



UNIVERSITÉ CATHOLIQUE DE LOUVAIN
ÉCOLE POLYTECHNIQUE DE LOUVAIN
INSTITUTE OF MECHANICS, MATERIALS AND CIVIL
ENGINEERING

CHOKING AND MIXING PHENOMENA IN SUPERSONIC EJECTORS

DOCTORAL DISSERTATION PRESENTED BY

OLIVIER LAMBERTS

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR THE DEGREE OF DOCTOR IN ENGINEERING SCIENCES

THESIS COMMITTEE:

Prof. Yann Bartosiewicz (UCL), supervisor
Prof. Philippe Chatelain (UCL), supervisor
Prof. Laurent Delannay (UCL), chairperson
Dr. Matthieu Duponcheel (UCL), secretary
Prof. Vincent Legat (UCL)
Prof. Sébastien Poncet (USherbrooke, Canada)
Prof. Adriano Milazzo (Università degli Studi di Firenze, Italy)

Louvain-la-Neuve, September 2018

ACKNOWLEDGEMENTS

Même si réaliser une thèse de doctorat est avant tout un processus individuel, on ne peut le mener à terme sans être bien entouré. C'est pourquoi je souhaiterais remercier l'ensemble des personnes qui m'ont aidé, de près ou de loin, pour arriver au bout de cette épreuve singulière.

Premièrement, je souhaiterais remercier très chaleureusement mes deux promoteurs – Yann Bartosiewicz et Philippe Chatelain – de m'avoir fourni l'impulsion initiale pour mes travaux de recherche et de m'avoir soutenu tout au long de ce voyage périlleux. Merci à eux pour leur enthousiasme, leurs questions, leurs suggestions et leurs encouragements: ils ont été très complémentaires dans l'encadrement de ce travail. Je tiens également à remercier Fulvio Scarano de m'avoir accordé du temps lorsqu'un avis extérieur était nécessaire. Un grand merci à Matthieu Duponcheel pour sa disponibilité et ses conseils avisés lorsque je n'y voyais plus très clair; ils furent très souvent d'une grande aide. Je souhaiterais d'ailleurs remercier Matthieu et les autres membres du jury d'avoir pris le temps de lire ce document et d'y avoir apporté un regard critique.

J'ai également reçu le soutien de nombreuses personnes pour la partie expérimentale de cette thèse: François, Lili, Franck, Julien, Pascal et Grégoire. Merci à eux pour leur aide au laboratoire. Sans eux, il n'y aurait sans doute pas de résultats expérimentaux dans ce manuscrit.

Même s'ils n'étaient pas directement impliqués dans mes activités de recherche, les autres membres du pôle TFL ont grandement contribué à ce que je puisse arriver au bout de cette thèse. En effet, en faisant de notre lieu de travail un endroit où il fait bon vivre, ils m'ont permis de garder une certaine joie à venir au bureau, même quand mes travaux de recherche n'avançaient plus.

Un merci particulier à Stéphane pour son aide dans la phase initiale de ce travail, lorsqu'il a fallu que je plonge dans le monde obscur de la simulation numérique. Je tiens à le remercier pour ses précieuses explications et sa patience alors que je découvrais ce qu'était un termi-

nal, un compilateur ou encore un cluster. Je souhaiterais aussi adresser un grand remerciement à Pierre, avec qui j'ai partagé mon bureau pendant les deux premières années de mon doctorat. Merci à lui pour sa disponibilité et son écoute attentive dans les bons et les moins bons moments de ces deux premières années. Après son départ, j'ai eu la chance de voir Nicolas s'installer en face de moi. Je souhaiterais le remercier chaleureusement pour toutes les discussions (au caractère scientifique ou non), ses relectures attentives de mes articles et tous les moments partagés à l'UCL ou en dehors; son visage restera définitivement associé à ce passage de ma vie.

Comment pourrais-je oublier de remercier ma famille, qui m'a transmis le sens de la curiosité et qui m'a toujours soutenu dans ce que j'entreprenais. Merci aussi à mes amis, ma deuxième famille, pour leur soutien et leur présence pendant toutes ces années. Un merci particulier à mes colocataires de la Bicocq et du MailingPot: ils m'ont accompagné au jour le jour tout au long de ce voyage aux milles humeurs.

Mon dernier et plus grand merci va à Anouchka, qui aura été à mes côtés du début à la fin. Une thèse de doctorat est souvent une épreuve pour celui ou celle qui l'entreprend, mais peut-être encore plus pour celui ou celle qui l'accompagne. Le mot *merci* n'est sans doute pas assez fort pour lui dire toute la gratitude que j'éprouve envers elle. Ce qui suit lui est dédié.

"Imagination is more important than knowledge."
Albert Einstein

CONTENTS

Contents	i
List of symbols and abbreviations	viii
1 Introduction	1
1.1 Context	1
1.2 Ejector phenomena	3
1.2.1 Ejector operation and performance	3
1.2.2 Simplified models	5
1.2.3 Computational Fluid Dynamics	8
1.3 Research questions and objectives	10
1.4 Structure of the manuscript	11
2 Materials and methods	15
2.1 Introduction	15
2.2 Experimental installation	15
2.2.1 Experimental setup	15
2.2.2 Schlieren system	17
2.2.3 Ejector geometry	18
2.3 Numerical methods	21
2.3.1 Compressible RANS equations	21
2.3.2 Closure of the RANS equations	23
2.3.3 Fluid properties	24
2.3.4 Flow solver	25
2.3.5 Computational Domain	26
2.3.6 Boundary conditions	28
2.4 Verification and validation	29
2.4.1 At global scale	29
2.4.2 At local scale	30
2.5 Numerical results exploited in the manuscript	33

3	New methods for analyzing transport phenomena in supersonic ejectors	35
3.1	Introduction	35
3.2	Definition of exergy efficiencies in supersonic ejectors . .	36
3.2.1	The concept of exergy	36
3.2.2	Global exergy balance in supersonic ejectors . . .	37
3.2.3	The reversible ejector	37
3.2.4	Exergy efficiencies	38
3.2.5	Application to a calorically perfect gas	39
3.2.6	Impact of the back-pressure on the exergy efficiencies	40
3.3	Transport tubes	41
3.3.1	Momentum and mean-flow kinetic energy tubes .	41
3.3.2	Mean-flow exergy tubes	44
3.4	Visualization of transport processes within the ejector . .	46
3.4.1	Transport of mass, momentum and mean-flow kinetic energy	46
3.4.2	Transport of mean-flow total exergy	51
3.5	Transfers between the primary and the secondary streams	58
3.5.1	Local transfers	58
3.5.2	Cumulative transfers	63
3.6	Exergy losses within the secondary stream	66
3.7	Global exergy balance for the secondary stream	68
3.8	Conclusions	74
4	Numerical and experimental evidence of the Fabri-pattern in a supersonic ejector	75
4.1	Introduction	75
4.2	Phenomenology of the Fabri-pattern	76
4.2.1	Sonic line and dividing streamline	76
4.2.2	Flow topology related to the Fabri-pattern	78
4.2.3	Aerodynamic throat in the secondary stream . . .	81
4.2.4	Y-shock wave pattern	81
4.2.5	Experimental evidence of the Y-shock wave pattern	82
4.3	Transfers and irreversibilities in the secondary stream upstream of the Fabri-pattern	84
4.3.1	Static pressure and thickness of the shear layer at the aerodynamic throat	84
4.3.2	Definition of the location of the Fabri-pattern . . .	86
4.3.3	Exergy transfers and losses in the secondary stream	89
4.4	Impact of the back-pressure on the Fabri-pattern	91
4.5	Conclusions	93

5	The compound-choking theory as an explanation of the entrainment limitation in supersonic ejectors	97
5.1	Introduction	97
5.2	Compound-choking theory	98
5.2.1	Assumptions	98
5.2.2	Compound-choking condition	99
5.2.3	Computational procedure for two-stream compound-compressible nozzle flows	100
5.2.4	Prediction of the entrainment ratio in supersonic ejectors with the compound-choking theory . . .	102
5.3	Numerical evidence of the compound-choking	102
5.3.1	Sonic line and dividing streamline	102
5.3.2	Wall static pressure	105
5.3.3	Equivalent Mach number	106
5.4	Comparison of the compound-choking theory with experimental data	108
5.4.1	Prediction of the entrainment ratio for various ejector configurations	108
5.4.2	Comparison with state-of-the-art simplified models	110
5.5	Numerical assessment of the limitations of the compound-choking theory	113
5.5.1	Impact of the primary stagnation pressure	113
5.5.2	Impact of the mixing duct cross-section	117
5.5.3	Impact of the inlet converging angle	118
5.6	Conclusions	120
6	Development of an upgraded experimental setup and comparison with 2D and 3D numerical simulations	123
6.1	Introduction	123
6.2	Upgraded experimental setup	124
6.3	Comparison of 2D numerical simulations with the experimental data	127
6.3.1	Selection of the operating conditions	127
6.3.2	Characteristic curve	128
6.3.3	On-design entrainment ratio	129
6.3.4	Flow structure at the exit of the primary nozzle .	132
6.3.5	Static pressure profile	135
6.4	Numerical investigation of three-dimensional effects . .	139
6.5	Conclusions	148
7	Conclusions and perspectives	151
7.1	Main achievements	152

7.2	Limitations and research perspectives	155
A	Compound waves	161
A.1	Compound Waves	161
	Bibliography	165

NOMENCLATURE

Abbreviations

CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
RANS	Reynolds-Averaged Navier-Stokes

Alpha numeric symbols

$\dot{m}$	Mass flow rate	[kg/s]
$\mathbf{q}$	Heat flux vector field	[J/(m ² s)]
$\mathbf{u}$	Velocity vector field	[m/s]
$\overline{\mathbf{F}}_\xi$	Total mean-flow total exergy flux vector field	[kg/s ³]
$\overline{\mathbf{F}}_K$	Total mean-flow kinetic energy flux vector field	[kg/s ³]
$\overline{\mathbf{F}}_m$	Total linear momentum flux vector field	[kg/(m s ²)]
$\overline{\mathbf{F}}_q$	Total heat flux vector field (exergy content)	[kg/(m s ²)]
A	Cross-sectional area	[m ²]
A_1	Cross-sectional area of the exit of the primary nozzle	[m ²]
A_m	Cross-sectional area of the mixing duct	[m ²]
A_t	Cross-sectional area of the throat of the primary nozzle	[m ²]
c_p	Heat capacity at constant pressure	[J/(kg K)]
c_v	Heat capacity at constant volume	[J/(kg K)]
e	Specific internal energy	[J/kg]
e_t	Specific total energy	[J/kg]

h	Specific static enthalpy	[J/kg]
H_1	Height of the exit of the primary nozzle	[m]
H_m	Height of the mixing duct	[m]
H_t	Height of the throat of the primary nozzle	[m]
h_t	Specific total enthalpy	[J/kg]
I_ξ	Cumulative transfer of mean-flow total exergy	[−]
I_K	Cumulative transfer of mean-flow kinetic energy	[−]
I_m	Cumulative transfer of momentum	[−]
I_q	Cumulative transfer of heat (exergy content)	[−]
k	Specific turbulent kinetic energy	[J/kg]
M	Mach number	[−]
M_{eq}	Equivalent Mach number of the compound-choking theory	[−]
p	Static pressure	[Pa]
p^*	Critical pressure	[Pa]
p_{out}^*	Critical back-pressure	[Pa]
p_t	Stagnation pressure	[Pa]
p_{eq}^*	Pressure corresponding to the compound-choking condition	[Pa]
R	Specific gas constant	[J/(kg K)]
s	Specific entropy	[J/(kg K)]
T	Static temperature	[K]
T_t	Stagnation temperature	[K]
x	Horizontal coordinate	[m]
x_b	Horizontal coordinate at the outlet of the ejector	[m]
x_c	Horizontal coordinate corresponding to $M_{eq} = 1$	[m]

x_f	Horizontal coordinate corresponding to $M_2 = 1$	[m]
y	Vertical coordinate	[m]
z	Span-wise coordinate	[m]
ER	Entrainment ratio	[–]

Greek letters

β	Compound-flow indicator	[m ²]
ζ	Constant unit vector	[–]
Δ	Gain in exergy of the secondary stream	[–]
γ	Specific heat ratio	[–]
κ	Thermal conductivity	[J/(s m K)]
μ	Dynamic viscosity	[Pa s]
ω	Specific turbulence dissipation rate	[1/s]
Π	Cumulative mean-flow total exergy losses related to viscous dissipation	[–]
π	Local mean-flow total exergy losses related to viscous dissipation	[–]
Π_Θ	Cumulative mean-flow total exergy losses related to heat transfer	[–]
π_Θ	Local mean-flow total exergy losses related to heat transfer	[–]
ρ	Density	[kg/m ³]
τ	Viscous stress tensor	[Pa]
ξ_t	Specific total exergy	[J/kg]

Operators

$\bar{\cdot}$	Time average
$\hat{\cdot}$	Density-weighted (or Favre) average
$\sim$	Value calculated from the mean-flow quantities

Subscripts

0	Reservoir conditions
1	Value associated to the primary stream
2	Value associated to the secondary stream
out	Value evaluated at the outlet of the ejector
ref	Reference conditions

CHAPTER 1

INTRODUCTION

1.1 Context

Supersonic ejectors are passive flow devices which have been exploited for a long time in several applications, notably to serve as vacuum generators in power stations, in the food industry or in aerospace engineering [13]. Their use is also considered in the field of propulsion [4, 5, 65, 80]. Nowadays, considerable attention is being paid to the enhancement of the performance of processes in a wide range of applications. In the areas of refrigeration and air-conditioning, heat-driven ejectors systems appear to be a promising solution [16, 78]. In such cycles, supersonic ejectors are used to compress a fluid by using low-grade heat release which would be wasted otherwise, or renewable energy sources such as solar energy [25, 44].

The concept of a solar air-conditioning cycle is shown in Fig. 1.1. As in common refrigeration systems, heat is absorbed at a certain temperature in an evaporator and released at a higher temperature level in a condenser. However, the compressor that is normally placed between both heat exchangers in vapor-compression systems has been removed and replaced by an ejector. While a compressor requires electricity as energy input, the ejector cycle offers the advantage of primarily consuming thermal energy. Indeed, as may be seen in Fig. 1.1, heat must be provided to a boiler (or generator) in order to achieve the evapo-

ration of a certain fraction of the refrigerant load. Logically, this additional evaporation process must take place at a temperature higher than the temperature prevailing in the condenser. Hence, ejector refrigeration cycles are sometimes referred to as three-thermal systems. Note that electrical energy is still needed in these installations as liquid fluid must be pumped at the exit of the condenser to feed the boiler. For off-grid configurations, electricity may, for example, be provided by a photovoltaic system (see Fig. 1.1), because liquids requires much less energy than gases to be compressed.

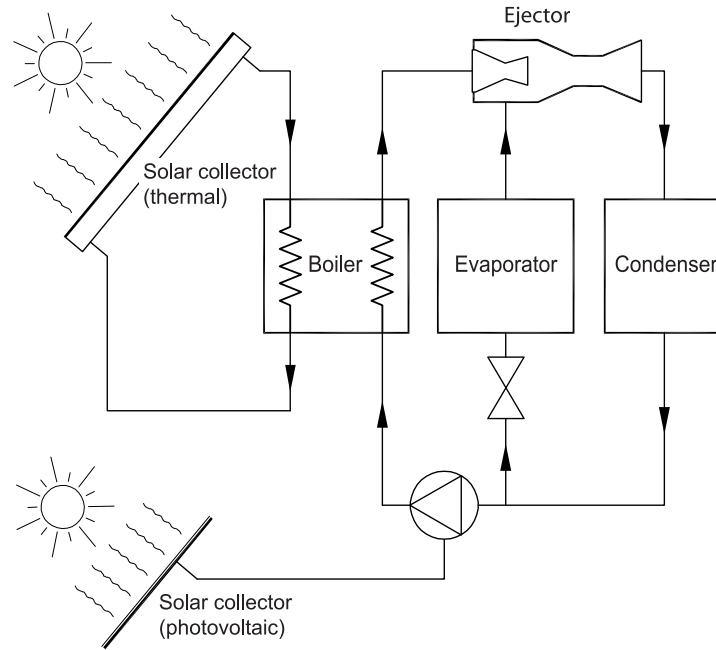


Figure 1.1: A supersonic ejector in a solar air-conditioning cycle [44].

The main advantages of ejector refrigeration systems lie in their mechanical simplicity, the limited maintenance needs, the low initial and operational costs and their ability to work with environmentally friendly refrigerants [16, 39]. Nevertheless, the low performance coefficient of ejector refrigeration cycles as well as the considerable deterioration of their performance for operating conditions departing from their design values have prevented ejector refrigerators to penetrate the market [16]. Due to its importance on the whole system, the ejector in itself is actually the main responsible for these shortcomings [25, 42, 77]. The fundamentals of ejector operation are presented below.

1.2 Ejector phenomena

1.2.1 Ejector operation and performance

Within a supersonic ejector, two fluids with different stagnation pressures are mixed to obtain a fluid with an intermediate stagnation pressure at the discharge. The stream with high stagnation pressure is known as the primary (or motive) stream while the entrained fluid with low stagnation pressure is referred to as the secondary (or suction) stream. In the present work, they are identified with the indices 1 and 2, respectively.

The typical layout of an ejector is shown in Fig. 1.2. It essentially consists in a primary nozzle placed in a secondary nozzle whose throat is extended, thereby forming a *mixing duct*, also termed *mixing chamber*. Depending on whether the exit section of the primary nozzle lies in the converging part or in the constant-area section of the secondary nozzle, one generally refers to *constant-pressure* or *constant-area* mixing ejectors, respectively [25].

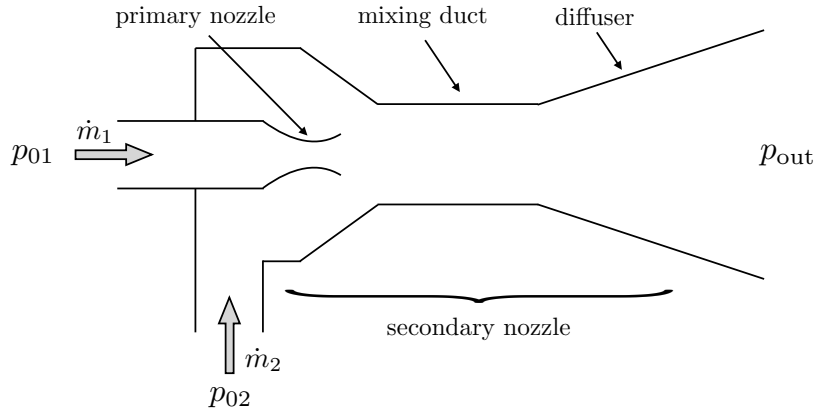


Figure 1.2: Typical layout of a supersonic ejector.

Although the geometrical structure of a supersonic ejector is relatively simple, the compressible turbulent mixing occurring within is rather complex [16, 42, 68]. In a similar way to a single nozzle flow, the primary stream reaches sonic conditions in the vicinity of the throat of the primary nozzle and the primary mass flow rate, $\dot{m}_1$, is therefore not sensitive to downstream conditions and depends only on the primary reservoir conditions. This limitation of the mass flow rate in a nozzle is generally referred to as the *choking* condition [73]. The primary

stream further accelerates in the diverging part of the primary nozzle and thus comes out as a supersonic jet with high velocity and low static pressure. The secondary stream accelerates towards this low pressure region and is entrained within the ejector through a turbulent transonic mixing layer.

Depending on the geometry of the ejector and on the operating conditions, the flow may also be choked within the secondary nozzle. As a consequence, for fixed reservoir conditions, the secondary mass flow rate, $\dot{m}_2$, may also reach a certain limit below a critical back-pressure, indicated as p_{out}^* . Under these circumstances, the ejector is said to operate in on-design (or critical) conditions. One also refers to double-choking operation since both flows are choked within the ejector. Above the critical pressure, the flow within the ejector is single-choked and the secondary mass flow rate is therefore sensitive to downstream conditions. It rapidly decreases with increasing back-pressures until it reaches zero when the back-pressure equals a breakdown value. Between the critical and the breakdown back-pressures, the ejector is said to operate in off-design (or sub-critical) conditions. This regime is logically also known as single-choking operation as solely the primary stream is choked for this mode of operation.

A convenient way to depict the different operating modes of a supersonic ejector for fixed reservoir conditions is to show the characteristic curve of the ejector which represents the evolution of the entrainment ratio, $ER = \dot{m}_2/\dot{m}_1$, versus the compression ratio p_{out}/p_{02} . A typical characteristic curve is shown in Fig. 1.3 where one may easily identify the different operating regimes outlined above. Considering the shape of the performance curve, it is logically preferable to operate the ejector in the vicinity of the critical back-pressure which corresponds to the highest compression ratio achievable with the on-design entrainment ratio [25].

Even though the limitation of the entrainment ratio for fixed reservoir conditions is a well-known phenomenon, there is still a lack of understanding of the complex flow phenomena at play within supersonic ejectors [42]. Hence, there is no generally accepted methodology for their design to this day [36]. This issue mainly stems from the difficulty of predicting the choked flow condition that exists in the ejector [24, 69]. Indeed, as pointed out in [24], the performance of supersonic ejectors is limited by this choking condition which is consequently critical in their design and operation. Therefore, further detailed knowledge and modeling of the choking process of the secondary flow is necessary [16, 59].

An additional limitation on the performance of supersonic ejectors can be traced to the mixing phenomena at play, as transfers and irre-

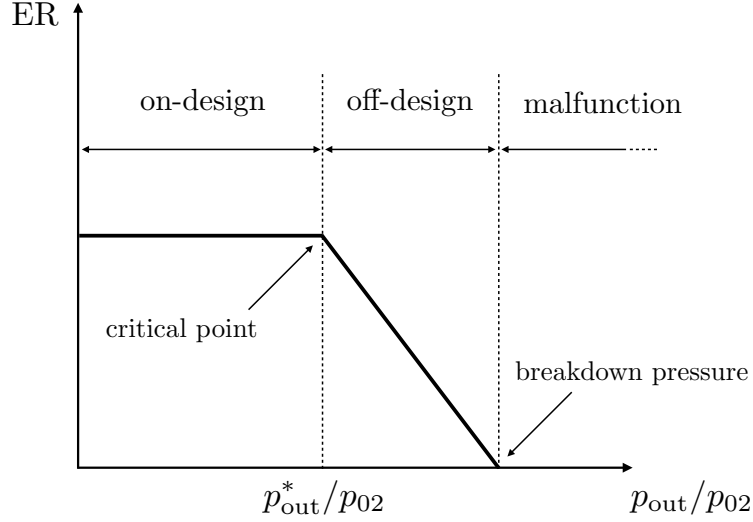


Figure 1.3: Typical characteristic curve of a supersonic ejector showing the entrainment ratio as a function of the compression ratio.

versibilities have a strong impact on the critical back-pressure [26, 81]. Hence, a better understanding of the mixing process is essential for improving the efficiency of supersonic ejectors [17, 32, 77].

From a general point of view, there are two main categories of models regarding the complex flow phenomena taking place within supersonic ejectors: one-dimensional models (1D) and models based on Computational Fluid Dynamics (CFD). A brief state-of-the-art is presented below for the two approaches, with a special emphasis on choking and mixing phenomena.

1.2.2 Simplified models

Studies resorting to simplified models are plentiful, some of which date back to the early 1950's when a control volume analysis was applied to the mixing of two streams using mass, momentum and energy conservation equations [50]. In order to derive analytical expressions, the mixing was assumed to occur either at constant pressure or in a duct of constant area. As mentioned by He et al. [42], the initial model of Keenan et al. [50] could, however, not give detailed information on the choking phenomena.

A few years later, based on shadowgrams of the flow within a rectangular ejector, two phenomenologies concerning the choking of the secondary stream were proposed by Fabri and Siestrunk [33]. According to the authors, when the stagnation pressure of the secondary fluid is sufficiently high, the maximum induced mass flow rate may be limited by the secondary section at the primary nozzle exit plane. In other words, in such cases, the secondary stream reaches sonic conditions at the location where it comes into contact with the primary stream. The authors named this phenomenology of choking the *saturated supersonic* flow pattern, which is also referred to as *choke I* in [24]. Obviously, the choking of the secondary flow in the saturated supersonic flow pattern does not require any model of the flow within the ejector to determine the maximum secondary mass flow rate as the latter may be easily computed using the classical 1D theory for nozzle flows.

More importantly, a second phenomenology of choking for the entrained flow was also introduced in Fabri and Siestrunk [33], describing the so-called *supersonic* flow pattern. In this regime, the induced secondary stream is assumed to accelerate as it flows along the primary jet and reaches unity Mach number at the section of maximum expansion of the primary flow, i.e. in a sort of aerodynamic throat located some distance downstream of the exit of the primary nozzle (see Fig. 1.4). Although this choking model has received the appellation *choke II* in [24], it will be referred to as the *Fabri-choking* in the present study, in accordance with other works in the literature [1, 54, 55, 56].

Based on their assumption of a sonic section in the secondary stream, so-called *aerodynamic* theories were proposed by Fabri and Siestrunk [33], in which the on-design entrainment ratio is determined by the compatibility conditions of two air streams flowing in the same duct using mass, momentum and energy conservation equations for both flows.

The idea of the Fabri-choking was taken up by many researchers developing theoretical models of the flow phenomena occurring in supersonic ejectors in on-design operation. It should be noted that the section where the secondary stream is assumed to reach sonic conditions within the ejector has received different names as *hypothetical throat* [42, 48, 51], *critical section* [30] or *effective area* [24, 26, 63, 88]. Nevertheless, all these different denominations actually refer to the same theoretical phenomenology regarding the choking of the secondary stream.

Despite the widespread use of the Fabri-choking model, the only experimental evidence of a sonic section in the secondary stream, to this day, consists in wall pressure distributions reported by Fabri and Siestrunk [33]. Indeed, the static pressure was found to fall to approxi-

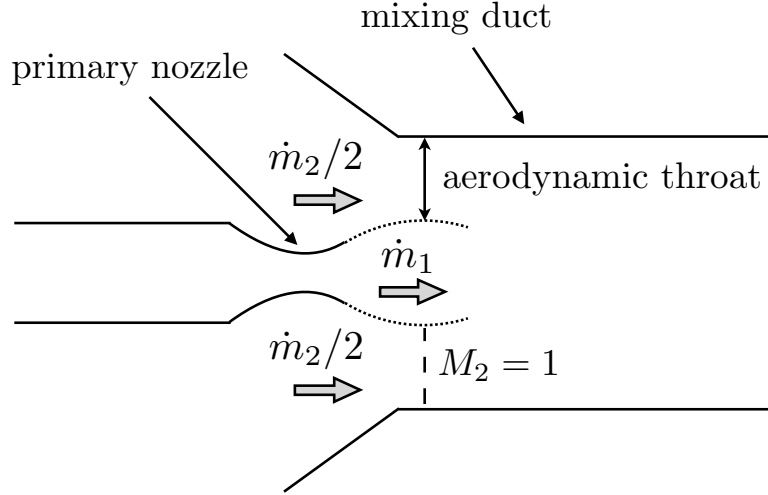


Figure 1.4: Schematic diagram for the model of the Fabri-choking.

mately 50% of the stagnation pressure of the secondary stream at reservoir conditions, which roughly corresponds to the pressure at sonic conditions under the assumptions of a 1D isentropic flow ($p/p_t = 0.528$ for $\gamma = 1.4$) [73]. Since then, this is actually the sole argument that has been proposed to attest the validity of the model of the Fabri-choking [26, 63].

At about the same time as Fabri and Siestrunck [33], another theory emerged in the propulsion field for the explanation of the entrainment limitation in supersonic boosters. Based on a wave velocity argument, a choked flow condition was derived in Pearson et al. [65] in the case of two perfect gases with different stagnation pressures flowing through a constant-area ejector. The obtained condition is such that the flow with the greater total pressure is supersonic while the other is subsonic. An equivalent result has been subsequently reported in Hoge and Segars [47]. In both studies, the measurement of the static pressure at the wall is used to corroborate the fact that the secondary stream remains subsonic as the pressure drop that it experiences is not sufficient for the gas to reach sonic velocities.

A few years later, based on the same assumptions as Pearson et al. [65], a so-called one-dimensional compound-compressible nozzle flow theory was developed by Bernstein et al. [14] for analyzing the behavior of one or more gas streams flowing through a single nozzle. The general conclusions of Bernstein et al. [14] were consistent with the re-

sults of Pearson et al. [65]. Note that the term *compound-choking* was introduced by Bernstein et al. [14] and is adopted in this study to designate the choking theory presented in their work.

Even though references to the works of Pearson et al. or Bernstein et al. are reported in some articles of the following decade related to ejectors [43, 53], the author of the present manuscript did not find recent studies referring to the compound-choking theory. Indeed, for an unknown reason, the Fabri-choking model is used as the foundation of almost all state-of-the-art simplified models of supersonic ejectors [20, 21, 30, 48, 57, 87, 88, 89].

An in-depth analysis of these models shows that they still have difficulty in predicting the entrainment ratio for on-design operating conditions. The average deviation is still close to 8 % in [88] when real gas effects are taken into account, even by using a fitting coefficient, whereas a maximum error of 17% is reported in del Valle et al. [30], who further models 2D features with a perturbation procedure.

Regarding the mixing process, it is generally assumed to occur downstream of the hypothetical throat, at constant pressure and/or in a duct of constant area, according to the early contribution of Keenan et al. [50]. In general, irreversibilities are accounted for using loss coefficients determined by experiments [42]. These coefficients are usually taken as constant values. As emphasized by Besagni et al. [16], this assumption probably limits the performance of current simplified models.

1.2.3 Computational Fluid Dynamics

The current limitations of 1D models and the development of computational power has lead many research groups since the mid-2000s to use Computational Fluid Dynamics (CFD) to simulate the flow in supersonic ejectors [9, 10, 11, 66, 70]. Even though first attempts at performing Large Eddy Simulations (LES) of supersonic ejectors may be found [18], numerical studies available in the literature are mainly confined to Reynolds-averaged Navier-Stokes (RANS) simulations [44, 74, 76]. As the RANS approach essentially relies on closure models, different turbulence models are compared in many papers and the comparison is generally based on the prediction of the entrainment ratio [9, 27, 31, 44, 59]. More recently, the suitability of several turbulence models for capturing local flow phenomena has been assessed through a qualitative comparison of the flow structure at the exit of the primary nozzle [35, 91].

Since the first attempts at performing numerical simulations of supersonic ejectors, CFD has been recognized as an interesting means

of gaining insights into the complex phenomena taking place in such devices [10, 28, 77]. However, in most cases, this technique is used to predict the global performance of the ejector under varying operating conditions and/or geometries [19, 23, 44, 49, 66, 69, 76, 81, 84, 85, 86, 90]. Recently, information concerning local flow features have been displayed through the analysis of iso-contours of Mach number [29, 45, 49, 51, 69] and/or pressure profiles [7, 15, 22, 28, 31, 60, 67, 86].

With regard to choking phenomena, a qualitative study of the choking process within a supersonic ejector was proposed by Bartosiewicz et al. [10] through the visualization of the iso-contour of Mach number unity, named the *sonic line*. It has been emphasized by Hemidi et al. [45] that looking at the sonic line along the ejector could help to find the location of the critical section. However, in their study, the local flow structure and the related phenomenology of the choking were not investigated as the analysis was limited to a qualitative comparison of the sonic line predicted by two different turbulence models. The idea of using the sonic line for the investigation of choking phenomena within the ejector was subsequently taken up by some authors but no special attention was paid to the phenomenology of the choking strictly speaking [22, 28, 49].

More interestingly, the Fabri-choking has actually never been observed in numerical studies. Indeed, in several works where the iso-contour of Mach number unity is shown for on-design operating conditions, the latter is found to progressively expand from the exit of the primary nozzle towards the wall of the ejector, with no apparent sonic section in the core of the secondary stream [28, 29, 45, 49, 60, 69]. Hence, based on many numerical simulations of the flow in supersonic ejectors, it seems that the choking process does not necessarily correspond to the Fabri-choking phenomenology, on which state-of-the-art simplified models are based. This issue has never been addressed in the literature. In fact, a thorough investigation regarding the choking mechanism within supersonic ejectors has not yet been proposed in the literature, neither numerically nor experimentally.

Regarding transport phenomena, the mixing process is analyzed in del Valle et al. [31] by introducing a scalar tracer for the mass fraction at the entrance of the secondary stream, but no information is given concerning the coefficient of diffusion that is used for the tracer. Furthermore, the authors indicate that the distinction between the primary and the secondary streams no longer makes sense once the mixing shear layer reaches the wall. Hence this method is not suitable for the quantification of the local transfers between both streams.

In addition to the analysis of the transfers, special attention should also be paid to the role played by irreversibilities within supersonic ejectors [39]. Although it is investigated in recent contributions [7, 28, 75], no distinction is made between the primary and the secondary streams and/or no information is given concerning the relative importance of the different sources of entropy generation. There is actually no study that really addresses in-depth the open question related to the quantification and the location of the transfers and the irreversibilities within the ejector, and their connection with the entrainment.

1.3 Research questions and objectives

As evidenced by the literature review, two fundamental issues are still unresolved regarding the flow physics in supersonic ejectors, namely :

- how to characterize local transport phenomena responsible for the mixing and losses in supersonic ejectors?
- what is the fundamental choking mechanism at the basis of the entrainment limitation in supersonic ejectors?

According to the author, the potential of the RANS approach has been, to this day, underexploited. Indeed, it is believed that a thorough analysis of the results of such simulations may contribute to a better understanding of both the mixing and the choking phenomena taking place in supersonic ejectors.

Therefore, this thesis aims at gaining insights into the complex flow physics in supersonic ejectors through numerical investigations with special emphasis on mixing and choking processes that are crucial for the limitation of their performance. More precisely, the main approach consists in exploiting the vast potential of 2D RANS numerical simulations to analyze the mixing and the choking phenomena in the post-processing phase of the CFD calculations. Experimental investigations are performed in order to validate these calculations both at the global and at the local scales.

As shown above, new methodologies to study the mixing process in supersonic ejectors are needed. Hence, the first objective of this work consists in developing original post-processing methods for the quantitative analysis of the transport phenomena in such devices. Two different and complementary approaches are presented and applied to the numerical results for fixed reservoir conditions in order to highlight the sole effects of the back-pressure on the transfers and the irreversibilities within the ejector.

