shown

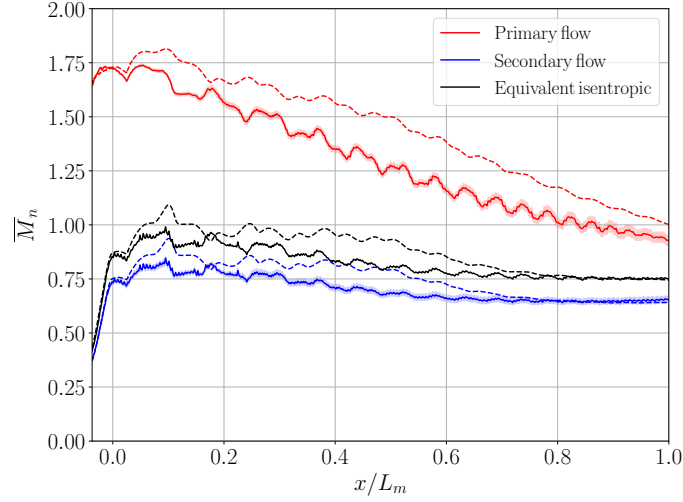


Figure 5.18: Average one-dimensional Mach number in the mixing pipe for the LES (solid lines) and RANS simulation (dashed lines).

(see B) in the general case that the equivalent Mach number with shear and heat transfer reads :

$$\overline{M}_{eq} = \left[\frac{A}{\sum_n \frac{\overline{A}_n}{\overline{M}_n^2} + \frac{1}{\frac{dp}{dx}} \sum_n \left(\frac{1+(\gamma-1)\overline{M}_n^2}{\overline{M}_n^2} f_{m,n} - \frac{\gamma-1}{\overline{u}_n} f_{h_{0,n}} \right)} \right]^{1/2}, \quad (5.1)$$

where $f_{m,n}$ and $f_{h_{0,n}}$ are the momentum and total enthalpy fluxes respectively, and defined as:

$$f_{m,n}(x) = - \int_{\Gamma_n} \left[\overline{\tau}_{ij} + \overline{\tau}_{ij}^t \right] t_i n_j ds, \quad (5.2)$$

$$f_{h_{0,n}}(x) = \int_{\Gamma_n} \left[\overline{q}_j + \overline{q}_j^t - \left(\overline{u}_i \overline{\tau}_{ij} + \tilde{u}_i \overline{\tau}_{ij}^t \right) + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''} \right] n_j ds, \quad (5.3)$$

where $\overline{\tau}_{ij}^t = -\overline{\rho u_i'' u_j''}$, $\overline{q}_j^t = \overline{\rho c_p T'' u_j''}$; n_j and $t_i = \frac{\overline{u}_i}{||\overline{u}||}$ are the components of the normal and tangent vectors to the stream inner and outer boundaries Γ_n , respectively. The presence of those two fluxes alone in the expression for the equivalent Mach number confirms the more intuitive proposition made in the previous chapter concerning the use of the total enthalpy flux and the balance between wall shear stress and shear

stress between streams (constituting the momentum flux) as metrics to evaluate the performance of ejectors.

The general equivalent Mach number obtained in Equation (5.1) allows to detect if and where the flow actually chokes, when $\overline{M}_{eq} = 1$. However, unrealistic values can be obtained when using this expression for $\overline{M}_{eq}$, coming from the square-root and the division of post-processed values. The new compound choking indicator β , related to the $\overline{M}_{eq}$, obtained in Appendix B and given by Equation (B.10):

$$\beta = \underbrace{\sum_n \frac{1 - \overline{M}_n^2}{\gamma \overline{M}_n^2} \overline{A}_n + \frac{1}{\frac{d\overline{p}}{dx}}}_{\beta^{is}} \underbrace{\sum_n \left(\frac{1 + (\gamma - 1) \overline{M}_n^2}{\gamma \overline{M}_n^2} f_{m,n} - \frac{\gamma - 1}{\gamma \overline{u}_n} f_{h_0,n} \right)}_{\beta^f}. \quad (5.4)$$

is hence used in Figure 5.19. which shows its normalized form, zoomed on the very beginning of the mixing pipe, i.e. around $x/L_m = 0$.

Figure 5.19 finally reconciles local and global choking diagnostic : the flow is found to be locally choked at about $x_c/L_m \simeq -0.0018$ and $x_c/L_m \simeq 0.0037$ for LES and RANS, respectively. According to the definition of β , it is expected that the choking section corresponds to the minimum cross-section, i.e. at $x/L_m = 0$. The small observed deviations can be attributed to post-processing errors.

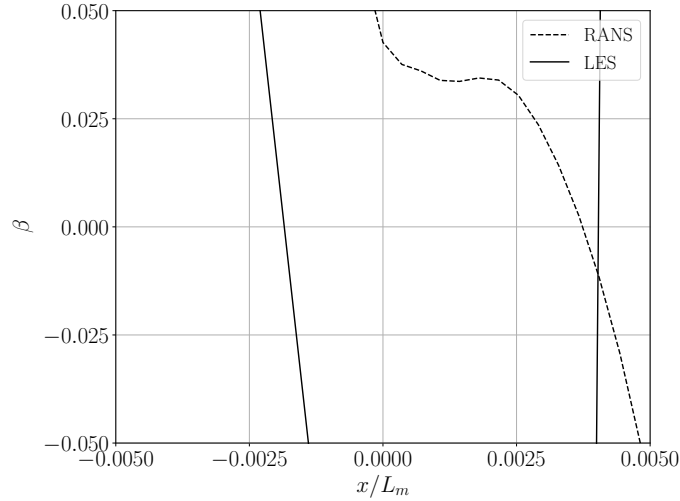


Figure 5.19: Evolution of the compound choking indicator at the beginning of the mixing pipe.

This can also lead to an explanation as to why the RANS simulations consistently underestimate the wall pressure in the beginning of the mixing pipe in the case of on-design operation when the ejector is compound-choked, whereas LES allows to capture this region more accurately. Indeed, Figure 5.16 shows that both pressure profiles start to diverge from the very beginning of the mixing pipe and according to the physical meaning of β , the compound choking indicator must reach 0 at the beginning of the mixing duct (choking section). This quantity is the sum of the term β^f due to the fluxes at the streams' boundaries and pressure gradient; and the "isentropic" part β^{is} , which solely depends on the Mach numbers and cross section areas of each stream. As the RANS simulations tend to underestimate the fluxes at the earlier development of the shear layers, as shown in Chapter 4 and highlighted in Figure 5.20 showing the evolution of the term only due to fluxes, this must be compensated either by the pressure gradient $\frac{d\bar{p}}{dx}$ and/or the isentropic term β^{is} . The evolution of those quantities is shown in Figure 5.21 but as only the balance between terms at the choking location is of importance in the present analysis, Table 5.1 displays their values measured at $x = x_c$ for each framework.

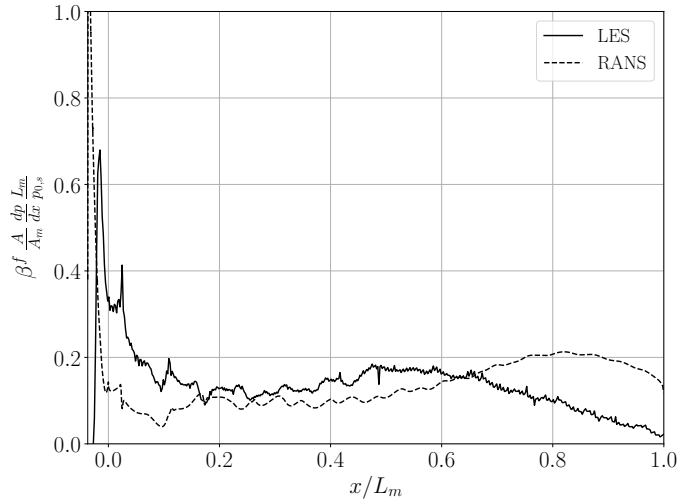


Figure 5.20: Evolution of the normalized flux term of β for LES (solid line) and RANS (dashed line).

This table shows that the underestimation of the flux term by the RANS is partially compensated by a smaller pressure gradient, which results in a 20 % underestimation of β^{is} . It was demonstrated by Bernstein et al. [6] that this term is decreased by decreasing the pressure of the flow. Thus, as a result of those compensations to satisfy the criterion

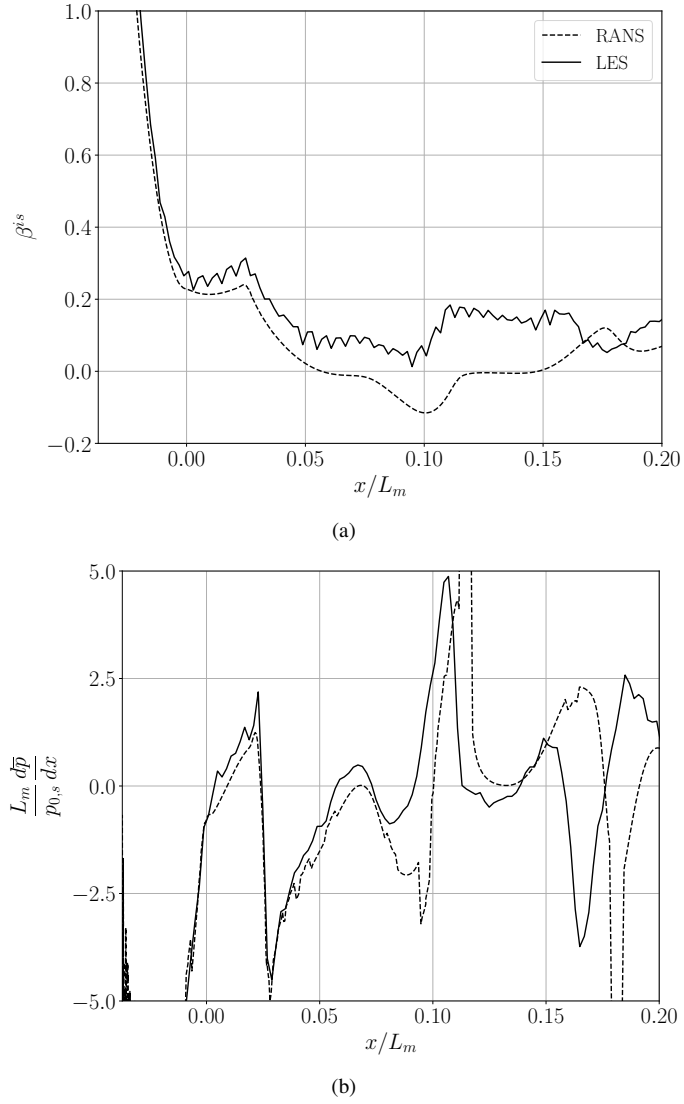


Figure 5.21: Evolution of (a) β^{is} and (b) the pressure gradient in the first part of the mixing pipe.

$\beta = 0$ at the choking sections, the RANS simulation tend to overexpand the flow, underestimating the pressure in that region. This would also provide an explanation to the fact that off-design pressure profiles are much more accurately predicted in

	x_c/L_m	$\frac{d\bar{p}}{dx} \frac{L_m}{p_{0,s}}$	β^{is}	$\beta^f \frac{A}{A_m} \frac{d\bar{p}}{dx} \frac{L_m}{p_{0,s}}$
RANS	0.0037	-0.56	0.22	0.12
LES	-0.0018	-1.44	0.28	0.34

Table 5.1: Values of the quantities relevant for the choking analysis, measured at the choking location x_c .

RANS, as the ejector is not compound-choked for these operating points, relaxing the constraint to reach $\beta = 0$. Further in the mixing pipe and up to $x/L_m \simeq 0.1$, as the pressure gradient in the RANS always underestimates that of the LES, the gap between pressure profiles continues to widen, resulting in the low pressure zone observed in Figure 5.16 in the RANS.

5.5 Conclusions and perspectives

The design of a new cylindrical supersonic air ejector was first presented. The geometry was obtained by maintaining the area ratios of the rectangular ejector designed by Lamberts [54], while optimizing the mixing pipe length and adapting the diffuser to the manufacturing constraints. The experimental setup includes mass flow-rate measurements at each inlet and outlet port of the ejector, as well as 64 pressure taps to obtain the wall pressure profile all along the ejector.

The characteristic curves of the ejector were experimentally obtained, and also using RANS simulations, for four total pressure ratios spanning the full range of operation of the ejector. This shows a fair agreement between the numerical and experimental results for the on-design entrainment ratio, with an error not exceeding 5 % for the entrainment ratio, as well as for the critical pressure where the error does not exceed 3 %. The wall pressure profiles were presented for two of the characteristic curves, and for five operating conditions. While off-design pressure profiles are in very good agreement, significant underestimations are amazingly observed between RANS simulations and experiments for on-design operations.

A high-fidelity LES of an on-design operating point near the critical point of one characteristic curve, was then also carried out. The entrainment ratio obtained from this simulation is very similar to that of the RANS simulation.

The instantaneous flow fields show an early transition of the shear layers to turbulence, as was observed in the rectangular case in Chapter 4. The dividing streamline is also computed using the LES results. It is shown to match that of RANS up to half the mixing pipe length. Subsequently, small deviations in the initial shock structures

induce larger differences in the dividing streamline position. This is also confirmed by the fields of averaged Mach number and averaged streamwise density gradient.

Furthermore, the wall-pressure profile obtained from the LES is compared to experimental and RANS simulation results. Although the LES profile still does not match the experimental profile, the agreement is much better in the choking region and along the first part of the mixing pipe (errors remaining below 5 %) than in RANS. The minimal pressure observed in experiment is also reproduced in the LES. The differences between the LES and RANS results are then further explained by analyzing the averaged Mach number in each stream, as well as the equivalent Mach number obtained from the compound-choking theory. This highlights the shortcomings of the initial theory, and a first attempt to extend it to a more general framework is made. This extension introduces a new compound choking indicator which is made of two components, (i) the usual indicator of the initial theory based on an isentropic form, β^{is} , (ii) a correction term which includes all the momentum and energy fluxes for each streams' boundaries. This updated indicator also allows to define an updated equivalent or compound Mach number $\overline{M}_{eq}$. This new indicator allows to determine if choking conditions occur and then the so-called choking section, taking into account the non-isentropic nature of the flow. According to the definition of this new indicator, this location is found to be around the very beginning of the mixing pipe, i.e. the minimal cross-section area in the ejector.

Based on this updated compound choking theory, we proposed an explanation for the observed discrepancies in the pressure profiles between RANS simulations and experiments at on-design condition. The lower pressure values predicted by RANS from the ejector throat, i.e. around the choking section, could be explained by the competition between β^{is} , β^f and the pressure gradient $\frac{d\bar{p}}{dx}$ to satisfy the constraint $\beta = 0$ at the geometrical throat in the case of compound-choked flows (on-design conditions). This balancing process would be responsible for providing a lower pressure for RANS compared to LES, and so larger discrepancies between RANS and experiments. Since the LES better predicts the pressure in this area, the major cause of the RANS failure is attributed to the less accurate evaluation of the flux terms.