The second goal of this thesis is to provide a better understanding of choking phenomena in supersonic ejectors. Hence, numerical results are thoroughly processed in order to shed light on the choking mechanism at play when the ejector operates in the on-design regime. The impact of the reservoir conditions and of the back-pressure on the choking are carefully examined. Throughout these investigations, theoretical considerations are used to give physical explanations of the observed phenomena. To the author's knowledge, such a detailed study of the choking process within supersonic ejectors has never been proposed before.

The last purpose of this work lies in the development of an upgraded experimental test rig for the investigation of mixing and choking phenomena in supersonic ejectors. The new installation aims at correcting the shortcomings of the initial setup referred to in the rest of the manuscript.

1.4 Structure of the manuscript

- **Chapter 2**

This chapter is devoted to a thorough description of both the experimental setup and the numerical methods used throughout the manuscript. First, the focus is put on the experimental installation, the visualization system and the ejector geometry. Then, the numerical tools are described in details, starting with the compressible RANS equations and their closure. The properties of the flow solver are subsequently presented as well as the structure of the computational domain. The verification and the validation of the 2D numerical results are also included in Chapter 2. As mentioned above, the validation is performed both globally and locally. At the global scale, two characteristic curves obtained by numerical simulations are compared to entrainment ratios measured experimentally. At the local level, an experimental schlieren image and a numerical pseudo-schlieren are juxtaposed in order to compare the flow structure at the exit of the primary nozzle. Finally, the numerical results which are effectively exploited in the following analyses are reported at the end of Chapter 2 in order to ease the reference to the different cases throughout the manuscript.

- **Chapter 3**

New post-processing methods aiming at characterizing transport phenomena in supersonic ejectors are presented in Chapter 3. Considering that the purpose of the ejector is to give compression work to the secondary stream, particular consideration should be given to exergy. Hence, the chapter starts by recalling the concept of exergy, as well as the different exergy efficiencies defined in supersonic ejectors. Then, two quantitative approaches are consecutively described and applied to the numerical results. First of all, recent analysis tools based on momentum and energy tubes [62] are extended to the present compressible flow context and applied to the mean-flow structure of turbulent flow within the ejector. Furthermore, the transport equation for the mean-flow total exergy is derived and exergy transport tubes are proposed as a tool for the investigation of transport phenomena within the ejector. In addition to this topological approach, a control volume analysis is performed for the region occupied on average by the secondary stream in order to quantitatively investigate transfers between the primary and the secondary streams all along the ejector. Finally, Chapter 3 also identifies the location of exergy losses as well as their origins. Throughout the chapter, new local and cumulative parameters related to transfers and irreversibilities are introduced. This chapter is based on the following paper:

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.

- **Chapter 4**

In Chapter 4, the numerical results are thoroughly processed in order to highlight the existence of a sonic section in the core of the secondary stream for fixed reservoir conditions. The adopted methodology essentially consists in bringing out the region of the ejector where the secondary stream reaches sonic velocities by constructing two fundamental lines in the ejector: the sonic line and the dividing streamline. The analysis reveals that, for the ejector geometry and the reservoir conditions under consideration, the secondary stream reaches sonic velocities in a sort of aerodynamic throat, as early postulated by Fabri and Siestrunk [33]. The similarities and the differences between the observed results and the established literature are highlighted. In addition, the complex flow signature brought to light by means of the

numerical simulations is evidenced through experimental shadowgraphy. The exergy analysis introduced in Chapter 3 is then applied to the secondary stream between the inlet and the sonic section in order to quantify the transfers and the irreversibilities upstream of this specific location. Finally, the impact of the backpressure on this particular flow feature is investigated. This chapter is based on the following paper:

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.

- **Chapter 5**

In Chapter 5, the focus is put on the choking mechanism responsible for the entrainment limitation in supersonic ejectors. The baseline configuration corresponds to the same geometry as in Chapter 4 but with different reservoir conditions. As the observed choking phenomenology is found to be properly described by the compound-choking theory, the fundamentals of this theory are recalled at the beginning of the chapter. The numerical results are then thoroughly post-processed with the same approach as in Chapter 4, namely by constructing the sonic line and the dividing streamline. Subsequently, the ability of the compound-choking theory to predict the on-design entrainment ratio is evaluated against extensive experimental data available in the literature. The performances of this theory are further assessed through a comparison with state-of-the-art 1D models based on the Fabri-choking assumption. Finally, the limitations of the compound-choking theory are investigated numerically, with special emphasis on the impacts of the operating conditions and the geometrical configuration on the ability of the compound-choking theory to predict the entrainment ratio in the on-design regime. This chapter is based on the following paper:

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.

- **Chapter 6**

Chapter 6 is devoted to the development of an upgraded experimental installation. The design of the new ejector is presented and the main differences with the initial setup are emphasized. Experimental results obtained with this second ejector facility are

compared to 2D numerical simulations. The comparison is performed both at the global and at the local scales and for different operating conditions. The potential sources of disagreement between the CFD and the experiment are discussed, including 3D effects which are investigated numerically for one operating point.

CHAPTER 2

MATERIALS AND METHODS

2.1 Introduction

In this chapter, the experimental methods and the numerical tools used in this work are presented in details. Section 2.2 is devoted to the description of the experimental installation, the visualization system and the ejector geometry. The focus is then put on the numerical methods in Section 2.3, including the compressible RANS equations, the flow solver and the computational domain. The verification and the validation of the numerical results are also included in the present chapter and are presented in Section 2.4. Finally, the numerical results effectively exploited in the other chapters are reported in Section 2.5 in order to ease the reference to the different cases throughout the manuscript.

2.2 Experimental installation

2.2.1 Experimental setup

The experimental setup used in the present work is shown schematically in Fig. 2.1. The working fluid is air for both the primary and the secondary streams. An industrial compressor (model CE46B - Ateliers François) continuously fills a 2 [m³] reservoir with chilled air at 16 [bar] from which the primary stream is taken, whereas the secondary stream is directly sucked from the atmosphere. The mixed stream is

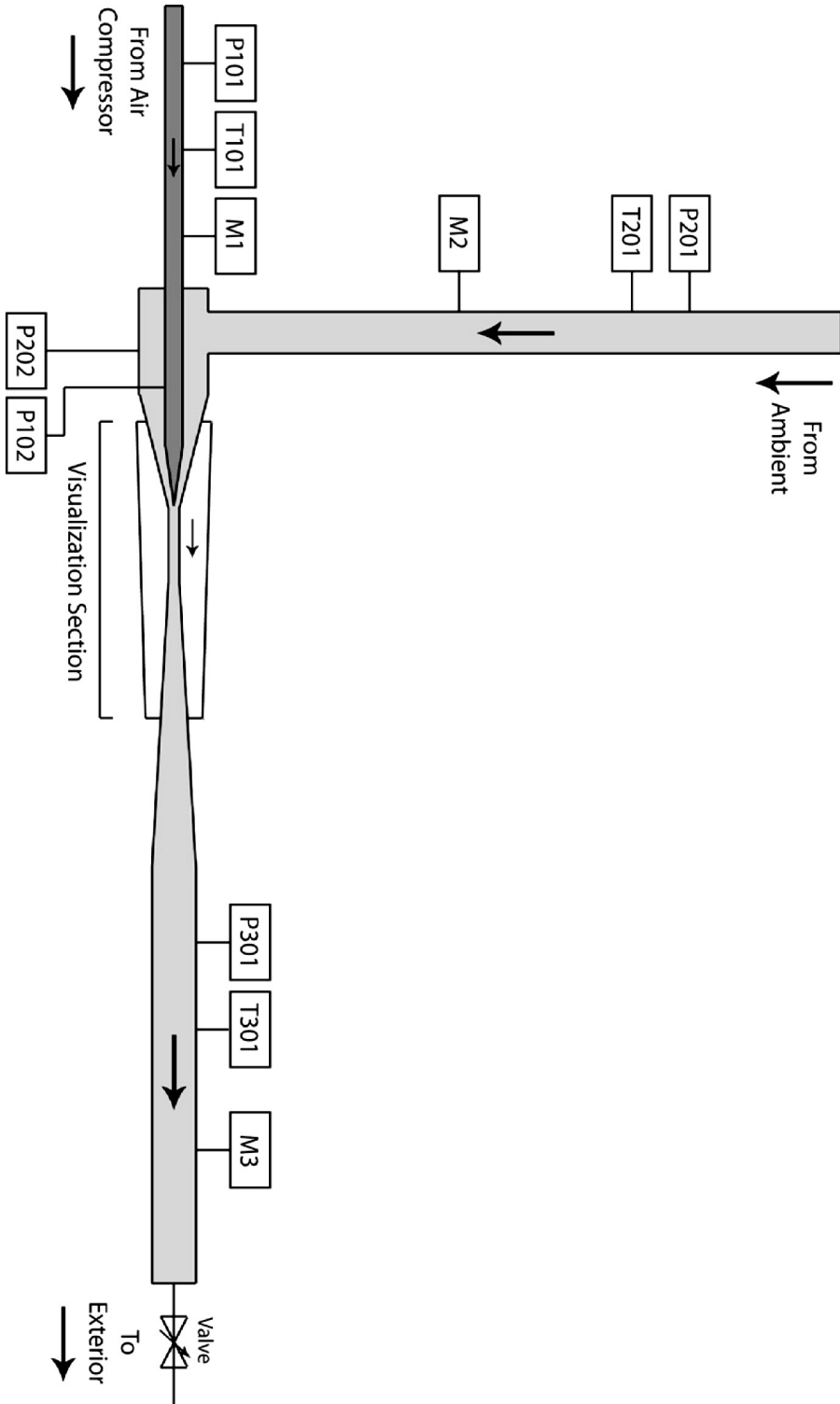


Figure 2.1: Schematic of overall experimental system indicating locations of pressure and temperature measurements, and orifice plates for mass flow rate measurements (taken from [59]).

discharged into the ambient air beyond a butterfly valve serving for the regulation of the back-pressure. The regulation of the primary stagnation pressure to the desired set point is ensured with a pneumatic valve (model Bellofram T-2000) not shown in Fig. 2.1.

The locations of pressure, temperature, and mass flow measurements are also shown in Fig. 2.1. Endress Hauser and Kistler pressure transducers (uncertainty $< \pm 300$ [Pa]) are used for pressure measurements and temperature measurements are taken using PT100 RTD temperature probes (uncertainty $< \pm 0.5$ [°C]). Mass flow rates are measured by means of the pressure drop across an orifice plate of known dimensions. The values and uncertainties for these measurements are determined according to ISO standard 5167 (uncertainty generally $< \pm 0.9\%$). As the pressure losses induced by the orifice plates slightly modify the reservoir conditions at both inlets of the ejector, additional pressure transducers are positioned downstream of each plate to yield the correct boundary conditions for the numerical simulations. The measurement of the mass flow rate at the outlet of the ejector is actually redundant but serves as an additional verification. For the experimental results reported here, the mass imbalance between both inflows and the outflow was always found to be less than 1%.

2.2.2 Schlieren system

At the local scale, the shock reflection pattern within the supersonic ejector is investigated through schlieren visualizations with a slightly modified Z-type schlieren setup. Figure 2.2 shows the spatial arrangement of the visualization system. Like for lens-type systems, the Z-type mirror schlieren system takes advantage of having a parallel light beam in the test region, but offers a larger field-of-a-view for a given cost [72].

A halogen light source of 150 [W] is used in combination with an optical fiber and an iris diaphragm to achieve a point source of high intensity light. The diameter of the mirrors is 200 [mm] and 300 [mm] for the parabolic and the plane mirrors, respectively. Both types of mirrors are characterized by a $\lambda/4$ surface flatness. The focal length of the parabolic mirrors is 1200 [mm].

Two different cameras are used in the present study. A Nikon D5000 camera is used at a pixel resolution of 4288x2848 with a focal length of 180 [mm] providing a spatial resolution close to 25 [pixel/mm]. With this camera, the aperture and the exposure time is set to f/5.6 and 1/1000 [s], respectively. For the visualization of oscillating flow structures, a HighSpeedStar 3G from LaVision is used at a pixel resolution of 1024x1024 with an exposure time of 1/15000 [s]. A spatial resolution

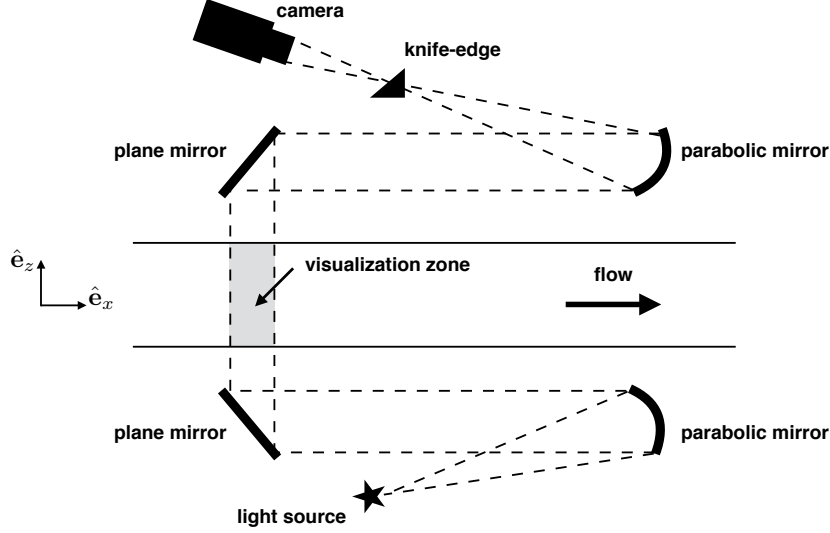


Figure 2.2: Layout of the visualization setup (top view).

of approximately 5.5 [pixel/mm] is obtained with a ZEISS 85mm f/1.4 lens.

2.2.3 Ejector geometry

The ejector used in the present work has a rectangular cross-section to avoid optical distortion with front and back surfaces made of Plexiglas windows. The primary nozzle has a converging-diverging geometry, as may be seen in Fig. 2.3 showing the definition of the different geometrical parameters that are used throughout the document.

The height of the primary nozzle throat, the height of the primary nozzle exit and the height of the mixing duct are noted H_t , H_1 and H_m , respectively. The cross-section at the corresponding location are indicated as A_t , A_1 and A_m , respectively. For the present ejector geometry, $A_1/A_t \simeq 1.33$ and $A_m/A_t \simeq 4.53$. Regarding the aspect ratio of the cross-section, it is close to 8 at the throat of the primary nozzle, whereas the nozzle exit is characterized by an aspect ratio slightly higher than 6. The aspect ratio of the cross-section of the mixing duct is approximately equal to 1.8 and diminishes towards the exit of the diffuser where it is only 0.4.

The Cartesian coordinate system used in this work is also shown in Fig. 2.3, as well as its origin, located on the ejector centerline at the entry of the mixing duct. Hence, the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$. The primary nozzle exit plane is located at

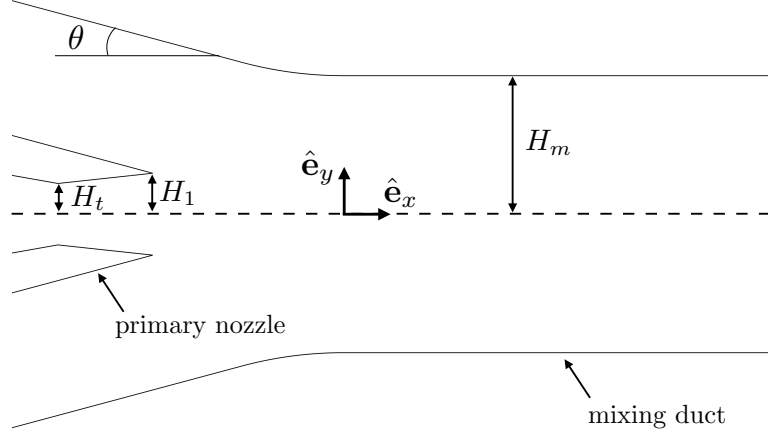


Figure 2.3: Definition of the geometrical parameters used throughout the manuscript.

$x/H_m \simeq -1.37$, characterizing the so-called NXP value in the literature. The inlet converging angle of the secondary nozzle, θ , is 15° . The detailed dimensioned drawing of the ejector used in this work is shown in Fig. 2.4.

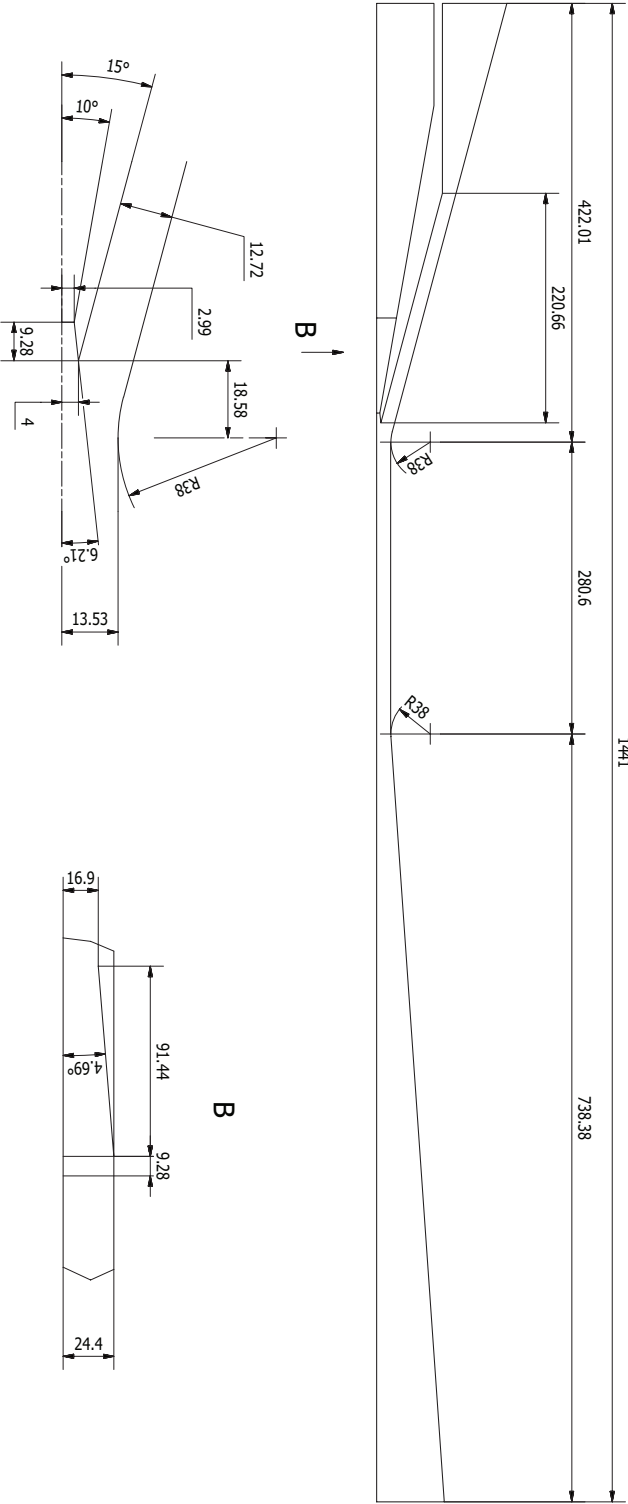


Figure 2.4: Dimensioned drawing of the ejector used for the experimental investigations and modeled in the numerical simulations. All dimensions are in mm.

2.3 Numerical methods

2.3.1 Compressible RANS equations

Navier-Stokes equations

Before describing the Reynolds-averaged Navier-Stokes (RANS) equations for turbulent compressible flows, the instantaneous Navier-Stokes equations are briefly recalled. In the absence of body forces, the conservative form of these equations may be written in a Cartesian frame of reference as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (u_j \rho) = 0 \quad (\text{mass}), \quad (2.1)$$

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (u_j \rho u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j} \quad (\text{momentum}), \quad (2.2)$$

$$\frac{\partial}{\partial t} (\rho e_t) + \frac{\partial}{\partial x_j} (u_j \rho h_t) = -\frac{\partial q_j}{\partial x_j} + \frac{\partial}{\partial x_j} (\tau_{ij} u_i) \quad (\text{energy}), \quad (2.3)$$

with e_t and h_t the specific total energy and total enthalpy of the fluid, defined as

$$e_t = e + \frac{u_j u_j}{2}, \quad (2.4)$$

$$h_t = e + p/\rho + \frac{u_j u_j}{2}, \quad (2.5)$$

respectively. In the Navier-Stokes equations, τ_{ij} represents the viscous stress tensor, defined for a Newtonian fluid (under Stokes' hypothesis) as

$$\tau_{ij} = \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right). \quad (2.6)$$

Regarding, the diffusive flux of heat, q_j , it is generally represented by Fourier's law

$$q_j = -\kappa \frac{\partial T}{\partial x_j}, \quad (2.7)$$

with κ the thermal conductivity of the fluid.

Reynolds and Favre average

When dealing with turbulent compressible flows, the RANS approach generally resorts to two different methods of averaging: the conventional time-average and the density-weighted (or Favre) average. Using the Favre averaging for non-constant density flows leads to much simpler expression.

For any quantity ϕ , the conventional time-average is defined by

$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_{-T/2}^{T/2} \phi(\mathbf{x}, t + \tau) d\tau . \quad (2.8)$$

The corresponding fluctuating quantity is indicated with ϕ' such that

$$\phi = \bar{\phi} + \phi' . \quad (2.9)$$

Based on these definitions, it directly follows that $\overline{\phi'} = 0$.

The density-weighted average of the instantaneous quantity φ is indicated with $\hat{\varphi}$ and defined by

$$\hat{\varphi} = \overline{\rho\varphi} / \bar{\rho} . \quad (2.10)$$

The fluctuating quantity with regard to the density-weighted average, φ'' , is directly related to φ and $\hat{\varphi}$ through

$$\varphi = \hat{\varphi} + \varphi'' , \quad (2.11)$$

such that $\overline{\rho\varphi''} = 0$. When averaging the convective terms of the Navier-Stokes equations, the mean of triple and quadruple products emerge, where one of the factor is the density. After computation, these products may be written in a general form as

$$\overline{\rho AB} = \bar{\rho} \hat{A} \hat{B} + \overline{\rho A'' B''} , \quad (2.12)$$

$$\overline{\rho ABC} = \bar{\rho} \hat{A} \hat{B} \hat{C} + \overline{\rho B'' C''} \hat{A} + \overline{\rho A'' C''} \hat{B} + \overline{\rho A'' B''} \hat{C} + \overline{\rho A'' B'' C''} . \quad (2.13)$$

Averaged equations

The compressible Reynolds-Averaged Navier-Stokes (RANS) equations are obtained by averaging the instantaneous Navier-Stokes equations given in Eqs. (2.1), (2.2) and (2.3). The resulting equations are

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j} (\hat{u}_j \bar{\rho}) = 0 , \quad (2.14)$$

$$\frac{\partial (\bar{\rho} \hat{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\hat{u}_j \bar{\rho} \hat{u}_i) = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\bar{\tau}_{ji} + \bar{\tau}_{ji}^t) , \quad (2.15)$$

$$\begin{aligned} \frac{\partial (\bar{\rho} \hat{e}_t)}{\partial t} + \frac{\partial}{\partial x_j} (\hat{u}_j \bar{\rho} \hat{e}_t) &= - \frac{\partial}{\partial x_j} (\bar{q}_j + \bar{q}_j^t) + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} \hat{u}_i + \bar{\tau}_{ij}^t \hat{u}_i) \\ &\quad + \frac{\partial}{\partial x_j} \left(\overline{\tau_{ij} u_i''} - \frac{1}{2} \overline{\rho u_i'' u_i'' u_j''} \right) . \end{aligned} \quad (2.16)$$

In Eq. (2.16), $\hat{e}_t$ represents the Favre-averaged specific total energy of the fluid, given by

$$\hat{e}_t = \hat{e} + \underbrace{\frac{\hat{u}_j \hat{u}_j}{2}}_{\triangleq K} + \underbrace{\frac{\overline{\rho u_j'' u_j''}}{2\bar{\rho}}}_{\triangleq k}, \quad (2.17)$$

where the specific mean-flow kinetic energy, K , and the specific kinetic energy of the fluctuating turbulent field, k , are defined as well. The Favre-averaged specific total enthalpy, $\hat{h}_t$, is

$$\hat{h}_t = \hat{e} + \frac{\bar{p}}{\bar{\rho}} + K + k. \quad (2.18)$$

2.3.2 Closure of the RANS equations

Most commonly, the following approximations are made for the time-averages of the viscous stress tensor and of the heat flux

$$\bar{\tau}_{ij} \simeq \mu(\hat{T}) \left(\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \hat{u}_k}{\partial x_k} \delta_{ij} \right), \quad (2.19)$$

$$\bar{q}_j \simeq -\kappa(\hat{T}) \frac{\partial \hat{T}}{\partial x_j}. \quad (2.20)$$

In Eqs. (2.15) and (2.16), $\bar{\tau}_{ij}^t$ represents the Reynolds stress tensor, whose elements are given by $-\rho u_i'' u_j''$, whereas $\bar{q}_j^t$ corresponds to the j^{th} component of the turbulent heat flux, i.e. $\bar{q}_j^t = c_p \overline{\rho u_j'' T''}$. The common Boussinesq approximation states that the Reynolds stress tensor may be written as

$$\bar{\tau}_{ij}^t \simeq \mu_t \left(\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \hat{u}_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \bar{\rho} k \delta_{ij}, \quad (2.21)$$

with μ_t the eddy viscosity provided by a turbulence model. Concerning the turbulent heat flux, it is usually modeled through a Reynolds analogy, i.e.

$$\bar{q}_j^t \simeq -\frac{c_p \mu_t}{\text{Pr}_t} \frac{\partial \hat{T}}{\partial x_j}, \quad (2.22)$$

where Pr_t is the turbulent Prandtl number, often taken to be constant [6, 8, 46, 64, 83]. Although there exist RANS models which are not based on the Reynolds analogy (i.e. Pr_t concept) such as the $k - \epsilon$, $k_\theta - \epsilon_\theta$ [58], the Pr_t concept has been shown to be valid for fluid having a $\text{Pr} \sim 1$ such as air. With regard to the value of Pr_t , the most commonly used

value is 0.9 which gives a quasi-perfect mapping between momentum and energy diffusion [83].

In general, additional simplifications are made. The contributions of the molecular diffusion, $\overline{\tau_{ij}u_i''}$, and of the turbulent transport of turbulence kinetic energy, $\frac{1}{2}\overline{\rho u_i''u_i''u_j''}$, are usually ignored in the energy equation (see Eq. (2.16)). This is usually a good approximation for flows with Mach numbers up to the supersonic range, which follows from the fact that $k \ll \hat{h}$ in most flows of engineering interest [83]. Furthermore, the $\frac{2}{3}\overline{\rho k\delta_{ij}}$ term is generally neglected in the expression of $\overline{\tau_{ij}^t}$ given in Eq. (2.21) and the turbulence kinetic energy is often ignored in the calculation of $\hat{e}_t$ and $\hat{h}_t$ (Eqs. (2.17) and (2.18)) [83]. In the present work, these terms were implemented and their influence was found to be negligible.

2.3.3 Fluid properties

Regarding the equation of state of the fluid, air is modeled with the ideal gas law

$$p = \rho RT , \quad (2.23)$$

where R represents the specific gas constant, defined as $R = \mathcal{R}/M_m$, with $\mathcal{R}$ the universal gas constant equal to 8.314 [J/(K·mol)]. The specific gas constant of air is taken as $R = 287.05$ [J/(kg·K)]. It should be noted that the ideal gas law also holds for averaged quantities

$$\bar{p} = \bar{\rho} R \hat{T} . \quad (2.24)$$

With regard to the thermodynamic properties of air, its specific heat ratio, i.e. $\gamma = c_p/c_v$, is assumed to be constant and equal to 1.4, which is a good approximation in view of the temperatures and the pressures prevailing within the ejector. In contrast, the temperature-dependence of its molecular viscosity, μ , and of its thermal conductivity, κ , is taken into account and approximated by Sutherland's law [79]

$$\begin{aligned} \mu(T) &\simeq A_s \frac{\sqrt{T}}{1 + T_s/T} , \\ \kappa(T) &\simeq \mu (1.32c_v + 1.77R) , \end{aligned}$$

where A_s and T_s are input parameters that depend on the considered fluid. For air, they are equal to $1.458 \cdot 10^{-6}$ [Pa·s/K^{1/2}] and 110.4 [K], respectively.

2.3.4 Flow solver

The present numerical simulations were produced by means of the open-source software for CFD, OpenFOAM 2.3 [82]. OpenFOAM relies on a Finite Volume method working with an unstructured discretization. Regardless of the solver, variables are colocated and defined at cell centres, and the governing equations are solved through a segregated approach.

In this work, the compressible RANS equations are solved by means of an unsteady density-based compressible flow solver, named *rhoCentralFoam*. The solver, described in details in the original paper of Green-shields et al. [41], uses an alternative approach to Riemann solvers based on the central-upwind schemes of Kurganov, Noelle and Petrova [52]. These schemes do not involve processes of reconstruction, evolution and projection and may actually be described as simple interpolation procedures [41]. As in most codes relying on a finite volume approach, solving the governing equations requires to evaluate fluxes at cell faces. These are generally calculated by interpolating the cell centre values onto the faces. Due to the inherent structure of OpenFOAM, the interpolation to a given face may only use the variables defined at the center of the two neighbouring cells. The specificity of the central-upwind schemes used in *rhoCentralFoam* lies in the treatment of the convective terms present in the governing equations, given in Eqs. (2.14-2.16). Indeed, for these terms, the flux at a given cell face is interpolated twice, using flux limiters computed for each neighbouring cell. The value that is ultimately retained for the flux at the cell face is obtained by combining the two interpolations with a weighting biased in the upwind direction based on local speeds of propagation [41]. With regard to the diffusive terms, they are discretized with central difference schemes.

Concerning the time discretization, the continuity equation that provides the density field (Eq. (2.14)) is solved explicitly while the resolution of the momentum and the energy equations (Eqs. 2.15 and 2.16, respectively) uses an operator-splitting approach: firstly, an explicit predictor equation is solved for the convection of *conserved* variables ($\bar{\rho}\hat{u}_i$ and $\bar{\rho}\hat{e}_t$), then the diffusion of *primitive* variables ($\hat{u}_i$ and $\hat{T}$) is taken into account through the resolution of an implicit corrector equation. For both explicit and implicit time integrations, the first order Euler method is used. Finally, the equation of state is used to obtain the pressure field. As the selected solver is unsteady, the solution is assumed to achieve a steady state as soon as the relative difference in mass flow

rates between the inlet and outlet boundaries remains below $5 \cdot 10^{-4}$. The value of the CFL number is set to 0.5 for all operating conditions.

For the present numerical results, the effects of turbulence are accounted for using the wall-resolved $k\text{-}\omega\text{-}SST$ turbulence model. This model has been found to yield the best agreement with the experimental measurements concerning both global and local flow quantities in many studies [9, 15, 27, 31, 59].

2.3.5 Computational Domain

The rectangular ejector is modeled as a two-dimensional geometry. Although Mazzelli et al. showed that, for the ejector under consideration, 3D simulations perform better than 2D simulations in off-design conditions, the errors for the prediction of the entrainment ratio are comparable in the on-design regime [59]. The investigation of the discrepancies between 2D and 3D numerical results is beyond the scope of this work, which rather aims at introducing novel post-processing approaches in order to shed light on the mixing phenomena as well as investigating the phenomenology of the choking of the secondary stream in on-design operating conditions. Hence, the analysis is almost exclusively confined to 2D simulations in this thesis. A preliminary investigation of 3D effects on an upgraded experimental setup is proposed at the end of Chapter 6 for a single operating point. It should be noted that the flow is also modeled as symmetrical with respect to the ejector centerline, which allows the simulation to be performed for only half of the ejector.

The grid structure of the numerical domain is shown in Fig. 2.5. The mesh has been generated via GMSH [37] and is essentially structured. The region extending from the exit of the primary nozzle up to the first quarter of the mixing duct is the only part of the mesh that is unstructured. The mesh is refined in the vertical direction in order to ensure that the centre of the cell adjacent to the wall is always at $y^+ \simeq 1$. In the unstructured zone, a structured layer is used close to the wall to ensure a uniform near-wall refinement. In the axial direction, the mesh is refined in the unstructured zone since it is precisely where both streams come into contact that gradients are expected to be important and that supersonic structures will appear in the core of the primary flow. The smallest cell of the mesh is located in the vicinity of the tip of the primary nozzle and has a characteristic length of nearly $3 \text{ } [\mu\text{m}]$. The axial refinement decreases smoothly from the entry of the mixing duct to the diffuser in the entire mixing section. In total, the mesh is composed of approximately 350,000 hexahedra.

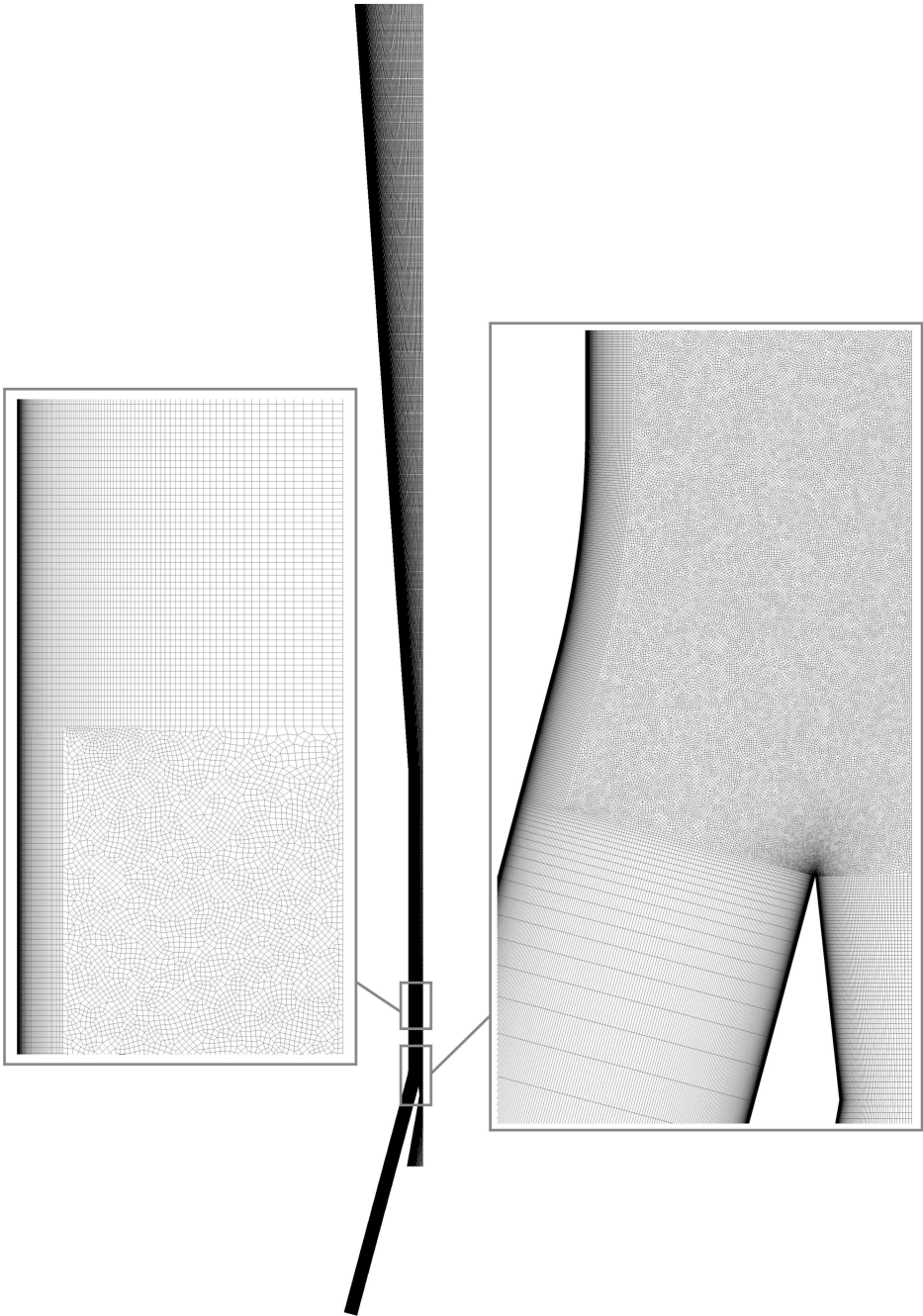


Figure 2.5: Grid structure of the computational domain for 2D numerical simulations.

The grid independence has been checked through the comparison of the profile of the Mach number on the axis of the ejector for the mesh described here above and a refined mesh. The finer mesh was obtained by further refinement only in the axial direction and solely in the structured zones, yielding a mesh of approximately 535,000 cells. Results are shown in Fig. 2.6. To simplify the comparison, profiles are shown only from the nozzle exit position. Note that for the operating conditions under consideration, the ejector operates in the on-design regime as the back-pressure is relatively low. In this case, the primary stream is still supersonic at the exit of the mixing duct and the flow is slowed down to subsonic velocities through a succession of oblique shock waves. The latter are evidenced by the wiggles at the entry of the diffuser in Fig. 2.6.

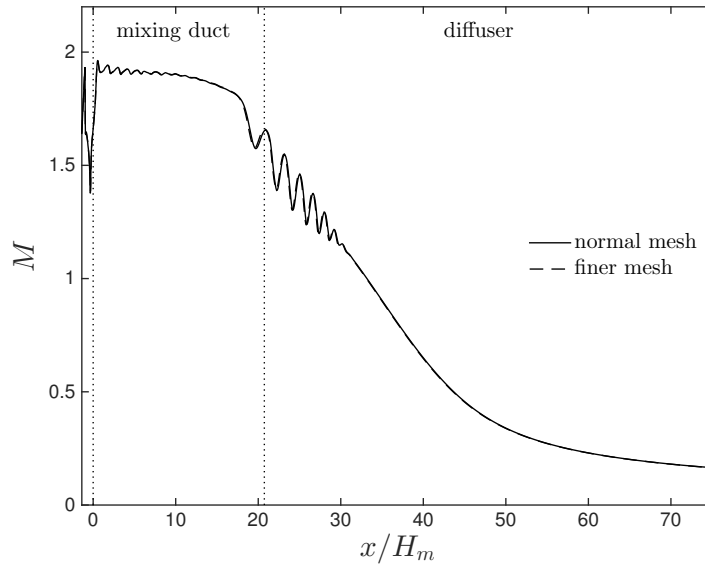


Figure 2.6: Mach Number profile along the axis of the ejector for grid independence verification.

2.3.6 Boundary conditions

The flow being subsonic at both inlets, the pressure and the temperature are specified as stagnation conditions, i.e. p_{01} and T_{01} for the primary stream and p_{02} and T_{02} for the secondary stream. At the outlet boundary, the sole variable that is specified is the static pressure, p_{out} .

For k and ω at both inlets, the value of the turbulence intensity (5%) and a specific mixing length are prescribed. This length is obtained for each inlet by $l = 0.07d_h$ with d_h the hydraulic diameter of the corresponding intake pipe, and then used to obtain ω through $\omega = \sqrt{k}/(C_\mu^{1/4}l)$ with C_μ a model coefficient set to 0.09 [34]. Walls are all assumed adiabatic.

2.4 Verification and validation

2.4.1 At global scale

In a first approach, the numerical results are validated at the global scale by comparing two characteristic curves obtained by means of CFD with the experimental data of Mazzelli et al. [59]. Hence, the boundary conditions at both inlets and at the outlet of the computational domain are based on experimental measurements.

As mentioned in the introduction, a characteristic curve shows the entrainment ratio, ER, as a function of the compression ratio, p_{out}/p_{02} , for fixed reservoir conditions. In the present work, two different values of the primary stagnation pressure are considered: $p_{01} = 3.50$ [bar] and $p_{01} = 5.00$ [bar]. The pneumatic valve used in the experiment for the regulation of the primary stagnation pressure ensures that it remains almost constant for all the operating points of the related characteristic curve. For the experimental results considered here, the maximum deviation was found to be less than 0.15% [59].

As the secondary stream is sucked from the atmosphere in the experiment, its stagnation pressure is close to the atmospheric pressure for all cases. However, the lack of regulation does not allow to ensure a constant secondary stagnation pressure for an entire characteristic curve. In particular, the orifice plate used for the measurement of the mass flow rate in the secondary intake (cfr. Section 2.2.1) induces a pressure drop which increase with the mass flow rate. As a consequence, the pressure measured in the experiment downstream of the orifice plate slightly varies with the entrained mass flow rate. For a given characteristic curve, as the primary and secondary reservoir conditions are fixed, the entrained mass flow rate and the resultant pressure drop in the secondary intake are only affected by the back-pressure once the latter exceeds the critical back-pressure. Hence, the stagnation pressure at the position coinciding with the secondary inlet in the numerical domain (downstream of the orifice plate) is constant in the experiment for on-design operation and slightly varies in the off-design

regime. The variation of p_{02} with the back-pressure has been taken into account for the validation of the numerical results presented below.

The comparison between the numerical and the experimental characteristic curves is shown in Fig. 2.7. In addition, the 2D simulations performed by Mazzelli et al. [59] with the same turbulence model but with a different solver are also shown in the figure. With regard to the entrainment ratio, the present numerical results are very close to those reported in [59] all over the range of the back-pressure.

For both values of p_{01} considered here, the predicted value of the entrainment ratio seems to be in good agreement with the experimental data in on-design conditions. The slight overestimation of the on-design entrainment ratio by the CFD model for the characteristic curve corresponding to $p_{01} = 5.00$ [bar] is close to 5.5% and may be nearly halved by performing 3D simulations [59]. As mentioned in the introduction, 3D effects on an upgraded ejector geometry will be investigated for one operating point in Chapter 6.

Regarding the critical back-pressure, the numerical simulations tend to overpredict it, thereby leading to some discrepancies in the off-design regime. Here again, 3D simulations lead to better results [59]. Note that the small bump noticeable in the vicinity of the critical back-pressure in the numerical characteristic curves is related to the increase of the stagnation pressure at the secondary inlet as outlined above.

2.4.2 At local scale

As demonstrated by Hemidi et al. [44], the link between a good validation in terms of a global performance parameter such as the entrainment ratio and a correct prediction of the local flow physics is not guaranteed. Therefore, the present numerical results are also validated with the experiment at the local scale for both primary stagnation pressures. More precisely, the shock reflection pattern at the exit of the primary nozzle has been visualized through schlieren techniques and compared to a numerical pseudo-schlieren. This validation at the local scale is performed for the operating point corresponding to the lowest value of the back-pressure of each characteristic curve. Hence, for both selected cases, the ejector operates in the on-design regime and the flow structure at the exit of the primary nozzle is relatively stable. The validation in the off-design regime is less straightforward because of flow instabilities with a jet flapping phenomenon and is beyond the scope of the present work.

In the experiment, some scratches on the plexiglas were found to soil the picture. In order to reduce their impact on the final images,

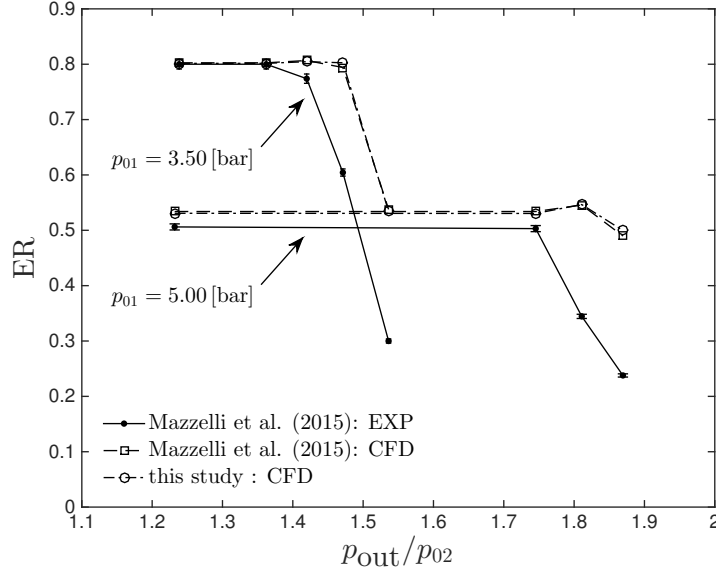
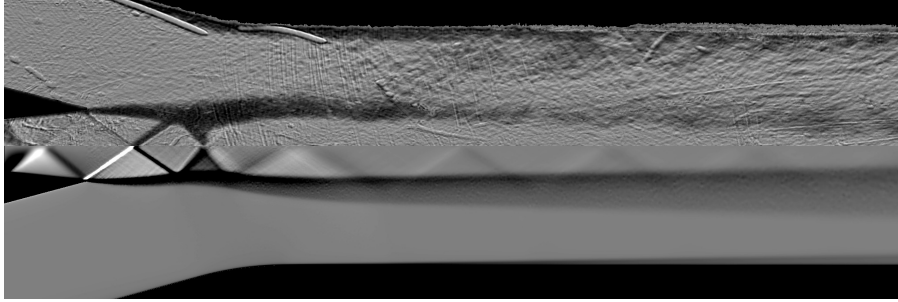


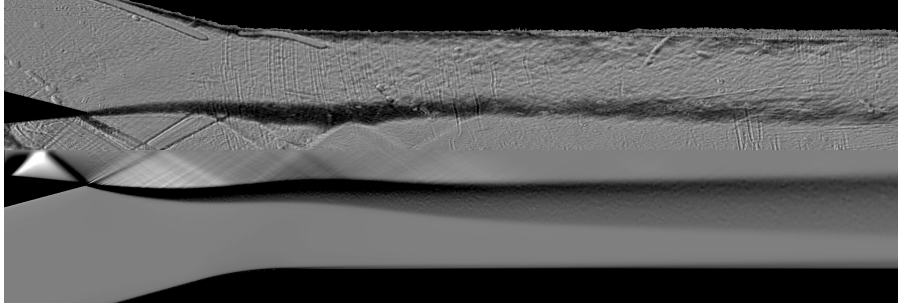
Figure 2.7: Comparison of experimental and numerical characteristic curves.

a no-flow picture has been subtracted from each raw image obtained in the presence of the flow before it was compared to the numerical pseudo-schlieren. Note that the latter was generated by computing the vertical gradient of the density field, as the knife-edge was held horizontally in the experiment. The comparison between the numerical and the experimental images is shown in Fig. 2.8. The reservoir conditions corresponding to $p_{01} = 3.50$ [bar] give rise to an overexpanded motive flow, as indicated by the oblique shock wave emanating from the lip of the primary nozzle in Fig. 2.8(a). For the operating point obtained with a higher primary stagnation pressure ($p_{01} = 5.00$ [bar]) the expansion waves in the core of the primary stream visible in Fig. 2.8(b) indicate that the motive flow is underexpanded at the nozzle exit plane.

Numerical results are found to be in fairly good agreement with the experiment for both operating points. Nevertheless, some subtle differences are apparent between the numerical and the experimental images. For the case corresponding to $p_{01}/p_{02} = 3.61$, shown in Fig. 2.8(a), the typical diamond pattern formed by the oblique shock waves within the core of the primary stream seems to extend further into the mixing duct in the numerical schlieren compared to the experiment. This discrepancy could be due to the sensitivity of the experimental picture



(a) $p_{01} = 3.50$ [bar] – $p_{\text{out}}/p_{02} = 1.238$.



(b) $p_{01} = 5.00$ [bar] – $p_{\text{out}}/p_{02} = 1.232$.

Figure 2.8: Comparison of experimental (top half of each image) and numerical (bottom half of each image) schlieren images for different primary stagnation pressures (horizontal knife-edge).

which is different than that of the numerical image. Indeed, the numerical colormap was chosen in order to highlight the structure of the flow at the exit of the primary nozzle. By doing so, some details in the flow may be either highlighted or masked compared to the experimental image.

Furthermore, it seems that the numerical simulation does not capture some features that may be observed in the zone occupied by the secondary stream in the experiment. These features may consist in shock-boundary layer interactions with the wall of the mixing duct and/or with the lateral windows. Note that the exposure time of the camera could also have an influence on the experimental image since the impact of the turbulence may be more or less smoothed in the picture. Finally, the instantaneous experimental image results from the ‘integration’ of flow features over the span-wise direction thus including eventual 3D effects, whereas the numerical schlieren is a time average of a 2D slice.

2.5 Numerical results exploited in the manuscript

As mentioned in the introduction, this work primarily aims at investigating mixing and choking mechanisms. In order to highlight the sole effect of the back-pressure on these complex phenomena, the reservoir conditions should be fixed for a given characteristic curve. Hence, the analyses presented in the following chapters are based on numerical characteristic curves obtained with a constant secondary stagnation pressure. The slight changes in the stagnation pressure at the secondary inlet observed in the experiment for off-design cases are thus neglected. For the experimental data under consideration [59], p_{02} is thus equal to 0.969 [bar] for the cases corresponding to $p_{01} = 3.50$ [bar] while p_{02} is set to 0.974 [bar] for the curve obtained with $p_{01} = 5.00$ [bar]. The resulting characteristic curves are shown in Fig. 2.9. Logically, the bumps that were noticeable in the numerical characteristic curves reported in Fig. 2.7 have disappeared. For the numerical results shown in Fig. 2.9 and exploited in the manuscript, since p_{02} is constant for a given characteristic curve, each curve is designated by the corresponding ratio of stagnation pressures, p_{01}/p_{02} , rather than p_{01} only. In addition, a certain letter has been associated to each operating point in order to ease the reference to the different cases throughout the manuscript.

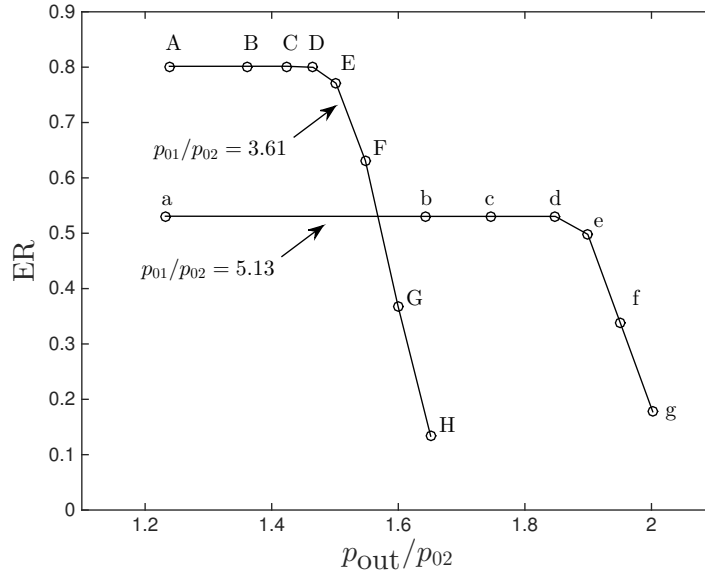


Figure 2.9: Numerical characteristic curves exploited in the manuscript.

CHAPTER
3

NEW METHODS FOR ANALYZING TRANSPORT PHENOMENA IN SUPERSONIC EJECTORS

This chapter is based on the following paper:

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.

3.1 Introduction

As evidenced by the literature review, there is no study that really addresses in-depth the open question related to the quantification and the location of the transfers and the irreversibilities within the ejector, and their connection with the entrainment. It is therefore necessary to develop new post-processing methods for analyzing transport phenomena in supersonic ejectors. In this chapter, a methodology is proposed to tackle this issue. The adopted perspective is based on the fundamental purpose of the ejector which consists in acting as a mechanical

compressor. Hence the main focus is put on the work done by the primary stream on the secondary stream.

Two different and complementary approaches are proposed. First of all, recent analysis tools based on momentum and energy tubes [62] are extended to the present compressible flow context and applied to the mean-flow structure of turbulent flow within the ejector. Since the ejector aims to give to the secondary stream the capacity to produce some work, particular consideration should be given to exergy. Hence the transport equation for the mean-flow total exergy is derived and exergy transport tubes are proposed as a tool for the investigation of transport phenomena within the ejector. In addition to this topological approach, an analysis based on classical stream tubes is performed in order to quantitatively investigate transfers between the primary and the secondary streams all along the ejector. The present work also identifies the location of exergy losses and their origins. Throughout this analysis, new local and cumulative parameters related to transfers and irreversibilities are introduced.

The chapter is structured as follows: as a preamble to the rest of the chapter, the concept of exergy and the definition of exergy efficiencies in supersonic ejectors are presented in Section 3.2. Subsequently, the post-processing methodology for the construction of momentum, energy and exergy tubes is described in Section 3.3. Section 3.4 is devoted to the application and the visualization of these transport tubes in the ejector for several operating points. The transfers between the primary and the secondary streams, the irreversibilities and the global exergy balance for the secondary stream are then investigated in Sections 3.5, 3.6 and 3.7, respectively. Finally, some conclusions and perspectives are drawn in Section 3.8.

3.2 Definition of exergy efficiencies in supersonic ejectors

3.2.1 The concept of exergy

The exergy is generally defined as the maximum theoretical work that can be obtained from an amount of energy [12]. The total specific exergy, ξ_t , is a state variable that may be defined as

$$\xi_t = (h_t - h_{\text{ref}}) - T_{\text{ref}} (s - s_{\text{ref}}) , \quad (3.1)$$

where h_t and s are the specific total enthalpy and the specific entropy, respectively. In Eq. (3.1), the subscript 'ref' refers to the reference state

(or *dead* state), generally defined by $T_{\text{ref}} = 300 \text{ [K]}$ and $p_{\text{ref}} = 1 \text{ [bar]}$. In the present study, it is proposed to consider instead the reservoir conditions of the secondary stream as the reference state, namely $T_{\text{ref}} = T_{02}$ and $p_{\text{ref}} = p_{02}$. For refrigeration applications, it would correspond to the temperature at the evaporator and the corresponding saturation pressure. By doing so, the exergy of the secondary stream at reservoir conditions, noted ξ_{02} , is equal to zero.

Note that the stagnation value of the enthalpy is considered in the definition of exergy given in Eq. (3.1) to account for the exergy content of the kinetic energy, which is often omitted in exergy analysis [28, 38]. When the kinetic energy is not included in the definition of the exergy, the latter is impacted by local acceleration or deceleration of the fluid, although these changes in velocity could simply be due to a changing cross-sectional flow area. By contrast, considering the kinetic energy as part of the exergy allows to highlight the variations in exergy due to transfer and/or destruction phenomena only, avoiding potential misinterpretation.

3.2.2 Global exergy balance in supersonic ejectors

If the flow within the supersonic ejector is steady and adiabatic, the first law of thermodynamics may be written for the entire process as follows

$$(\dot{m}_1 + \dot{m}_2) h_{\text{out}} = \dot{m}_1 h_{01} + \dot{m}_2 h_{02} , \quad (3.2)$$

where h_{01} and h_{02} represent the enthalpy at reservoir conditions of the primary and the secondary streams, respectively, while h_{out} is the total enthalpy of the mixed stream at the outlet of the ejector. Regarding the second law of thermodynamics, it may be expressed as

$$(\dot{m}_1 + \dot{m}_2) s_{\text{out}} \geq \dot{m}_1 s_{01} + \dot{m}_2 s_{02} . \quad (3.3)$$

Multiplying Eq. (3.3) by T_{ref} and subtracting it from Eq. (3.2) yields the global exergy balance of the process taking place within the ejector

$$(\dot{m}_1 + \dot{m}_2) \xi_{\text{out}} \leq \dot{m}_1 \xi_{01} + \dot{m}_2 \xi_{02} , \quad (3.4)$$

where ξ_{out} is the total exergy of the mixed stream at the outlet of the ejector.

3.2.3 The reversible ejector

It may be interesting to investigate the hypothetical case of a reversible ejector in order to determine the theoretical limit of its performance. If

the process taking place within the ejector is reversible, there will be no generation of entropy and thus no exergy losses. In this situation, the equality sign must hold in Eq. (3.4). The resulting equation may be used to determine the maximum exergy that could be reached at the outlet of the ejector through a reversible mixing, noted $\xi_{\text{out}}^{\text{rev}}$, for known reservoir conditions and a given entrainment ratio

$$\xi_{\text{out}}^{\text{rev}} = \frac{\xi_{01} + \text{ER}\xi_{02}}{1 + \text{ER}}. \quad (3.5)$$

By imposing that there is no exergy losses within the ejector, one may also determine the entrainment ratio corresponding to a reversible process, ER^{rev} , for specified operating conditions, including the total exergy of the mixed stream at the outlet of the ejector, ξ_{out} . Indeed, enforcing the equality in Eq. (3.4) leads to the following expression for the reversible entrainment ratio

$$\text{ER}^{\text{rev}} = \frac{\xi_{01} - \xi_{\text{out}}}{\xi_{\text{out}} - \xi_{02}}. \quad (3.6)$$

3.2.4 Exergy efficiencies

Based on the global exergy balance expressed in Eq. (3.4), several exergy efficiencies may be defined for supersonic ejectors. Indeed, as pointed out in [61], the identification of the exergy inputs and outputs in such devices is somewhat subjective. The two main perspectives are illustrated in Fig. 3.1.

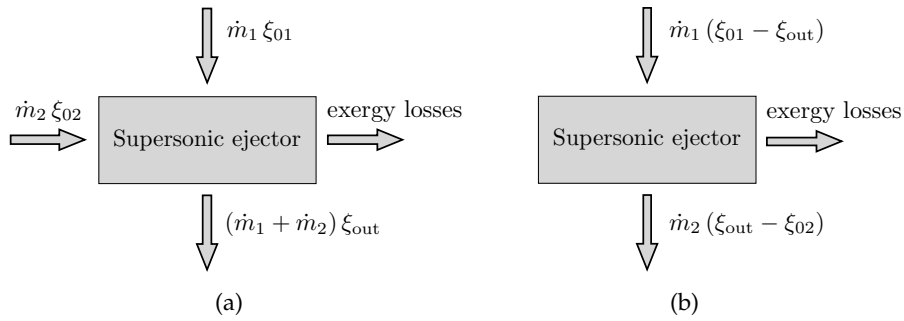


Figure 3.1: The two main perspectives for the exergy balance in a supersonic ejector.

In the first option, shown in Fig. 3.1(a), the flow within the ejector is essentially seen as a mixing process. Hence, the exergy fluxes are

associated with the inputs and outputs in terms of mass, leading to the following definition of the exergy efficiency

$$\eta_m = \frac{(\dot{m}_1 + \dot{m}_2)\xi_{\text{out}}}{\dot{m}_1\xi_{01} + \dot{m}_2\xi_{02}} = \frac{(1 + \text{ER})\xi_{\text{out}}}{\xi_{01} + \text{ER}\xi_{02}}. \quad (3.7)$$

This definition is often adopted in the literature, for example in [28, 71]. It may easily be shown that η_m actually represents the ratio of the exergy at the outlet of the ejector in the real application, ξ_{out} , to the maximum theoretical value, $\xi_{\text{out}}^{\text{rev}}$, defined in Eq. (3.5), i.e.

$$\eta_m = \frac{\xi_{\text{out}}}{\xi_{\text{out}}^{\text{rev}}}. \quad (3.8)$$

The second definition of the exergy efficiency that is used for supersonic ejectors consists in identifying such a device as a compressor, whose purpose is to transfer exergy to a fluid [3, 61]. Seen from this perspective, illustrated in Fig. 3.1(b), the change in exergy of the secondary stream is considered as the exergy output of the process taking place within the ejector, while the exergy input is the change in exergy of the primary stream. This approach leads to the following expression of the ejector efficiency

$$\eta_c = \frac{\dot{m}_2 (\xi_{\text{out}} - \xi_{02})}{\dot{m}_1 (\xi_{01} - \xi_{\text{out}})} = \text{ER} \frac{(\xi_{\text{out}} - \xi_{02})}{(\xi_{01} - \xi_{\text{out}})}. \quad (3.9)$$

From the definition of ER^{rev} given in Eq. (3.6), it directly follows that η_c also reflects the ratio of the actual entrainment ratio to the theoretical entrainment ratio corresponding to a reversible process, i.e.

$$\eta_c = \frac{\text{ER}}{\text{ER}^{\text{rev}}}. \quad (3.10)$$

Even though this relation is said to be valid for ideal gas only in [61], it is shown here that it must hold whatever the nature of the fluid.

3.2.5 Application to a calorically perfect gas

For a calorically perfect gas, the total specific exergy of the fluid, defined in Eq. (3.1), takes the following expression

$$\xi_t = c_p(T_t - T_{\text{ref}}) - T_{\text{ref}} \left[c_p \ln \left(\frac{T_t}{T_{\text{ref}}} \right) - R \ln \left(\frac{p_t}{p_{\text{ref}}} \right) \right]. \quad (3.11)$$

Considering that, by definition, entropy remains unchanged between the static and the total states, ξ_t is also equal to

$$\xi_t = c_p(T_t - T_{\text{ref}}) - T_{\text{ref}} \left[c_p \ln \left(\frac{T_t}{T_{\text{ref}}} \right) - R \ln \left(\frac{p_t}{p_{\text{ref}}} \right) \right]. \quad (3.12)$$

Hence, for a calorically perfect gas, Eq. (3.5) may be used to yield an expression for the total pressure at the outlet of the ejector corresponding to a reversible process, noted $p_{\text{out}}^{\text{rev}}$

$$\frac{p_{\text{out}}^{\text{rev}}}{p_{\text{ref}}} = \exp \left\{ \frac{1}{T_{\text{ref}} R} \left(\frac{\xi_{01} + \text{ER} \xi_{02}}{1 + \text{ER}} \right) - \frac{\gamma}{\gamma - 1} \left[\left(\frac{T_{\text{out}}}{T_{\text{ref}}} - 1 \right) - \ln \left(\frac{T_{\text{out}}}{T_{\text{ref}}} \right) \right] \right\}, \quad (3.13)$$

where T_{out} is the total temperature at the outlet of the ejector, fixed by the first law of thermodynamics given in Eq. (3.2). For a calorically perfect gas, this equation leads to

$$T_{\text{out}} = \frac{T_{01} + \text{ER} T_{02}}{1 + \text{ER}}. \quad (3.14)$$

Regarding the reversible entrainment ratio, ER^{rev} , it may be calculated with the following expression in the case of a calorically perfect gas [61]

$$\text{ER}^{\text{rev}} = \frac{\frac{\gamma}{\gamma-1} \ln \left(\frac{T_{01}}{T_{\text{out}}} \right) - \ln \left(\frac{p_{01}}{p_{\text{out}}} \right)}{\frac{\gamma}{\gamma-1} \ln \left(\frac{T_{02}}{T_{\text{out}}} \right) - \ln \left(\frac{p_{02}}{p_{\text{out}}} \right)}. \quad (3.15)$$

3.2.6 Impact of the back-pressure on the exergy efficiencies

Both exergy efficiencies defined in Section 3.2.4 are calculated for the numerical characteristic curve corresponding to $p_{01}/p_{02} = 3.61$ shown in Fig. 2.9. As proposed in Section 3.2.1, the reference state for the definition of the exergy is taken as the reservoir conditions of the secondary stream. Hence, ξ_{02} is equal to zero for all the operating points of the characteristic curve. The total exergy corresponding to the primary reservoir conditions, ξ_{01} , and the total exergy of the mixed stream at the outlet of the ejector, ξ_{out} , are calculated with Eq. (3.12). The total pressure of the mixed stream at the outlet of the ejector is approximated by the value used as boundary condition in the numerical simulations as the difference between static and total pressures is negligible at the exit of the diffuser. Regarding the outlet total temperature, T_{out} , it may be determined with Eq. (3.14) since walls are assumed adiabatic in the numerical model. The exergy efficiencies η_m and η_c are finally computed with Eqs. (3.7) and (3.9), respectively, for all the operating points of the characteristic curve. Results are shown in Fig. 3.2.

Both efficiencies have essentially the same pattern, with an optimum (case E) corresponding to a value of the back-pressure slightly higher than the critical back-pressure. This trend could have been expected since ξ_{out} increases with the back-pressure, while ER starts decreasing with p_{out} once it exceeds the critical back-pressure. Hence, the

best performance of the ejector should be reached near the critical operating point. For the ejector geometry and reservoir conditions under consideration, the maximum value of η_m and η_c is 35.70% and 56.05%, respectively. The following analysis will shed light on the local flow phenomena related to the global exergy balance of the supersonic ejector and its variation with the back-pressure.

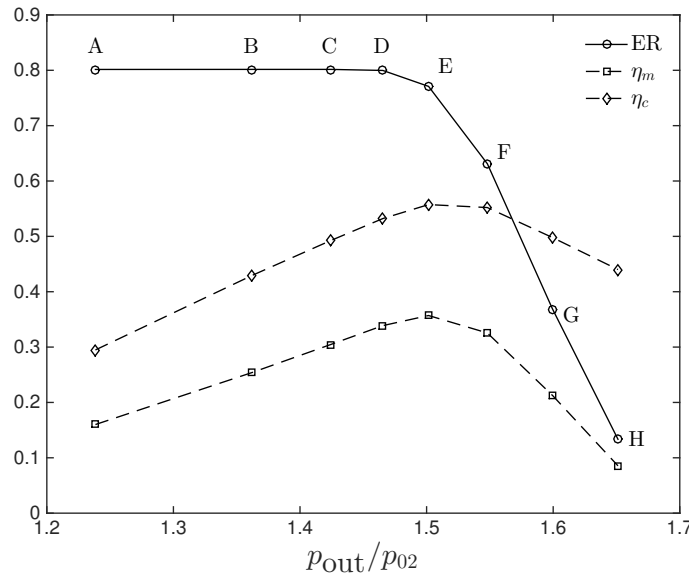


Figure 3.2: Impact of the back-pressure on the exergy efficiencies – $p_{01}/p_{02} = 3.61$.

3.3 Transport tubes

3.3.1 Momentum and mean-flow kinetic energy tubes

In Meyers and Meneveau [62], the concept of momentum and energy transport tubes is introduced as a transport visualization tool. These tubes are a generalization of the mass-flux based classical stream tubes. Just as there is, by construction, no average mass flux through the boundary of a stream tube, momentum and energy transport tubes are constructed as surfaces with zero fluxes of the corresponding quantities. The construction of these tubes is therefore not based on the velocity vector field, but rather by considering the vector fields formed by the total fluxes of momentum or energy.

Figure 3.3 illustrates the concept of a transport tube. The construction of the tube is based on transport lines that are at any point tangent to the local total flux of the considered physical quantity, $\bar{F}$. In the present study, this quantity may be mass (classical stream tube), momentum, mean-flow kinetic energy or exergy. Having defined a closed curve, C , which is nowhere tangent to this vector field, the transport tube originating from this curve is simply constructed by considering the bundle of all transport lines passing through that curve. By doing this, there is no net flux of the corresponding quantity through the tube surface. Hence the difference between the flux of the considered quantity on the tube's cross-sections A_{in} and A_{out} is only due to the integral effects of sources and sinks in the tube.

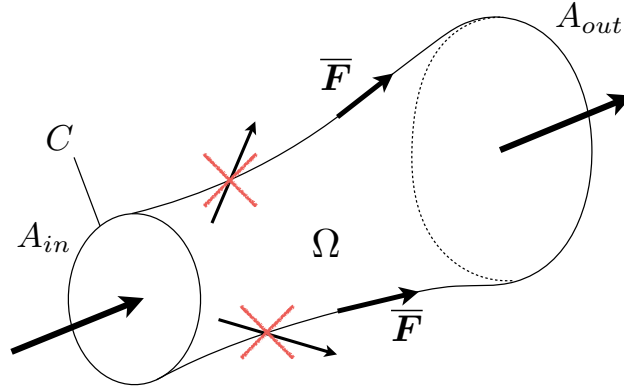


Figure 3.3: Illustration of the concept of a transport tube.