More generally, those results show how challenging it is to predict local flow quantities within a supersonic ejector, as even the LES wall pressure profile does not yet satisfactorily agree with measures. Further simulations are then required at other conditions and even higher resolution to further confirm the importance of accurately capturing the fluxes through each stream to better reproduce the wall-pressure profile along the ejector. Moreover, this study also questions the use of the wall pressure along

the ejector as a metric to assess the level of accuracy of more global quantities such as the secondary flow rate (entrainment ratio) or critical pressure predicted by numerical simulations.

Finally, the new choking criterion could be used to produce improved 1D models to predict the choking of an ejector. This however involves a proper modeling or closure of the momentum and total enthalpy fluxes between streams and at walls.

Impact on the ejector performance of the addition of chevrons to the primary nozzle

6.1 Introduction

The purposeful generation of vortices in shear flows to increase performance is extensively studied in many different areas. In subsonic propulsion applications, the most recent engine nacelles used on some Boeing aircraft, such as the 737 MAX, the 787 Dreamliner, and the 747-8, are serrated at the aft section as shown in Figure 6.1.



Figure 6.1: Picture of the inaugural flight of the Boeing 747-8 [77].

This design is claimed to reduce the engine noise by 15 dB, allowing subsequent sound insulation reduction, thus decreasing the mass of the aircraft, and finally reducing its fuel consumption.

In supersonic aircraft propulsion, ejectors have historically been used to produce additional thrust. For example, on the Pratt & Whitney J-58 engine mounted on the Lockheed SR-71 Blackbird and schematized in Figure 6.2, it was responsible for 29 % of the thrust at the maximum cruising speed of Mach 3 [81]. This application field is still currently studied [74]; Thirumurthy et al. [78] proposed a preliminary design of a primary nozzle with chevrons in a propulsion application and performed RANS simulations to evaluate its performance. They observed a decrease in the recirculation zone between primary and secondary flows and which, in their is caused by the detachment of the primary flow and a decrease in ejector entrainment ratio that could not be explained. On the contrary, Kong et al. [53] observed an increase in entrainment ratio by almost 15 % and a pressure recovery increase of 8 % in supersonic ejectors. The increase of the entrainment ratio was however largely dictated by a decrease in primary mass flow-rate as only a convergent nozzle was used.

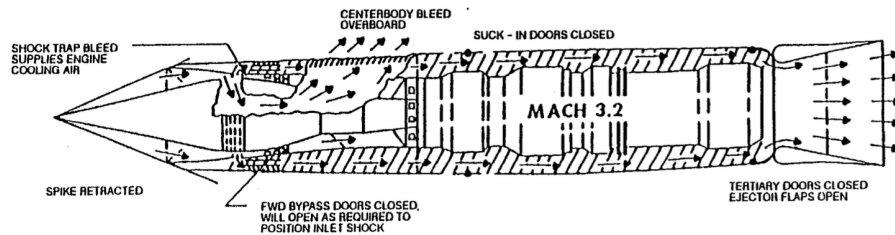


Figure 6.2: Schematic representation of the configuration of the Pratt & Whitney J-58 engine at maximum cruising speed, reproduced from Urie et al. [81].

The use of chevrons in supersonic jets was also investigated to reduce noise. Tide and Srinivasan [79] found experimentally that there was a significant impact of the number of lobes used, as well as their taper angle on the noise reduction, reaching up to a 4 dB noise level decrease. They also found that the characteristic screech of supersonic jets at specific nozzle pressure ratios was eliminated for most chevron configurations.

Finally, in external flows, vortex generators are used to control and delay boundary layer separation in aeronautics and high-speed ground travel applications.

In this chapter, we investigate the effect of using a chevroned primary nozzle for the cylindrical ejector of Chapter 5. All the tools presented in the two previous chapters

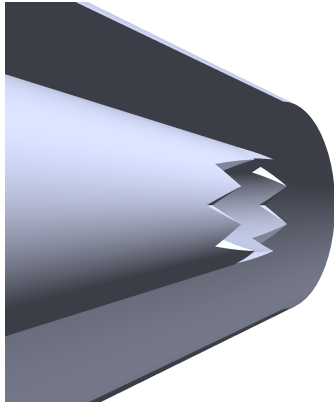
will be used to evaluate the impact of this geometric modification on the local flow characteristics, as well as on the overall performance of the ejector. This is done through a single LES of the chevroned ejector, at the same operating conditions as the LES performed on the base ejector. First, the chevron geometry will be exposed and the new numerical setup presented. Then, the main performance metrics will be compared to the reference configuration. The instantaneous flow fields, as well as averaged fields, will also be presented and compared. Finally, as it was shown to have a significant impact on performance, the total exergy fluxes through the dividing surface will be compared between both configurations.

6.2 Geometry and numerical setup

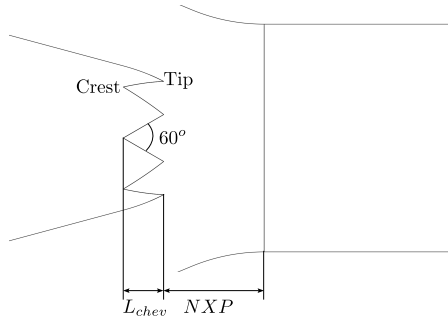
6.2.1 Chevron geometry

The geometry and number of chevrons used in this preliminary study is inspired by the study performed by Tide and Srinivasan [79]. The chevron count is thus set to $N_{chev} = 8$, with triangular shape and opening angle of $\alpha_{chev} = 60^\circ$ so that the chevron length is

$$\frac{L_{chev}}{R_e} = \frac{\sin\left(\frac{\pi}{N_{chev}}\right)}{\tan\left(\frac{\alpha_{chev}}{2}\right)} = \sqrt{3} \sin\left(\frac{\pi}{8}\right).$$



(a) Geometry of the primary nozzle exit and secondary inlet.



(b) Side view of the chevrons used in this study. The shape irregularities come from the distortion due to the conical shape of the secondary inlet inner wall.

Figure 6.3: Chevron geometry.

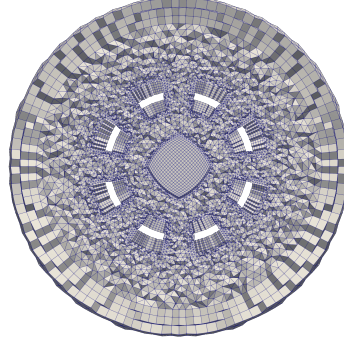
The tip of the chevron is located at the same location as the primary nozzle exit in the base design, so that the NXP is constant and corresponds to the location where no obstruction subsists between both flows. The new geometry is shown in Figure 6.3(a) with a side view of the chevrons in Figure 6.3(b), showing their dimensions.

6.2.2 Mesh

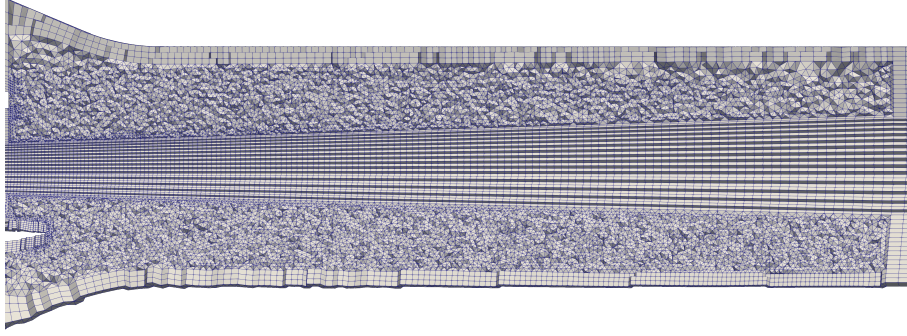
The mesh is similar to that of the base geometry but was created using the Ansys Meshing software. The structured parts are essentially identical to those generated using GMSH, with the same streamwise normal section meshing as that shown in Figure 5.10(a) but the transition to Ansys Meshing allows to obtain a better unstructured mesh around the chevrons and in the beginning of the mixing pipe, shown in Figure 6.4. However, the second order elements created by this mesh generator are incomplete, i.e. they lack the face and volume nodes, and those were subsequently added to be compatible with ArgoDG. This creates a slightly less accurate geometry than the complete second order elements but, since the geometric shapes of the ejector are relatively simple, this is not significant. The resulting mesh is constituted of around 650 thousand hexahedra, 2.7 million tetrahedra, 135 thousand prisms and 20 thousand pyramids. The prisms are mainly located in the boundary layers around the chevrons and around the unstructured part of the mixing pipe, as the mesh generator does not allow to create surface quadrangles on those boundaries. With the interpolation polynomials used in the solver, this mesh results in a total of 105 million degrees of freedom. The simulation is still performed on 1536 AMD EPYC 7763 cores but with doubled memory to accommodate the increase of elements.

6.2.3 Boundary conditions

The boundary conditions used for this simulation are similar to those used for the reference simulation. Thus, the inlet total pressure ratio is $p_{0,p}/p_{0,s} = 4$ and the outlet pressure is set to $p_{out}/p_{0,s} = 1.48$. The total temperature ratio is still unitary and the wall-friction is imposed through the wall model presented in Chapter 3 for all the outer walls and the inner walls, including the chevron walls. A free-slip boundary condition is imposed on the side-walls of the chevrons as no wall model could be applied in this region because of the unstructured mesh. The impact of this approximation should however be negligible as the flow in this region will essentially be constituted of a recirculating flow.



(a) Cross-section slice in the middle of the chevrons.



(b) Full unstructured region. The top (resp. bottom) half shows the mesh in a slice along a chevron crest (resp. tip) as defined in Figure 6.3(b).

Figure 6.4: Details of the mesh in the unstructured region of the mixing pipe.

6.3 Simulation results

The flow statistics obtained from the LES of the presented ejector were averaged over about $4t_c$. The entrainment ratio was measured to be $ER = 0.757$, with a peak-to-peak deviation of 1.55 % during the averaging period. The entrainment ratio is decreased by less than 1 % compared to the base ejector which indicates that the ejector is still likely in the on-design regime. This is confirmed by the choking criterion developed in the previous chapter and shown in Figure 6.5, which crosses zero at the beginning of the mixing pipe at around $x_c/L_m \simeq 0.0047$. This result is expected as the chevrons are only supposed to promote mixing and it was observed in Section

5.2 that the mixing duct length, and thus the mixing, did not impact the on-design entrainment ratio but had a strong impact on the critical pressure (compression ability of the ejector). It might thus be expected that the enhanced mixing obtained from the chevrons should push the critical pressure to higher levels, without changing the on-design entrainment ratio.

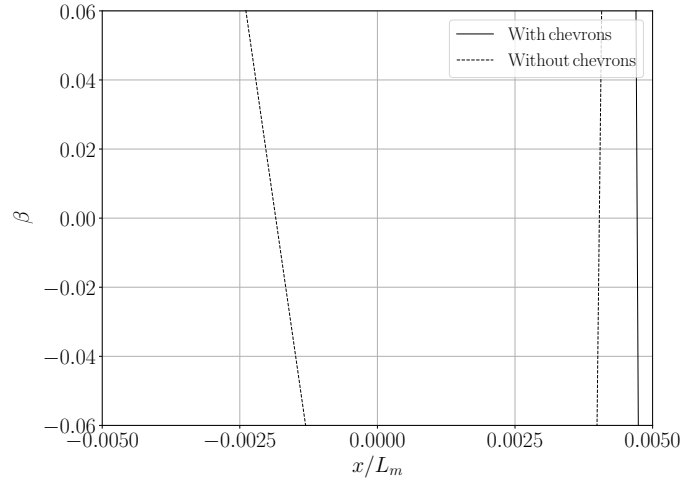
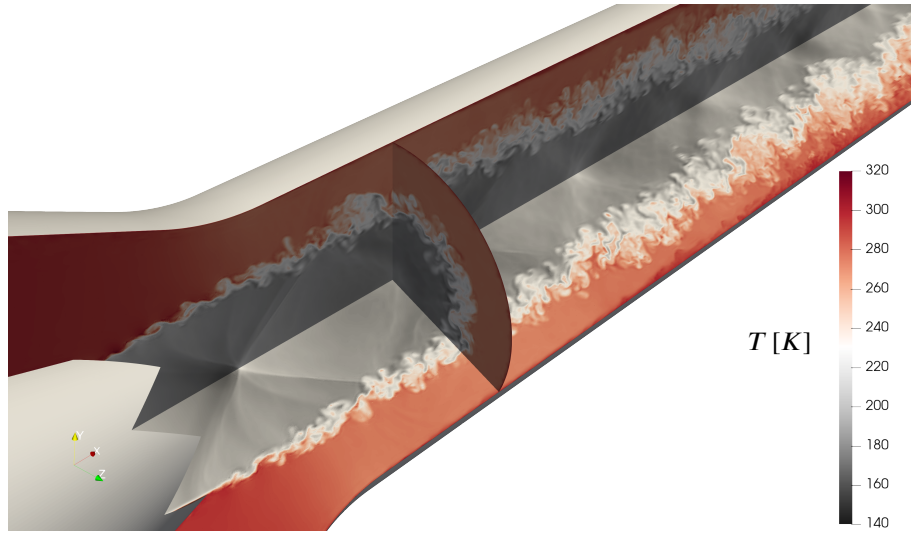


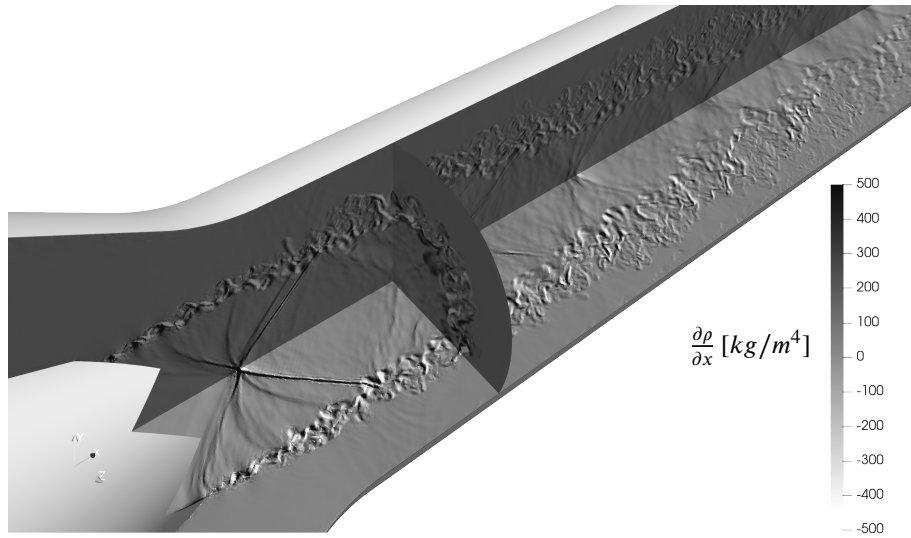
Figure 6.5: Evolution of the compound flow indicator at the beginning of the mixing pipe.