Since linear momentum is a vector quantity, it is proposed to first choose a direction, characterised by a constant unit vector ζ (and components ζ_i), and then to solely consider the linear momentum in the ζ direction, i.e. $\bar{\rho}\hat{u}_i\zeta_i$. The analysis developed by Meyers and Meneveau [62] for incompressible flows is here extended to compressible flows and in the absence of body forces. For statistically steady flows, the transport equation of the momentum in the ζ direction is given by

$$\frac{\partial}{\partial x_j} \left(\underbrace{\hat{u}_j \bar{\rho} (\hat{u}_i \zeta_i) - \bar{\tau}_{ij} \zeta_i - \bar{\tau}_{ij}^t \zeta_i}_{\bar{F}_{m,j}} \right) = - \frac{\partial \bar{p}}{\partial x_i} \zeta_i . \quad (3.16)$$

The left-hand side of the equation is the divergence of the vector field formed by the total flux of linear momentum in the ζ direction, $\bar{F}_m$, consisting of advective, viscous and turbulent fluxes, respectively. For the flow within the ejector, the linear momentum is nearly aligned with the axis of the ejector, hence ζ is chosen as $\hat{e}_x$. For a 2D flow, the total flux of linear momentum aligned with the horizontal direction will have the following expression

$$\bar{F}_{m,x} = (\hat{u}\bar{\rho}\hat{u} - \bar{\tau}_{xx} - \bar{\tau}_{xx}^t) , \quad (3.17)$$

$$\bar{F}_{m,y} = (\hat{v}\bar{\rho}\hat{u} - \bar{\tau}_{xy} - \bar{\tau}_{xy}^t) . \quad (3.18)$$

Considering the flow topology within the ejector, one may expect that the horizontal component of this vector field, $\bar{F}_{m,x}$, will be mainly determined by the contribution of the advective flux (i.e. $\hat{u}\bar{\rho}\hat{u}$). By contrast, for the vertical component, $\bar{F}_{m,y}$, it is likely that the viscous and the turbulent fluxes (i.e. $-\bar{\tau}_{xy} - \bar{\tau}_{xy}^t$) prevail. In what follows, the reader should bear in mind that the term *momentum* will refer to the linear momentum aligned with the axis of the ejector.

For the determination of energy tubes, the transport equation for the mean-flow kinetic energy, $K = \bar{\rho}\hat{u}_i\hat{u}_i/2$, is used. The latter is obtained by the projection of the averaged momentum equation given in Eq. (2.15) onto the mean velocity vector, $\hat{u}$, and may be written as

$$\frac{\partial}{\partial x_j} (\bar{F}_{K,j}) = -\hat{u}_j \frac{\partial \bar{p}}{\partial x_j} - \bar{\tau}_{ij} \frac{\partial \hat{u}_i}{\partial x_j} - \bar{\tau}_{ij}^t \frac{\partial \hat{u}_i}{\partial x_j} , \quad (3.19)$$

where $\bar{F}_{K,j}$ is the j^{th} component of the vector field formed by the total flux of mean-flow kinetic energy, consisting of

$$\bar{F}_{K,j} = K\hat{u}_j - \bar{\tau}_{ij}\hat{u}_i - \bar{\tau}_{ij}^t\hat{u}_i . \quad (3.20)$$

As pointed out in [62], such transport tubes are an interesting means of visualizing where the momentum and the energy in the flow are transported to, especially when transport processes are dominated by turbulence instead of mean-flow convection. Hence it may be very useful for the complex flow within a supersonic ejector. However, since the fundamental purpose of the ejector, as a mechanical compressor, consists in conferring to the secondary stream the capacity to produce some work, it is also interesting to define and investigate *exergy* transport tubes.

3.3.2 Mean-flow exergy tubes

In order to perform an analysis in the post-processing phase of a CFD calculation, the mean-flow total exergy, $\tilde{\xi}_t$, is defined by

$$\tilde{\xi}_t = (\tilde{h}_t - h_{\text{ref}}) - T_{\text{ref}}(\tilde{s} - s_{\text{ref}}) , \quad (3.21)$$

where the symbol $\sim$ indicates that the variable is calculated with its instantaneous definition but from the averaged flow quantities obtained by numerical simulations. Hence the mean-flow total enthalpy, $\tilde{h}_t$, is defined according to Eq. (2.5) as follows

$$\tilde{h}_t = \hat{e} + \frac{\bar{p}}{\bar{\rho}} + \frac{\hat{u}_j \hat{u}_j}{2} . \quad (3.22)$$

In the present study, the turbulence kinetic energy, k , is assumed to be not recoverable any more as mean-flow kinetic energy but not yet recoverable as heat. Hence, it is not taken into account in the definition of exergy.

Before constructing mean-flow total exergy transport tubes, a transport equation for $\tilde{\xi}_t$ must be obtained. Since, for the solver used in the present study, k is neglected in the calculation of $\hat{h}_t$ in Eq. (2.18), no distinction is made between the Favre-averaged total enthalpy, $\hat{h}_t$, and the mean-flow total enthalpy, $\tilde{h}_t$. Hence, in this case, the transport equation of the mean-flow total enthalpy may be obtained easily from Eq. (2.16), neglecting the last term of this equation in accordance with the above

$$\frac{\partial (\bar{\rho} \tilde{h}_t)}{\partial t} + \frac{\partial}{\partial x_j} (\hat{u}_j \bar{\rho} \tilde{h}_t) = \frac{\partial \bar{p}}{\partial t} - \frac{\partial}{\partial x_j} (\bar{q}_j + \bar{q}_j^t) + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} \hat{u}_i + \bar{\tau}_{ij}^t \hat{u}_i) . \quad (3.23)$$

The transport equation of the mean-flow specific entropy, $\tilde{s}$, defined with the classical definition of the entropy but with the averaged flow quantities, i.e.

$$\hat{T} d\tilde{s} = d\hat{e} - \frac{\bar{p}}{\bar{\rho}^2} d\bar{\rho} ,$$

may be written

$$\bar{\rho} \hat{T} \left(\frac{\partial \tilde{s}}{\partial t} + \hat{u}_j \frac{\partial \tilde{s}}{\partial x_j} \right) = \bar{\rho} \left(\frac{\partial \hat{e}}{\partial t} + \hat{u}_j \frac{\partial \hat{e}}{\partial x_j} - \frac{\bar{p}}{\bar{\rho}^2} \left(\frac{\partial \bar{\rho}}{\partial t} + \hat{u}_j \frac{\partial \bar{\rho}}{\partial x_j} \right) \right) .$$

Using mass conservation (see Eq. (2.14)), this equation may be rewritten as

$$\hat{T} \left(\frac{\partial (\bar{\rho} \tilde{s})}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j (\bar{\rho} \tilde{s})] \right) = \frac{\partial (\bar{\rho} \hat{e})}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j (\bar{\rho} \hat{e})] + \bar{p} \frac{\partial \hat{u}_j}{\partial x_j} .$$

The underlying transport equation of $\hat{e}$ may be obtained by subtracting Eq. (3.19) from Eq. (3.23) and reads

$$\frac{\partial(\bar{\rho}\hat{e})}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j(\bar{\rho}\hat{e})] = -\bar{p}\frac{\partial\hat{u}_j}{\partial x_j} - \frac{\partial}{\partial x_j} [\bar{q}_j + \bar{q}_j^t] + (\bar{\tau}_{ij} + \bar{\tau}_{ij}^t) \frac{\partial\hat{u}_i}{\partial x_j}. \quad (3.24)$$

Hence the transport equation of $\tilde{s}$ may be written as

$$\hat{T} \left(\frac{\partial(\bar{\rho}\tilde{s})}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j(\bar{\rho}\tilde{s})] \right) = -\frac{\partial}{\partial x_j} [\bar{q}_j + \bar{q}_j^t] + (\bar{\tau}_{ij} + \bar{\tau}_{ij}^t) \frac{\partial\hat{u}_i}{\partial x_j}. \quad (3.25)$$

By using Eqs. (3.23) and (3.25), the transport equation for the mean-flow total specific exergy, $\tilde{\xi}_t$, can be obtained

$$\begin{aligned} \frac{\partial(\bar{\rho}\tilde{\xi}_t)}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j(\bar{\rho}\tilde{\xi}_t)] &= \frac{\partial\bar{p}}{\partial t} - \frac{\partial}{\partial x_j} [\bar{q}_j + \bar{q}_j^t] + \frac{\partial}{\partial x_j} [\bar{\tau}_{ij}\hat{u}_i + \bar{\tau}_{ij}^t\hat{u}_i] \\ &\quad + \frac{T_{\text{ref}}}{\hat{T}} \frac{\partial}{\partial x_j} [\bar{q}_j + \bar{q}_j^t] - \frac{T_{\text{ref}}}{\hat{T}} \left[\bar{\tau}_{ij} \frac{\partial\hat{u}_i}{\partial x_j} + \bar{\tau}_{ij}^t \frac{\partial\hat{u}_i}{\partial x_j} \right]. \end{aligned} \quad (3.26)$$

Rearranging the terms that involve the mean and the turbulent heat fluxes, this equation may finally be written as

$$\begin{aligned} \frac{\partial(\bar{\rho}\tilde{\xi}_t)}{\partial t} + \frac{\partial}{\partial x_j} [\hat{u}_j(\bar{\rho}\tilde{\xi}_t)] &= \frac{\partial\bar{p}}{\partial t} - \frac{\partial}{\partial x_j} \left[(\bar{q}_j + \bar{q}_j^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) \right] \\ &\quad + \frac{\partial}{\partial x_j} [\bar{\tau}_{ij}\hat{u}_i + \bar{\tau}_{ij}^t\hat{u}_i] + \frac{T_{\text{ref}}}{\hat{T}^2} (\bar{q}_j + \bar{q}_j^t) \left(\frac{\partial\hat{T}}{\partial x_j} \right) \\ &\quad - \frac{T_{\text{ref}}}{\hat{T}} \left[\bar{\tau}_{ij} \frac{\partial\hat{u}_i}{\partial x_j} + \bar{\tau}_{ij}^t \frac{\partial\hat{u}_i}{\partial x_j} \right] \end{aligned} \quad (3.27)$$

For statistically steady flows, the transport equation of $\tilde{\xi}_t$ simply reads

$$\frac{\partial}{\partial x_j} \left[\underbrace{\hat{u}_j(\bar{\rho}\tilde{\xi}_t) + (\bar{q}_j + \bar{q}_j^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) - \bar{\tau}_{ij}\hat{u}_i - \bar{\tau}_{ij}^t\hat{u}_i}_{\bar{F}_{\xi,j}} \right] = -T_{\text{ref}} \left(\frac{\Phi}{\hat{T}} + \frac{\Phi_{\Theta}}{\hat{T}^2} \right), \quad (3.28)$$

where $\bar{F}_{\xi,j}$ is the j^{th} component of the vector field formed by the total flux of mean-flow total exergy, and

$$\Phi = \bar{\tau}_{ij} \frac{\partial\hat{u}_i}{\partial x_j} + \bar{\tau}_{ij}^t \frac{\partial\hat{u}_i}{\partial x_j}, \quad (3.29)$$

$$\Phi_{\Theta} = -(\bar{q}_j + \bar{q}_j^t) \left(\frac{\partial\hat{T}}{\partial x_j} \right), \quad (3.30)$$

are positive terms related to entropy generation by viscous dissipation and by heat transfer, respectively. Actually it appears that the expressions of Φ and Φ_Θ correspond exactly to models mentioned in the literature for the calculation of mean entropy production [2]. Note that the terms in the RHS of Eq. (3.29) may be identified as the mean-flow viscous dissipation and the production of turbulence kinetic energy, respectively. The latter may not be considered as entropy generation itself, but this kinetic energy will ultimately be dissipated, hence generating entropy.

Applying Eq. (3.28) to a mean-flow total exergy transport tube (see Fig. 3.3 with $\bar{\mathbf{F}} = \bar{\mathbf{F}}_\xi$) leads to the following equation

$$\iint_{A_{in}} \bar{F}_{\xi,j} n_j ds + \iint_{A_{out}} \bar{F}_{\xi,j} n_j ds = -T_{ref} \iiint_{\Omega} \left(\frac{\Phi}{\hat{T}} + \frac{\Phi_\Theta}{\hat{T}^2} \right) dV, \quad (3.31)$$

where $\mathbf{n}$ is the outward directed normal to the control volume, Ω . Note that there is no average transport of mean-flow total exergy over the tube surface, since by construction $\bar{F}_{\xi,j} n_j = 0$ there. Hence the difference between the flux of mean-flow total exergy on the tube cross-sections A_{in} and A_{out} is only due to the integral effects of sinks of total exergy in the tube. Compared to momentum and kinetic energy transport tubes, the distinctive feature of total exergy transport tubes is that there is no positive source term. In other words, the flux of mean-flow total exergy going out of a transport tube may not exceed the flux that enters the tube. This is a real advantage because it allows unambiguous interpretation of an exergy transport tube compared to momentum and mean-flow kinetic energy tubes whose interpretation may be difficult due to the fact that the pressure gradient acts as a source term [62].

3.4 Visualization of transport processes within the ejector

3.4.1 Transport of mass, momentum and mean-flow kinetic energy

Figure 3.4 illustrates the visualization of the transport of mass, momentum and energy within the ejector. For all the figures presented in this section, the ejector is depicted with a twofold stretching in the vertical direction for improved readability. For 2D flows, a transport tube may be fully pictured by two transport lines: the tube surface simply consisting in the extrusion of these lines. For the visualization of a transport line, the corresponding vector field is first computed in each cell

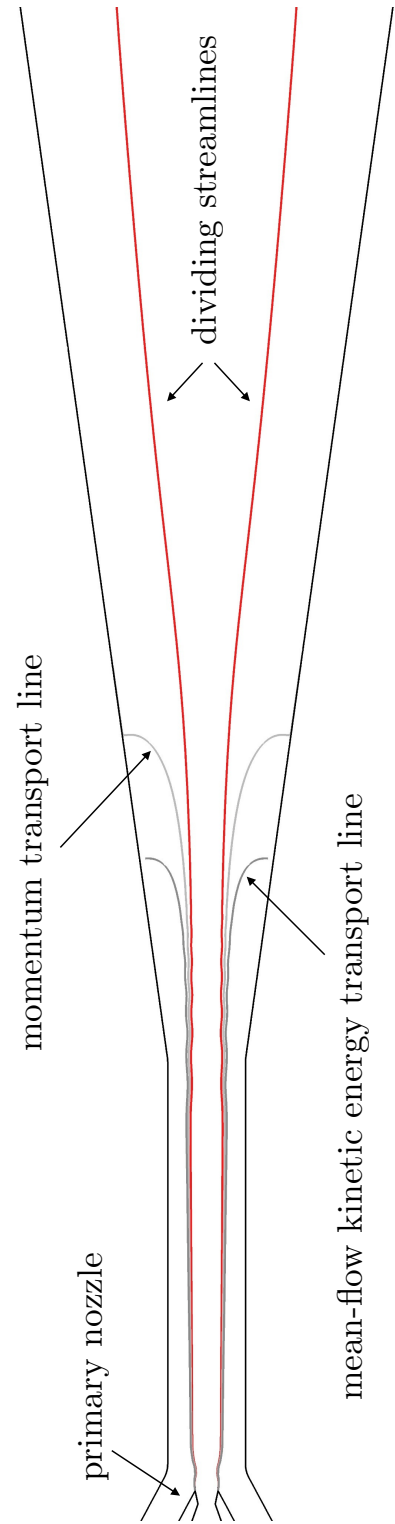


Figure 3.4: Illustration of the visualization of the transport of mass (in red), momentum (in light grey) and mean-flow kinetic energy (in dark grey) within the ejector.

of the computational domain and the line is then build up with an integrator using a Runge-Kutta method of the fourth order.

As a first step, for each operating condition, the mass flux based classical stream tube surrounding the primary stream is shown in order to evaluate which part of the ejector is occupied by the primary stream and which part is occupied by the secondary stream. The boundary of this stream tube, named the *dividing* surface, may be represented by the *dividing* streamline, defined as the streamline passing through the point located in the nozzle exit plane at $y = H_1 + \varepsilon$ with $\varepsilon \ll H_1$, i.e. just above the tip of the primary nozzle, in order to avoid problems that would be caused by a potential separation of the flow within the primary nozzle. Since, by construction, there is no average mass flux passing through the dividing surface, the dividing streamline may be interpreted as a virtual separation between the primary and the secondary streams all along the ejector.

Superimposed on these streamlines, a momentum and a mean-flow kinetic energy transport tube are illustrated. For the flow within the ejector, a question that may arise is 'where is the momentum and the kinetic energy of the primary stream transported to?'. Looking for an answer to this question, the corresponding transport tube surrounding the primary stream in the nozzle exit plane should be highlighted, since transport processes between the primary and the secondary streams start there. Hence, the transport line originating from the point located in the nozzle exit plane at $y = H_1 + \varepsilon$ is considered, both for the momentum and the kinetic energy, and is depicted on both halves of the ejector.

As expected, the transport lines are directed from the dividing streamline towards the wall as the momentum and the kinetic energy are transported from the primary stream to the secondary stream. The slope of the line is the image of the relative importance of viscous and turbulent fluxes (nearly aligned with $\hat{e}_y$) compared to the advective flux (nearly aligned with $\hat{e}_x$).

Concerning the momentum, note that a momentum flux is absorbed at the wall due to the wall shear stress. Actually, the value of the local flux of horizontal momentum crossing the wall is equal to the horizontal component of the wall shear stress. Hence, the position at which the momentum transport line reaches the wall may be interpreted as the location from where all the remaining momentum within the ejector originates from the primary stream. As a consequence, at that point, all the momentum contained in the secondary stream at the nozzle exit position has been removed from the ejector. This momentum is either

absorbed at the wall or removed from the secondary stream by the pressure gradient.

The situation is the same for the mean-flow kinetic energy. Indeed, the position at which the kinetic energy transport line reaches the wall may be interpreted as the location from where all the remaining mean-flow kinetic energy within the ejector originates from the primary stream. As a consequence, at that point, all the mean-flow kinetic energy contained in the secondary stream at the nozzle exit position has been removed from the ejector. This kinetic energy is either dissipated or removed from the secondary stream by the work done by the pressure gradient. At the wall, $\bar{F}_{K,j} = 0$, since the magnitude of the velocity is zero. Hence there is no kinetic energy flux crossing the wall as no work is being done on the wall.

Figure 3.5 shows the momentum and mean-flow kinetic energy transport tubes for five different operating conditions. The considered cases correspond to cases A, C, D, E and G of the characteristic curve obtained with $p_{01}/p_{02} = 3.61$ (see Fig. 2.9, p.33). The first two cases lie in the on-design region, the third one corresponds to a back-pressure that is close to the critical back-pressure, and the ejector operates in off-design conditions in the last two illustrations.

Two remarks may be made. Firstly, it clearly appears that, depending on the back-pressure, the transport of momentum and kinetic energy from the primary stream towards the secondary stream takes place in a different location within the ejector. The global trend is that when the back-pressure is higher, the transport lines start to deviate from the dividing streamline at a location closer to the nozzle exit position. This trend is logically related to the flow structure within the ejector, in particular the position of the second shock train that moves towards the nozzle exit position as the back-pressure increases [28, 77], as evidenced by the slight oscillations of the dividing streamlines. The fact that the kinetic energy transport line does not reach the wall in Fig. 3.5 for cases A and G reveals the presence of a flow detachment and an associated zone of recirculation, at the entrance of the diffuser for case A and at the entrance of the mixing duct for case G, as shown in Fig. 3.6. A closer inspection (not shown here) reveals that the stagnation locus of mean-flow kinetic energy transport lines actually consists in a line within the zone of recirculation and thus, does not correspond to specific points of attraction. Secondly, the slope of the transport lines seems to be more important for these extreme cases, and more gentle for a value of the back-pressure that is close to the critical back-pressure (case D). In other words, the transfers seem to be smoother for the latter case. These two observations will be analyzed further.

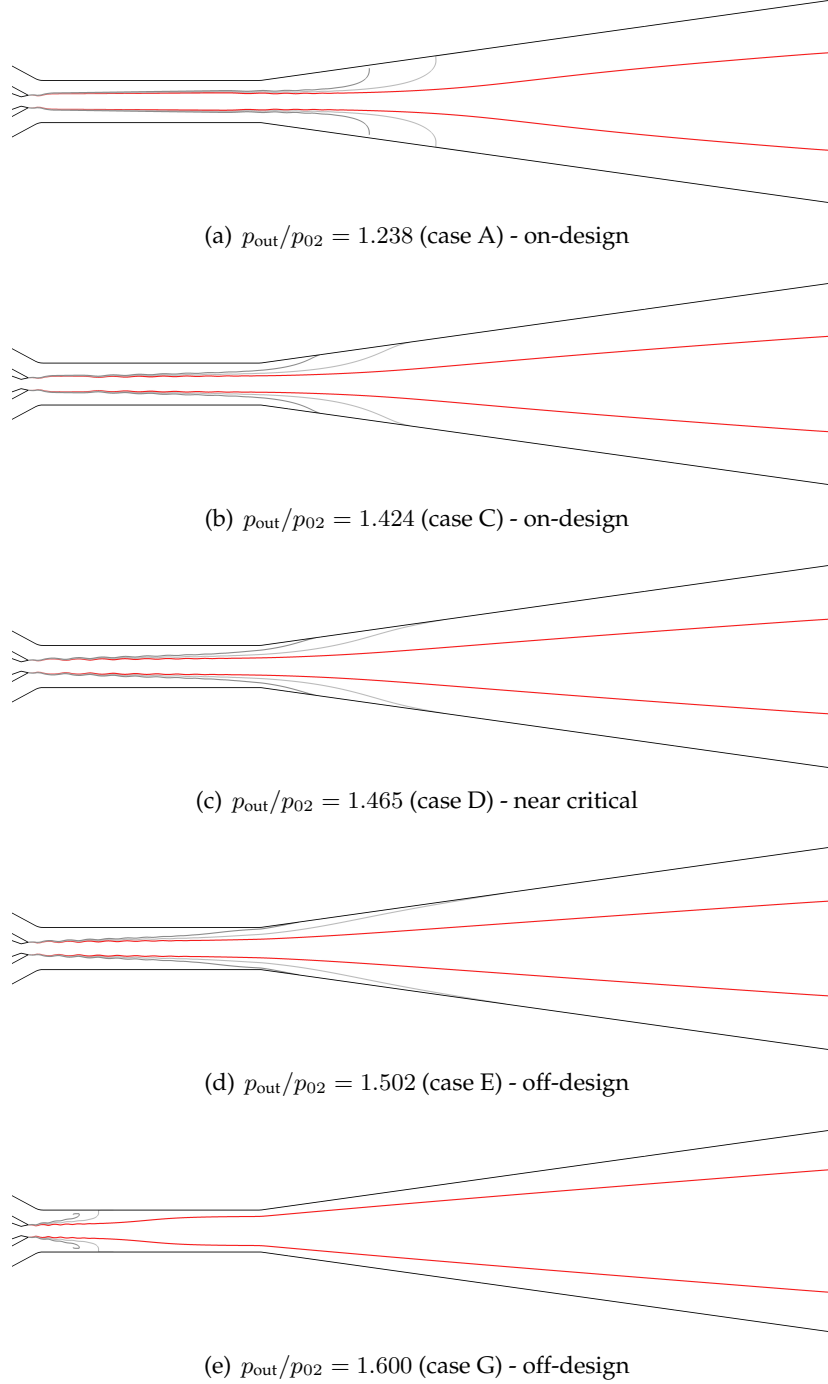


Figure 3.5: Stream tubes (in red), momentum (in light grey) and mean-flow kinetic energy (in dark grey) tubes for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

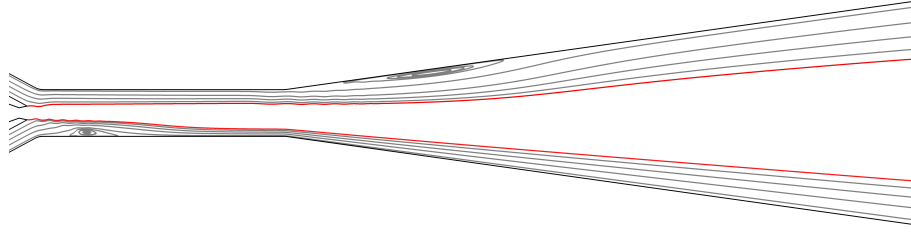


Figure 3.6: Streamlines in the secondary stream for case A (top half) and for case G (bottom half). The corresponding dividing streamline is shown in red.

3.4.2 Transport of mean-flow total exergy

The transport of exergy within the ejector is now investigated through the visualization of mean-flow total exergy transport tubes. Note that, unlike the advective fluxes of momentum and kinetic energy that are supposed to decrease near the outlet of the ejector as both streams are decelerated, the advective flux of total exergy that leaves the diffuser is obviously still important. Hence, by constructing backwards the exergy tube surrounding the secondary stream at the outlet, it is possible to answer the fundamental question “where does the exergy contained in the secondary stream at the exit of the ejector come from?”. This tube will be depicted in the top half of the ejector as illustrated in Fig. 3.7.

It should be noted that the reference state (i.e. T_{ref} and p_{ref}) will have an influence on the geometry of the tube. In order to observe the provenance of the gain in exergy of the secondary stream within the ejector, the reservoir conditions of the secondary stream are taken as the reference state, namely $T_{\text{ref}} = T_{02}$ and $p_{\text{ref}} = p_{02}$. For refrigeration applications, it would correspond to the temperature at the evaporator and the corresponding saturation pressure.

The mean-flow total exergy transport tube constructed in this way reveals the transport of the exergy that is ultimately useful for the secondary stream. It should be noted that there is a transfer of exergy from the primary stream to the secondary stream upstream of the location where the transport tube enters the secondary stream. But this exergy does not flow out of the ejector. In fact, it is dissipated within the secondary stream and used to overcome the head losses. Furthermore, the transport tube also shows the portion of the exergy contained in the primary stream that is not transferred to the secondary stream. This exergy either remains in the primary stream at the outlet, or is dissipated within the ejector.

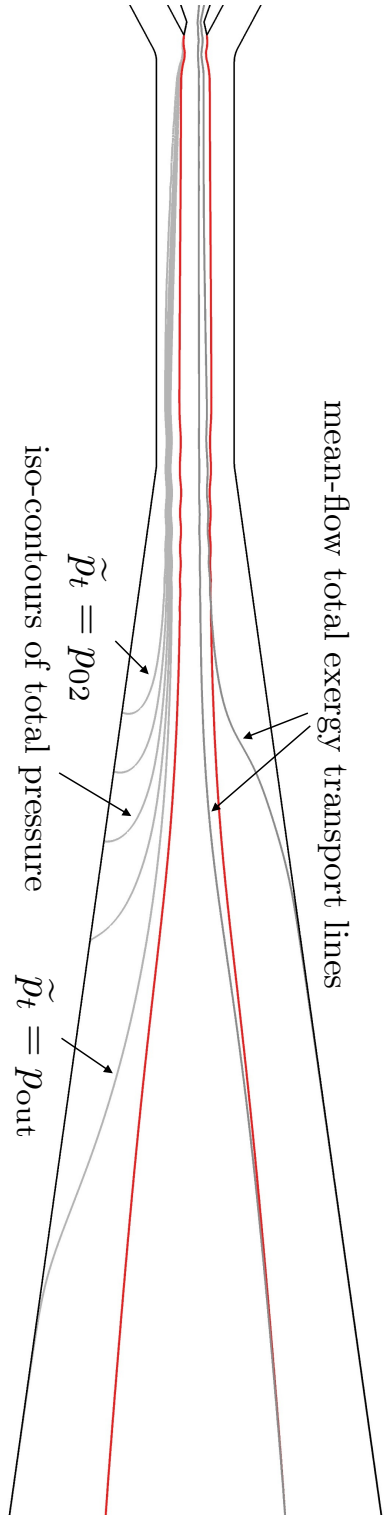


Figure 3.7: Illustration of the visualization of transport of mean-flow total exergy (in dark grey - top half) and iso-contours of stagnation pressure (in light grey - bottom half). The dividing streamlines are shown in red.

In addition, in order to illustrate the increase in the stagnation pressure of the secondary stream, iso-contours of mean-flow stagnation pressure, $\tilde{p}_t$, are shown in the bottom half of the ejector. The latter is defined as

$$\tilde{p}_t = \bar{p} \left(1 + \frac{(\gamma - 1)}{2} \tilde{M}^2 \right)^{\frac{\gamma}{\gamma - 1}}, \quad (3.32)$$

where $\tilde{M}$ represents the mean-flow Mach number

$$\tilde{M} = \frac{|\hat{\mathbf{u}}|}{\sqrt{\gamma R \hat{T}}}. \quad (3.33)$$

Five iso-contours of the mean-flow stagnation pressure are represented here, equally distributed from $\tilde{p}_t = p_{02}$ to $\tilde{p}_t = p_{\text{out}}$. This last iso-contour should logically reach the wall at the outlet of the ejector as the static pressure is p_{out} and the velocity is zero at that point. This methodology is applied to the ejector for the same operating conditions as those used in Fig. 3.5.

The results are shown in Fig. 3.8. The mean-flow exergy transport tubes and iso-contours of stagnation pressure are consistent with the observations made above, both with regard to the location of the transfers and the slope of the transport lines. In particular, for some cases, the transfer of mean-flow total exergy may be not completely finished at the entrance of the diffuser. This is especially true for case A: iso-contours of stagnation pressure then indicate that a certain fraction of the secondary stream enters the diffuser with a stagnation pressure lower than its reservoir condition, since the iso-contour $\tilde{p}_t = p_{02}$ reaches the wall in the diffuser. In other words, at the entry of the diffuser, the compression has not even begun for this fraction of the secondary stream. Hence, for low values of the back-pressure, a substantial part of the exergy transfer occurs in the diffuser.

Note that the sensitivity of the integration of the exergy transport line emanating from the wall at the outlet of the ejector is assessed in Fig. 3.9 for case A and case G: the five lines seeded in the five first cells near the wall are essentially superimposed.

In order to provide a more detailed understanding of the interpretation of the exergy transport tube shown above, it could be interesting to focus on the form of the tube at the nozzle exit position, for different values of the back-pressure. Hence the exergy footprint of the secondary stream within the primary stream is defined as the projected surface of the exergy tube issued from the secondary stream outlet at

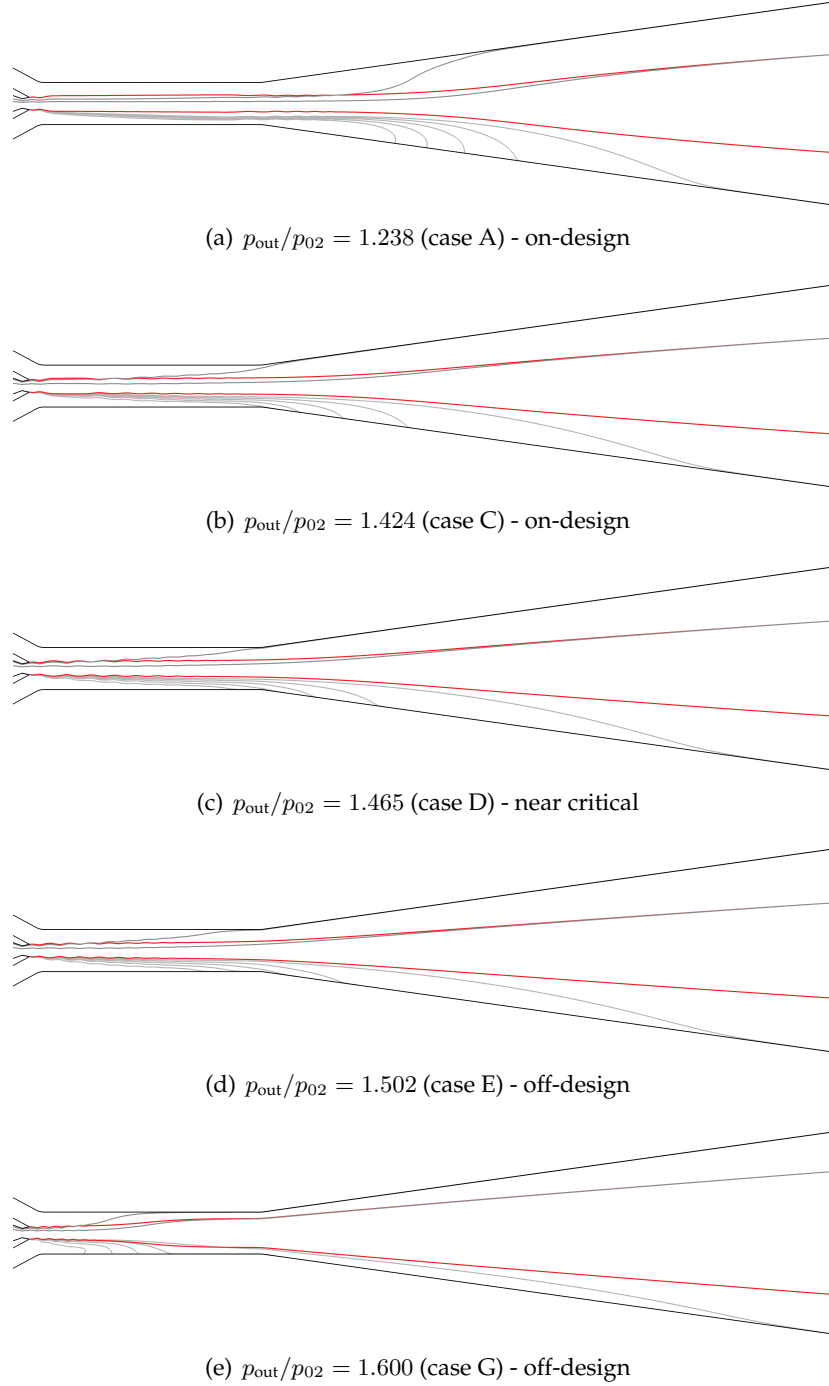


Figure 3.8: Mean-flow total exergy tubes (in dark grey - top half) and iso-contours of stagnation pressure (in light grey - bottom half) for different values of the back-pressure – $p_{01}/p_{02} = 3.61$. The dividing streamlines are shown in red.

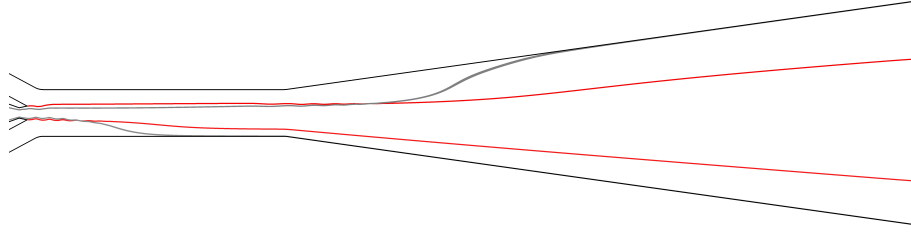


Figure 3.9: Sensitivity analysis of the integration of the exergy transport line emanating from the wall at the outlet for case A (top half) and for case G (bottom half). The corresponding dividing streamline is shown in red.

the nozzle exit position. The concept is illustrated in Fig. 3.10(a) for a particular value of the back-pressure. As the ejector considered in this study is rectangular and symmetrical, the pattern is the same for the bottom half of the ejector and is thus not shown here. The interpretation of this figure is the following: for a given operating point, the exergy that passes through the exergy footprint corresponds to the maximum amount of exergy that may be contained in the secondary stream at the outlet. Obviously, the exergy contained in the secondary stream at the exit of the ejector is lower due to the integral effects of sinks of total exergy in the tube. With regard to the exergy that passes *above* the exergy footprint, it does not go out of the ejector since there is no average flux of mean-flow total exergy through the tube surface. Hence, this flux of exergy corresponds to a loss of exergy and is dissipated within the ejector, either in the primary or in the secondary stream. Finally, the exergy flowing out of the primary nozzle *below* the exergy footprint will be partially dissipated within the primary stream, and the remaining exergy at the outlet of the diffuser will be contained in the primary stream.