6.3.1 Instantaneous results

The instantaneous fields of temperature and streamwise density gradient are shown in Figure 6.6. The cuts in the domain are such that the flow in a slice along one chevron tip and one chevron crest is visible. This shows that the first shock cell is constituted of multiple oblique shocks emanating from the chevrons. This is in contrast to the very well-defined and conical primary oblique shock observed in the reference simulation. The clarity of the shocks is however retrieved quickly and axisymmetry is globally retrieved beyond the first reflection. The effect of the chevrons on the shear layers is visible on the cross-sectional slice shown in both instantaneous fields and placed at $x/L_m = 0.1$ as it appears to be more turbulent than what was shown in Figure 5.12 at the same location. Moreover, the beginning of the shear layers appears to produce more complex instabilities from the nozzle tip, which also differs from the base geometry where a short distance of laminar shear layer was visible.



(a) Temperature



(b) Longitudinal density gradient

Figure 6.6: Instantaneous fields in the region around the primary nozzle exit.

Figure 6.7 shows the iso-contour of the Q-criterion, colored by the azimuthal vorticity. This shows the very high level of turbulence created by the chevrons. A comparison with Figure 5.13 illustrates that impact. Indeed, in the base ejector, the iso-contour of the Q-criterion was constituted of larger structures at the beginning of the shear layer. With the addition of chevrons, the shear layers become directly fully turbulent because of the streamwise component of vorticity generated by the chevrons. We stress that the value chosen to show the Q-criterion iso-contour is identical in Figures 5.13 and 6.7 allowing to perform a fair comparison between both configurations.

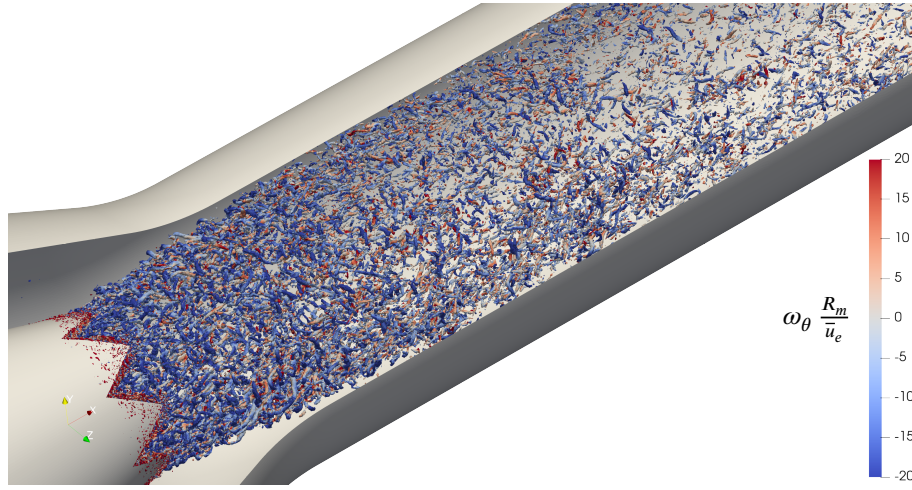


Figure 6.7: Iso-contour of the Q-criterion around the NXP, colored by the azimuthal vorticity.

The earlier transition to turbulence is further confirmed in Figure 6.8 which shows the instantaneous temperature field for the baseline geometry and the modified one in a slice of the domain, at the beginning of the mixing pipe. The addition of chevrons at the nozzle tip also modifies the position and angle of the shocks in the primary supersonic jet, which is expected to have an impact on the position of the dividing stream-surface, and thus on the stream-averaged flow quantities.

6.3.2 Averaged fields

To better analyze the structure of the flow, the averaged Mach number and stream-wise density gradient fields are shown in Figure 6.9 and compared to the baseline

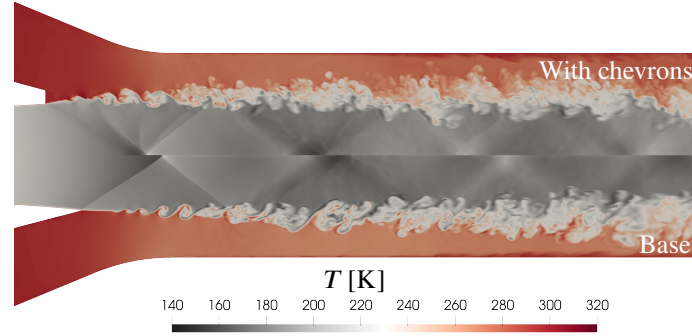
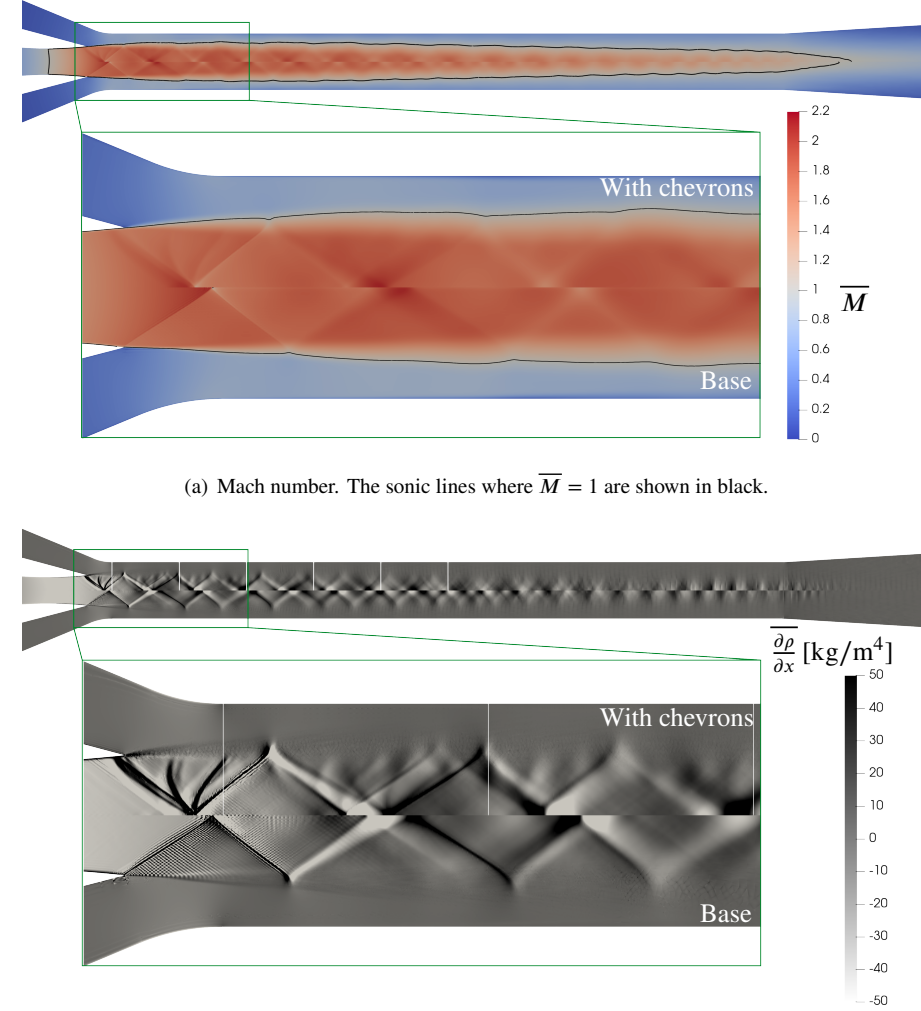


Figure 6.8: Instantaneous temperature field in a slice of the ejector for the geometry with chevrons (top half) and the base geometry (bottom half).

geometry. This figure only shows the averaged fields in a slice along a chevron tip, as it was observed that the axisymmetry of the averaged fields was rapidly retrieved in the mixing pipe and that no significant difference persisted in the flow fields after the first shock structure. However, for the sake of being complete, the comparison of the flow in slices along chevron tip and crest is shown in Figure 6.10, confirming the global axisymmetry of the flow in most of the mixing pipe.

As expected from the instantaneous temperature field of Figure 6.8, the position of the shocks is quite different between both configurations, which causes the position of the sonic line to be slightly modified as well, see Figure 6.9(a). It is observed to expand more into the secondary flow at the very beginning of the shear layer, which might translate into a faster acceleration of the secondary flow, compared to the base geometry. Moreover, the first oblique shock is largely modified due to the presence of chevrons, as presented in Figure 6.9(b). Indeed, a first oblique shock is observed slightly before the NXP and is caused by the shock emanating from the chevron crest. Then, additional shocks can be seen and are caused by the complex interaction between the chevron geometry, the primary shock and the shear layers. After the first reflection, the shock structure is much closer to that of the axisymmetric geometry even though the shift caused by the first shock cell persists. This shift is observed to be present up to the end of the mixing duct, causing the supersonic region of the flow to extend slightly further than in the base geometry, as highlighted by the end of the sonic line in Figure 6.9(a).



(b) Streamwise density gradient. The vertical lines show the location of the slices presented in Figure 6.11.

Figure 6.9: Average fields in a slice of the modified geometry, along a chevron tip (top half) and for the base geometry (bottom half). Full mixing duct and zoom on the region near the primary nozzle exit.

The streamwise density gradient field shown in Figure 6.9(b) also highlights the evolution of the shear layers as it grows much faster from the nozzle tip for the serrated geometry, compared to the base nozzle. This is caused by the faster transition

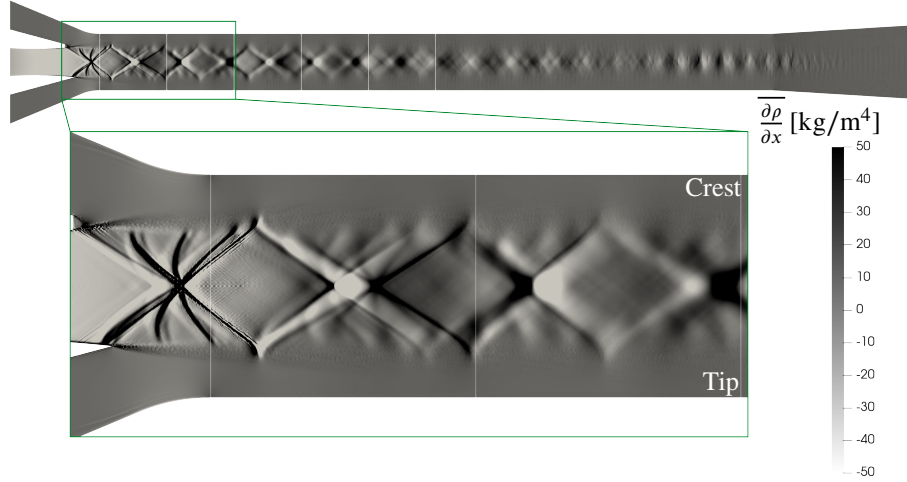


Figure 6.10: Average streamwise density gradient field in a slice of the modified geometry, along a chevron tip (top) and a chevron crest (bottom). Full mixing duct and zoom on the region near the primary nozzle exit. The vertical lines show the location of the slices presented in Figure 6.11.

to turbulence in this case, whereas it was observed that the shear layer of the base geometry remained laminar before transitioning to turbulence.

Cross-sectional slices of the average temperature field are presented in Figure 6.11, highlighting the evolution of the asymmetry of the flow. It can indeed be observed that at the beginning of the mixing pipe, i.e. at $x/L_m = 0.0$, the average temperature field presents a strong azimuthal variability in the form of lobes in the region around the dividing streamline. There are twice as many lobes as there are chevrons, one primary lobe being produced by the chevron itself, and one secondary lobe caused by the outward radial flow, consequence of the vortex generated by the chevron. This lobe is thus located between two consecutive chevrons. Still at the beginning of the mixing pipe, the core of the supersonic jet also presents strong asymmetries. These are produced by the multiple oblique shocks caused by the chevrons and which impact the flow down to the center of the supersonic jet. This can be traced back to Figure 6.10 where two different shocks are seen to be crossed by the slice taken at $x/L_m = 0.0$.

Further down the mixing pipe, the asymmetries progressively fade out as the direct effect of the chevron on the averaged flow is lost due to turbulence. However, although the secondary lobes completely vanish from $x/L_m = 0.1$, the primary lobes still persist, though rather faintly, causing the flow to present an octagonal shape around

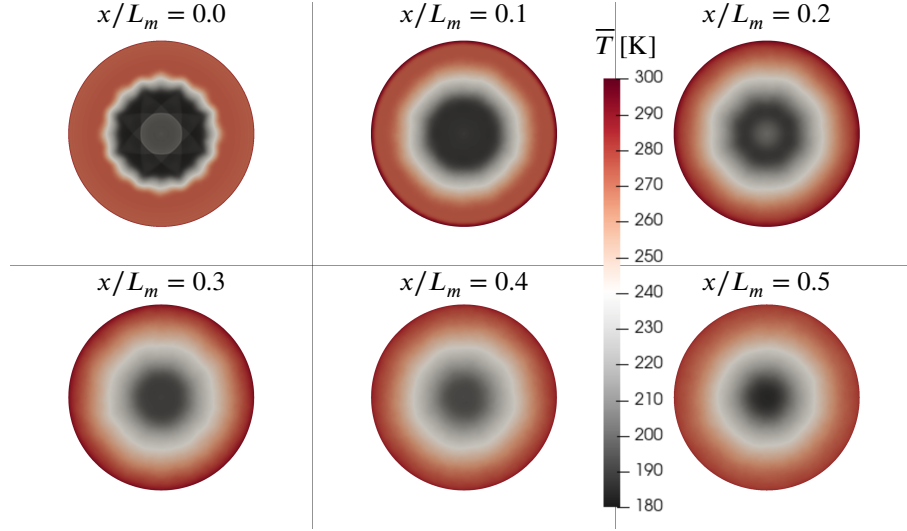


Figure 6.11: Average temperature field in streamwise slices along the mixing duct.

the primary jet. This octagonal shape is also visible in the lower temperature region surrounding the symmetry axis at $x/L_m = 0.2$. This characteristic however gradually dissipates and, at the half of the mixing pipe (i.e., in the last slice), the flow almost completely retrieves axisymmetry.

6.3.3 One-dimensional analysis

Dividing surface computation

The method used to compute the position of the dividing surface is similar to that used for the baseline ejector. However, as the position of the starting point of the dividing stream-surface depends on the azimuthal coordinate in the region around the chevrons, the beginning of the integration must be adapted. The procedure used is schematized in Figure 6.12 and goes as follows:

1. The dividing stream-surface is initialized as N_θ points equally spaced in the azimuthal direction and placed along the chevron edges.
2. Step 1: the Runge-Kutta 4 integration is performed only for the points located on the chevron crest and up to the x -coordinate of the next points on the chevron edges.