The exergy footprint is shown in Fig. 3.10(b) for different values of the back-pressure. In view of the above, it seems logical that the region below the exergy footprint increases with the back-pressure, as the constant primary mass flow rate flows out of the ejector with more exergy when the back-pressure is higher. By contrast, the upper edge of the exergy footprint is the highest for case D, suggesting that the exergy dissipated within the secondary stream or used for its entrainment is lower for these operating conditions. As a consequence, the exergy footprint is maximum in the vicinity of the critical back-pressure. It indicates that the gain in exergy of the secondary stream should reach a maximum close to this operating point, unless the exergy losses within

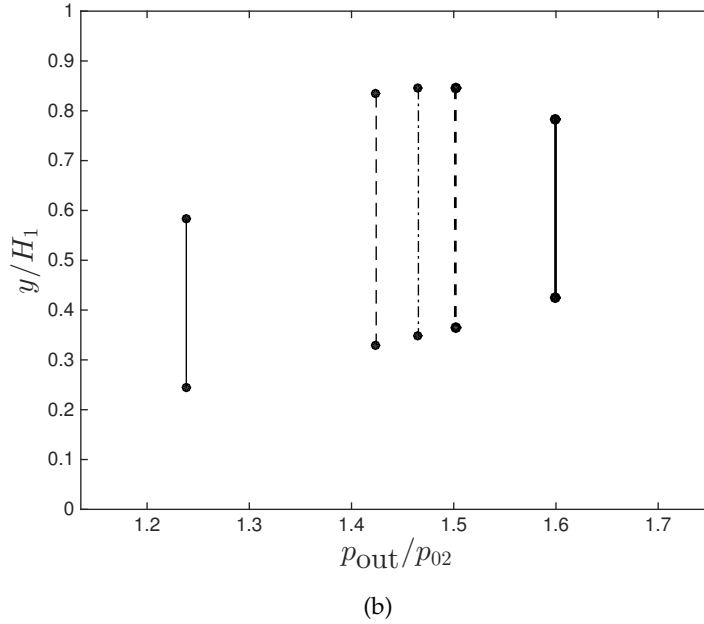
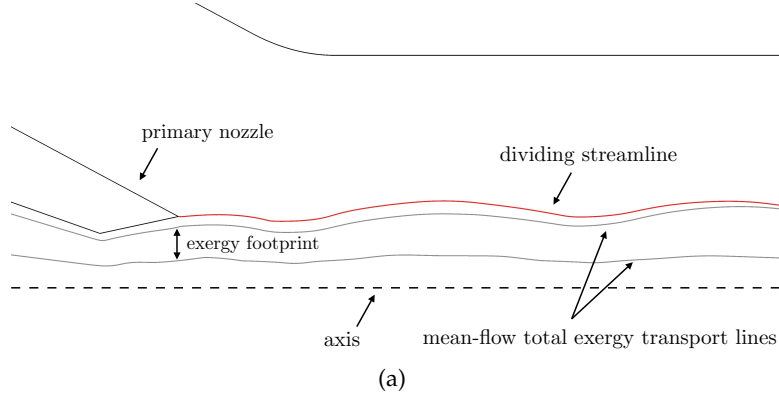


Figure 3.10: Exergy footprint of the secondary stream within the primary stream at the nozzle exit position: (a) illustration and (b) application to different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

the tube are significantly higher in this situation. This maximum could be expected since, at the critical back-pressure, the compression ratio has the highest possible value before the ejector enters the off-design regime, which leads to a decrease of the secondary mass flow rate.

Finally, in order to illustrate the impact of the sink terms, the profile of the cross-sectional flux of mean-flow total exergy in the tube is

shown in Fig. 3.11 (solid line) for case D. This flux is normalized by the flux entering the tube at the nozzle exit position, i.e. by the exergy flowing through the exergy footprint. The integral effects of sink terms (see Eq. (3.31)) are also normalized and shown in the figure. The position where the exergy tube enters the secondary stream - i.e. the upper transport line crosses the dividing streamline - is highlighted. In light of Eq. (3.31), it should be clear that the cross-sectional flux of mean-flow total exergy in the tube can not increase with x . For the set of operating conditions under consideration, this flux has approximately decreased by a factor two between the nozzle exit position and the exit of the ejector. As may be seen in Fig. 3.11, this is mainly due to the exergy losses related to viscous dissipation (dashed line), i.e. the first term of the RHS in Eq. (3.31).

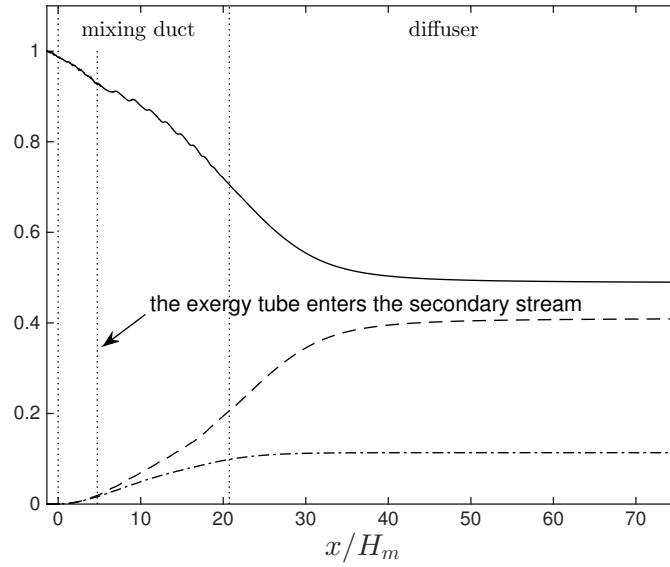


Figure 3.11: Normalized exergy flux and losses in the mean-flow total exergy transport tube for $p_{\text{out}}/p_{02} = 1.465$ (case D). Solid line = cross-sectional flux of mean-flow total exergy, dashed line = cumulative exergy losses due to viscous dissipation, dashed-dotted line = cumulative exergy losses due to heat transfer.

This kind of analysis actually opens a new path for the investigation of transfers within supersonic ejectors and could bring new perspective: geometry optimization, understanding the effect of local geometrical/flow features on the overall operation, defining new performance criteria,... However, as mentioned in the introduction of this chapter,

the objective of the present study is to address one of the main issues regarding supersonic ejectors, namely the characterization of the transfers between both streams all along the ejector. This requires the use of classical stream tubes as a way to identify each stream within the ejector and is performed in the next section.

3.5 Transfers between the primary and the secondary streams

3.5.1 Local transfers

Basically, the primary and the secondary streams exchange on average three fundamental physical quantities: momentum (mainly in the horizontal direction), kinetic energy and heat. These transfers essentially occur through the shear layer. In the present investigation, a control volume analysis focusing on the secondary stream is performed, as illustrated in Fig. 3.12. The transfers between the primary and the secondary streams are thus quantified through the dividing surface.

By noting $\mathbf{n}$ the normal vector to the dividing surface and pointing *towards* the secondary stream (see Fig. 3.12), the local flux of momentum that crosses the dividing surface per unit area is simply given by $\bar{F}_{m,j}n_j$ and is positive if the momentum is transported from the primary towards the secondary stream. Naturally, by construction of the dividing surface, the contribution of the advective flux is null, and only the viscous and turbulent fluxes are taken into account when calculating $\bar{F}_{m,j}n_j$.

For the quantification of the transfers within the ejector, new indicators are proposed. The dimensionless local flux of momentum across the dividing surface, $\bar{f}_m$, is defined as follows

$$\bar{f}_m = \frac{A_m}{\dot{m}_1 \sqrt{2\xi_{01}}} \bar{F}_{m,j}n_j, \quad (3.34)$$

where ξ_{01} is the exergy of the primary stream at reservoir conditions. Therefore, the value given by $\sqrt{2\xi_{01}}$ represents the maximum speed that the primary stream could reach by converting all its exergy into kinetic energy. The corresponding quantities for the local fluxes of mean-flow kinetic energy and mean-flow total exergy are, respectively,

$$\bar{f}_K = \frac{A_m}{\dot{m}_1 \xi_{01}} \bar{F}_{K,j}n_j, \quad (3.35)$$

and

$$\bar{f}_\xi = \frac{A_m}{\dot{m}_1 \xi_{01}} \bar{F}_{\xi,j}n_j. \quad (3.36)$$

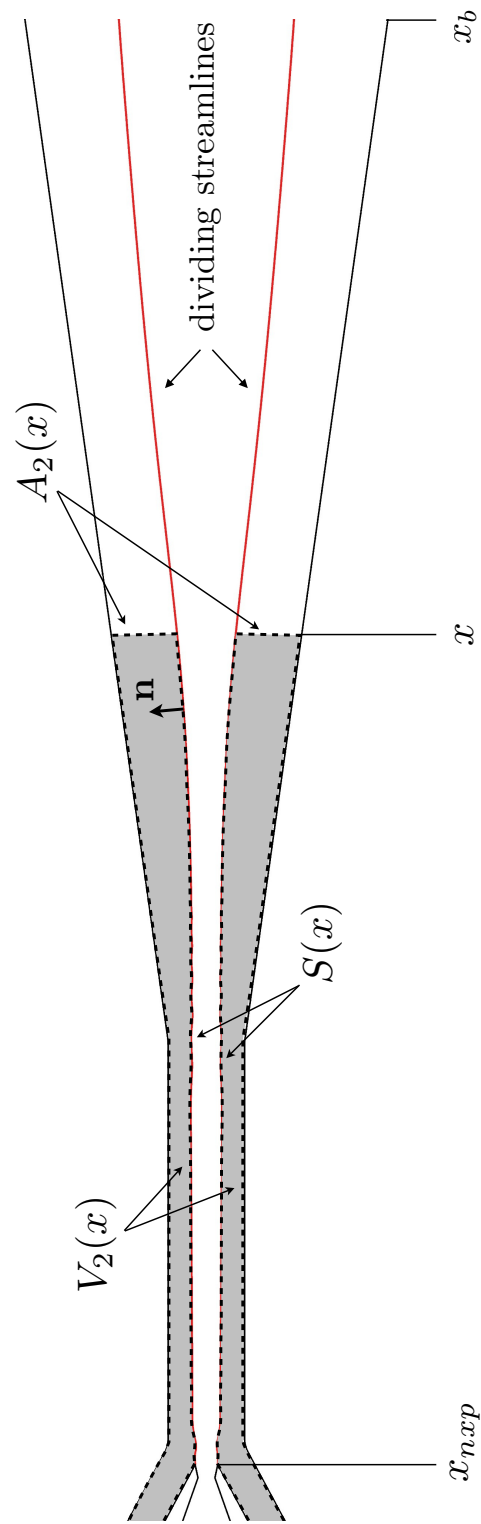


Figure 3.12: Control volume analysis.

Regarding the heat transfer between the primary and the secondary stream, the focus should be put on its exergy content. Hence the flux of exergy associated with the mean heat flux and the turbulent transport of heat is defined as

$$\overline{F}_{q,j} = (\overline{q}_j + \overline{q}_j^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right), \quad (3.37)$$

such that the flux of exergy related to the exergy content of heat crossing the dividing surface per unit area is given by $\overline{F}_{q,j}n_j$. In accordance with the above, the dimensionless version of $\overline{F}_{q,j}n_j$ is given by

$$\overline{f}_q = \frac{A_m}{\dot{m}_1 \xi_{01}} \overline{F}_{q,j}n_j. \quad (3.38)$$

It should be noted that, based on Eq. (3.28), it follows that

$$\overline{f}_\xi = \overline{f}_K + \overline{f}_q. \quad (3.39)$$

The profile of the local dimensionless transfers across the dividing surface are shown in Figs. 3.13 and 3.14 for different operating conditions. A first remark to be made on these figures is that, for a given operating point, transfers of momentum, mean-flow kinetic energy and exergy have essentially the same spatial pattern. Obviously this pattern is related to the complex flow structure within the ejector. By contrast, the profile of the flux of exergy related to the exergy content of heat is different. This is due to the temperature-correction factor (see Eq. (3.37)) whose magnitude is more important at low temperature, thus at high speed. This is why this transfer is relatively more important at the exit of the primary nozzle compared to the other fluxes. Note that the magnitude of $\overline{f}_q$ is relatively low yet non negligible compared to the flux of mean-flow kinetic energy, $\overline{f}_K$. Hence the transfer of exergy between the primary and the secondary streams is mainly due to a transfer of kinetic energy.

Moreover, one may directly observe that when the ejector is operating in the on-design regime, the peak of the local transfer of momentum and mean-flow kinetic energy is located in the diffuser. This means that the flow is not uniform at the exit of the mixing duct. In these cases, the mixing between both streams is thus not yet completed when they enter the diffuser. On the other hand, as it could be seen with transport tubes, it appears that, in off-design operations, transfers take place almost entirely in the mixing duct. Finally, for a value of p_{out} close to the critical back-pressure (case D), it seems that the peak of the local transfers is the lowest, hence transfers are more evenly distributed all along the mixing duct.

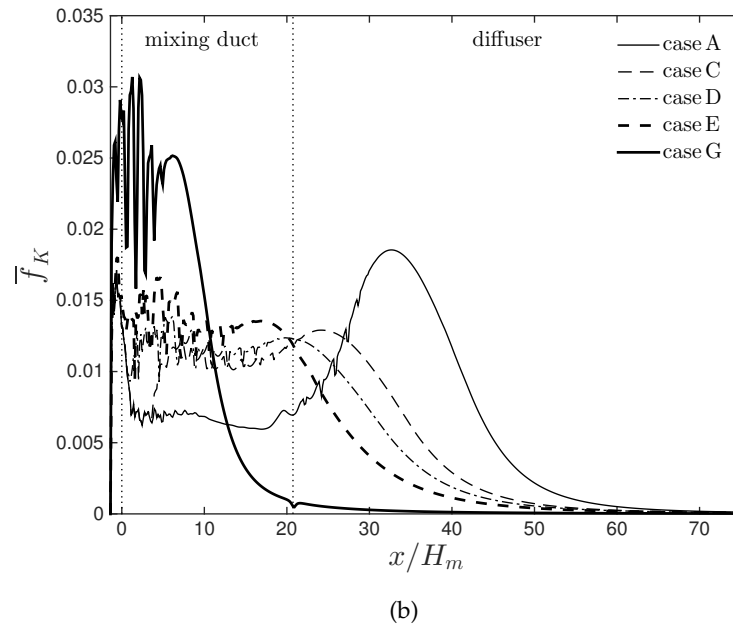
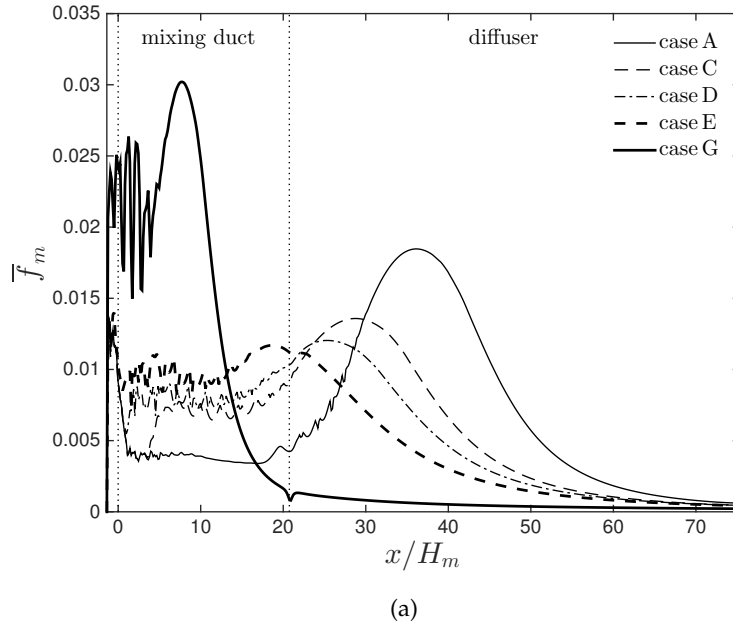


Figure 3.13: Dimensionless local flux of (a) momentum and (b) mean-flow kinetic energy crossing the dividing surface for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

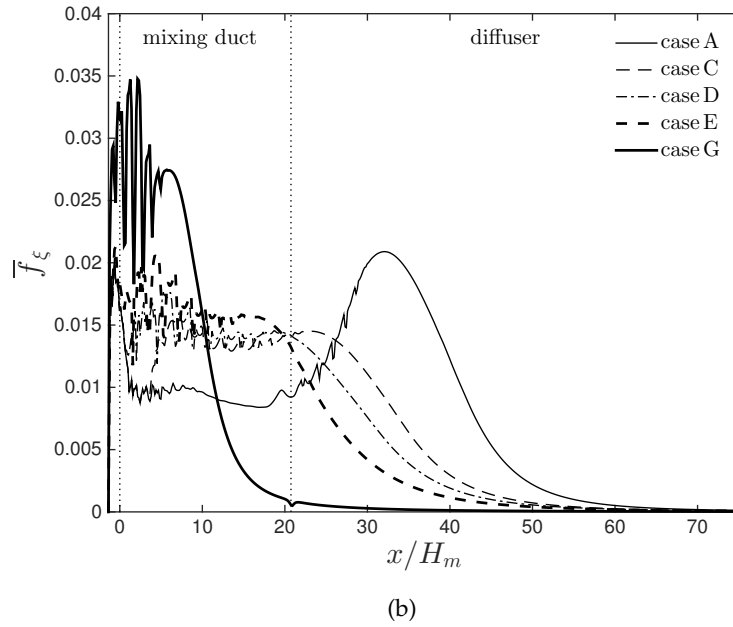
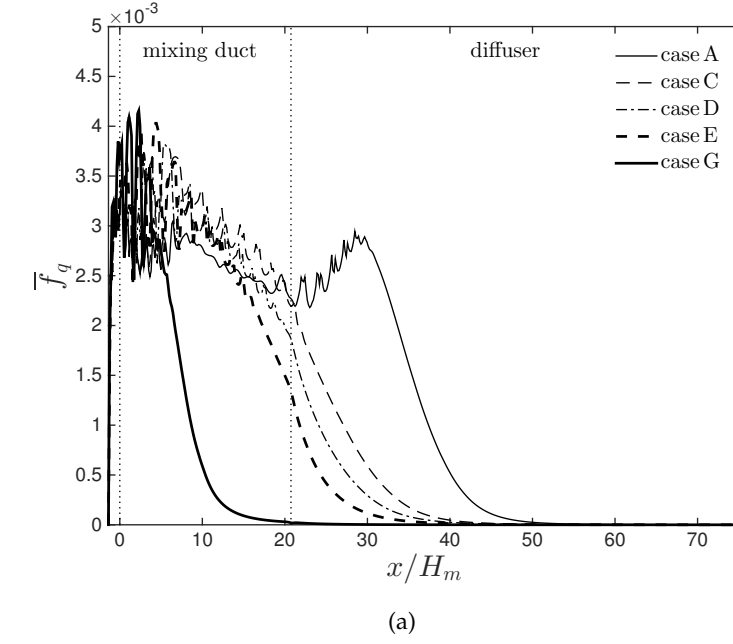


Figure 3.14: Dimensionless local flux of (a) heat (exergy content) and (b) mean-flow total exergy crossing the dividing surface for different values of the back-pressure $-p_{01}/p_{02} = 3.61$.

3.5.2 Cumulative transfers

The dimensionless cumulative transfer of momentum between the primary and the secondary stream, $I_m(x)$, is simply defined as

$$I_m(x) = \frac{1}{A_m} \iint_{S(x)} \bar{f}_m ds, \quad (3.40)$$

where $S(x)$ represents the portion of the dividing surface between the nozzle exit position, x_{np} , and x , taking into account both halves of the ejector, as illustrated in Fig. 3.12 (p. 59). The corresponding quantity for the mean-flow kinetic energy may be expressed as

$$I_K(x) = \frac{1}{A_m} \iint_{S(x)} \bar{f}_K ds, \quad (3.41)$$

and for the exergy content of heat

$$I_q(x) = \frac{1}{A_m} \iint_{S(x)} \bar{f}_q ds. \quad (3.42)$$

The dimensionless cumulative transfer of mean-flow total exergy between the primary and the secondary stream is simply given by

$$I_\xi(x) = \frac{1}{A_m} \iint_{S(x)} \bar{f}_\xi ds = I_K(x) + I_q(x). \quad (3.43)$$

The profiles of $I_m(x)$, $I_K(x)$, $I_q(x)$ and $I_\xi(x)$ are shown in Figs. 3.15 and 3.16 for different operating conditions. As mentioned above, for on-design operations, it clearly appears that transfers may be not completed at the entry of the diffuser. Furthermore, apart from case G, the value of $I_m(x_b)$ is very close to the value of $I_K(x_b)$ for a given operating point, although both profiles are different.

Finally, as may be observed in Figs. 3.15 and 3.16, the global transfers increase when the back-pressure decreases. It should be noted that the value of $I_\xi(x_b)$ may exceed 50% in cases A and C, meaning that more than half of the exergy available in the primary stream is transferred to the secondary stream. It should be emphasized that the maximization of $I_\xi(x_b)$ does not imply the maximization of the net gain in exergy of the secondary stream as the exergy losses occurring within the secondary stream vary with the operating conditions, as shown in the next section.

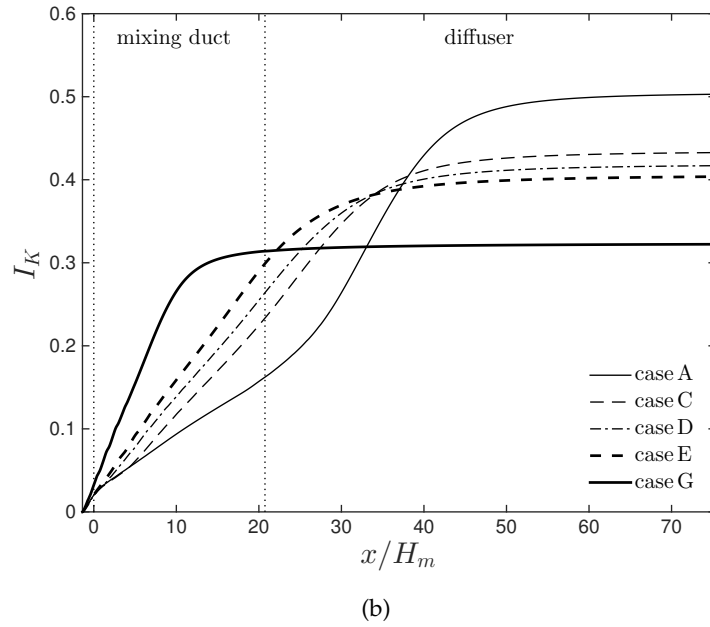
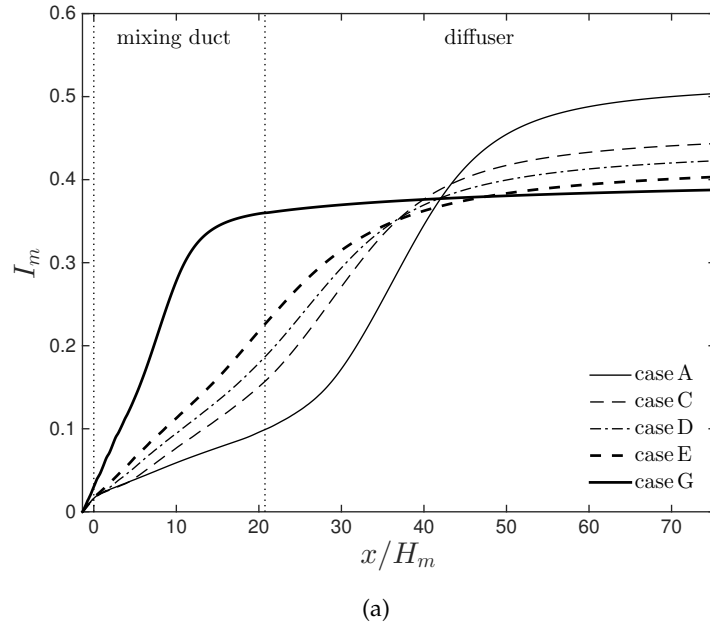
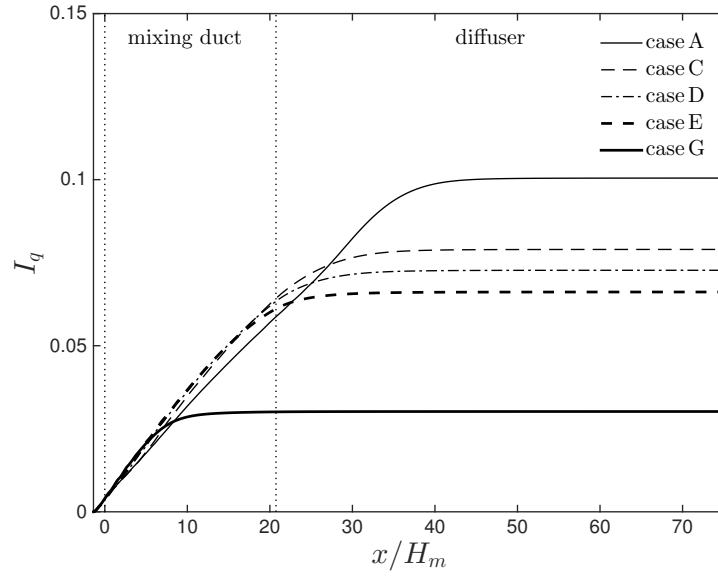
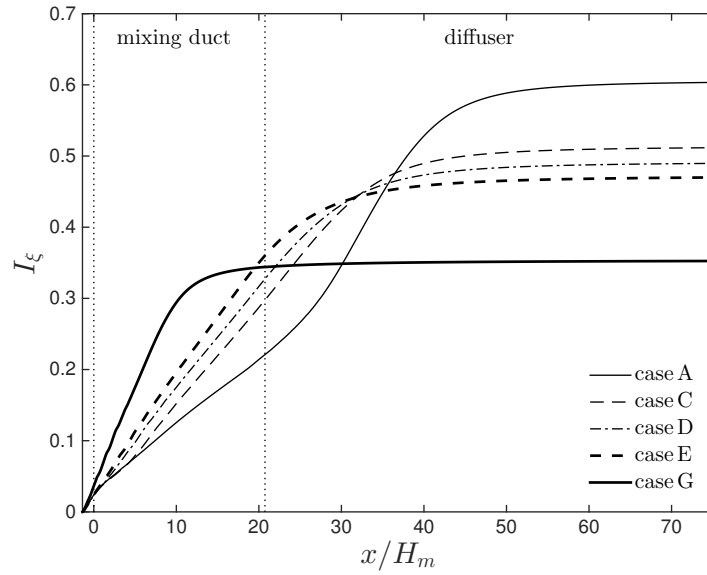


Figure 3.15: Dimensionless cumulative flux of (a) momentum and (b) mean-flow kinetic energy crossing the dividing surface for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.



(a)



(b)

Figure 3.16: Dimensionless cumulative flux of (a) heat (exergetic content) and (b) mean-flow total exergy crossing the dividing surface for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

3.6 Exergy losses within the secondary stream

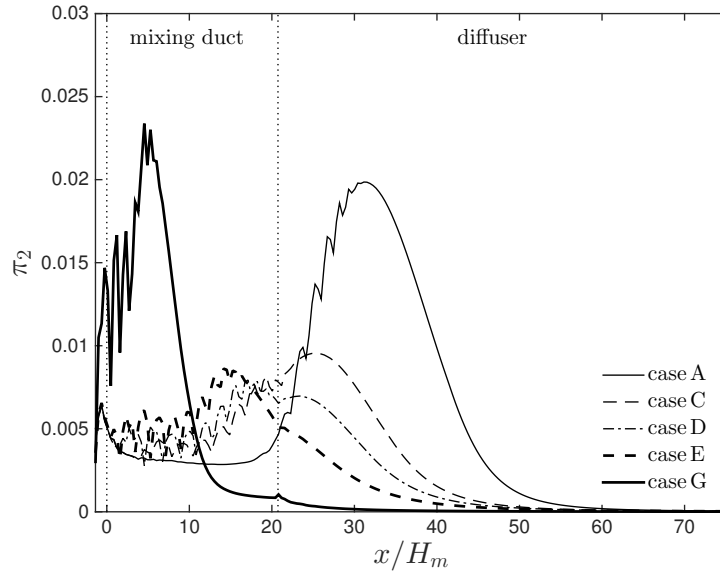
As mentioned above, Eq. (3.28) implies that the mean-flow total exergy may be dissipated by phenomena related to viscous dissipation (through the term $T_{\text{ref}} \Phi / \hat{T}$) or related to heat transfer (through the term $T_{\text{ref}} \Phi_{\Theta} / \hat{T}^2$). In order to assess the spatial distribution of these sink terms, their integrals over the secondary stream cross-section, $A_2(x)$, are first computed in a dimensionless form, as follows

$$\pi_2(x) = \frac{H_m T_{\text{ref}}}{\dot{m}_1 \xi_{01}} \iint_{A_2(x)} \frac{\Phi}{\hat{T}} ds, \quad (3.44)$$

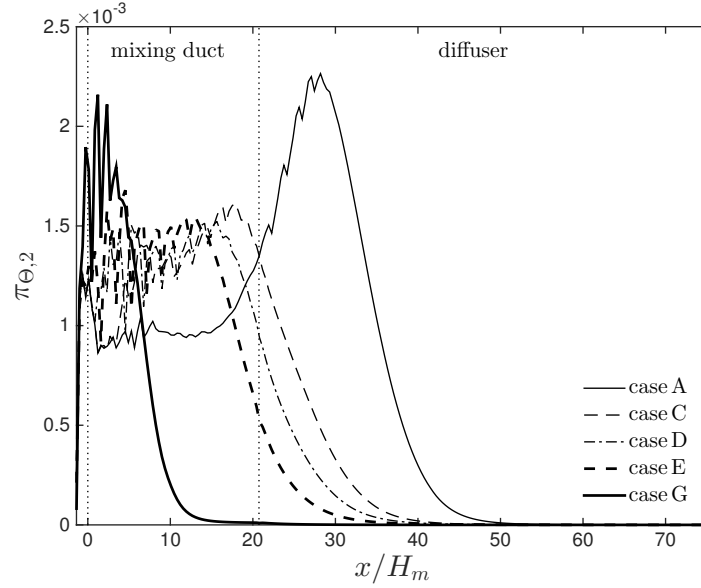
$$\pi_{\Theta,2}(x) = \frac{H_m T_{\text{ref}}}{\dot{m}_1 \xi_{01}} \iint_{A_2(x)} \frac{\Phi_{\Theta}}{\hat{T}^2} ds. \quad (3.45)$$

Note that $A_2(x)$ is calculated as the cross-section delimited by the dividing surface and through which the secondary stream virtually flows (see Fig. 3.12). The profiles of $\pi_2(x)$ and $\pi_{\Theta,2}(x)$ are shown in Fig. 3.17. The irreversibilities related to viscous dissipation are more important than exergy losses related to heat transfer. Moreover, these profiles are closely related to the profiles of the local transfers. This could have been expected as the viscous and turbulent fluxes of momentum or kinetic energy are intimately connected with the dissipation of total exergy. The large peak in the profile of $\pi_2(x)$ for case A and case G is related to the flow detachment and the corresponding zone of recirculation highlighted above (see Fig. 3.6, p.51).

With regard to the exergy losses related to shock waves, it should be noted that, for weak shocks, the entropy increase scales with the third power of the shock strength (see for example [73]), the latter being defined as the ratio of the static pressure increase to the initial static pressure. Hence the irreversibility related to shock waves will often be negligible within supersonic ejectors. To illustrate this point, it is possible to analytically derive the exergy losses through the oblique shock wave emanating from the lip of the primary nozzle by assuming that both streams are flowing isentropically between their respective reservoir and the primary nozzle exit plane (not shown here). By doing so, the theoretical flow properties upstream and downstream of the first oblique shock wave and hence the entropy increase can be found. When the ejector is operating at on-design conditions, the exergy losses through this weak shock, $T_{\text{ref}} \Delta s$, are found to represent less than 0.05% of the exergy at primary reservoir conditions, ξ_{01} . Hence, the irreversibility due to this shock wave is negligible. Moreover, when focusing on the secondary stream, exergy losses due to shock waves



(a)



(b)

Figure 3.17: Dimensionless local mean-flow total exergy destruction related to (a) viscous dissipation and (b) heat transfer in the secondary stream for different values of the back-pressure $-p_{01}/p_{02} = 3.61$.

are even more limited as the flow does not reach high supersonic Mach number, even when the ejector operates at on-design conditions.

In order to compare quantitatively these losses of mean-flow total exergy with the transfer of exergy between the primary and the secondary streams, the dimensionless cumulative exergy destruction related to viscous dissipation in the secondary stream is defined as

$$\Pi_2(x) = \frac{T_{\text{ref}}}{\dot{m}_1 \xi_{01}} \iiint_{V_2(x)} \frac{\Phi}{\hat{T}} dV, \quad (3.46)$$

where $V_2(x)$ is the volume occupied by the secondary stream between the secondary inlet and x , taking into account both halves of the ejector, as illustrated in Fig. 3.12 (p. 59). The dimensionless cumulative exergy destruction related to heat transfer in the secondary stream is simply given by

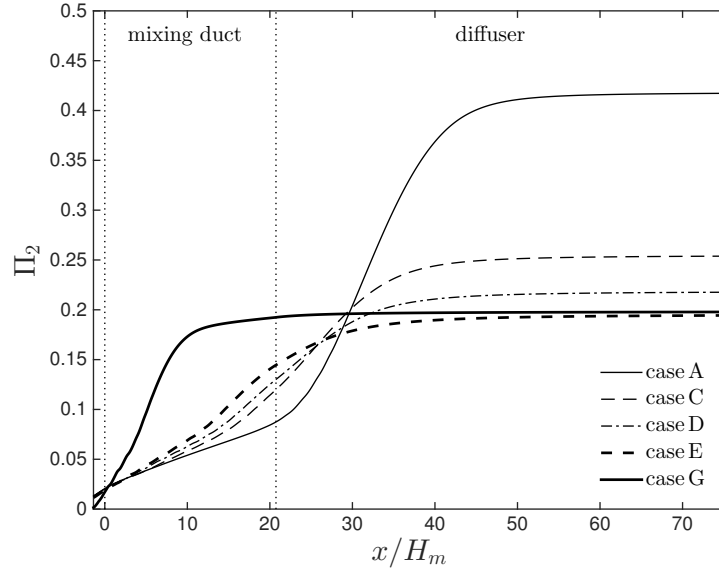
$$\Pi_{\Theta,2}(x) = \frac{T_{\text{ref}}}{\dot{m}_1 \xi_{01}} \iiint_{V_2(x)} \frac{\Phi_{\Theta}}{\hat{T}^2} dV. \quad (3.47)$$

The profiles of $\Pi_2(x)$ and $\Pi_{\Theta,2}(x)$ are shown in Fig. 3.18, starting from the nozzle exit position. Firstly, as expected for the flow within the ejector, the global exergy losses due to the mean heat flux and the turbulent transport of heat, corresponding to $\Pi_{\Theta,2}(x_b)$, are much lower than those related to mean-flow viscous dissipation and the production of turbulence kinetic energy, reflected by $\Pi_2(x_b)$. Furthermore, although $\Pi_{\Theta,2}(x_b)$ seems to increase with decreasing back-pressure, this is not the case for $\Pi_2(x_b)$. Indeed, $\Pi_2(x_b)$ has a minimum which corresponds approximately to case E. So far, the transfer of exergy between the primary and the secondary streams as well as the exergy losses have been investigated. The following analysis will focus on the global exergy balance for the secondary stream.

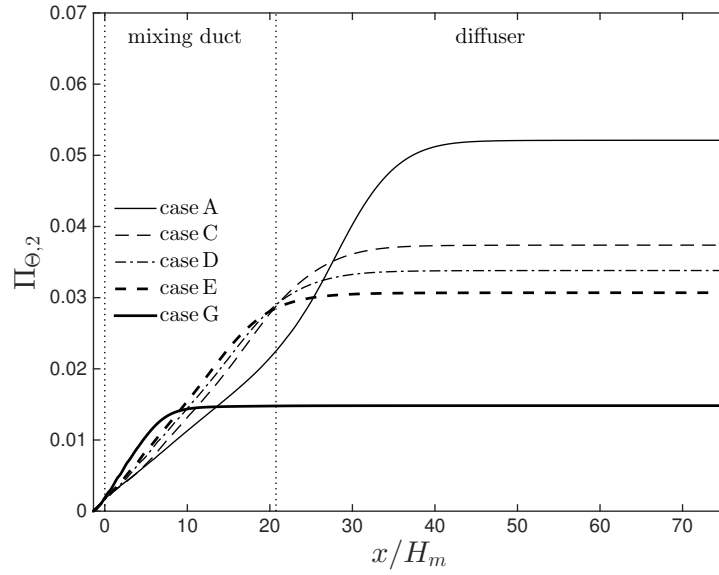
3.7 Global exergy balance for the secondary stream

The dimensionless net gain in exergy of the secondary stream at a certain position x is noted $\Delta_2(x)$ and is defined as

$$\Delta_2(x) = \frac{\dot{m}_2 (\xi_{t2}(x) - \xi_{02})}{\dot{m}_1 \xi_{01}}, \quad (3.48)$$



(a)



(b)

Figure 3.18: Dimensionless cumulative mean-flow total exergy destruction related to (a) viscous dissipation and (b) heat transfer in the secondary stream for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

where $\xi_{t_2}(x)$ is the mass flow-averaged total exergy flux associated with the secondary stream, i.e.

$$\xi_{t_2}(x) = \frac{\iint_{A_2(x)} \bar{\rho} \tilde{\xi}_t \hat{u}_j n_j ds}{\dot{m}_2} . \quad (3.49)$$

It should be noted that, considering the definition of the different dimensionless parameters given above, the following fundamental equation must be verified

$$\Delta_2(x) = I_K(x) + I_q(x) - \Pi_2(x) - \Pi_{\Theta,2}(x) . \quad (3.50)$$

Actually, it simply consists in Eq. (3.28) integrated in the control volume $V_2(x)$. The profile of $\Delta_2(x)$ is shown in Fig. 3.19 for several operating conditions. It clearly appears that, unless the ejector is run at a off-design regime, the net gain in exergy of the secondary stream is not completed at the entry of the diffuser. This fact confirms that a certain part of the transfers between both streams takes place in the diffuser.

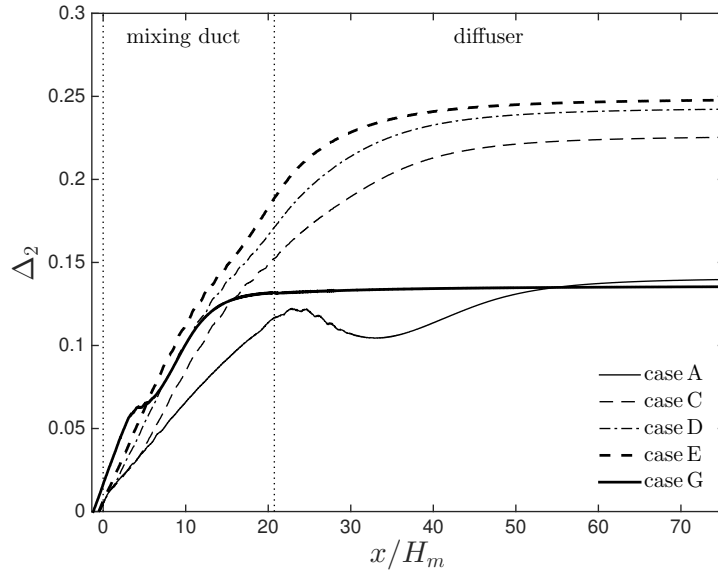


Figure 3.19: Gain in exergy of the secondary stream for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

Moreover, this exergy analysis reveals additional information about the quality of the mixing. Although the global exergy transfer, $I_\xi(x_b)$, is

maximum for case A (see Fig. 3.16(b)), it appears that the corresponding net gain in exergy is relatively low due to the large exergy losses related to viscous dissipation, $\Pi_2(x_b)$ (see Fig. 3.18(a)). As may be seen in Fig. 3.19, the net gain in exergy of the secondary stream is actually maximum for a value of the back-pressure slightly higher than the critical back-pressure (case E), although the global transfer of exergy is not maximum in this situation (see Fig. 3.16(b)). As mentioned above, this optimum could be expected since, at the critical back-pressure, the compression ratio has the highest possible value before the ejector enters the off-design regime, which leads to a decrease of the secondary mass flow rate. Hence $\Delta_2(x_b)$ should reach a maximum close to the critical back-pressure. In light of the analysis of the exergy losses, one may state that the net gain in exergy of the secondary stream is maximum for a value of the back-pressure close to the critical pressure not because the transfers of exergy are maximum but because the exergy losses related to viscous dissipation are significantly lowered. Based on these observations, one may conclude that the sole analysis of the transfers of momentum, kinetic energy and exergy is not sufficient to capture the net gain in exergy of the secondary stream. The reason behind this surprising result lies in the large contributions of irreversibilities.

To summarise the achieved results, Fig. 3.20 shows the evolution with the back-pressure of the different cumulative dimensionless quantities that have been introduced in the present study. As the purpose of this study is to draw a global picture of the ejector, only the final value of these parameters is considered in this figure. In this way, it is possible to analyze the entrainment, the global transfers and the cumulative losses in a single image. Note that $I_q(x_b)$ is not shown to avoid overloading the figure. Of course, it may be represented as the difference between $I_\xi(x_b)$ and $I_K(x_b)$. It clearly appears in Fig. 3.20 that the global transfer of mean-flow total exergy decreases with increasing back-pressure. Hence, as mentioned above, the maximum net gain in exergy of the secondary stream, $\Delta_2(x_b)$, is related to a significant decrease of exergy losses related to viscous dissipation, $\Pi_2(x_b)$. Concerning the global transfer of momentum, it is consistent with the global transfer of kinetic energy and both parameters give the same information for on-design operations. However, as may be observed in Fig. 3.20, it seems that, from a certain value of the back-pressure in the off-design regime, $I_m(x_b)$ and $I_K(x_b)$ start to differ. The global transfer of momentum is approximately constant for a certain range of back-pressures in off-design operations, while the global transfer of mean-flow kinetic energy decreases.

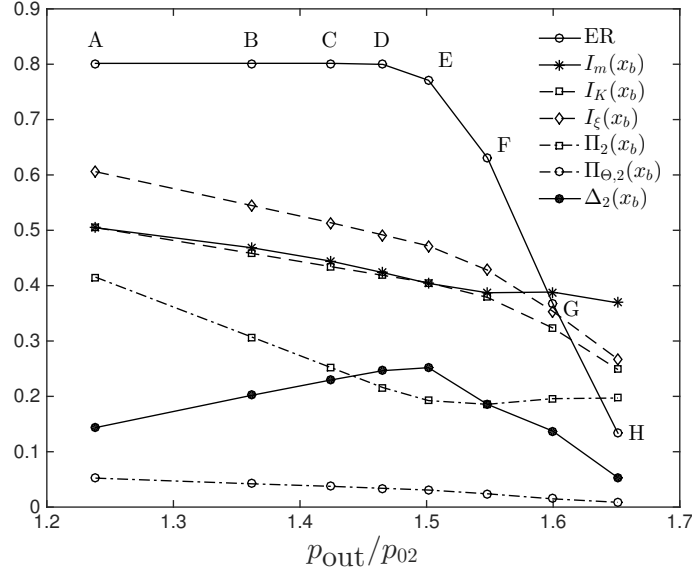


Figure 3.20: Global value of the different indicators – $p_{01}/p_{02} = 3.61$.

Finally, a Sankey diagram illustrating the fluxes of mean-flow total exergy in the secondary stream is presented in Fig. 3.21, for three different values of the back-pressure. Concerning the flux of exergy coming from the primary stream, the distinction is made between the transfer of exergy due to the transfer of mean-flow kinetic energy, $I_K(x_b)$, and due to the exergy content of the heat transfer, $I_q(x_b)$. In addition, the contributions of $\Pi_2(x_b)$ and $\Pi_{\Theta,2}(x_b)$ to the destruction of mean-flow total exergy within the secondary stream are also highlighted separately. Of course, the net gain in exergy of the secondary stream within the ejector is assessed through the parameter $\Delta_2(x_b)$, with the following balance $\Delta_2(x_b) = I_K(x_b) + I_q(x_b) - \Pi_2(x_b) - \Pi_{\Theta,2}(x_b)$.

When the ejector operates in the on-design regime (Fig. 3.21(a)), the transfer of exergy is very important but a significant part of this transfer is dissipated through phenomena related to viscous dissipation. Hence, for $p_{out}/p_{02} = 1.238$ (case A), the latter represent more than 40% of the flux of exergy coming from the primary reservoir. Close to the critical pressure (case D), although the transfers of exergy between both streams are lower, the exergy losses are also reduced, hence the net gain in exergy of the secondary stream is higher and may reach 24.2% of the flux of exergy originating from the primary reservoir (see Fig. 3.21(b)). Note that for this set of operating conditions close to the

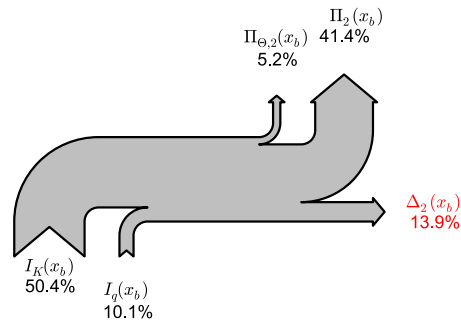
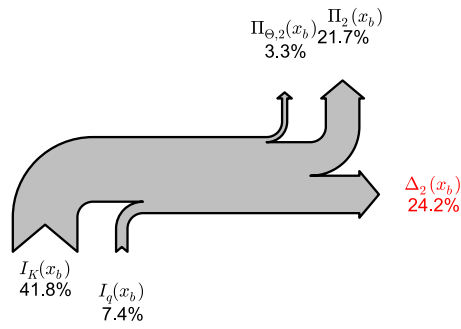
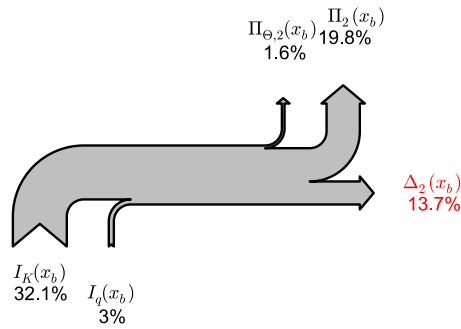
(a) $p_{\text{out}}/p_{02} = 1.238$ (case A) - on-design(b) $p_{\text{out}}/p_{02} = 1.465$ (case D) - near critical(c) $p_{\text{out}}/p_{02} = 1.600$ (case G) - off-design

Figure 3.21: Sankey diagrams of the fluxes of mean-flow total exergy within the secondary stream for different values of the back-pressure – $p_{01}/p_{02} = 3.61$.

critical point, more than half of the exergy that is given by the primary stream remains in the secondary stream at the outlet of the ejector. Finally, $\Delta_2(x_b)$ is only 13.7% for $p_{\text{out}}/p_{02} = 1.600$ (case G) which corresponds to off-design operations, as the global transfer of exergy is relatively low and represents less than one third of the flux of exergy flowing out of the primary reservoir (see Fig. 3.21(c)).

3.8 Conclusions

In this chapter, two different and complementary approaches have been proposed to provide novel insights into the quantification and the location of the transfers and the irreversibilities within supersonic ejectors, and their connection with the entrainment. First of all, the momentum and energy tube analysis tools introduced by Meyers and Meneveau [62] have been applied to investigate the key transport phenomena occurring along a supersonic ejector. This framework has also been extended to the concept of exergy to construct the corresponding transport tubes. These exergy tubes allow to unequivocally identify the topology of the exergy transfers.

Subsequently, an analysis based on classical stream tubes has been performed in order to quantitatively investigate transfers between the primary and the secondary streams all along the ejector, as well as isolate the exergy losses by heat transfer and viscous dissipation within the secondary stream. This approach offers some novel phenomenological insights but also leads to the definition of new quantitative indicators to assess these transfers and losses.

The application of these tools to the present ejector geometry and reservoir conditions has shown that the mixing between both streams is not yet completed at the entry of the diffuser for some cases lying in the on-design regime. Moreover, the operating point resulting in the highest global gain in exergy of the secondary stream actually corresponds to the optimization of two effects: minimized losses and maximized exergy transfer. Hence, this does not correspond to the point where the net transfers through the dividing streamline are the highest. Logically, it appears that this point nearly matches with the critical point of the characteristic curve.

CHAPTER

NUMERICAL AND EXPERIMENTAL EVIDENCE OF THE FABRI-PATTERN IN A SUPERSONIC EJECTOR

This chapter is based on the following paper:

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.

4.1 Introduction

Besides mixing processes, a better understanding of choking phenomena in supersonic ejectors is also necessary to improve their performances. In this chapter, the existence of a sonic section in the core of the secondary stream caused by an aerodynamic throat is evidenced both numerically and experimentally. It will be shown in Chapter 5 that this flow feature is not responsible for the entrainment limitation, but rather a particular flow pattern. Therefore, the acceleration of the secondary stream to supersonic velocities through an aerodynamic throat is referred to as the Fabri-pattern instead of the Fabri-choking. The nu-

merical results exploited in this chapter correspond to the characteristic curve obtained with $p_{01}/p_{02} = 3.61$ (see Fig. 2.9, p. 33).

The structure of this chapter is the following: in Section 4.2, the numerical results are thoroughly processed for a fixed value of the back-pressure corresponding to on-design operation in order to gain insights into the phenomenology of the Fabri-pattern. In Section 4.3, after having defined a quantitative criterion for the location of the Fabri-pattern, the exergy analysis introduced in Chapter 3 is performed in order to confirm the validity of the assumption that mixing may be neglected between the nozzle exit position and this particular location. Finally, the impact of the back-pressure on the Fabri-pattern is investigated in Section 4.4.

4.2 Phenomenology of the Fabri-pattern

4.2.1 Sonic line and dividing streamline

As this study mainly aims at investigating the Fabri-pattern in the post-processing phase of a CFD calculation, usual physical quantities such as the Mach number or the stagnation pressure are calculated for the mean-flow with their instantaneous definitions but from the averaged flow quantities obtained by numerical simulations, and are indicated with the symbol $\sim$. The mean-flow stagnation pressure, $\tilde{p}_t$ and the mean-flow Mach number, $\tilde{M}$, have already been defined in Eqs. (3.32) and (3.33), respectively (see p.53). Although in most numerical studies reported in the literature, the mathematical definition of the Mach number obtained after post-processing of the results is not explicitly stated, it is highly likely that it corresponds to the mean-flow Mach number. Of course, the mean-flow Mach number defined in Eq. (3.33) is different from the time-averaged Mach number, $\overline{M}$, but it is the sole definition of the Mach number that is directly calculable with results of classical CFD calculations. Furthermore, $\tilde{M}$ is a good approximation of $\overline{M}$ if the density and temperature fluctuations are limited. In the following analysis, the reader should bear in mind that the term *Mach number* will refer to the *mean-flow* Mach number, defined in Eq. (3.33).

Figure 4.1 shows the Mach number field (supersonic region only, i.e. $\tilde{M} \geq 1$) for operating conditions in the on-design regime ($p_{\text{out}}/p_{02} = 1.238$, case A in Fig. 2.9). It should be emphasized that for improved readability, the ejector is depicted with a twofold stretching in the vertical direction for all the figures presented in this chapter.

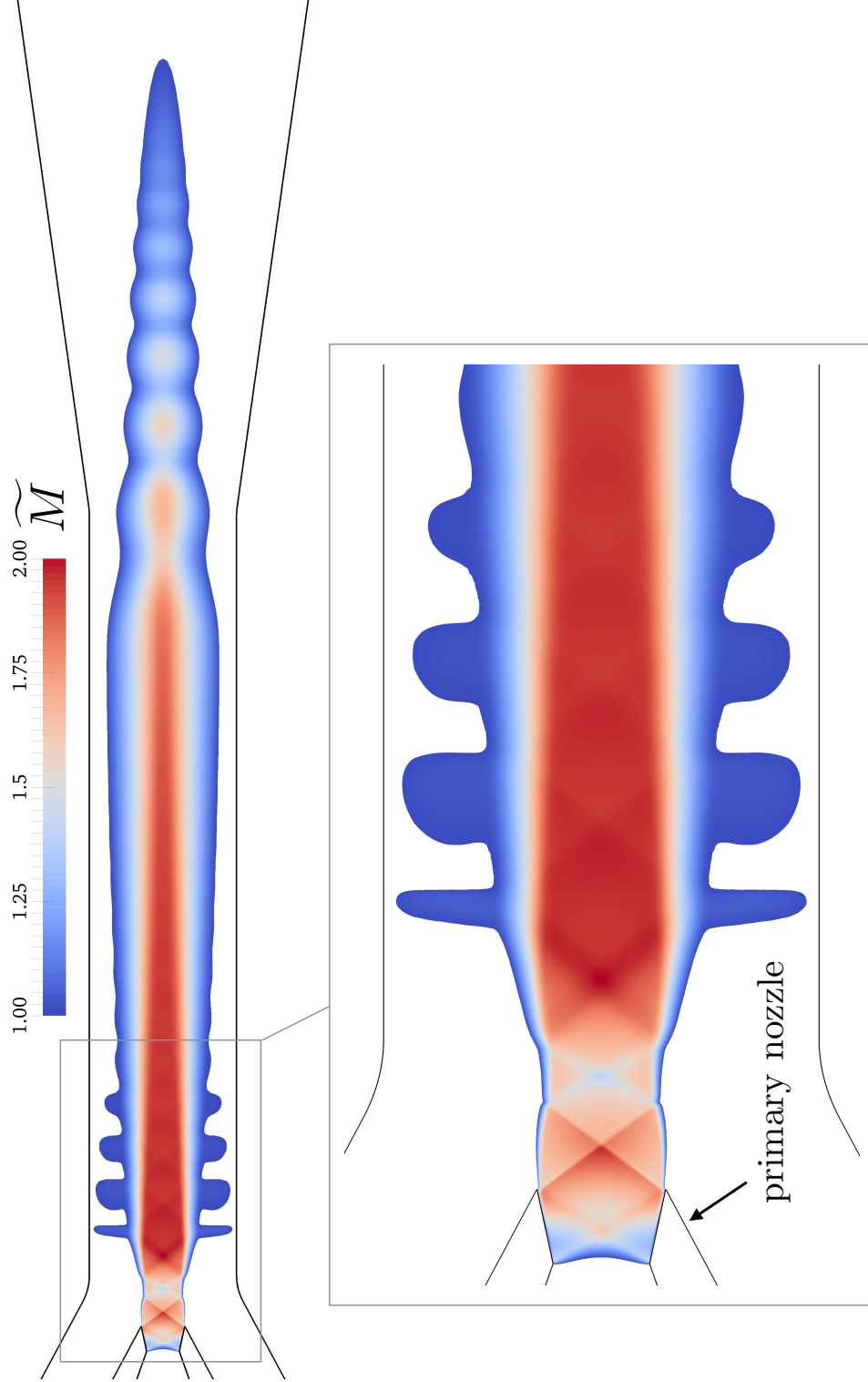


Figure 4.1: Mach number field (supersonic region only) - $p_{01}/p_{02} = 3.61 - p_{out}/p_{02} = 1.238$ (case A). For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

As the value of the back-pressure is relatively low, two distinct shock trains are logically discernible in the core of the flow: the first one at the exit of the primary nozzle and the second one at the entry of the diffuser. This observation corresponds to previous results reported in several numerical studies [28, 45, 77]. In contrast, to the knowledge of the author, the surprising aspect of the Mach number field at the entry of the mixing duct, brought to the forefront in Fig. 4.1, has never been highlighted before. Actually, this alternation of supersonic and subsonic regions observed in Fig. 4.1 is related to a complex flow phenomenology that is thoroughly analyzed below.

To investigate the transition from subsonic to supersonic velocities, the following analysis will focus on the *sonic line*, illustrated in Fig. 4.2, rather than the entire Mach number field. The sonic line is defined as the iso-contour of mean-flow Mach number unity (i.e. $\widetilde{M} = 1$) and is the most straightforward way to distinguish between subsonic and supersonic regions of the flow based on CFD results. As mentioned in the introduction, Hemidi et al. [45] already introduced the idea that looking at the sonic line could help to find the location of the critical cross-section.

Then, in order to isolate the region of the ejector occupied on average by the secondary stream, a classical stream tube based on the velocity vector field $\hat{u}$ and surrounding the primary stream is constructed, according to the methodology proposed in Chapter 3. The boundary of this stream tube, named the *dividing surface*, may be represented by the *dividing streamline*, illustrated in Fig. 4.2.

In the light of both lines, it appears that the primary stream remains entirely supersonic at the entry of the mixing duct. Indeed, the alternation of sub-/supersonic regions is exclusively located in the core of the secondary stream. A physical explanation of this particular flow feature is provided in the next section.

4.2.2 Flow topology related to the Fabri-pattern

In comparison with a common overexpanded nozzle (see for example [92]), the two types of boundary conditions that must be met by the supersonic jet at the exit of the primary nozzle are the same. On one hand, from symmetry aspects, a central streamline must exist. On the other hand, pressure equilibrium must exist across the jet boundary. However, in contrast with the exhaust of a classical nozzle, in a supersonic ejector, the outer pressure is not constant along the jet. Indeed, as the primary jet is confined by the walls of the secondary nozzle, the shape of the supersonic jet influences the outer pressure acting on it. There-

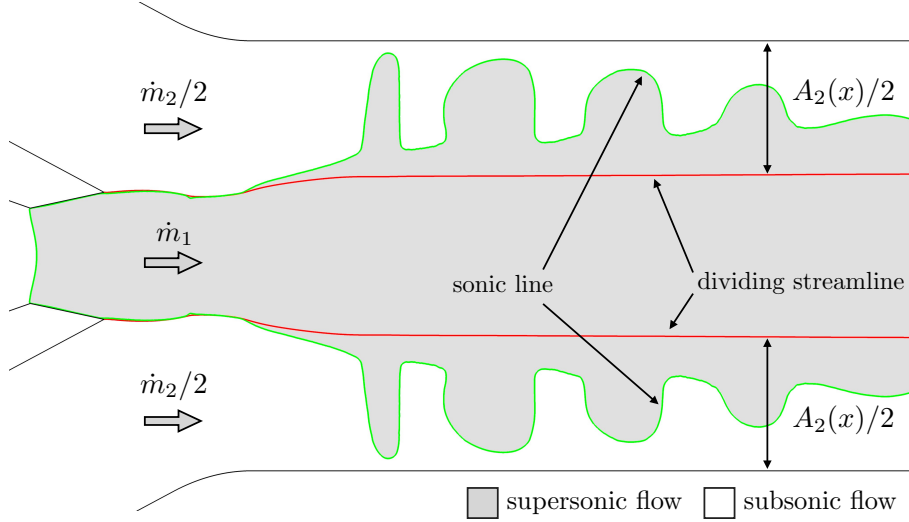


Figure 4.2: Interpretation of the sonic line and the dividing streamline. For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

fore, there is a coupling — intrinsic to supersonic ejectors — between the shape of the motive stream at the exit of the primary nozzle and the outer static pressure, and thereby with the entrained mass flow rate.

Figure 4.3 represents schematically the flow topology in the vicinity of the nozzle exit. Up to the region indicated by the number 4, the supersonic structures in the core of the primary stream are very close to the overexpanded nozzle described in a common textbook [92]. As the region 4 is close to the entry of the mixing duct where the total flow passage reaches its minimum value, the coupling between the primary and the secondary streams becomes more important. More precisely, the Prandtl-Meyer expansion fan in the primary jet between regions 4 and 5 implies an increase of the primary flow section. Therefore, the section available for the secondary mass flow rate decreases. As a consequence, the secondary stream, that is still subsonic, accelerates and its static pressure falls. This causes the primary jet to further expand, which in turn amplifies the constriction of the secondary stream. Due to this interaction, the drop in the static pressure in the secondary stream and hence acting on the jet boundary in region 6 may be significant.

At a point of maximum expansion of the primary stream, the secondary stream reaches sonic conditions in a sort of aerodynamic throat, as postulated by Fabri and Siestrunk [33]. The location of this throat is highlighted in the top half of Fig. 4.3 at the first sonic section in the

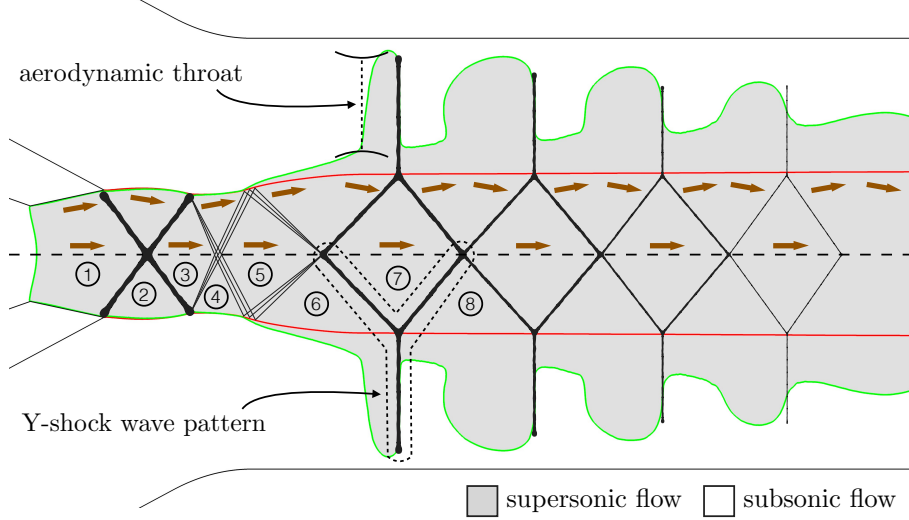


Figure 4.3: Flow topology related to the Fabri-pattern. For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

core of the secondary stream, which implies a passage from subsonic to supersonic velocities. Throughout the rest of the document, the terms sonic section, aerodynamic throat and Fabri-pattern are used as synonyms to designate this particular flow feature.

In the study of Fabri and Siestrunck [33], nothing is said regarding the downstream flow topology. Actually, the secondary stream further accelerates to supersonic velocities in a virtual diverging flow section. This causes the flow structure in the primary stream to deviate from the free jet configuration [92]. Indeed, the oblique shock wave in the core of the primary stream between regions 6 and 7 that would reflect on the jet boundary as a Prandtl-Meyer expansion fan in the case of a free jet, propagates here into the supersonic secondary stream and turns into a normal shock wave in its core, thereby decelerating the flow to subsonic velocities, as attested by the sonic line in Fig. 4.3. But this normal shock wave in the core of the secondary stream also results in an increase of the outer static pressure acting on the jet boundary. Hence, another oblique shock wave is required in the core of the primary stream to ensure pressure equilibrium with the secondary stream, thereby forming a particular shock wave pattern highlighted in the bottom half of Fig. 4.3. Due to its structure, it is proposed to name this pattern a *Y-shock wave (pattern)*. To the knowledge of the authors, this Y-shock wave pattern has never been reported in the literature. It should not be

confused with the *lambda foot* observed in interactions between shock waves and turbulent boundary layers [40] that may appear in the primary nozzle of a supersonic ejector [59]. In region 8, the situation is approximately the same as in region 6. The primary stream forms another aerodynamic throat in the core of the secondary stream resulting in a second Y-shock wave pattern. Ideally, this pattern would continue infinitely but the turbulent shear layer progressively dissipates it.

4.2.3 Aerodynamic throat in the secondary stream

Through the post-processing of the numerical results, it may be shown that the first sonic section in the core of the secondary stream essentially corresponds to an aerodynamic throat, although it is not very clear when observing the dividing streamline or other secondary streamlines in the bottom half of Fig. 4.4. Actually, it is more clearly highlighted with the iso-contours of zero vertical velocity, shown in the top half of the same figure. Indeed, these lines are useful to bring out changes in the vertical orientation of the flow, as illustrated with arrows in Fig. 4.4. Based on these iso-contours, it clearly appears that the secondary stream changes its direction from upwards to downwards as it passes for the first time through the sonic line, due to a similar behavior of the dividing streamline. Hence, even though it is not very pronounced, the cross-section delimited by the dividing streamline through which the secondary stream virtually flows, noted $A_2(x)$ in Fig. 4.2, exhibits a local minimum close to the first sonic section occurring in the secondary stream.

4.2.4 Y-shock wave pattern

As explained above, most of the secondary mass flow rate reverts to a subsonic regime a short distance downstream of the first sonic section through a normal shock wave that forms, with two oblique shock waves in the core of the primary stream, a Y-shock wave pattern (see Fig. 4.3, p. 80). The latter may be observed in Fig. 4.5 showing the overlay of the sonic line with the horizontal gradient of the density field (pseudo-schlieren) within the ejector. The vertical portion of the sonic line outlined in the top half of the figure is clearly related to a sharp increase of the density, typical of a shock wave. Compared to the pseudo-schlieren shown in the bottom half of Fig. 2.8(a) (p. 32), the numerical results shown in Fig. 4.5 reveal more flow features, although both images refer to the same operating conditions. This is mainly due to the fact that these two figures illustrate a different component of the

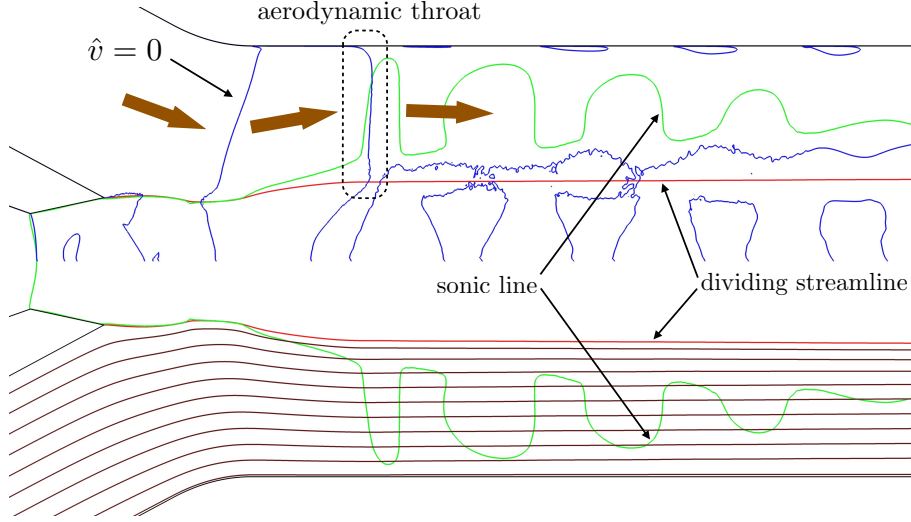


Figure 4.4: Direction of the flow (top half) and streamlines (bottom half) in the secondary stream – $p_{01}/p_{02} = 3.61$ – $p_{\text{out}}/p_{02} = 1.238$ (case A). For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

gradient of the density field. While its vertical component is shown in Fig. 2.8(a) to bring to light the shear layer, the horizontal gradient of the density field is shown in Fig. 4.5 in order to highlight a normal shock wave in the secondary stream. Regarding the oblique shock waves in the primary stream, they are more visible in Fig. 4.5 because of the saturation settings (saturation amplified by a factor 4 compared to Fig. 2.8(a)).

4.2.5 Experimental evidence of the Y-shock wave pattern

Experimentally, this Y-shock wave pattern has been investigated with shadowgraphy, with the high-speed camera described in Section 2.2.1. An instantaneous image of the flow at the entry of the mixing duct is shown in Fig. 4.6 for $p_{\text{out}}/p_{02} = 1.238$ (case A). The first Y-shock wave pattern may be clearly observed, in particular in the bottom half of the figure where it is highlighted by a white dashed line.