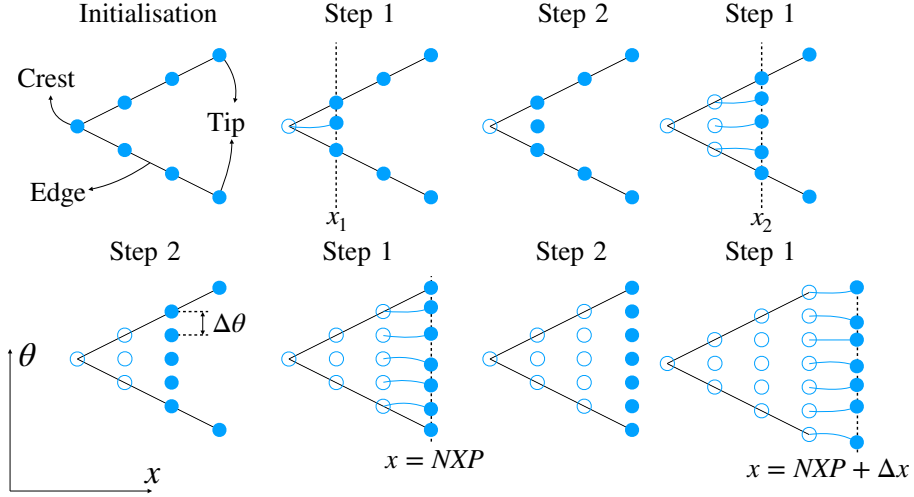
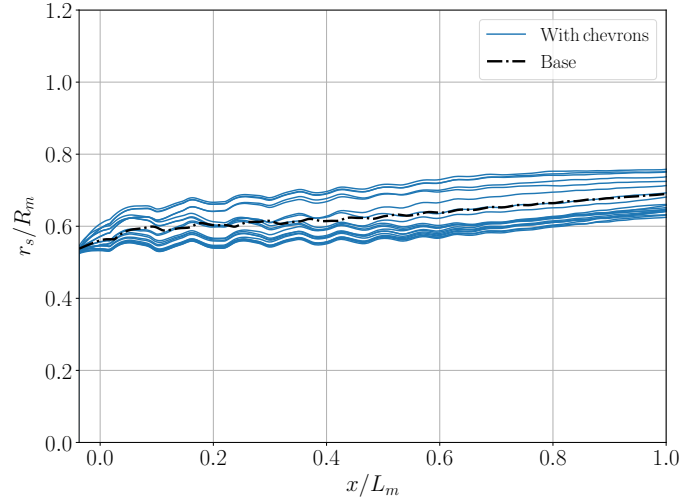


Figure 6.12: Starting procedure to compute the position of the dividing stream-surface. Note that this figure shows only four points while in reality ten points per chevron edge are used to compute the dividing stream-surface.

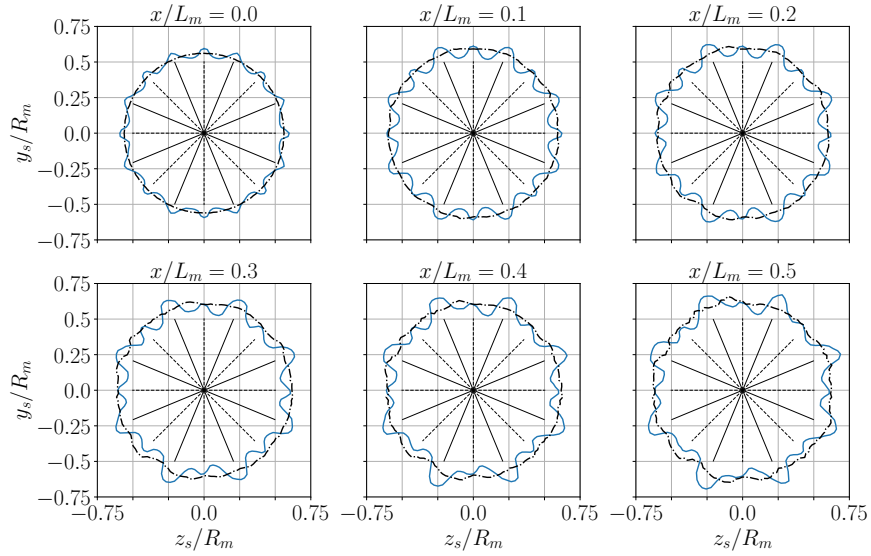
3. Step 2: the obtained streamline locations are re-interpolated using the points on the chevron edges with the same streamwise coordinate and to have equally spaced points in the azimuthal direction.
4. These edge-points are added to the points that must be used to perform step 1
5. Steps 1 and 2 are alternated up to the tip of the chevron.
6. From the chevron tip, the dividing stream-surface spans the whole azimuthal space and can be integrated as for the base cylindrical ejector.

Since the radial distance between primary and secondary flows is non-zero at the chevron crest, the dividing stream-surface can be slightly different at the beginning of the mixing duct and the integration is thus performed twice: once starting from the edges of the chevrons on the primary nozzle wall, and once starting from those on the secondary inlet wall. These two stream-surfaces are observed to coincide after a short distance after the chevron tips. Only the stream-surface emanating from the primary wall will thus be shown in the following.

The resulting dividing stream-surface position is shown in Figure 6.13 along the mixing duct. Each line shown in Figure 6.13(a) is obtained by averaging the integrated



(a) Streamwise evolution for equidistant points in the azimuthal direction



(b) Cross-sectional view of the dividing stream-surface at six streamwise positions in the mixing pipe. The black solid lines and dashed lines represent the location of the chevron tips and crests, respectively.

Figure 6.13: Evolution of the position of the dividing stream-surface. Comparison between chevroned (solid lines) and base (dash-dotted lines) geometries.

dividing streamlines emanating from the same location on each chevron. One dividing streamline shown in Figure 6.13(a) is thus the average of N_{chev} dividing streamlines. This shows the intrinsic asymmetry of the obtained flow. As expected, the dividing streamlines emanating from the primary and secondary walls almost perfectly match from the NXP, which corresponds to the lower limit of the x-axis. Moreover, the comparison with the dividing streamline obtained from the axisymmetric configuration shows that there is indeed a difference in shock patterns, as the oscillations observed in the dividing streamline position do not match between both configurations.

Figure 6.13(b) shows the location of the dividing stream-surface at several stream-wise positions in the mixing pipe of the ejector. This more clearly shows the differences between chevroned and base geometries, as well as the lobes caused by the addition of the chevrons, already observed in Figure 6.11. The dissipation of the secondary lobes, aligned with the chevron crests is also highlighted.

Stream-average profiles

As was done for previous simulations to further analyze the results, a one-dimensional view is obtained by averaging each of the flow quantities across the individual streams. The obtained profiles are compared to the LES results of the base cylindrical ejector.

First of all, the averaged Mach number is shown in Figure 6.14. This figure shows that the primary flow Mach number is slightly smaller in the case of the chevroned ejector at the exit of the primary nozzle. This is caused by the fact that the primary nozzle was locally shortened to create the chevrons. This causes a reduction of the effective outlet area of the primary nozzle and thus a reduction of the averaged outlet Mach number. The reduction of primary Mach number yields a higher pressure in the mixing pipe which reduces the suction effect of the ejector and thus explains the observed small reduction of entrainment ratio for the modified geometry. Furthermore, the primary flow Mach number does not reach values as high as for the plain nozzle ejector, which is likely caused by the multiple oblique shocks present in the primary jet close to the NXP, decreasing the averaged Mach number. Moreover, due to the turbulence generated at the primary nozzle exit, the shear layer is thicker at the beginning of the mixing pipe compared to the base geometry, which causes a reduction of the available cross-section area to expand the jet. The behavior of the primary flow Mach number from $x/L_m \simeq 0.2$ is however very similar for both configurations. For the secondary flow Mach number, a slight shift appears around $x/L_m \simeq 0.1$ and remains until the end of the mixing pipe, the Mach number measured in the modified geometry

being always smaller than that of the base ejector. This deviation however shrinks and grows to follow the oscillations of the Mach number which are slightly out of phase, as observed previously.

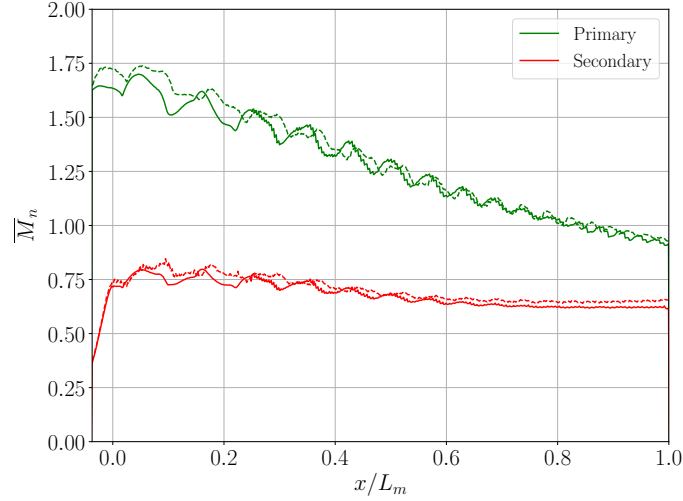


Figure 6.14: Average one-dimensional Mach number in the mixing pipe for the chevrone (solid lines) and base (dashed lines) ejectors.

The averaged total exergy is presented in Figure 6.15 similarly to what was done in Figure 4.16 and with

$$\dot{m} \xi_0(x) = \int_A \rho u \xi_0 dA \quad [\text{W}] . \quad (6.1)$$

This shows that the evolution of the total exergy convected by each stream is almost independent of the configuration of the ejector. Both curves match for both streams except at the beginning of the shear layers where the total exergy for the modified geometry tends to evolve faster than for the base geometry. This evolution is however compensated in the following of the mixing pipe and, at $x/L_m \simeq 1$, the secondary flow contains exactly the same amount of total exergy in both configurations, while the primary flow holds slightly more total exergy in the ejector with chevrons. This aligns with higher total exergy fluxes at the beginning of the shear layer and a reduction of the sink terms within the streams.

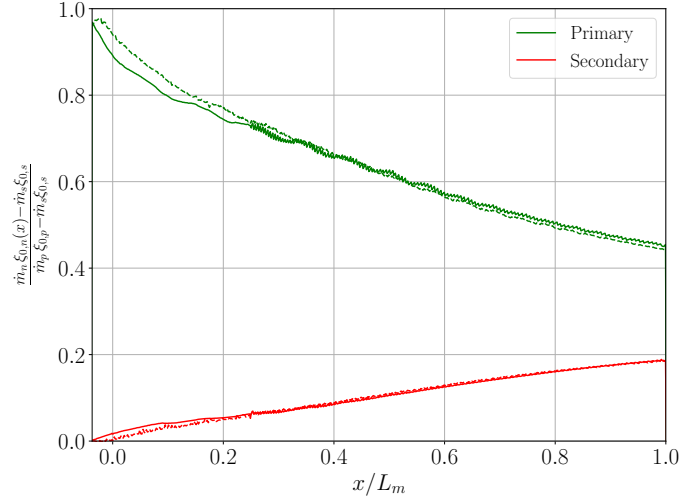


Figure 6.15: Normalized total exergy for each stream in the mixing pipe : ejector with serrated (solid lines) and plain (dashed lines) primary nozzle.

Flux of total exergy through the dividing streamlines

The components of the total exergy flux through the dividing stream-surface are presented in Figure 6.16 and compared to the plain nozzle case. This figure confirms an increase of the total exergy flux and of each of its individual components right after the NXP, causing a more rapid rise (resp. decrease) of the secondary (resp. primary) flow total exergy, as highlighted in Figure 6.15. However, upon reaching $x/L_m = 0.0$, all fluxes across the dividing stream-surface are reduced to values similar to those observed in the base ejector case. The fluxes are further decreased and in the rest of the mixing pipe, the total exergy flux components are lower for the modified geometry, compared to the base geometry. In Figure 6.15, a good match was observed between both configurations from $x/L_m \simeq 0.3$ whereas the total exergy flux is consistently smaller for the nozzle with chevrons. This suggests that less total exergy is transferred from the primary flow to the secondary flow but that this total exergy is transported more efficiently in the secondary flow, i.e. there is less dissipation.

The profiles of dissipation in each stream are presented in Figure 6.17 for both ejectors. Those profiles correspond to the sink terms presented in Equations (4.14)

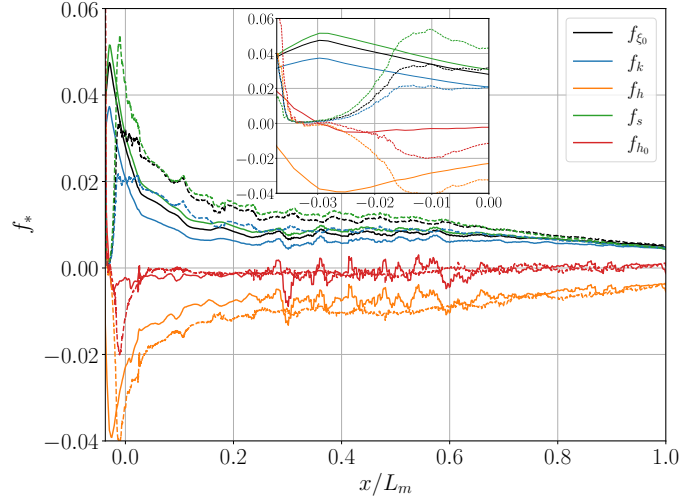


Figure 6.16: Total exergy flux components through the dividing stream-surface. Comparison between serrated (solid lines) and plain nozzle (dashed lines).

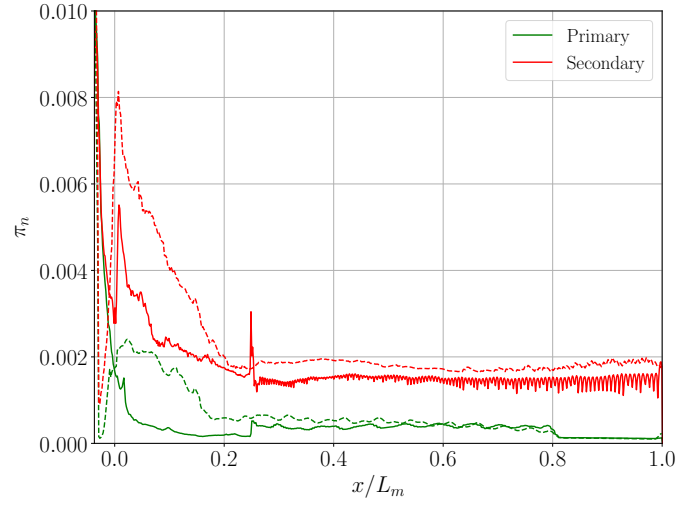
and (4.15), integrated over each stream n and non-dimensionalized, giving:

$$\pi_n = \frac{R_m}{\dot{m}_p \xi_{0,p}} T_{ref} \int_{A_n} \frac{\Phi}{T} dA, \quad (6.2)$$

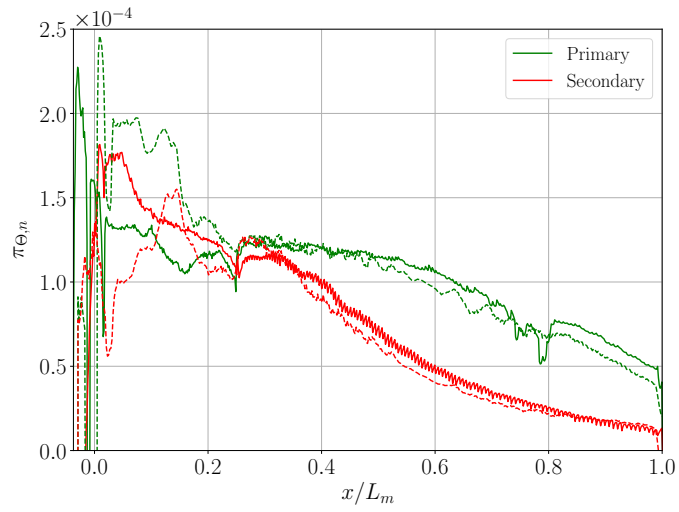
$$\pi_{\Theta,n} = \frac{R_m}{\dot{m}_p \xi_{0,p}} T_{ref} \int_{A_n} \frac{\Phi_{\Theta}}{T^2} dA. \quad (6.3)$$

These figures confirm the reduced dissipation in the mixing pipe for the modified geometry. It is however observed that only the sink term due to viscous dissipation is consistently reduced with the chevrons, whereas the sink term due to heat conduction presents certain variations. At the beginning of the mixing pipe and up to $x/L_m \simeq 0.2$, that term is decreased in the primary flow whereas it is increased in the secondary flow. In the second part of the mixing pipe, both profiles are close, with slightly higher values for the new geometry. However, given the significant difference in order of magnitude between the viscous dissipation and the heat conduction terms, the decrease of viscous dissipation dominates and causes the overall total exergy sink term to be reduced for the case with chevrons.

This reduction in dissipation has an impact on the total pressure that can be recovered at the end of the mixing pipe. This is however shown to be rather insignificant in Figure 6.18 as the increase is only of the order of 1 %. This slight increase is only due



(a) Due to viscous dissipation



(b) Due to heat conduction

Figure 6.17: Evolution of the sink terms in the mixing pipe for the serrated (solid lines) and plain (dashed lines) primary nozzle.

to the equivalent increase in primary total pressure left in this stream, as the secondary streams exhibit exactly the same total pressure in both configurations.

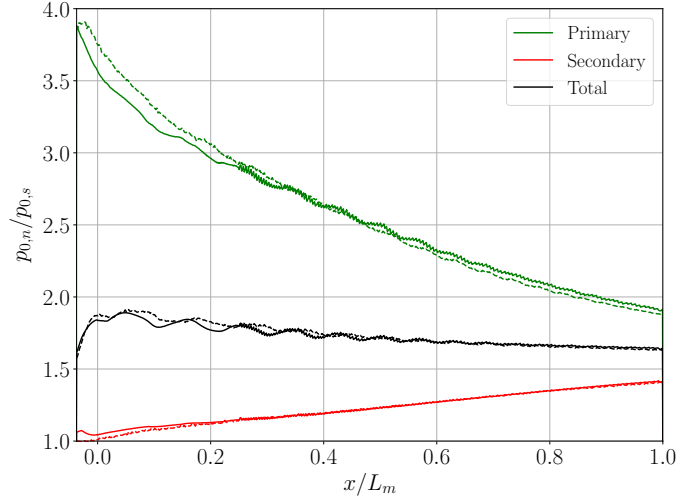


Figure 6.18: Evolution of the averaged total pressure in the mixing pipe

Mixing pipe length

With the present results, the final possible difference that can be investigated concerns the mixing pipe length. Indeed, it was shown in Chapter 4 that the maximum critical pressure of the ejector was obtained when the balance between wall shear stress and shear stress between streams was reached. This can be found to be equivalent to requiring a zero net momentum flux for the secondary stream. Figure 6.19 shows the ratio between shear stress and wall friction for both cylindrical ejectors considered.

From this figure, it is clear that the LES predicts that the chosen mixing pipe length is slightly sub-optimal. However, as an extrapolation of the ratio obtained at the end of the mixing pipe is found to produce a length increase similar to that of the rectangular ejector case in Figure 4.25, it is expected that the improvement on the critical pressure for this particular pressure ratio should be around 1 %. Then, it can be seen that both configurations produce the same shear stress ratio at the end of the mixing pipe, meaning that, from the present simulation results, there is no evidence that the optimal mixing pipe length is changed by the addition of chevrons to the primary nozzle. This does however not give any indication as to the actual value of the critical pressure; only a full characteristic curve allowing to predict that value.

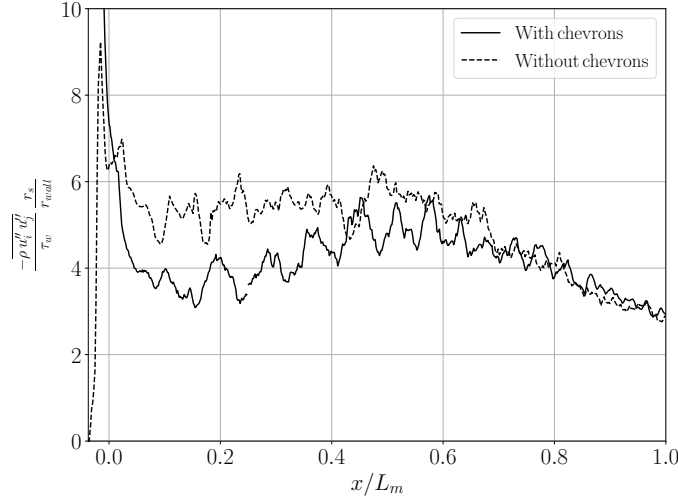


Figure 6.19: Evolution of the ratio of shear stress between streams and wall shear stress.

6.4 Conclusions and perspectives

In this chapter, the addition of chevrons to the primary nozzle was proposed to promote the mixing in the beginning of the mixing pipe of the cylindrical ejector presented in the previous chapter. The design of those chevrons is simple and triangular, and eight such chevrons were added to the nozzle. Although a different meshing tool was used, the mesh is very similar to that used for the base cylindrical ejector, combining a structured mesh in most of the ejector and an unstructured mesh at the beginning of the mixing pipe.

First, the global performance parameters are presented and compared to the baseline geometry. It is observed that the entrainment ratio is slightly reduced, but not significantly. This is due to the average Mach number $\overline{M}$ at the primary nozzle exit being slightly smaller, and hence the pressure at the beginning of the mixing pipe being slightly higher, hence producing a bit less suction. Moreover, the ejector is shown to be in the on-design regime, meaning that the critical pressure for the new geometry is, at worst, equal to that of the base geometry. However, with a single simulation, it is not possible to obtain a more precise estimation of the critical pressure.

Then, instantaneous three-dimensional fields are presented, showing a significantly earlier transition to turbulent shear layers for the chevroned ejector compared to the base geometry. The impact of the chevrons on the flow in the beginning of the mixing pipe

is thus clearly visible. Moreover, the addition of those chevrons changes the behavior of the oblique shocks emanating from the primary nozzle lip as the axisymmetry is broken.

Two-dimensional averaged fields are then presented and compared to the base ejector simulation results, showing this shift of the shock structures in the whole mixing pipe. Moreover, lobes are observed in the averaged temperature field in a cross-sectional slice at $x/L_m = 0$. These lobes disappear rapidly and are no longer clearly visible at $x/L_m = 0.1$. However, in this slice, and all further sections up to half of the mixing pipe, an octagonal shape can be observed in the time-averaged supersonic jet, which is directly caused by the presence of the eight chevrons at the primary nozzle exit.

The dividing stream-surface of the time-averaged flow is then computed, also highlighting the presence of the lobes observed in the temperature field, but in this case they persist much further downstream due to the streamwise evolution of the dividing stream-surface. The splitting of streams then also allows to compute one-dimensional averaged profiles, which gives significant insight to analyze ejector performance. The chevroned ejector is here observed to produce slightly lower Mach number flows in both streams compared to the baseline ejector, and the shift in shock structure is also visible. The lower Mach number persists up to the end of the mixing pipe for the secondary flow, whereas the primary flow Mach number reaches essentially the same value.

The stream-averaged total exergy is also presented, showing a similar behavior as in the base geometry for the secondary flow. For the primary flow, a faster decrease of total exergy is observed at the beginning of the mixing pipe. This is however compensated further downstream and the modified geometry presents slightly more total exergy at the end of the mixing pipe than the base geometry.

The slightly higher total exergy content of the flows is further shown not to be caused by the total exergy flux between streams, as all the flux components are significantly lower than in the base geometry. However, the presence of chevrons is shown to produce less dissipation in the mixing pipe, thus better conserving the total exergy. The total pressure is also shown to be slightly increased in the modified geometry, but this increase remains very small.

Finally, the balance between the shear stress between streams and the wall shear stress is analyzed and shown to be very different in the first half of the mixing pipe, yet both geometries present similar results in the second half of the mixing pipe. The presence of chevrons is thus foreseen to not impact the optimal mixing pipe length.

Overall, the presence of chevrons is observed to modify the flow significantly locally but the developed tools and diagnostics fail to predict a meaningful change in the global performance of the modified ejector. To further study the differences arising from using a serrated primary nozzle, multiple LES should be performed to obtain better estimates of all performance parameters. Complementary three-dimensional RANS simulations could also be performed, but their accuracy is bound to be rather limited given the highly unsteady nature of the flow at the beginning of the mixing pipe. Finally, the experimental setup presented in Chapter 5 could be fitted with the proposed nozzle to obtain the full characteristic curves and quantify the actual impact of this modification. In addition, the geometry and number of chevrons could also be analyzed, and their effect on the global performance could be quantified.

Conclusions and perspectives

The intensive study of the internal flow phenomena of supersonic ejectors in the last years has greatly improved the overall understanding of those devices. More accurate models were recently implemented, relying on more physically sound theories and supported by high quality RANS simulations. However, the intrinsic unsteadiness of the flow within supersonic ejectors was rarely taken into account in ejector studies as this requires scale-resolving simulations, which poses a large number of challenges.

In this context, this work aims at paving the way to perform high-fidelity scale-resolving simulations of supersonic ejectors by developing methods to reduce the intrinsic cost of the simulation (through wall-modeling), by studying the impact of the boundary conditions on the flow (through turbulence injection) and by developing accurate post-processing tools to efficiently study the flow in more details and to allow a thorough comparison to lower fidelity simulations and to experimental results.

Main achievements

The first major achievement of this work is the development of a simple wall model for the LES of compressible flows for a wide range of Mach numbers. This wall model uses a new scaling of the flow quantities, before computing the wall shear stress using Reichardt formula, which approximates the law of the wall for incompressible flows. This method was shown to maintain the accuracy of the wall shear stress prediction up to Mach numbers around 1.5, whereas an unscaled wall model showed significant increase in error, and up to 15 % for the highest Mach numbers. This wall-model was also validated at high Reynolds number.

Then, Chapter 4 presented the results of the first high-fidelity LES of a spanwise periodic supersonic air ejector. This simulation included a turbulent channel flow co-simulation to provide the correct profile of turbulence at the primary nozzle inlet, hence better reproducing the actual turbulent content of the flow when it reaches the end of the primary nozzle. Moreover, the wall resolution used in the setup allowed the development of turbulent boundary layers in the secondary inlet channels, where no inlet turbulence was supplied. A study of the extrusion depth requirement to properly capture the spectrum of the turbulent flow in the shear layers between flows was also conducted. Combined with a highly refined mesh around the shear layers, this allowed to capture the early transition of the shear layers to turbulence and this was shown to produce a shear layer evolution qualitatively matching that observed experimentally by Lamberts [54].

Subsequently, the post-processing tools developed by Lamberts et al. [55] for RANS simulations and based on total exergy fluxes were adapted to the LES framework. This included the derivation of the total exergy evolution equation and its exact averaged form. The total exergy fluxes were computed directly in simulation, allowing to accurately compute their average, contrary to the RANS framework which only allowed to compute the total exergy fluxes of the averaged flow. This subtle difference was however observed to not be the main reason for the significant underestimation of the fluxes in RANS compared to the LES. Indeed, the deviation is shown to be largely due to the consistent under-evaluation of the turbulent shear stress and heat flux density in the RANS. These quantities, which are computed in the LES but modeled in the RANS simulation, were also used to show that the turbulent Prandtl number in the region surrounding the shear layers varies significantly, from the value imposed in the RANS model of 0.85, down to 0.4.

The final achievement of Chapter 4 is the proposition of two criteria based on fluxes across the dividing streamline to characterize the performance of an ejector. The first is based on the total enthalpy flux to quantify the full mixing of the flows, as this flux was observed to reach zero when other flow quantities indicated a possible mixing of the flows. The second criterion states that the maximum critical pressure is obtained for a mixing duct length allowing to balance the driving shear stress between flows and the resistive wall friction in the secondary stream. This corresponds to requiring a zero net momentum flux in the secondary stream.

Then, the most prominent achievement presented in Chapter 5 is the complete design and building of a new cylindrical air ejector with mass flow rate and high-resolution wall pressure measurement capabilities. This ejector was based on the

rectangular ejector used in Chapter 4, with an optimized mixing pipe length to produce the maximum critical pressure. The characteristic curves of this ejector were obtained experimentally over a large range of inlet total pressure ratios and were shown to be quite accurately predicted by RANS simulations, with errors on the on-design entrainment ratio and on the critical pressure not exceeding 5 % over the considered range.

The wall-pressure evolution in the mixing pipe of the ejector was also measured experimentally and in the RANS for many different operating points. This showed that, for the on-design conditions, the numerical results systematically underestimated the experimental data. This was already observed by Lamberts [54] in the rectangular case and was largely attributed to three-dimensional effects, due to the presence of side-walls in the experiment, whereas the simulation considered an infinite span. However, although their three-dimensional RANS simulation allowed to largely reduce the error made on the wall-pressure profile, it was still significant. Moreover, the use of a cylindrical geometry for the newly built ejector was partly proposed to remove the difficultly predictable three-dimensional effects. Finally, it was observed that the wall-pressure obtained in the RANS for off-design conditions predicted the experimental data quite accurately. For all those reasons, it was concluded that a more fundamental physical explanation would explain the underestimation of the RANS wall-pressure profile.

Thus, to better understand the physical phenomena at play within the ejector and potentially impacting the accuracy of the RANS wall-pressure profile, the LES of this cylindrical ejector was performed for operating conditions near the critical point. This LES produced very similar global statistics but the wall pressure profile predicted in this framework showed an increased accuracy compared to the RANS simulation results. These discrepancies between RANS and LES wall-pressure profiles despite almost identical mass flow rates led to the derivation of a new criterion for the choking of the flow based on the compound-choking theory, but including both the momentum and the energy transfers at the boundaries of each stream.

This new choking criterion showed that for the flow to be choked, a balance between two terms must be reached at the beginning of the mixing pipe : (1) the classical isentropic compound flow indicator, obtained by Bernstein et al. [6] and used by Lamberts et al. [57] to predict the entrainment ratio of an ejector and (2) a term including the fluxes at the streams' boundaries. This balancing process finally provided an explanation to the discrepancies in wall-pressure between the LES and the RANS; and thus also to the consistent underestimation of the RANS wall-pressure profile in

on-design conditions compared to the experiment. Indeed, the flux term is under-evaluated in the RANS, as already observed in other configurations, which forces the isentropic term to be reduced to preserve the balance between both terms. This thus causes a reduction of the pressure as the isentropic term decreases with pressure.