It should be emphasized that the experimental visualization shown in Fig. 4.6 essentially serves as a qualitative validation of the Fabri-pattern as it primarily aims to evidence the existence of a normal shock wave in the secondary stream and the related Y-shock wave pattern. A quantitative validation of the entire sonic line pattern is beyond the

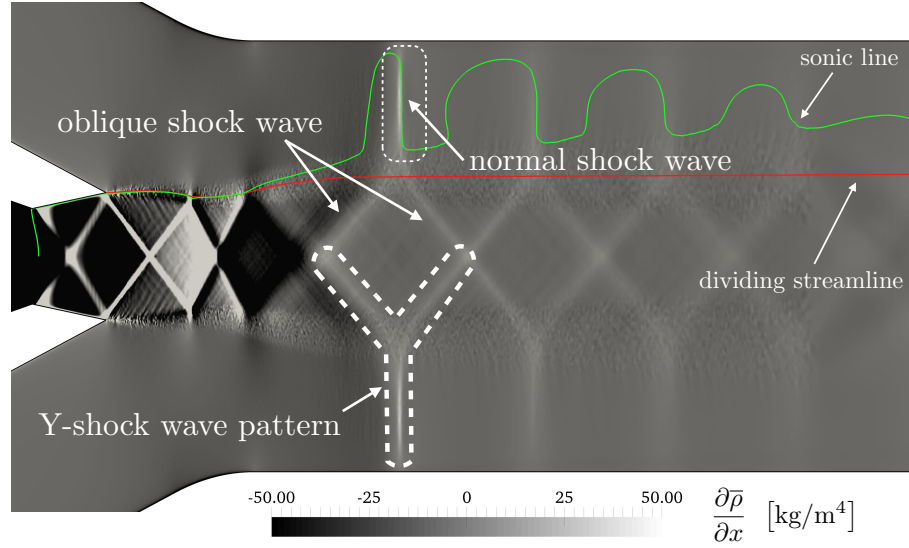


Figure 4.5: Horizontal gradient of the density field highlighting the Y-shock wave pattern – $p_{01}/p_{02} = 3.61 - p_{out}/p_{02} = 1.238$ (case A). For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

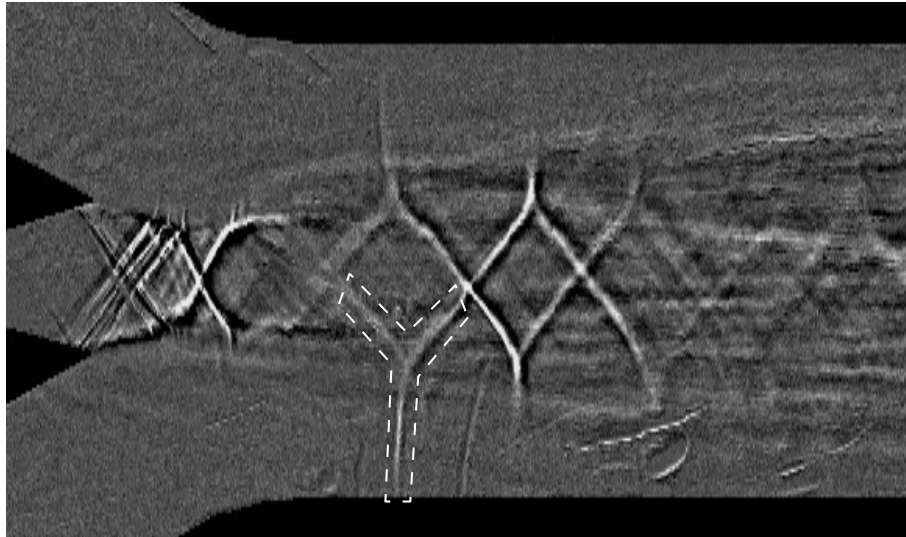


Figure 4.6: Experimental observation of the Y-shock wave pattern through shadowgraphy – $p_{01}/p_{02} = 3.61 - p_{out}/p_{02} = 1.238$ (case A). For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

scope of the present work and is experimentally very challenging due to the oscillating behavior of this complex flow signature as well as its dissipation induced by turbulence and mixing effects.

Nevertheless, the agreement between the CFD and the experimental results is actually fairly good. Indeed, the axial position of the first normal shock wave in the secondary stream is roughly the same in the pseudo-schlieren as in the experimental image. Moreover, the number of shock cells in the core of the primary stream that may be distinguished upstream of the first Y-shock wave pattern in the CFD is also consistent with the experimental visualization. This agreement in terms of topology thus also brings confidence about the Mach numbers achieved between the discontinuities. The Y-shock wave pattern may be considered as the signature of the Fabri-pattern and its observation based both on numerical (see Fig. 4.5) and experimental (see Fig. 4.6) results evidences the existence of this particular flow feature.

4.3 Transfers and irreversibilities in the secondary stream upstream of the Fabri-pattern

This section checks the validity of the assumption that transfers between the primary and the secondary streams are negligible upstream of the Fabri-pattern. This hypothesis is often used in the literature and was already postulated by Fabri and Siestrunk [33]. To this end, the static pressure and the thickness of the shear layer in the vicinity of the aerodynamic throat are first investigated. Then, the location of the Fabri-pattern is defined by a new quantitative criterion and the transfers of exergy between the nozzle exit position and this particular point are finally quantified by an exergy analysis using the methodology proposed in Chapter 3.

4.3.1 Static pressure and thickness of the shear layer at the aerodynamic throat

Regarding the static pressure in the vicinity of the aerodynamic throat, it could be expected to be close to the sonic state reached with the secondary reservoir conditions through a 1D isentropic nozzle flow. This particular value of the static pressure is generally referred to as the critical pressure (to not be confused with the critical back-pressure) and noted p^* in common textbooks [73, 92]. For an ideal gas, the critical

pressure is given by

$$\frac{p^*}{p_{02}} = \left(1 + \frac{(\gamma - 1)}{2}\right)^{-\frac{\gamma}{(\gamma-1)}}. \quad (4.1)$$

This expression takes a value close to 0.528 for $\gamma = 1.4$. Hence the iso-contour of the static pressure corresponding to this particular value is shown in the top half of Fig. 4.7. It appears indeed that the mean static pressure prevailing at the first sonic section corresponds nearly to the above-mentioned value as its iso-contour almost overlaps that of the sonic line.

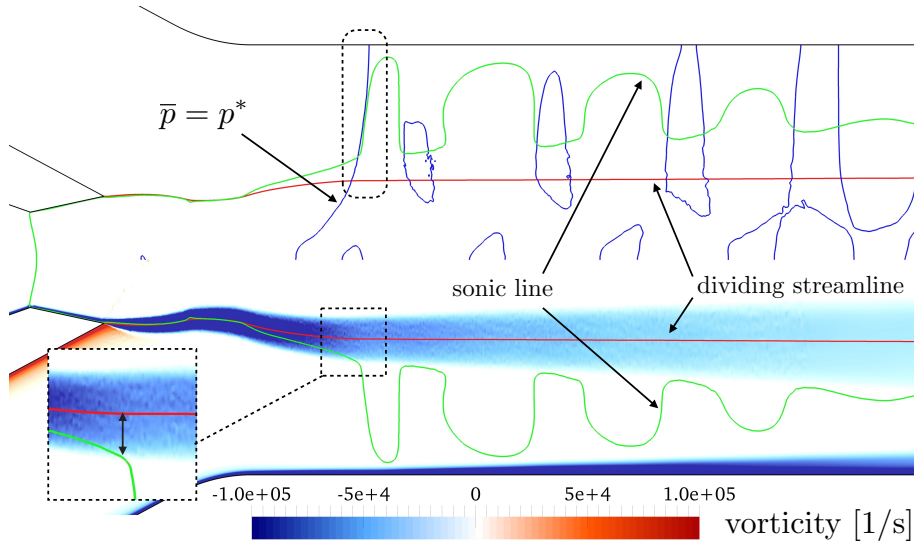


Figure 4.7: Static pressure (top half) and vorticity (bottom half) – $p_{01}/p_{02} = 3.61 - p_{\text{out}}/p_{02} = 1.238$ (case A). For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

The small part of the secondary stream that is confined between the sonic line and the dividing streamline just upstream of the Fabri-pattern (highlighted with an arrow in the bottom half of Fig. 4.7) has already accelerated to supersonic velocities before reaching the aerodynamic throat. Actually, when looking at the span-wise vorticity field in the bottom half of Fig. 4.7, it appears clearly that this fraction of the secondary stream is located in the shear layer. It has therefore already received some exergy from the primary stream, resulting in higher velocities in this part of the secondary stream. These transfers are quantified below, after having defined the location of the Fabri-pattern.

4.3.2 Definition of the location of the Fabri-pattern

A new parameter, $M_2(x)$, corresponding to a 1D-equivalent Mach number for the secondary stream, is proposed for a quantitative investigation of the Fabri-pattern. The parameter $M_2(x)$ is calculated in the post-processing phase of the CFD calculation as the ratio of a mean velocity to a mean speed of sound in the secondary stream. Hence, for a perfect gas, it may be defined as

$$M_2(x) = \frac{U_2(x)}{\sqrt{\gamma R T_2(x)}} , \quad (4.2)$$

with $U_2(x)$ a mean velocity

$$U_2(x) = \frac{\dot{m}_2}{\iint_{A_2(x)} \bar{\rho} ds} , \quad (4.3)$$

and $T_2(x)$ a mean temperature over the secondary cross-section

$$T_2(x) = \frac{1}{A_2(x)} \iint_{A_2(x)} \hat{T} ds . \quad (4.4)$$

Readers are reminded that $A_2(x)$ is the virtual cross-section through which the secondary stream flows, i.e. the section between the ejector boundary and the dividing surface (see Fig. 4.2, p. 79). The profile of $M_2(x)$ in the first half of the mixing duct is illustrated in Fig. 4.8(a) for $p_{\text{out}}/p_{02} = 1.238$ (case A). Logically, $M_2(x)$ increases at the entry of the mixing duct as the secondary stream accelerates in a reducing flow passage, and reaches the value of approximately 1.05 before oscillating just above unity. These oscillations in the profile of $M_2(x)$ in Fig. 4.8(a) correspond to the alternation of sub-/supersonic regions in the core of the secondary stream observed above (see Fig. 4.2).

It is proposed to define the location of the Fabri-pattern, x_f , as the first position where $M_2(x)$ reaches unity. For the present geometry and operating conditions, this axial position corresponds to $x_f/H_m \simeq 1.02$ and is in very good agreement with the first sonic section in the core of the secondary stream, as shown in Fig. 4.8(b).

Having defined the location of the Fabri-pattern, it is interesting to assess the consistency of this position with the profile of the virtual cross-section through which the secondary stream flows, $A_2(x)$. The latter is shown in Fig. 4.9 in the vicinity of $x = x_f$. It may be observed that the position of the Fabri-pattern defined above nearly corresponds

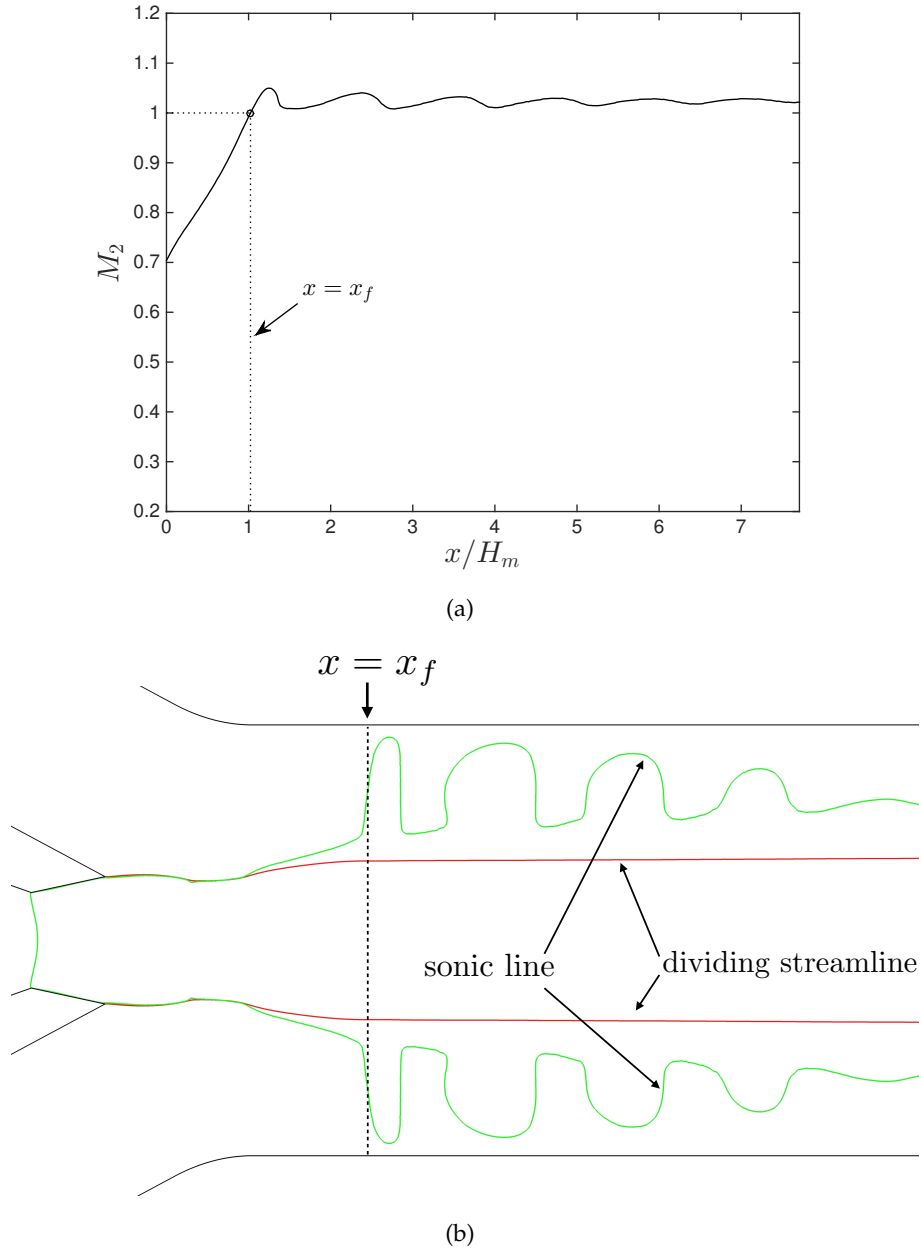


Figure 4.8: Definition of the location of the Fabri-pattern: (a) profile of $M_2(x)$ in the first half of the mixing duct and (b) corresponding location – $p_{01}/p_{02} = 3.61 - p_{\text{out}}/p_{02} = 1.238$ (case A).

to a local minimum for $A_2(x)$. It is logically located at a small distance upstream of this minimum as the definition of $M_2(x)$ includes the contribution of the fraction of the secondary mass flow rate that is accelerated to supersonic velocities through the shear layer.

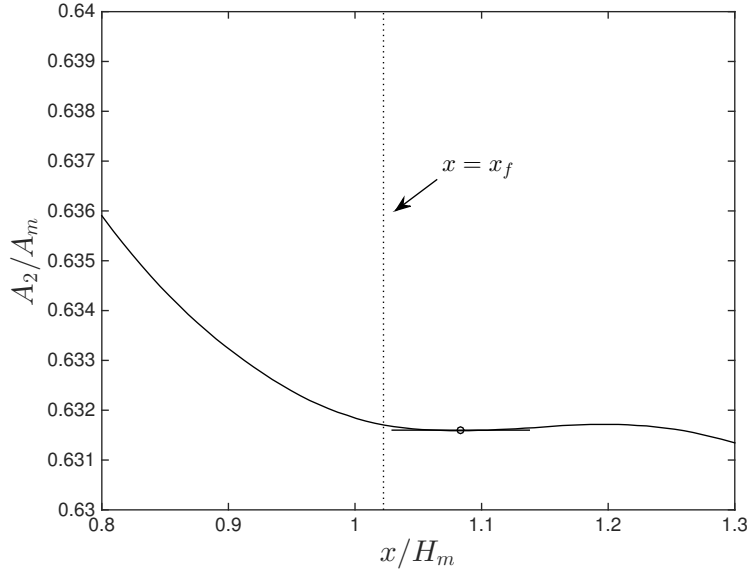


Figure 4.9: Profile of $A_2(x)$ in the vicinity of the choking position (A_m is the cross-section of the mixing duct) – $p_{01}/p_{02} = 3.61$ – $p_{\text{out}}/p_{02} = 1.238$ (case A).

Furthermore, Fig. 4.9 further confirms that the local minimum in the profile of $A_2(x)$ is not very pronounced, as mentioned earlier (see Section 4.2). As a consequence, the Fabri-pattern will be very elusive and highly sensitive to operating conditions and 3D effects.

At the aerodynamic throat, the secondary stream occupies the larger portion of the cross-section of the mixing duct (approximately 63%) although the entrainment ratio is lower than unity. This is due to the fact that both the velocity and the density are still higher in the primary stream at this location.

The profile of the mean-flow Mach number at the Fabri-pattern position is shown in Fig. 4.10, showing the larger velocities in the core of the primary stream compared to the secondary stream. It consistently appears that the flow in the core of the secondary stream is essentially uniform at the aerodynamic throat, with $\widetilde{M} \simeq 1$. Nevertheless, at both edges of the secondary stream, the flow differs from the core. First of

all, there is logically a region of low velocities in the boundary layer, in the vicinity of the wall. Then, close to the dividing streamline (i.e. near the primary stream), a small but significant fraction of the secondary mass flow rate ($\simeq 20\%$) has already reached supersonic velocities at the choking position. As mentioned above, the acceleration of this small part of the secondary stream is due to a transfer of exergy with the primary stream occurring through the shear layer.

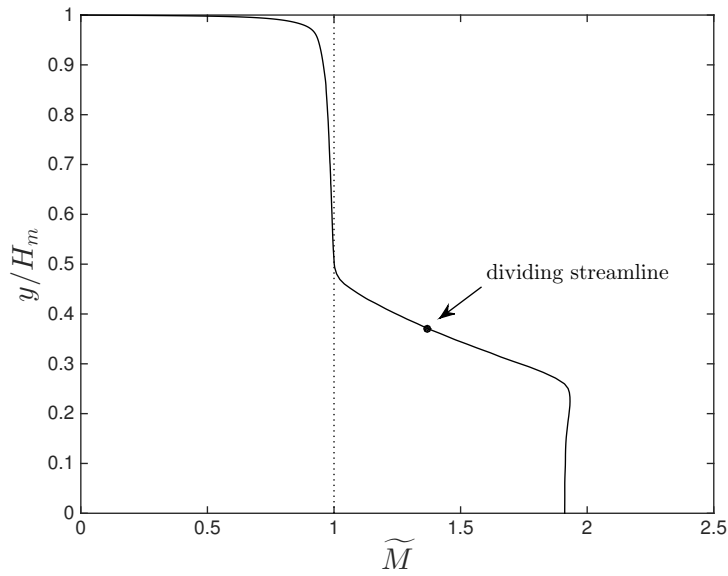


Figure 4.10: Profile of the mean-flow Mach number at the location of the Fabri-pattern in the top half of the ejector (The center and the top wall of the mixing duct correspond to $y/H_m = 0$ and $y/H_m = 1$, respectively) – $p_{01}/p_{02} = 3.61$ – $p_{\text{out}}/p_{02} = 1.238$ (case A).

4.3.3 Exergy transfers and losses in the secondary stream

The quantitative exergy analysis introduced in Chapter 3 may be applied to the secondary stream between the secondary inlet and the location of the Fabri-pattern, x_f . The value of the different dimensionless parameters evaluated at $x = x_f$ is shown in Table 4.1. In order to put these results in perspective, the value achieved at the outlet of the ejector ($x = x_b$) is also reported for each parameter. As may be observed in Table 4.1, the cumulative transfer of mean-flow total exergy between the primary and the secondary stream, $I_\xi(x)$, is relatively low at the first sonic section as it represents only 3.9% of the flux of exergy com-

ing from the primary reservoir. By comparison, $I_\xi(x)$ reaches 60.5% at the outlet of the ejector for the operating point under consideration. Due to exergy losses in the secondary stream reaching 2.7% ($\Pi_2(x_f)$ and $\Pi_{\Theta,2}(x_f)$), the dimensionless net gain in exergy of the secondary stream at $x = x_f$, $\Delta_2(x_f)$, is even lower, and barely reaches 1.3%. As mentioned above, this increase of exergy is not evenly distributed within the secondary stream but is rather concentrated in the region of the secondary stream lying in the shear layer. Nevertheless, in view of the small value of $\Delta_2(x_f)$, it appears that the secondary stream arrives at the aerodynamic throat with almost its reservoir conditions in terms of mean-flow total exergy.

	$x = x_f$	$x = x_b$
I_ξ [%]	3.9	60.5
Π_2 [%]	2.4	41.4
$\Pi_{\Theta,2}$ [%]	0.3	5.2
Δ_2 [%]	1.3	13.9

Table 4.1: Value of the exergy dimensionless parameters at the location of the Fabri-pattern ($x = x_f$) and at the outlet of the ejector ($x = x_b$) – $p_{01}/p_{02} = 3.61 - p_{\text{out}}/p_{02} = 1.238$ (case A).

In view of the above, it appears that an important part of the secondary stream reaches sonic conditions at nearly the same axial position, in a sort of aerodynamic throat and with almost its reservoir conditions. This flow feature is proposed to be referred to as the Fabri-pattern since it was early postulated by Fabri and Siestrunk [33]. Even though the assumption of a sonic section in the secondary stream was made for an underexpanded primary nozzle in [33], it is observed for an overexpanded nozzle in the present work. Of course, this observation remains consistent with the model of Fabri and Siestrunk as the sonic section is located in the vicinity of an expansion zone in the primary jet.

Thus far, only one operating point corresponding to the lowest value of the back-pressure, i.e. case A, has been considered for the analysis. In the following section, the impact of the back-pressure on the Fabri-pattern is investigated.

4.4 Impact of the back-pressure on the Fabri-pattern

The values of the back-pressure considered in this analysis correspond to the five lowest values of the numerical characteristic curve shown in Fig. 2.9 (p. 33), i.e. cases A to E. The cases A and B clearly correspond to on-design operation. As may be observed in Fig. 4.11 showing a zoom on the numerical results for the three cases located close to the critical back-pressure, the ejector still operates in the on-design regime for case C while the case D seems slightly higher than the critical back-pressure. The case E evidently lies in the off-design region.

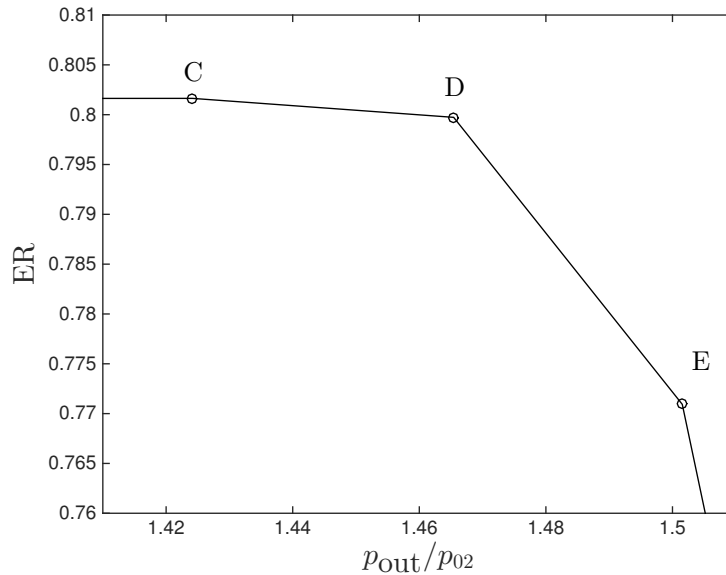


Figure 4.11: Zoom on the numerical characteristic curve corresponding to $p_{01}/p_{02} = 3.61$ (shown in Fig. 2.9, p. 33) close to the critical back-pressure.

First of all, the sonic line at the entry of the mixing duct is shown in Fig. 4.12 for the selected values of the back-pressure. Note that the same information is depicted in both halves of the ejector for easier readability. As a first observation in Fig. 4.12, it logically appears that the primary stream leaves the primary nozzle as a supersonic jet for all operating conditions. Actually, for the cases under study, all the sonic lines are almost superimposed from the throat of the primary nozzle to

the entry of the mixing duct. Further downstream, each sonic line begins to deviate from the sonic line corresponding to case A at a different axial position. Logically, the overlap is the largest for the closest value of the back-pressure (i.e. case B) and then decreases with increasing back-pressure.

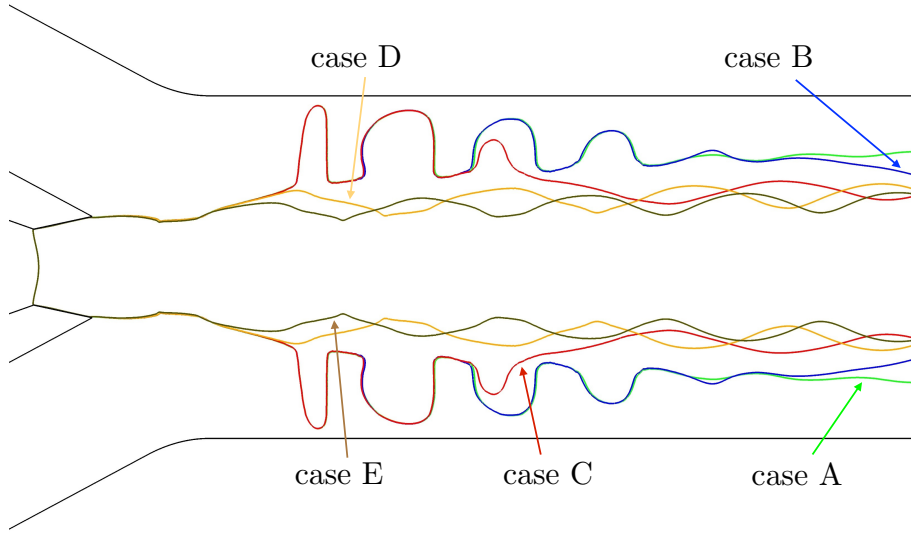


Figure 4.12: Sonic line at the entry of the mixing duct for $p_{01}/p_{02} = 3.61$ and different values of the back-pressure. For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

More importantly, for the three cases corresponding to on-design operation (cases A, B and C), the sonic lines are superimposed beyond the aerodynamic throat. Hence the secondary stream reaches sonic velocities in the ejector for these three operating points and the flow structure is logically the same. In contrast, above the critical back-pressure, the sonic line does not penetrate deep into the secondary stream and there is consequently no sonic section, thus no Fabri-pattern.

Figure 4.13 shows the profile of the parameter $M_2(x)$ in the first half of the mixing duct for the five values of the back-pressure under consideration. The profiles of the three cases in the on-design regime are logically superimposed beyond the Fabri-pattern position defined above. For the case D, the profile of $M_2(x)$ is almost common with the

aforementioned values of the back-pressure but starts deviating at a small distance upstream of $x = x_f$ and reaches a maximum value close to 0.9. Note that the entrainment ratio for this case is very close to the plateau attained in the on-design regime (see Fig. 4.11). The profile of $M_2(x)$ is logically different and oscillates around a lower value (close to 0.65) for case E.

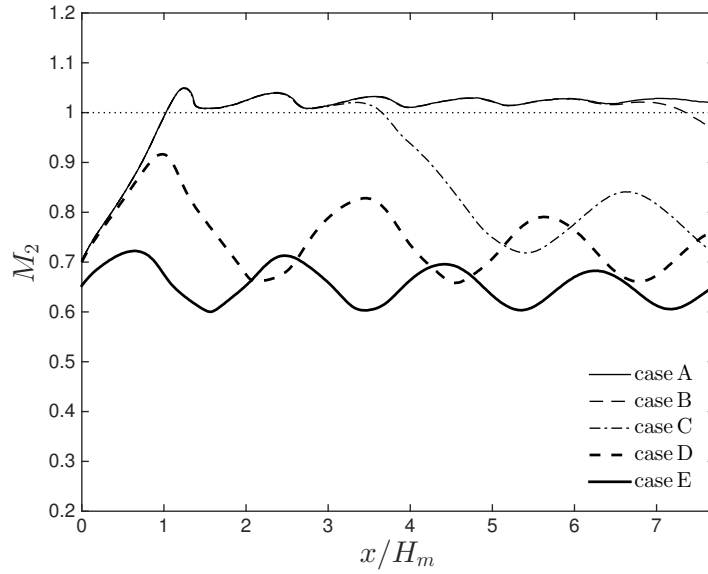


Figure 4.13: Profile of $M_2(x)$ in the first half of the mixing duct for different values of the back-pressure – $p_{01}/p_{02} = 3.61$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

Finally, for the five cases under study, the profile of the static pressure at the wall, $\bar{p}_{\text{wall}}$, is shown in Fig. 4.14, in the first half of the mixing duct only. Once again, in on-design operation, the flow is the same at least until the first sonic section. Furthermore, it appears that, once the ejector enters the off-design regime, the static pressure at the wall does not fall below the critical pressure, p^* , as the secondary stream does not reach sonic velocities.

4.5 Conclusions

In this chapter, the existence of a sonic section in the core of the secondary stream has been evidenced numerically and experimentally. The first analysis consisted in the visualization of two fundamental

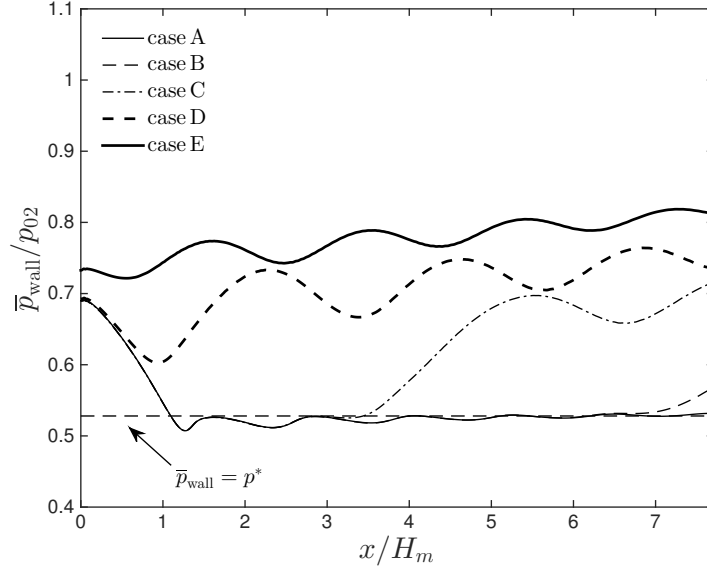


Figure 4.14: Static pressure profile in the first half of the mixing duct for different values of the back-pressure – $p_{01}/p_{02} = 3.61$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

lines that are constructed in the post-processing phase of a CFD calculation: the *dividing streamline* which is a virtual separation between the primary and the secondary stream and the *sonic line* that corresponds to the iso-contour of mean-flow Mach number unity. A detailed physical explanation of the observed flow structure has then been provided.

The subsequent analysis has confirmed that a significant fraction of the secondary mass flow rate reaches sonic conditions through a restriction of its streamlines. This is effectively an aerodynamic throat for the secondary stream where the static pressure is roughly equal to the sonic state reached with the secondary reservoir conditions through a 1D isentropic flow. In addition to this, a flow signature, never identified within a supersonic ejector until now, forming a so-called Y-shock wave pattern and being repeated axially, has been numerically highlighted with a pseudo-schlieren and corroborated by means of experimental visualization. This is the first work which brings a clear evidence both numerically and experimentally of the existence of a sonic section in the secondary stream, postulated almost 60 years ago by Fabri and Siestrunk [33]. However, it will be shown in the next chapter that this flow feature is not responsible for the entrainment limitation. There-

fore, it is proposed to name it the Fabri-pattern rather than the Fabri-choking. The reason why the Fabri-pattern has never been highlighted in the literature is because it requires very peculiar conditions (having an aerodynamic throat and pressure mismatch between both streams) that are not generally met in supersonic ejectors. Furthermore, the visualization methodology (experimental or numerical) requires a very fine resolution as it basically consists in shock waves produced in conditions very close to $M = 1$.

Then, a new parameter corresponding to a 1D-equivalent Mach number for the secondary stream has been proposed for the definition of the location of the Fabri-pattern, and an exergy analysis based on the methodology proposed in Chapter 3 has been applied to the secondary stream. It appears that the entrained mass flow rate reaches sonic velocities with practically the exergy corresponding to its reservoir conditions. Indeed, both the transfer of mean-flow total exergy with the primary stream and the irreversibilities within the secondary stream were found to be negligible between the inlet and the first sonic section.

Finally, the impact of the back-pressure on the Fabri-pattern has been investigated through the visualization of the sonic lines for five different values of the back-pressure. For the ejector geometry and reservoir conditions under consideration, it seems that the Fabri-pattern exists for all the on-design operating points and disappears when the ejector operates in the off-design regime. Nevertheless, as mentioned above, it will be shown in the next chapter that the choking mechanism at the basis of the entrainment limitation is not related to this specific flow feature.

CHAPTER

THE COMPOUND-CHOKING THEORY AS AN EXPLANATION OF THE ENTRAINMENT LIMITATION IN SUPERSONIC EJECTORS

This chapter is based on the following paper:

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.

5.1 Introduction

In the present chapter, it is shown that the compound-choking theory proposed by Bernstein et al. [14] yields an explanation for both the absence of the Fabri-pattern in most numerical results and the difficulties of state-of-the-art 1D models to predict the on-design entrainment ratio.

Its structure is the following: the fundamentals of the compound-choking theory are briefly exposed in Section 5.2. In Section 5.3, the post-processing methodology introduced in the previous chapter to in-

investigate the acceleration of the secondary stream to supersonic velocities is applied to the same ejector geometry but with other reservoir conditions. The operating points that are considered here correspond to the characteristic curve obtained with $p_{01}/p_{02} = 5.13$ (see Fig. 2.9, p. 33). Section 5.4 is devoted to the application of the compound-choking theory to predict the on-design entrainment ratio of published experimental ejector configurations, as well as a comparison with state-of-the-art 1D models. The main conclusions and perspectives of the present investigation are summarized at the end of the chapter.