Finally, the last chapter proposed a modification of the geometry of the designed cylindrical ejector in order to improve mixing between flows by adding triangular chevrons at the end of the primary nozzle. This was observed to increase turbulence at the beginning of the mixing pipe, as it was expected. A method to compute the position of the dividing stream-surface was proposed and the total exergy flux components between streams were computed. A comparison with the base geometry shows that each of those components are reduced in the presence of chevrons, except at the very beginning of the shear layer. On the other hand, it was observed that the total exergy convective flux is slightly higher for the modified ejector, meaning that the dissipation within the streams is reduced.

However, the diagnostics developed in this work failed to highlight significant improvements in the global performance of the ejector.

Limitations and perspectives

First of all, the wall model developed in this work was validated up to Mach numbers of around 1.5, showing consistent accuracy. The actual accuracy limit of the proposed wall model should be further investigated. However, an increase of the centerline Mach number, with a constant friction Reynolds number $Re_\tau = \frac{h \sqrt{\rho_c \tau_w}}{\mu_c}$, causes a decrease of the friction Reynolds number evaluated using quantities measured at the wall, i.e.

$$Re_\tau^* = \frac{h \sqrt{\rho_w \tau_w}}{\mu_w}, \quad (6.4)$$

as the ratio of centerline to wall density increases and that of viscosity decreases with an increase in freestream Mach number. The friction Reynolds number Re_τ should then be increased to ensure a stable fully turbulent channel flow simulation. A reference wrLES should thus be run at higher Re_τ , which requires large computational resources.

Moreover, as the target ejector application requires adiabatic walls, only this type of boundary conditions was investigated. However, many applications involving supersonic flows use cooled walls as the wall temperature rapidly rises with freestream Mach number. Thus, to be relevant in more use cases, an extension of this wall model to other thermal boundary conditions should be investigated. In this regard,

recent publications propose wall-models valid for highly compressible flows with also non-adiabatic walls, such as Griffin et al. [38].

Then, the LES of different supersonic ejector geometries, whether it be spanwise periodic or cylindrical, showed that there is still a need to bring the cost of LES down sufficiently for it to become a viable design tool. Indeed, even though the wall-modeling allowed to perform the simulation, each of the presented simulations were performed during multiple months on more than 1500 CPUs to be statistically converged and then to properly accumulate the flow statistics. This thus represents a very significant computational cost for a single operating point. To overcome this limitation, a GPU accelerated version of the code seems to be the best way forward. This should also be complemented with more efficient initialization and initial transient evacuation, in the form of non-reflecting boundary conditions for example.

The post-processing tools presented could be further improved to allow the analysis of fully unsteady simulations. There is indeed an interest in using LES to study the transition between different design points, especially from on- to off-design operation, and this unsteadiness renders the use of streamlines inadequate. Furthermore, the data generated in the presented simulations could be used to develop or tune better RANS closure models, especially around the shear layers as it was shown that in that region the turbulence modeling deviated significantly from the real turbulence data. RANS simulations with such a model could then be performed to evaluate the impact on the accuracy of this method.

In the experimental domain, the newly built ejector setup will be used to further study the full possible operation range. This includes higher total pressure ratios as well as cases where the secondary port is completely closed to study the ejector performance as a vacuum pump. Moreover, the possibility to add flow visualization capabilities to the current setup will be studied. This will be in the form of Particle Image Velocimetry (PIV) and/or Background-Oriented Schlieren (BOS) to further study the complete flow field inside the ejector. This however requires very accurately machined transparent walls for the mixing pipe of the ejector.

Finally, the LES of an ejector with serrated primary nozzle failed to demonstrate a clear improvement in the performance of the device. The study performed here was however primarily destined to prove the feasibility of such a simulation with the methods available. Multiple additional simulations should be performed to obtain a full characteristic curve. This would allow to have a much more accurate image of the impact of the addition of chevrons on the performance of the ejector. Moreover, different chevron geometries could be investigated, also including swirl within the

primary nozzle to further promote mixing. These modified primary nozzles could also be built and tested experimentally thanks to the modularity of the designed experimental setup.

Bibliography

- [1] D. N. Arnold, F. Brezzi, B. Cockburn, L. D. Marini, Unified Analysis of Discontinuous Galerkin Methods for Elliptic Problems, *SIAM Journal on Numerical Analysis* 39 (5) (2002) 1749–1779, doi: 10.1137/S0036142901384162.
- [2] P. Bader, M. Pschernig, W. Sanz, J. Woisetschlager, F. Heitmeir, W. Meile, G. Brenn, Experimental Investigation of Boundary Layer Relaminarization in Accelerated Flow, *Journal of Fluids Engineering* 140 (081201), doi: 10.1115/1.4039257.
- [3] E. Balaras, C. Benocci, U. Piomelli, Two-layer approximate boundary conditions for large-eddy simulations, *AIAA Journal* 34 (6) (1996) 1111–1119, doi: 10.2514/3.13200.
- [4] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, Y. Mercadier, Numerical and experimental investigations on supersonic ejectors, *International Journal of Heat and Fluid Flow* 26 (1) (2005) 56–70, doi: 10.1016/j.ijheatfluidflow.2004.07.003.
- [5] I. H. Bell, J. Wronski, S. Quoilin, V. Lemort, Pure and Pseudo-pure Fluid Thermophysical Property Evaluation and the Open-Source Thermophysical Property Library CoolProp, *Industrial & Engineering Chemistry Research* 53 (6) (2014) 2498–2508, doi: 10.1021/ie4033999.
- [6] A. Bernstein, W. H. Heiser, C. Hevenor, Compound-Compressible Nozzle Flow, *Journal of Applied Mechanics* 34 (3) (1967) 548–554, doi: 10.1115/1.3607742.
- [7] G. Besagni, R. Mereu, F. Inzoli, Ejector refrigeration: A comprehensive review, *Renewable and Sustainable Energy Reviews* 53 (2016) 373–407, doi: 10.1016/j.rser.2015.08.059.

- [8] BIPM, IEC, IECC, ILAC, ISO, IUPAC, IUPAP, OIML, Evaluation of measurement data — Guide to the expression of uncertainty in measurement, Joint Committee for Guides in Metrology, JCGM 100:2008, 2008.
- [9] S. Bocquet, P. Sagaut, J. Jouhaud, A compressible wall model for large-eddy simulation with application to prediction of aerothermal quantities, *Physics of Fluids* 24 (6) (2012) 065103, doi: 10.1063/1.4729614.
- [10] A. Bouhangel, P. Desevaux, E. Gavignet, Visualization of flow instabilities in supersonic ejectors using Large Eddy Simulation, *Journal of Visualization* 18 (1) (2015) 17–19, doi: 10.1007/s12650-014-0231-4.
- [11] M. Boxho, M. Rasquin, T. Toulorge, G. Dergham, G. Winckelmans, K. Hillewaert, Data-driven wall shear stress model for Large Eddy Simulations applied to flow separation, DLES13, Udine, Italy, 2022.
- [12] C. Brun, M. Petrovan Boiarciuc, M. Haberkorn, P. Comte, Large eddy simulation of compressible channel flow: Arguments in favour of universality of compressible turbulent wall bounded flows, *Theoretical and Computational Fluid Dynamics* 22 (3-4) (2008) 189–212, doi: 10.1007/s00162-007-0073-y.
- [13] M. Catchirayer, J.-F. Boussuge, P. Sagaut, M. Montagnac, D. Papadogiannis, X. Garnaud, Extended integral wall-model for large-eddy simulations of compressible wall-bounded turbulent flows, *Physics of Fluids* 30 (6) (2018) 065106, doi: 10.1063/1.5030859.
- [14] A. Chambers, R. Antonia, L. Fulachier, Turbulent Prandtl number and spectral characteristics of a turbulent mixing layer, *International Journal of Heat and Mass Transfer* 28 (8) (1985) 1461–1468, doi: 10.1016/0017-9310(85)90248-0.
- [15] W. Chen, M. Liu, D. Chong, J. Yan, A. B. Little, Y. Bartosiewicz, A 1D model to predict ejector performance at critical and sub-critical operational regimes, *International Journal of Refrigeration* 36 (6) (2013) 1750–1761, doi: 10.1016/j.ijrefrig.2013.04.009.
- [16] H. Choi, P. Moin, Grid-point requirements for large eddy simulation: Chapman’s estimates revisited, *Physics of Fluids* 24 (1) (2012) 011702, doi: 10.1063/1.3676783.
- [17] D. Chong, M. Hu, W. Chen, J. Wang, J. Liu, J. Yan, Experimental and numerical analysis of supersonic air ejector, *Applied Energy* 130 (2014) 679–684, doi: 10.1016/j.apenergy.2014.02.023.

- [18] D. Chung, D. I. Pullin, Large-eddy simulation and wall modelling of turbulent channel flow, *Journal of Fluid Mechanics* 631 (2009) 281–309, doi: 10.1017/S0022112009006867.
- [19] S. Croquer, O. Lamberts, S. Poncet, S. Moreau, Y. Bartosiewicz, Large Eddy Simulation of a supersonic air ejector, *Applied Thermal Engineering* 209 (2022) 118177, doi: 10.1016/j.applthermaleng.2022.118177.
- [20] L. Davidson, S. H. Peng, Hybrid LES-RANS modelling: a one-equation SGS model combined with a $k-\omega$ model for predicting recirculating flows, *International Journal for Numerical Methods in Fluids* 43 (9) (2003) 1003–1018, doi: 10.1002/fld.512.
- [21] C. C. de Wiart, A. Frère, K. Hillewaert, Towards Wall-Modelled Implicit LES with a Discontinuous Galerkin Method, *ERCOFTAC*, 2014.
- [22] C. C. de Wiart, K. Hillewaert, L. Bricteux, G. Winckelmans, Implicit LES of free and wall-bounded turbulent flows based on the discontinuous Galerkin/symmetric interior penalty method, *International Journal for Numerical Methods in Fluids* 78 (6) (2015) 335–354, doi: 10.1002/fld.4021.
- [23] J. W. Deardorff, A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers, *Journal of Fluid Mechanics* 41 (2) (1970) 453–480, doi: 10.1017/S0022112070000691.
- [24] R. Debroeyer, M. Rasquin, T. Toulorge, Y. Bartosiewicz, G. Winckelmans, A wall model for large-eddy simulation of highly compressible flows based on a new scaling of the law of the wall, *Journal of Fluid Mechanics* 980 (2024) A9, doi: 10.1017/jfm.2023.1086.
- [25] R. Debroeyer, T. Toulorge, M. Rasquin, G. Winckelmans, Y. Bartosiewicz, Wall-resolved LES of turbulent flow in a supersonic nozzle, *Rhodes, Greece*, 2021.
- [26] R. Debroeyer, T. Toulorge, M. Rasquin, G. Winckelmans, Y. Bartosiewicz, Analysis of Entrainment and Mixing in a Supersonic Air Ejector Using Large-Eddy Simulation, *International Journal of Heat and Fluid Flow* doi: 10.2139/ssrn.4877056, [Under review].
- [27] R. Debroeyer, G. Winckelmans, Y. Bartosiewicz, Numerical and experimental characterization of a cylindrical supersonic air ejector, *International Journal of Heat and Fluid Flow* doi: 10.2139/ssrn.4992378, [Under review].

- [28] C. Duprat, G. Balarac, O. Métais, P. M. Congedo, O. Brugière, A wall-layer model for large-eddy simulations of turbulent flows with/out pressure gradient, *Physics of Fluids* 23 (1) (2011) 015101, doi: 10.1063/1.3529358.
- [29] I. Eames, S. Aphornratana, H. Haider, A theoretical and experimental study of a small-scale steam jet refrigerator, *International Journal of Refrigeration* 18 (6) (1995) 378–386, doi: 10.1016/0140-7007(95)98160-M.
- [30] J. Fabri, R. Siestrunk, Supersonic Air Ejectors, in: *Advances in Applied Mechanics*, vol. 5, Elsevier, 1–34, doi: 10.1016/S0065-2156(08)70016-4, 1958.
- [31] A. Frère, C. Carton de Wiart, K. Hillewaert, P. Chatelain, G. Winckelmans, Application of wall-models to discontinuous Galerkin LES, *Physics of Fluids* 29 (8) (2017) 085111, doi: 10.1063/1.4998977.
- [32] A. Frère, K. Hillewaert, P. Chatelain, G. Winckelmans, High Reynolds Number Airfoil: From Wall-Resolved to Wall-Modeled LES, *Flow, Turbulence and Combustion* 101 (2) (2018) 457–476, doi: 10.1007/s10494-018-9972-9.
- [33] J. García del Valle, J. M. Sáiz Jabardo, F. Castro Ruiz, J. San José Alonso, A one dimensional model for the determination of an ejector entrainment ratio, *International Journal of Refrigeration* 35 (4) (2012) 772–784, doi: 10.1016/j.ijrefrig.2011.11.020.
- [34] G. Gassner, D. A. Kopriva, A Comparison of the Dispersion and Dissipation Errors of Gauss and Gauss–Lobatto Discontinuous Galerkin Spectral Element Methods, *SIAM Journal on Scientific Computing* 33 (5) (2011) 2560–2579, doi: 10.1137/100807211.
- [35] C. Geuzaine, J.-F. Remacle, Gmsh: A 3-D finite element mesh generator with built-in pre- and post-processing facilities: THE GMSH PAPER, *International Journal for Numerical Methods in Engineering* 79 (11) (2009) 1309–1331, doi: 10.1002/nme.2579.
- [36] E. Gillyns, S. Buckingham, G. Winckelmans, Implementation and Validation of an Algebraic Wall Model for LES in Nek5000, *Flow, Turbulence and Combustion* 109 (4) (2022) 1111–1131, doi: 10.1007/s10494-022-00378-y.
- [37] K. P. Griffin, L. Fu, P. Moin, Velocity transformation for compressible wall-bounded turbulent flows with and without heat transfer, *Proceedings of the National Academy of Sciences* 118 (34) (2021) e2111144118, doi: 10.1073/pnas.2111144118.

- [38] K. P. Griffin, L. Fu, P. Moin, Near-wall model for compressible turbulent boundary layers based on an inverse velocity transformation, *Journal of Fluid Mechanics* 970 (2023) A36, doi: 10.1017/jfm.2023.627.
- [39] A. Hemidi, F. Henry, S. Leclaire, J.-M. Seynhaeve, Y. Bartosiewicz, CFD analysis of a supersonic air ejector. Part II: Relation between global operation and local flow features, *Applied Thermal Engineering* 29 (14-15) (2009) 2990–2998, doi: 10.1016/j.applthermaleng.2009.03.019.
- [40] K. Hillewaert, Development of the discontinuous Galerkin method for high-resolution, large scale CFD and acoustics in industrial geometries, Ph.D. thesis, Ecole Polytechnique de Louvain, 2013.
- [41] L. Howarth, Concerning the effect of compressibility on laminar boundary layers and their separation, in: Royal Society of London, A, 1947.
- [42] B. Huang, J. Chang, C. Wang, V. Petrenko, A 1-D analysis of ejector performance, *International Journal of Refrigeration* 22 (5) (1999) 354–364, doi: 10.1016/S0140-7007(99)00004-3.
- [43] P. G. Huang, G. N. Coleman, Van Driest transformation and compressible wall-bounded flows, *AIAA Journal* 32 (10) (1994) 2110–2113, doi: 10.2514/3.12259.
- [44] P. G. Huang, G. N. Coleman, P. Bradshaw, Compressible Turbulent Channel Flows: Dns Results and Modelling, *Journal of Fluid Mechanics* 305 (1995) 185–218, doi: 10.1017/S0022112095004599.
- [45] International Organization for Standardization, ISO 5167-1:2022 - Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full - Part 1: General principles and requirements, 2022.
- [46] International Organization for Standardization, ISO 5167-2:2022 - Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full - Part 2: Orifice plates, 2022.
- [47] B. Kader, Temperature and concentration profiles in fully turbulent boundary layers, *International Journal of Heat and Mass Transfer* 24 (9) (1981) 1541–1544, doi: 10.1016/0017-9310(81)90220-9.

- [48] S. K. Karthick, S. M. V. Rao, G. Jagadeesh, K. P. J. Reddy, Passive scalar mixing studies to identify the mixing length in a supersonic confined jet, *Experiments in Fluids* 58 (5) (2017) 59, doi: 10.1007/s00348-017-2342-x.
- [49] G. Karypis, V. Kumar, A Fast and High Quality Multilevel Scheme for Partitioning Irregular Graphs, *SIAM Journal on Scientific Computing* 20 (1) (1998) 359–392, doi: 10.1137/S1064827595287997.
- [50] S. Kawai, J. Larsson, Wall-modeling in large eddy simulation: Length scales, grid resolution, and accuracy, *Physics of Fluids* 24 (1) (2012) 015105, doi: 10.1063/1.3678331.
- [51] J. H. Keenan, E. P. Neumann, F. Lustwerk, An Investigation of Ejector Design by Analysis and Experiment, *Journal of Applied Mechanics* 17 (3) (1950) 299–309, doi: 10.1115/1.4010131.
- [52] K. Kitamura, E. Shima, Improvements of simple low-dissipation AUSM against shock instabilities in consideration of interfacial speed of sound, in: *ECCOMAS CFD 2010*, Lisbon, Portugal, 2010.
- [53] F. Kong, Y. Jin, T. Setoguchi, H. D. Kim, Numerical analysis of Chevron nozzle effects on performance of the supersonic ejector-diffuser system, *Journal of Thermal Science* 22 (5) (2013) 459–466, doi: 10.1007/s11630-013-0648-4.
- [54] O. Lamberts, Choking and Mixing Phenomena in Supersonic Ejectors, Ph.D. thesis, UCLouvain, Louvain-la-Neuve, Belgium, 2018.
- [55] O. Lamberts, P. Chatelain, Y. Bartosiewicz, New methods for analyzing transport phenomena in supersonic ejectors, *International Journal of Heat and Fluid Flow* 64 (2017) 23–40, doi: 10.1016/j.ijheatfluidflow.2017.01.009.
- [56] O. Lamberts, P. Chatelain, Y. Bartosiewicz, Numerical and experimental evidence of the Fabri-choking in a supersonic ejector, *International Journal of Heat and Fluid Flow* 69 (2018) 194–209, doi: 10.1016/j.ijheatfluidflow.2018.01.002.
- [57] O. Lamberts, P. Chatelain, N. Bourgeois, Y. Bartosiewicz, The compound-choking theory as an explanation of the entrainment limitation in supersonic ejectors, *Energy* 158 (2018) 524–536, doi: 10.1016/j.energy.2018.06.036.
- [58] J. Lee, M. Cho, H. Choi, Large eddy simulations of turbulent channel and boundary layer flows at high Reynolds number with mean wall shear stress boundary condition, *Physics of Fluids* 25 (11) (2013) 110808, doi: 10.1063/1.4819342.

- [59] M. Lee, R. D. Moser, Direct numerical simulation of turbulent channel flow up to $Re_\tau \approx 5200$, arXiv:1410.7809 [physics] ArXiv: 1410.7809.
- [60] N. Maheu, V. Moureau, P. Domingo, F. Duchaine, G. Balarac, Large-eddy simulations of flow and heat transfer around a low-Mach number turbine blade, in: CTR Summer Program, Center for Turbulence Research, NASA Ames/Stanford Univ., 2012.
- [61] A. Metsue, R. Debroeyer, S. Poncet, Y. Bartosiewicz, An improved thermodynamic model for supersonic real-gas ejectors using the compound-choking theory, *Energy* 238 (2022) 121856, doi: 10.1016/j.energy.2021.121856.
- [62] A. Metsue, S. Poncet, Y. Bartosiewicz, A homogeneous relaxation model algorithm in density-based formulation with novel tabulated method for the modeling of CO₂ flashing nozzles, *International Journal of Refrigeration* 161 (2024) 188–201, doi: 10.1016/j.ijrefrig.2024.02.020.
- [63] D. Modesti, S. Pirozzoli, Reynolds and Mach number effects in compressible turbulent channel flow, *International Journal of Heat and Fluid Flow* 59 (2016) 33–49, doi: 10.1016/j.ijheatfluidflow.2016.01.007.
- [64] J. T. Munday, D. F. Bagster, A New Ejector Theory Applied to Steam Jet Refrigeration, *Industrial & Engineering Chemistry Process Design and Development* 16 (4) (1977) 442–449, doi: 10.1021/i260064a003.
- [65] G. I. Park, P. Moin, An improved dynamic non-equilibrium wall-model for large eddy simulation, *Physics of Fluids* 26 (1) (2014) 015108, doi: 10.1063/1.4861069.
- [66] P.-O. Persson, J. Peraire, Sub-Cell Shock Capturing for Discontinuous Galerkin Methods, in: 44th AIAA Aerospace Sciences Meeting and Exhibit, American Institute of Aeronautics and Astronautics, Reno, Nevada, doi: 10.2514/6.2006-112, 2006.
- [67] U. Piomelli, E. Balaras, Wall-Layer Models for Large-Eddy Simulations, *Annual Review of Fluid Mechanics* 34 (1) (2002) 349–374, doi: 10.1146/annurev.fluid.34.082901.144919.
- [68] U. Piomelli, J. Ferziger, P. Moin, J. Kim, New approximate boundary conditions for large eddy simulations of wall-bounded flows, *Physics of Fluids A: Fluid Dynamics* 1 (6) (1989) 1061–1068, doi: 10.1063/1.857397.

- [69] S. B. Pope, Turbulent Flows, Cambridge University Press, 1 edn., doi: 10.1017/CBO9780511840531, 2000.
- [70] M. Rasquin, A. Bauer, K. Hillewaert, Scientific *post hoc* and *in situ* visualisation of high-order polynomial solutions from massively parallel simulations, International Journal of Computational Fluid Dynamics 33 (4) (2019) 171–180, doi: 10.1080/10618562.2019.1618453.
- [71] H. Reichardt, Vollständige Darstellung der turbulenten Geschwindigkeitsverteilung in glatten Leitungen, ZAMM - Zeitschrift für Angewandte Mathematik und Mechanik 31 (7) (1951) 208–219, doi: 10.1002/zamm.19510310704.
- [72] E. Rogdakis, G. Alexis, Design and parametric investigation of an ejector in an air-conditioning system, Applied Thermal Engineering 20 (2) (2000) 213–226, doi: 10.1016/S1359-4311(99)00013-7.
- [73] U. Schumann, Subgrid scale model for finite difference simulations of turbulent flows in plane channels and annuli, Journal of Computational Physics 18 (4) (1975) 376–404, doi: 10.1016/0021-9991(75)90093-5.
- [74] L. Shi, G. Zhao, Y. Yang, D. Gao, F. Qin, X. Wei, G. He, Research progress on ejector mode of rocket-based combined-cycle engines, Progress in Aerospace Sciences 107 (2019) 30–62, doi: 10.1016/j.paerosci.2019.03.003.
- [75] K. Stewartson, Correlated Incompressible and Compressible Boundary Layers, in: Proceedings of the Royal Society of London, A, Royal Society of London, 84–100, 1949.
- [76] J. Telega, R. Szwaba, P. Doerffer, Shock Waves Asymmetry in a Symmetric Nozzle, Symmetry 11 (12) (2019) 1477, doi: 10.3390/sym11121477.
- [77] The Boeing Company, 747-8, available online at <https://www.boeing.com/commercial/747-8>; Accessed : 05/09/2024., 2010.
- [78] D. Thirumurthy, G. Blaisdell, A. Lyrantzis, J. Sullivan, Preliminary Design and Computational Analysis of an Ejector Nozzle with Chevrons, in: 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, American Institute of Aeronautics and Astronautics, Orlando, Florida, doi: 10.2514/6.2011-918, 2011.

- [79] P. Tide, K. Srinivasan, Effect of chevron count and penetration on the acoustic characteristics of chevron nozzles, *Applied Acoustics* 71 (3) (2010) 201–220, doi: 10.1016/j.apacoust.2009.08.010.
- [80] A. Trettel, J. Larsson, Mean velocity scaling for compressible wall turbulence with heat transfer, *Physics of Fluids* 28 (2) (2016) 026102, doi: 10.1063/1.4942022.
- [81] D. Urie, H. G. Hornung, P. Germain, L. A. D. Company, G. A. Laboratories, Course AE107 Case Studies in Engineering: The SR-71 Blackbird, 2020.
- [82] J. Van den Berghe, M. A. Mendez, Y. Bartosiewicz, An extension of the compound flow theory with shear and friction, arXiv:2401.07236 [physics], 2024.
- [83] J. Van Den Berghe, J. B. Vemula, Y. Bartosiewicz, M. A. Mendez, A Machine Learning-Based Calibration of a 1D Ejector Model from CFD, in: 36th International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems (ECOS 2023), ECOS 2023, Las Palmas De Gran Canaria, Spain, 424–435, doi: 10.52202/069564-0039, 2023.
- [84] E. R. Van Driest, Turbulent Boundary Layer in Compressible Fluids, *Journal of the Aeronautical Sciences* 18 (3) (1951) 145–160, doi: 10.2514/8.1895.
- [85] J. B. Vemula, M. Duponcheel, Y. Bartosiewicz, Numerical study of the complete operating map of ejectors in ultra high bypass ratio engine bleed, in: Proceeding of 8th Thermal and Fluids Engineering Conference (TFEC), Begellhouse, College Park, USA, 1–4, doi: 10.1615/TFEC2023.aer.046042, 2023.
- [86] S. B. Verma, C. Manisankar, Origin of flow asymmetry in planar nozzles with separation, *Shock Waves* 24 (2) (2014) 191–209, doi: 10.1007/s00193-013-0492-1.
- [87] P. S. Volpiani, P. S. Iyer, S. Pirozzoli, J. Larsson, Data-driven compressibility transformation for turbulent wall layers, *Physical Review Fluids* 5 (5) (2020) 052602, doi: 10.1103/PhysRevFluids.5.052602.
- [88] A. Walz, *Boundary layers of flow and temperature*, M.I.T. Press, Cambridge, Mass., 1969.
- [89] G. Winckelmans, M. Duponcheel, Modeling the law of the wake using an offset from the wall, *Physical Review Fluids* 6 (6) (2021) 064606, doi: 10.1103/PhysRevFluids.6.064606.

-
- [90] P. Wu, J. Meyers, A constraint for the subgrid-scale stresses in the logarithmic region of high Reynolds number turbulent boundary layers: A solution to the log-layer mismatch problem, *Physics of Fluids* 25 (1) (2013) 015104, doi: 10.1063/1.4774344.
- [91] J. Yao, F. Hussain, Turbulence statistics and coherent structures in compressible channel flow, *Physical Review Fluids* 5 (8) (2020) 084603, doi: 10.1103/PhysRevFluids.5.084603.
- [92] Y.-S. Zhang, W.-T. Bi, F. Hussain, X.-L. Li, Z.-S. She, Mach-Number-Invariant Mean-Velocity Profile of Compressible Turbulent Boundary Layers, *Physical Review Letters* 109 (5) (2012) 054502, doi: 10.1103/PhysRevLett.109.054502.
- [93] Z. Zhou, G. He, X. Yang, A wall model based on neural networks for LES of turbulent flows over periodic hills doi: 10.48550/ARXIV.2011.04157, publisher: arXiv Version Number: 1.

Total exergy fluxes in the RANS framework

The expression for the total exergy flux obtained in Section 4.3.3 contains terms that are not available in the RANS framework. It is thus necessary to derive the expressions of the fluxes in this framework, as was done by Lamberts et al. [55] and is reminded here.

The total exergy of the mean flow is

$$\hat{\xi}_0 = (\tilde{h} + \tilde{k}) - h_{ref} - T_{ref} (\hat{s} - s_{ref}), \quad (\text{A.1})$$

where the mean kinetic energy is $\tilde{k} = \frac{1}{2} \tilde{u}_k \tilde{u}_k + \frac{1}{2} \overline{u_k'' u_k''}$. The total exergy differential is

$$d\hat{\xi}_0 = d(\tilde{h} + \tilde{k}) - T_{ref} d\hat{s}. \quad (\text{A.2})$$

The entropy differential is obtained from

$$\tilde{T} d\hat{s} = d\tilde{h} - \frac{1}{\tilde{\rho}} d\tilde{p}, \quad (\text{A.3})$$

which is not an exact averaging of the Gibbs formula but again it gives the entropy based on the mean-flow variable (mean temperature, mean pressure, mean density).

The mean kinetic energy equation is obtained by averaging its conservation equation reminded in Equation (2.6) and neglecting the same terms as in the total internal energy equation

$$\frac{\partial}{\partial t} (\tilde{\rho} \tilde{k}) + \frac{\partial}{\partial x_j} (\tilde{\rho} \tilde{k} \tilde{u}_j) = -\overline{u_i \frac{\partial p}{\partial x_i}} + \frac{\partial}{\partial x_j} \left(\tilde{u}_i (\hat{\tau}_{ij} + \hat{\tau}_{ij}^t) \right) - \overline{\tau_{ij} d_{ij}}. \quad (\text{A.4})$$

The evolution equation for the mean enthalpy is obtained by using the total internal energy equation and the kinetic energy equation

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{h}) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{h} \tilde{u}_j) = \frac{\partial \bar{p}}{\partial t} - \frac{\partial}{\partial x_j} (\hat{q}_j + \bar{q}_j^t) + \overline{u_i \frac{\partial p}{\partial x_i}} + \overline{\tau_{ij} d_{ij}}, \quad (\text{A.5})$$

giving

$$\frac{\partial}{\partial t} (\bar{\rho} \hat{s}) + \frac{\partial}{\partial x_j} (\bar{\rho} \hat{s} \tilde{u}_j) = -\frac{1}{\tilde{T}} \frac{\partial}{\partial x_j} (\hat{q}_j + \bar{q}_j^t) + \frac{1}{\tilde{T}} \overline{\tau_{ij} d_{ij}}, \quad (\text{A.6})$$

$$\frac{\partial}{\partial t} (\bar{\rho} \hat{s}) + \frac{\partial}{\partial x_j} (\bar{\rho} \hat{s} \tilde{u}_j) = -\frac{\partial}{\partial x_j} \left(\frac{1}{\tilde{T}} (\hat{q}_j + \bar{q}_j^t) \right) - \frac{1}{\tilde{T}^2} (\hat{q}_j + \bar{q}_j^t) \frac{\partial \tilde{T}}{\partial x_j} + \frac{1}{\tilde{T}} \overline{\tau_{ij} d_{ij}}, \quad (\text{A.7})$$

for the evolution equation of the entropy. The evolution equation of the total exergy is finally given by

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho} \hat{\xi}_0) + \frac{\partial}{\partial x_j} (\bar{\rho} \hat{\xi}_0 \tilde{u}_j) &= \frac{\partial}{\partial x_j} (\tilde{u}_i (\hat{\tau}_{ij} + \bar{\tau}_{ij}^t)) - \frac{\partial}{\partial x_j} \left(\left(1 - \frac{T_{ref}}{\tilde{T}} \right) (\hat{q}_j + \bar{q}_j^t) \right) \\ &\quad - T_{ref} \left(-\frac{1}{\tilde{T}^2} (\hat{q}_j + \bar{q}_j^t) \frac{\partial \tilde{T}}{\partial x_j} + \frac{1}{\tilde{T}} \overline{\tau_{ij} d_{ij}} \right), \end{aligned} \quad (\text{A.8})$$

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho} \hat{\xi}_0) + \frac{\partial}{\partial x_j} (\bar{\rho} \hat{\xi}_0 \tilde{u}_j - (\tilde{u}_i (\hat{\tau}_{ij} + \bar{\tau}_{ij}^t))) &+ \left(1 - \frac{T_{ref}}{\tilde{T}} \right) (\hat{q}_j + \bar{q}_j^t) \\ &= -T_{ref} \left[\frac{\Phi}{\tilde{T}} + \frac{\Phi_\Theta}{\tilde{T}^2} \right], \end{aligned} \quad (\text{A.9})$$

where

$$\frac{\Phi^{RANS}}{\tilde{T}} = \frac{1}{\tilde{T}} \overline{\tau_{ij} d_{ij}}, \quad (\text{A.10})$$

$$\frac{\Phi_\Theta^{RANS}}{\tilde{T}^2} = -\frac{1}{\tilde{T}^2} (\bar{q}_j + \bar{q}_j^t) \frac{\partial \tilde{T}}{\partial x_j}, \quad (\text{A.11})$$

are total exergy production terms due to viscous dissipation and heat transfer irreversibilities respectively. Note the difference in viscous dissipation term compared to Lamberts et al. [55] as they used the evolution equation of the kinetic energy of the mean flow instead of the mean evolution equation of the kinetic energy, thus adding the turbulent kinetic energy production term $\hat{\tau}_{ij}^t \hat{d}_{ij}$ in their dissipation term.

The left hand side of Equation A.9 contains the expression for the total exergy flux in the RANS framework

$$\hat{F}_{\xi_0}^{RANS} = \bar{\rho} \hat{\xi}_0 \tilde{u}_j - \left(\tilde{u}_i \left(\hat{\tau}_{ij} + \hat{\tau}_{ij}^t \right) \right) + \left(1 - \frac{T_{ref}}{\tilde{T}} \right) \left(\hat{q}_j + \hat{q}_j^t \right). \quad (\text{A.12})$$

As for the LES framework, only the flux crossing the dividing streamline will be analysed and it will be decomposed into three components: the total exergy flux due to

(1) enthalpy:

$$\mathcal{F}_h^{RANS} = \left[\hat{q}_j + \hat{q}_j^t \right] n_j, \quad (\text{A.13})$$

(2) kinetic energy:

$$\mathcal{F}_k^{RANS} = \left[-\tilde{u}_i \left(\hat{\tau}_{ij} + \hat{\tau}_{ij}^t \right) \right] n_j, \quad (\text{A.14})$$

(3) entropy:

$$\mathcal{F}_s^{RANS} = -T_{ref} \left[\frac{1}{\tilde{T}} \left(\hat{q}_j + \hat{q}_j^t \right) \right] n_j. \quad (\text{A.15})$$

The main differences between those fluxes and the exact fluxes obtained in section 4.3.3 (except for the averaging operators and the neglected term in the kinetic energy equation) comes from the entropy flux and the heat transfer irreversibility. Indeed, in the exact averaged equation, neither of those terms contain a contribution of the turbulent heat flux. These discrepancies come from the approximated averaged entropy equation and can be partially retrieved by linearizing the entropy-velocity correlation.

Derivation of the compound Mach number for a compound flow with general transfers between streams

Bernstein et al. [6] proposed a compound flow indicator β which allows to determine if a compound flow, i.e. a flow constituted of several separate streams, is supersonic, subsonic or sonic. This compound flow indicator was used by Lamberts et al. [57] to predict the entrainment ratio of a supersonic ejector in the case of the isentropic flow of a perfect gas, as was also done in Bernstein et al. [6].

The assumption of isentropic flow was removed by Metsue et al. [61] by adding polytropic efficiencies for each of the flows. This gave rise to a new expression of β which was shown to correspond to a mass flow-rate maximization in the case of an ejector, proving that this compound-choking indicator is an image of the choking of a flow, when properly defined.

All those compound choking theories however lack the exchange of heat and momentum through the dividing streamlines. An attempt to add momentum transfer through shear was recently proposed by Van den Berghe et al. [82] for a two-stream flow.

In the following, the compound choking theory is extended for the case with momentum and heat transfer through the dividing streamlines. First, the conservation equations on a control volume are expressed for a single-stream flow. It is then extended to the case with multiple streams.

The three conservation equations on a control volume with potential momentum and heat flux terms read :

$$\frac{d\rho}{\rho} + \frac{du}{u} + \frac{dA}{A} = 0 \quad (\text{Mass}) \quad (\text{B.1})$$

$$\rho u du + dp + \frac{f_m}{A} dx = 0 \quad (\text{Momentum}) \quad (\text{B.2})$$

$$\rho u dh_0 + \frac{f_{h_0}}{A} dx = 0 \quad (\text{Total enthalpy}) \quad (\text{B.3})$$

where $dh_0 = c_p dT + u du$ for a perfect gas. The general expression for f_m and f_{h_0} are obtained as the flux terms of the corresponding local evolution equations and are :

$$f_m = - \int_{\Gamma} [\bar{\tau}_{ij} + \bar{\tau}_{ij}^t] t_i n_j ds, \quad (\text{B.4})$$

$$f_{h_0} = \int_{\Gamma} [\bar{q}_j + \bar{q}_j^t - (\bar{u}_i \bar{\tau}_{ij} + \bar{u}_i^t \bar{\tau}_{ij}^t) + \frac{1}{2} \overline{\rho u_k'' u_k'' u_j''}] n_j ds, \quad (\text{B.5})$$

where $\bar{\tau}_{ij}^t = -\overline{\rho u_i'' u_j''}$, $\bar{q}_j^t = \overline{\rho c_p T'' u_j''}$; n_j and $t_i = \frac{\bar{u}_i}{||\bar{u}||}$ are the components of the normal and tangent vectors to the control volume boundary Γ , respectively. Note that the conservation equations assume a steady flow and thus, for the sake of readability, no averaging operators are used. Combining the mass and total enthalpy conservation equations, as well as the perfect gas law gives

$$(\rho u c_p T + \rho u u^2) \frac{du}{u} + \rho u c_p T \frac{dp}{p} + \rho u c_p T \frac{dA}{A} + \frac{f_{h_0}}{A} dx = 0, \quad (\text{B.6})$$

which, when injected into the momentum conservation gives the relation between pressure and area differentials :

$$\frac{p(1-M^2)}{1+(\gamma-1)M^2} \frac{dp}{p} - \frac{p\gamma M^2}{1+(\gamma-1)M^2} \frac{dA}{A} + \left(f_m - \frac{(\gamma-1)M/c}{1+(\gamma-1)M^2} f_{h_0} \right) \frac{dx}{A} = 0. \quad (\text{B.7})$$

For a multi-stream flow, this equation must hold true for each stream n . To obtain the compound flow indicator β , similarly to that introduced by Bernstein et al. [6], only the assumption that pressure is constant across streams is necessary. Thus, the pressure gradient is also constant across streams. Finally, and by construction, the sum of individual stream areas must be equal to the total area of the flow, which is thus also true for the area gradients. We then obtain:

$$\frac{dA}{dx} = \frac{1}{p} \frac{dp}{dx} \sum_n \frac{1-M_n^2}{\gamma M_n^2} A_n + \frac{1}{p} \sum_n \left(\frac{1+(\gamma-1)M_n^2}{\gamma M_n^2} f_{m,n} - \frac{\gamma-1}{\gamma u_n} f_{h_0,n} \right). \quad (\text{B.8})$$

The compound flow indicator for a multiple stream flow, with general interaction between streams is then

$$\beta = \frac{\frac{1}{A} \frac{dA}{dx}}{\frac{1}{p} \frac{dp}{dx}} \quad (\text{B.9})$$

$$= \sum_n \frac{1 - M_n^2}{\gamma M_n^2} \frac{A_n}{A} + \frac{1}{A \frac{dp}{dx}} \sum_n \left(\frac{1 + (\gamma - 1) M_n^2}{\gamma M_n^2} f_{m,n} - \frac{\gamma - 1}{\gamma u_n} f_{h_0,n} \right). \quad (\text{B.10})$$

Note that the definition of β is modified compared to that of Bernstein et al. [6] to obtain a dimensionless quantity.

This expression reduces to the isentropic variant when no source terms are considered in the streams, i.e. $f_{m,n} = f_{h_0,n} = 0$ and it can be verified to correspond to the expression of β obtained by Metsue et al. [61], with polytropic efficiencies used to represent the momentum exchange. In that case, $f_{m,n} = A_n (\eta_{pol,n} - 1) \frac{dp}{dx}$ and $f_{h_0} = 0$. As this expression was analytically shown to correspond to a maximum of mass flow-rate in the case of a two-stream flow, this validates the physical validity of the momentum flux contribution in this equation.

To also validate that the total enthalpy flux term leads to a maximization of the mass flow-rate, the equations for an isothermal flow are developed hereafter. It must then be proved that the maximization of the mass flow-rate in this case corresponds to the expression obtained in B.10 for the appropriate total enthalpy flux. In this case, the three conservation equations are, for a single stream :

$$\frac{d\rho}{\rho} + \frac{du}{u} + \frac{dA}{A} = 0 \quad (\text{Mass}) \quad (\text{B.11})$$

$$\rho u du + dp = 0 \quad (\text{Momentum}) \quad (\text{B.12})$$

$$dT = 0 \quad (\text{Energy}) \quad (\text{B.13})$$

The constant temperature assumption, along with the perfect gas law, yields : $d\rho/\rho = dp/p$. Combining mass and momentum conservation equations then gives

$$(\rho u^2 - p) \frac{du}{u} - p \frac{dA}{A} = 0 \quad (\text{B.14})$$

$$\frac{dA}{A} = (\gamma M^2 - 1) \frac{du}{u}. \quad (\text{B.15})$$

These equations, obtained for a single stream, are then used to maximize the mass flow-rate in a two-stream supersonic ejector by computing the derivative of the

secondary mass flow-rate with respect to the pressure p . In this case, the total enthalpy fluxes between streams and at the wall are non zero and ensure that each of the streams is isothermal. Since the primary flow is supersonic, its mass flow rate is fixed.

$$\frac{\partial \dot{m}_s}{\partial p} = \frac{\dot{m}_s}{A_s} \left(\frac{1}{\rho_s} \frac{\partial \rho_s}{\partial p} A_s + \frac{1}{u_s} \frac{\partial u_s}{\partial p} A_s + \frac{\partial A_s}{\partial p} \right). \quad (\text{B.16})$$

The momentum conservation equation gives the partial derivative of velocity with respect to pressure and since the total flow area is independent on the pressure,

$$\frac{\partial A_s}{\partial p} = -\frac{\partial A_p}{\partial p} \quad (\text{B.17})$$

$$= \frac{1}{\rho_p} \frac{\partial \rho_p}{\partial p} A_p + \frac{1}{u_p} \frac{\partial u_p}{\partial p} A_p, \quad (\text{B.18})$$

where the fact that the primary mass flow rate is constant was used. The density derivative is obtained from the perfect gas law and thus, the derivative of secondary mass flow rate with respect to pressure is

$$\frac{\partial \dot{m}_s}{\partial p} = \frac{\dot{m}_s}{A_s p} \left(A_s \left(1 - \frac{1}{\gamma M_s^2} \right) + A_p \left(1 - \frac{1}{\gamma M_p^2} \right) \right) \quad (\text{B.19})$$

$$= \frac{\dot{m}_s}{A_s p} \left(A_s \frac{\gamma M_s^2 - 1}{\gamma M_s^2} + A_p \frac{\gamma M_p^2 - 1}{\gamma M_p^2} \right). \quad (\text{B.20})$$

In the case of an isothermal nozzle flow, the total enthalpy equation would read

$$\rho u^2 du + \frac{f_{h_0}}{A} dx = 0 \quad (\text{B.21})$$

$$\Rightarrow f_{h_0} = -A \rho u^2 \frac{du}{dx} \quad (\text{B.22})$$

$$= A u \frac{dp}{dx}. \quad (\text{B.23})$$

Equation B.10 thus becomes:

$$\beta = \frac{1}{A} \sum_n \frac{1 - M_n^2}{\gamma M_n^2} A_n - \frac{\gamma - 1}{\gamma} \sum_n \frac{A_n}{A} \quad (\text{B.24})$$

$$= \sum_n \frac{1 - \gamma M_n^2}{\gamma M_n^2} \frac{A_n}{A}. \quad (\text{B.25})$$

This equation, similarly to what was obtained by Metsue et al. [61] shows that the maximization of the secondary mass flow rate and the compound flow indicator β , when taken for two streams are linked by

$$\frac{\partial \dot{m}_s}{\partial p} = -\frac{A}{p} \frac{\dot{m}_s}{A_s} \beta, \quad (\text{B.26})$$

showing that the newly found expression for the compound choking indicator is indeed the correct expression to ensure mass flow-rate maximization and thus choking of the compound flow for the general case. This criterion is true globally but also locally, i.e. to find the position of the choking section. As the mean momentum flowing through the ejector is :

$$(\rho u)_{tot} = \frac{\dot{m}_p + \dot{m}_s}{A}, \quad (\text{B.27})$$

maximizing the momentum in the ejector is equivalent to finding the location within the ejector where $\beta = 0$. Since the location of maximum momentum is by definition the choking section, finding $\beta = 0$ gives the choking section of the ejector.

It is also convenient to define an equivalent or compound Mach number to have a direct correspondence with classical one-stream theory. This Mach number is defined by inverting the compound indicator for an isentropic flow:

$$M_{eq} = \left(\frac{1}{\gamma \beta + 1} \right)^{1/2}. \quad (\text{B.28})$$

Using eq (B.10) for the compound choking indicator yields:

$$M_{eq} = \left(\frac{\frac{dp}{dx} A}{\frac{dp}{dx} \sum_n \frac{A_n}{M_n^2} + \sum_n \left(\frac{1+(\gamma-1) M_n^2}{M_n^2} f_{m,n} - \frac{\gamma-1}{u_n} f_{h_{0,n}} \right)} \right)^{1/2}. \quad (\text{B.29})$$