5.2 Compound-choking theory

5.2.1 Assumptions

The compound-choking theory of Bernstein et al. [14] has been derived from the concept of a one-dimensional compound-compressible nozzle flow, illustrated in Fig. 5.1. This theory is based on the following

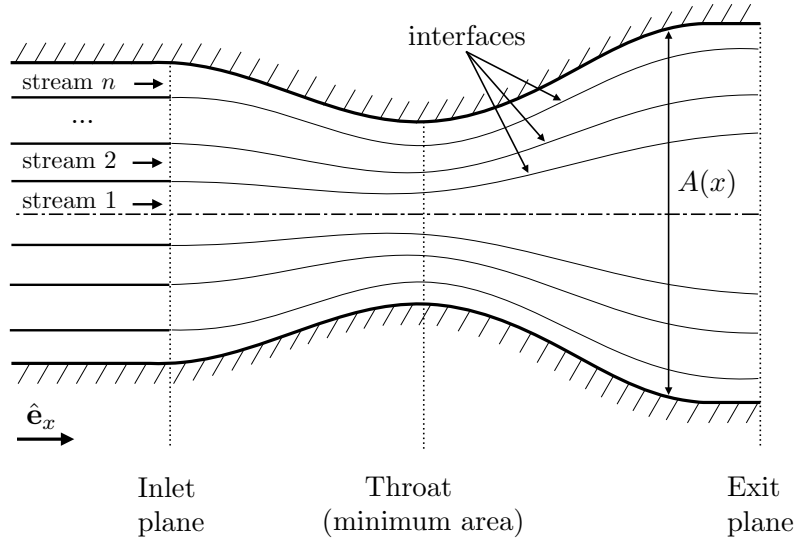


Figure 5.1: Schematic drawing of the compound-compressible nozzle flow (adapted from [14]).

assumptions:

1. the flow in each stream is one-dimensional, steady and isentropic;

2. the static pressure can vary only along the nozzle, implying that it cannot change from stream-to-stream across the nozzle;
3. each fluid is a perfect gas with constant thermodynamic properties.

Note that mixing and real gas effects are thus ignored in the compound-choking theory of Bernstein et al. [14]. Although less explicitly stated, the pioneering contribution of Pearson et al. [65] essentially relied on the aforementioned assumptions. In what follows, in addition to these assumptions, all the streams are assumed to have identical thermodynamic properties, namely equal specific heat ratios and specific gas constants, noted γ and R , respectively. As a consequence, in the present analysis, the streams differ from each other only with regard to their reservoir conditions. The general case of perfect gases with different thermodynamic properties is presented in [14].

5.2.2 Compound-choking condition

By deriving the velocity of a weak pressure disturbance – a so-called compound wave – Bernstein et al. arrived at the conclusion that the *compound-flow indicator*, $\beta(x)$, defined as

$$\beta(x) = \sum_{i=1}^n \frac{A_i(x)}{\gamma} \left(\frac{1}{M_i^2(x)} - 1 \right) , \quad (5.1)$$

is the key indicator to determine the *compound-flow regime*, i.e. the nature of the flow [14]. In Eq. (5.1), $A_i(x)$ and $M_i(x)$ represent the local flow area and the local Mach number of the i th stream, respectively. More precisely, it is shown in [14] that the velocity in the upstream direction of a compound wave must always have the same sign as the compound flow indicator, β . Hence, $\beta > 0$, $\beta = 0$ and $\beta < 0$ corresponds to the so-called *compound-subsonic*, *compound-sonic* and *compound-supersonic* regime, respectively. The demonstration of this result may be found in Appendix A.

Actually, the compound-choking theory of Bernstein et al. [14] considers an aerodynamic-based choking just as Fabri and Siestrunk [33], but based on another criterion for the blockage of the pressure waves. While, in the Fabri-choking theory, the focus is put on the secondary stream only, the compound-choking theory takes both streams under consideration to analyze the choking process, as the combination of both streams may behave like a sonic stream [47].

In addition, Bernstein et al. also demonstrated that the compound-choking of the flow can only occur at the nozzle minimum area, i.e. at the location where $A(x)$ is minimum and equal to A_{throat} , for it is there only that β can equal zero [14]. The interested reader is referred to the original publication for more details in deriving these results.

A closer inspection of Eq. (5.1) leads to the conclusion that, when a compound flow is choked ($\beta = 0$), the individual stream Mach numbers at the throat will not necessarily be equal to one [14]. More importantly, for flows involving two streams at different stagnation pressures, one stream must be supersonic, yielding $1/M_i^2 < 1$ in Eq. (5.1), while the other must be subsonic, i.e. $1/M_i^2 > 1$, to have $\beta = 0$. As the static pressure is assumed to be the same in both streams, the supersonic stream will correspond to the fluid with the greater total pressure.

As proposed by Hedges et al. [43], when all the streams have the same thermodynamic properties, it is convenient to define an equivalent Mach number, $M_{eq}(x)$, such that

$$M_{eq}(x) = \left(\gamma \frac{\beta(x)}{A(x)} + 1 \right)^{-1/2}, \quad (5.2)$$

where A is the local total flow area (see Fig. 5.1). This equivalent Mach number offers the advantage of being analogous with the Mach number in single-stream flows with regard to its interpretation. Indeed, M_{eq} will be unity for compound-sonic flow ($\beta = 0$), less than unity for compound-subsonic flow ($\beta > 0$) and greater than unity for compound-supersonic flow ($\beta < 0$) [43].

5.2.3 Computational procedure for two-stream compound-compressible nozzle flows

Another achievement of the work of Bernstein et al. consists in proposing a computational procedure in the case of two-stream convergent-divergent nozzles [14]. First, under the assumptions reported in Section 5.2.1, the mass flow rate of the i th stream may be written as a function of the local pressure $p(x)$ and the local flow area $A_i(x)$ only

$$\dot{m}_i = \frac{A_i(x)p_{0i}}{\sqrt{T_{0i}}} \left(\frac{p(x)}{p_{0i}} \right)^{\frac{1}{\gamma}} \sqrt{\frac{2}{R} \left(\frac{\gamma}{\gamma-1} \right) \left[1 - \left(\frac{p(x)}{p_{0i}} \right)^{\frac{\gamma-1}{\gamma}} \right]} \quad (i = 1, 2). \quad (5.3)$$

The derivation of this equality may be found in classical textbooks like [73] or [92]. Hence, the geometrical compatibility condition, i.e. $A_1(x) +$

$A_2(x) = A(x)$, may be written at any point in the nozzle as¹

$$\begin{aligned} \frac{\dot{m}_2}{\dot{m}_1} \sqrt{\frac{T_{02}}{T_{01}}} = & \left\{ \frac{A(x)}{A_1^*} \left[\left(\frac{2}{\gamma-1} \right) \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma+1}{\gamma-1}} \right]^{1/2} \right. \\ & \left. - \left(\frac{p_{01}}{p(x)} \right)^{1/\gamma} \left[1 - \left(\frac{p(x)}{p_{01}} \right)^{\frac{\gamma-1}{\gamma}} \right]^{-1/2} \right\} \\ & \times \left[1 - \left(\frac{p(x)}{p_{02}} \right)^{\frac{\gamma-1}{\gamma}} \right]^{1/2} \left(\frac{p_{02}}{p_{01}} \right) \left(\frac{p(x)}{p_{02}} \right)^{1/\gamma}, \quad (5.4) \end{aligned}$$

where

$$A_1^* = \frac{\dot{m}_1 \sqrt{T_{01}}}{p_{01}} \sqrt{\frac{R}{\gamma} \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma+1}{\gamma-1}}}. \quad (5.5)$$

Under choked flow conditions, a second equation arises from the fact that β , defined in Eq. (5.1), must fall to zero at the nozzle geometric throat (compound-sonic condition). By using the relation between the Mach number and the ratio of total to static pressures of the i th stream for an isentropic flow, i.e.

$$M_i^2(x) = \frac{2}{\gamma-1} \left[\left(\frac{p_{0i}}{p(x)} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right] \quad (i = 1, 2), \quad (5.6)$$

the condition $\beta = 0$ may be rewritten as

$$\begin{aligned} \frac{\dot{m}_2}{\dot{m}_1} \sqrt{\frac{T_{02}}{T_{01}}} = & \left(\frac{p_{02}}{p_{01}} \right)^{\frac{\gamma-1}{\gamma}} \frac{\left\{ \frac{\gamma-1}{2} \left[\left(\frac{p_{eq}^*}{p_{01}} \right)^{\frac{1-\gamma}{\gamma}} - 1 \right]^{-1} - 1 \right\} \left[1 - \left(\frac{p_{eq}^*}{p_{02}} \right)^{\frac{\gamma-1}{\gamma}} \right]^{1/2}}{\left\{ 1 - \frac{\gamma-1}{2} \left[\left(\frac{p_{eq}^*}{p_{02}} \right)^{\frac{1-\gamma}{\gamma}} - 1 \right]^{-1} \right\} \left[1 - \left(\frac{p_{eq}^*}{p_{01}} \right)^{\frac{\gamma-1}{\gamma}} \right]^{1/2}}, \quad (5.7) \end{aligned}$$

where p_{eq}^* represents the static pressure corresponding to the compound-sonic regime ($\beta = 0$). The four dimensionless ratios $\dot{m}_2 \sqrt{T_{02}} / \dot{m}_1 \sqrt{T_{01}}$, p_{01}/p_{02} , A_{throat}/A_1^* and p_{eq}^*/p_{02} are thus the fundamental dimensionless quantities in the compound-choking theory. As explained in [14], if two of these ratios are known, solving Eqs. 5.4 and 5.7 will provide the value of the two others under choked flow conditions, for any value of γ .

¹Please note that the minus sign in the '-1/2' exponent of the first line in Eq. (5.4) is missing in the original paper of [14], which is probably a mere oversight.

5.2.4 Prediction of the entrainment ratio in supersonic ejectors with the compound-choking theory

By identifying the flow in supersonic ejectors as a two-stream compound compressible nozzle flow, the computational procedure described above may be used to predict the on-design entrainment ratio for known reservoir conditions and a given geometrical configuration. Indeed, if the flow is assumed to be isentropic in the primary nozzle, A_1^* will be equal to the area of the throat of the primary nozzle, A_t . In addition, the cross-section of the mixing duct of a supersonic ejector, A_m , may be identified as the geometrical throat of the compound-compressible nozzle flow considered in [14]. Hence, for specified A_m/A_t , p_{01}/p_{02} , T_{01}/T_{02} and γ , the simultaneous resolution of Eqs. (5.4) and (5.7) provides a certain entrainment ratio for choked conditions according to the compound-choking theory. The value of p_{eq}^*/p_{02} is the second output of the procedure.

5.3 Numerical evidence of the compound-choking

5.3.1 Sonic line and dividing streamline

In order to highlight the choking process, the analysis of the numerical results is based on the method proposed in Chapter 4. It essentially consists in bringing out the region of the ejector where the secondary stream reaches sonic velocities by constructing two fundamental lines in the ejector: the *sonic line* and the *dividing streamline*. The interpretation of both lines is reminded in Fig. 5.2. The sonic line, defined as the iso-contour of Mach number unity, allows to make a distinction between the subsonic and the supersonic regions of the flow [10, 45], whereas the dividing streamline aims at separating the primary and the secondary streams. This line is defined in Chapter 3 as the streamline constructed from the velocity vector field and passing through the point located just above the tip of the primary nozzle. For 2D flows, this streamline actually represents the surface of a stream tube surrounding the primary stream. Since, by construction, there is no average mass flux passing through the surface of a stream tube, the dividing streamline may be interpreted as a virtual separation between the primary and the secondary streams all along the ejector.

Figure 5.3 shows the sonic line and the dividing streamline for different values of the back-pressure. The reader is reminded that all these results have been obtained with the same reservoir conditions (cfr. Section 2.3.6). For improved readability, the ejector is depicted

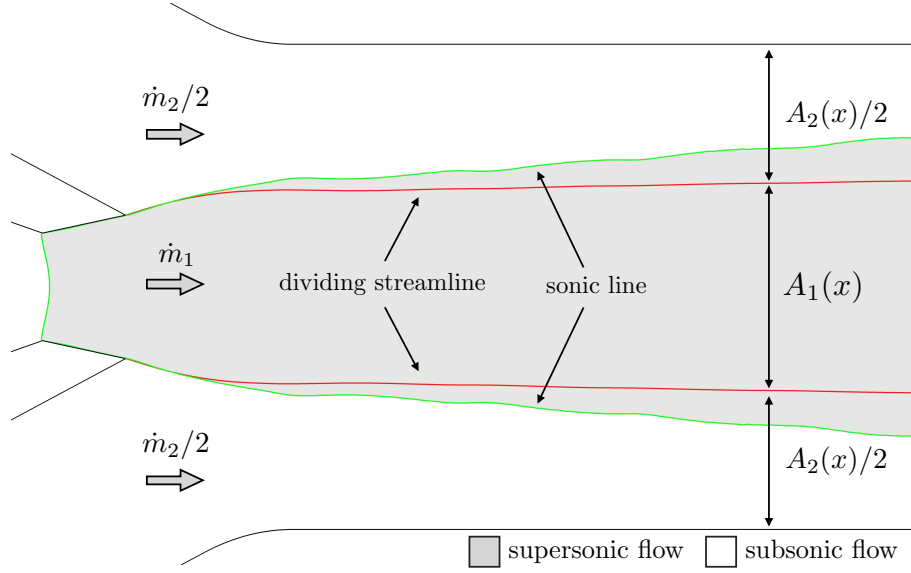


Figure 5.2: Interpretation of the sonic line and the dividing streamline. For improved readability, the ejector is depicted with a twofold stretching in the vertical direction.

with a twofold stretching in the vertical direction in Fig. 5.3. As may be observed in this figure, for on-design operating conditions, there is no sudden transition from subsonic to supersonic velocities in the secondary stream as the sonic line does not abruptly penetrate it. For cases a and b, the sonic line progressively deviates from the dividing streamline starting from the exit of the primary nozzle and gets very close to the wall near the entry of the diffuser. Hence, for these cases, the secondary stream gradually reaches supersonic velocities by being accelerated through the shear layer. For case e, the ejector operates in the off-design regime and the main portion of the secondary stream logically does not reach supersonic velocities (see Fig. 5.3(e)).

More interestingly, for cases c and d, a significant part of the secondary stream remains subsonic within the entire ejector, even though these operating conditions lie in the on-design regime as indicated by the characteristic curve corresponding to $p_{01}/p_{02} = 5.13$ shown in Fig. 2.9 (p. 33). The following analysis will give more insights into the choking process at play.

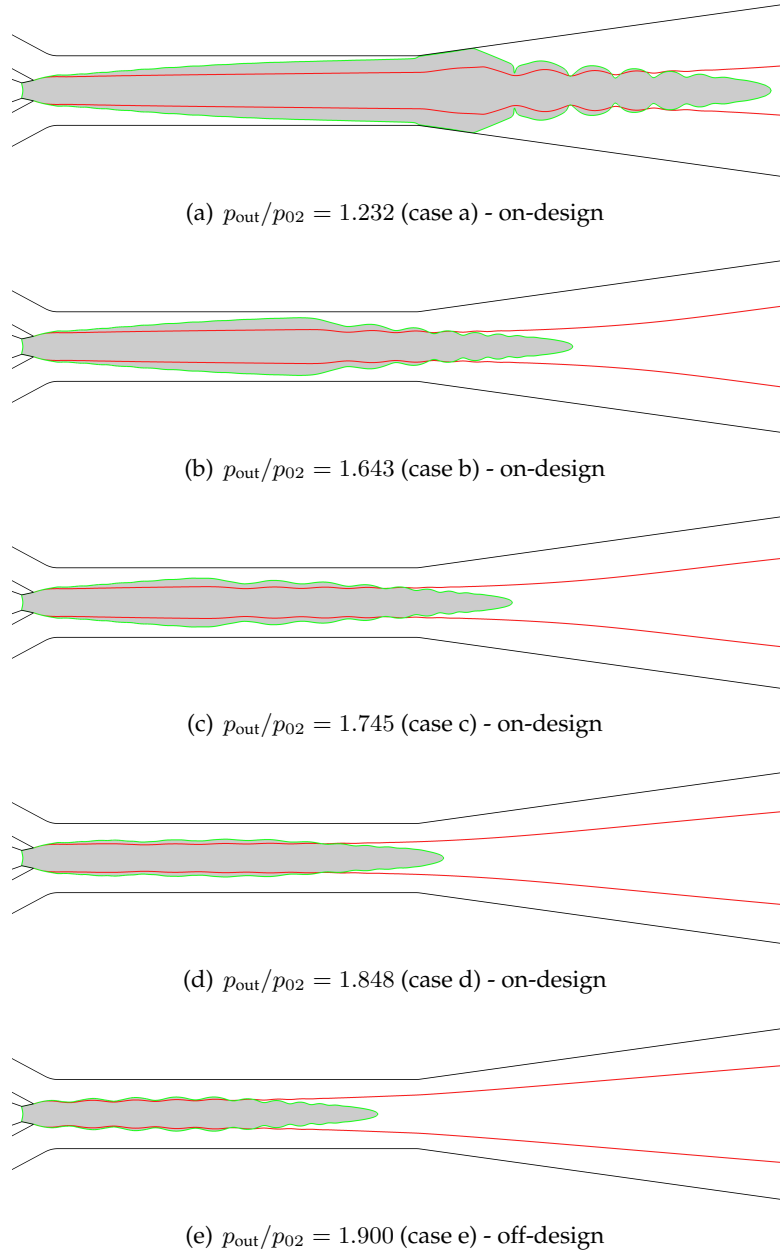


Figure 5.3: Sonic line (in green) and dividing streamline (in red) for different values of the back-pressure – $p_{01}/p_{02} = 5.13$. The supersonic region is coloured in grey.

5.3.2 Wall static pressure

The profile of the static pressure at the wall, $\bar{p}_{\text{wall}}$, between the nozzle exit plane and the end of the mixing duct is shown in Fig. 5.4, for the five values of the back-pressure considered above. The significant drop in the static pressure between the nozzle exit and the entry of the mixing duct reflects the contraction and the related acceleration of the secondary stream due to the expansion of the primary stream. Nevertheless, apart from case a, the static pressure does not fall low enough for the secondary stream to reach sonic velocities, at least through an isentropic process, for which it should fall below the critical pressure, p^* , defined in Eq. (4.1). This particular value is highlighted by a dashed horizontal line in Fig. 5.4. In contrast, for all the on-design cases, the pressure drop experienced by the secondary stream is consistent with the compound-choking theory as $\bar{p}_{\text{wall}}$ reaches the theoretical value corresponding to the compound-sonic regime, p_{eq}^* , calculated through the procedure described in Section 5.2.4 (see the dash-dotted horizontal line in Fig. 5.4).

It clearly appears in Fig. 5.4 that the flow in the ejector is identical up to a certain axial location for the four on-design configurations (cases a-d), although not easily detected in Fig. 5.3. Indeed, the four curves corresponding to the cases a-d are actually superimposed upstream of the mixing duct entry, and start deviating from each other at different positions downstream in the mixing duct. This is consistent with the fact that the flow is choked for all these operating points. For case d, which corresponds to a back-pressure near the critical value, the pressure profile starts deviating from the three other curves shortly after the entry of the mixing duct. Results shown in Fig. 5.4 thus imply that the choking process, i.e. the blockage of the secondary mass flow rate, takes place in the vicinity of the entry of the mixing duct, where the secondary stream is almost entirely subsonic (see Fig. 5.3 (a) to (d)). Hence, in the present situation, the sonic line is not a good criterion to assess if the flow is choked within the ejector.

These observations are consistent with the compound-choking theory which states that the flow may be choked with a subsonic secondary stream if the primary stream is supersonic. To further confirm that the choking phenomenology is well described by this theory, an equivalent Mach number is calculated all along the ejector by post-processing the numerical results in the next section.

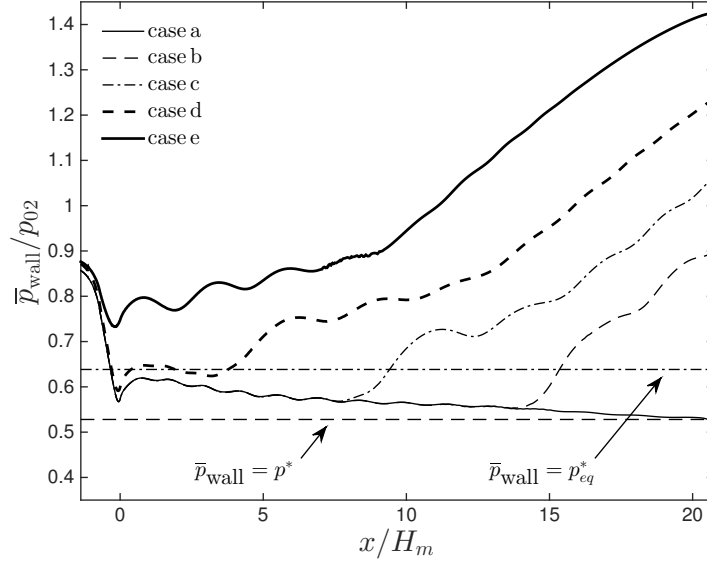


Figure 5.4: Static pressure profile for different values of the back-pressure – $p_{01}/p_{02} = 5.13$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

5.3.3 Equivalent Mach number

The equivalent Mach number is obtained through Eq. (5.2) by post-processing the numerical results. For the calculation of $\beta(x)$, Eq. (5.1) is used, where the local Mach number of the i th stream is computed according to the definition given in Eq. (4.2) (p. 86), i.e.

$$M_i(x) = \frac{U_i(x)}{\sqrt{\gamma R T_i(x)}} \quad (i = 1, 2), \quad (5.8)$$

with $U_i(x)$ a mean velocity

$$U_i(x) = \frac{\dot{m}_i}{\iint_{A_i(x)} \bar{\rho} ds} \quad (i = 1, 2), \quad (5.9)$$

and $T_i(x)$ a mean temperature

$$T_i(x) = \frac{1}{A_i(x)} \iint_{A_i(x)} \hat{T} ds \quad (i = 1, 2). \quad (5.10)$$

In these equations, $A_i(x)$ represents the cross-section of the i th stream, delimited by the dividing streamline (see Fig. 5.2, p. 103).

The profile of $M_{eq}(x)$ is shown in Fig. 5.5 for the five cases considered above, from the exit of the primary nozzle to the entry of the diffuser. For all the on-design cases (cases a-d), the equivalent Mach number reaches unity. In addition, the first location where $M_{eq} = 1$ consistently lies close to the entry of the mixing duct, in the region of the flow that is common to the four operating points. In contrast, the equivalent Mach number logically remains lower than one for the case e which is off-design. Hence, the profiles of the equivalent Mach number are consistent with the compound-choking theory.

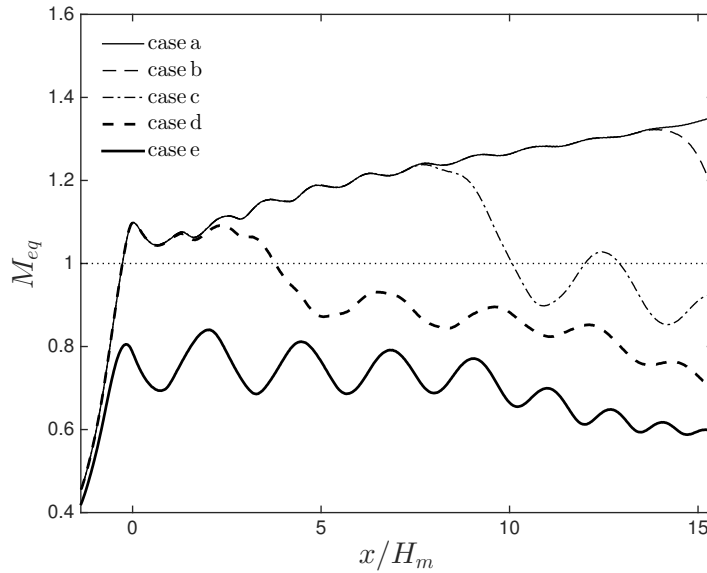


Figure 5.5: Equivalent Mach number obtained in the post-processing phase of the CFD calculations for different values of the back-pressure – $p_{01}/p_{02} = 5.13$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

Note that the position where the equivalent Mach number reaches unity for on-design cases, noted x_c , corresponds to $x_c/H_m \simeq -0.27$. It is thus not exactly located at the geometrical throat of the ejector but rather a small distance upstream of the mixing duct entry. Yet the total flow area at $x = x_c$ is very close to the mixing duct cross-section. The fact that $M_{eq} = 1$ before the entry of the mixing duct is probably due to non-isentropic effects which are not taken into account in the compound-choking theory, as well as the assumption of a uniform pressure profile in the entire cross-section.

For the present ejector geometry and operating conditions, the phenomenology of the choking thus closely corresponds to the compound-choking theory introduced by Bernstein et al. [14]. Even though a sonic section in the core of the secondary stream (Fabri-pattern) has been evidenced in Chapter 4 for other reservoir conditions, these two observations are compatible. Indeed, even when the Fabri-pattern is present, the compound-choking may be responsible for the entrainment limitation. As mentioned in the introduction, a sonic line pattern similar to the one obtained here for case (a) may be found in several numerical studies [28, 29, 45, 49, 60, 69], while the Fabri-pattern has never been reported in the literature. This fact corroborates the idea that the compound-choking theory is well suited to model the choking mechanism in supersonic ejectors and that the Fabri-pattern is unusual in common ejector configurations. The following analysis brings a second argument in favour of the compound-choking theory as the explanation of the entrainment limitation in supersonic ejectors.

5.4 Comparison of the compound-choking theory with experimental data

5.4.1 Prediction of the entrainment ratio for various ejector configurations

The experimental data of Huang et al. [48] using R-141b as refrigerant often serve for the validation of simplified models [21, 30, 89]. In their experiment, eleven axisymmetric ejectors were tested by combining two distinct primary nozzles with eight different secondary nozzles. The two primary nozzles differed from each other with regard to their throat and exit diameters, whereas the geometrical differences between the secondary nozzles lied in the diameter of the constant area region and the inlet converging angle. Considering all the configurations reported in [48], A_m/A_t ranges from 6.44 to 10.64. In what follows, the ejector geometry is characterized by two capital letters, referring to the primary nozzle (A or E) and the secondary nozzle (A to H), respectively, in line with Huang et al. [48]. The interested reader is referred to the original article for more information regarding the geometry of the different ejectors.

The reservoir conditions were also varied in the experimental investigation of Huang et al. [48]: the primary stagnation pressure was set to four different values, ranging from 4 to 6.04 [bar], combined with two distinct secondary stagnation pressures: 0.4 and 0.47 [bar]. The ra-

tio of stagnation pressures at reservoir conditions, p_{01}/p_{02} , thus varies between 8.51 and 15.1. At both inlets, the stagnation temperature corresponded to the saturated temperature of the respective stagnation pressure since the refrigerant was in a saturated vapor state.

In total, 39 different configurations were reported in [48], specifying for each case the on-design entrainment ratio as well as the critical back-pressure. Here, only the information regarding the entrainment ratio is used in order to assess the ability of the compound-choking theory to predict the entrainment ratio in on-design conditions, according to the method described in Section 5.2.4. As a reminder, the four dimensionless quantities A_m/A_t , p_{01}/p_{02} , T_{01}/T_{02} and γ are the inputs of the procedure which yields ER and p_{eq}^* as outputs. The specific heat ratio of R-141b is taken as $\gamma = 1.135$, in accordance with Zhu et al. [89] who used this value to test their simplified model on the experimental data reported in [48].

The prediction of the entrainment ratio with the compound-choking theory is reported in Table 5.1 (p. 111) for the 39 cases of Huang et al. [48] and compared to the experimental results. As may be noted from Table 5.1, the maximum relative deviation of the model with respect to the experimental data regarding the entrainment ratio is less than 15%. In addition, considering the 39 cases, the arithmetic mean of the relative error is positive and close to 4%, implying that the compound-choking theory tends to slightly overestimate the experimental entrainment ratio. This may be partially due to real gas effects which are neglected by the model as the compound-choking theory is currently limited to ideal gas. Accounting for real gas effects when the working fluid is R141b generally tends to lower the predicted entrainment ratio [30] and could thus further enhance the performance of the compound-choking theory.

To ease the comparison between the model and the experiment, the entrainment ratio predicted by the compound-choking theory is plotted against the experimental data in Fig. 5.6. As can be seen, most predictions are in the $\pm 10\%$ range when compared to the experimental data. As mentioned in Section 5.2.1, the hypothesis of an isentropic process implies that the compound-choking theory neglects the mixing. Of course, in real applications, transfers between the primary and the secondary streams occur through the shear layer, starting from the nozzle exit plane. These transfers probably slightly enhance the entrainment ratio of the ejector. This may cause the compound-choking theory to underestimate the entrainment ratio, in particular when the latter is low. As may be observed in Fig. 5.6, all cases in which the theory underestimates the entrainment ratio (i.e. negative error) cor-

respond indeed to relatively low values of the entrainment ratio. The no-mixing assumption is thus the most likely explanation as to why the theoretical prediction of the entrainment ratio may underestimate the experimental value.

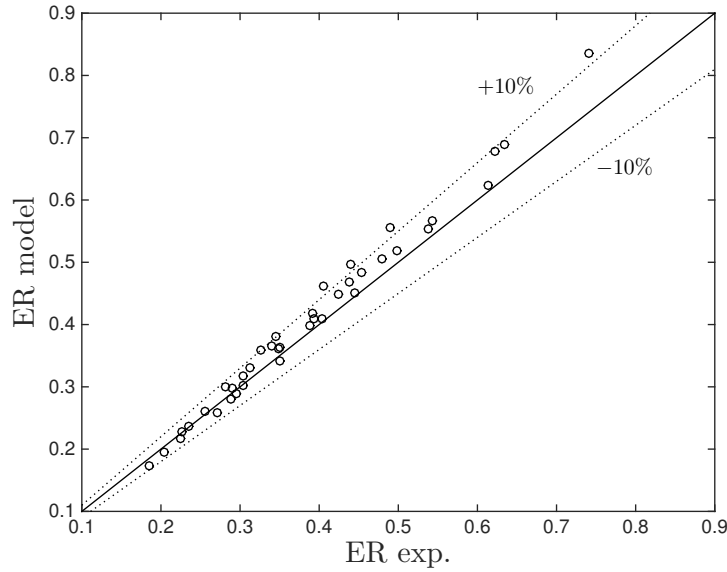


Figure 5.6: Comparison of the compound-choking theory with the experimental data reported in [48].

5.4.2 Comparison with state-of-the-art simplified models

The good performances of the compound-choking theory for the prediction of the entrainment ratio in on-design conditions may be further assessed through a comparison with state-of-the-art 1D models based on the Fabri-choking assumption. In order to make the comparison possible, four models validated against the experimental data of Huang et al. [48] have been considered here: [21, 30, 48, 89]. Note that these models are more sophisticated than the procedure considered here. More precisely, the model of del Valle et al. [30] uses a perturbation procedure of linearized and axisymmetric supersonic flow to model 2D effects, whereas the models of Chen et al. [21], Huang et al. [48] and Zhu et al. [89] rely on fitting coefficients. Furthermore, real gas effects are taken into account in Chen et al. [21] and del Valle et al. [30]. The comparison, based on the error distribution of the different models, is shown in Fig. 5.7.

Case	Ejector	A_m/A_t	p_{01} [bar] (T_{sat} [°C])	p_{02} [bar] (T_{sat} [°C])	ER (exp)	ER (model)	Error [%]
1	EH	10.64	6.04 (95)	0.4 (8)	0.4377	0.4691	7.16
2	EF	9.83	6.04 (95)	0.4 (8)	0.3937	0.4103	4.21
3	AD	9.41	6.04 (95)	0.4 (8)	0.3457	0.3800	9.92
4	EE	9.17	6.04 (95)	0.4 (8)	0.3505	0.3628	3.50
5	AC	8.28	6.04 (95)	0.4 (8)	0.2814	0.2994	6.38
6	ED	8.25	6.04 (95)	0.4 (8)	0.2902	0.2972	2.43
7	EC	7.26	6.04 (95)	0.4 (8)	0.2273	0.2280	0.32
8	AG	7.73	6.04 (95)	0.4 (8)	0.2552	0.2607	2.15
9	EG	6.77	6.04 (95)	0.4 (8)	0.2043	0.1945	-4.80
10	AA	6.44	6.04 (95)	0.4 (8)	0.1859	0.1722	-7.35
11	AD	9.41	5.38 (90)	0.4 (8)	0.4446	0.4513	1.50
12	AC	8.28	5.38 (90)	0.4 (8)	0.3488	0.3604	3.32
13	AG	7.73	5.38 (90)	0.4 (8)	0.3040	0.3166	4.16
14	AB	6.99	5.38 (90)	0.4 (8)	0.2718	0.2586	-4.86
15	AA	6.44	5.38 (90)	0.4 (8)	0.2246	0.2162	-3.76
16	AD	9.41	4.65 (84)	0.4 (8)	0.5387	0.5545	2.93
17	AC	8.28	4.65 (84)	0.4 (8)	0.4241	0.4491	5.88
18	AG	7.73	4.65 (84)	0.4 (8)	0.3883	0.3982	2.55
19	AB	6.99	4.65 (84)	0.4 (8)	0.3117	0.3305	6.02
20	AA	6.44	4.65 (84)	0.4 (8)	0.2880	0.2808	-2.52
21	AD	9.41	4.00 (78)	0.4 (8)	0.6227	0.6788	9.01
22	AC	8.28	4.00 (78)	0.4 (8)	0.4889	0.5562	13.77
23	AG	7.73	4.00 (78)	0.4 (8)	0.4393	0.4970	13.13
24	AB	6.99	4.00 (78)	0.4 (8)	0.3922	0.4179	6.54
25	AA	6.44	4.00 (78)	0.4 (8)	0.3257	0.3596	10.42
26	EF	9.83	6.04 (95)	0.47 (12)	0.4989	0.5187	3.98
27	EE	9.17	6.04 (95)	0.47 (12)	0.4048	0.4626	14.28
28	AD	9.41	6.04 (95)	0.47 (12)	0.4541	0.4830	6.36
29	AG	7.73	6.04 (95)	0.47 (12)	0.3503	0.3416	-2.50
30	EC	7.26	6.04 (95)	0.47 (12)	0.3040	0.3026	-0.45
31	AA	6.44	6.04 (95)	0.47 (12)	0.2350	0.2357	0.32
32	AD	9.41	5.38 (90)	0.47 (12)	0.5422	0.5676	4.68
33	AG	7.73	5.38 (90)	0.47 (12)	0.4034	0.4085	1.26
34	AA	6.44	5.38 (90)	0.47 (12)	0.2946	0.2889	-1.95
35	AD	9.41	4.65 (84)	0.47 (12)	0.6350	0.6898	8.63
36	AG	7.73	4.65 (84)	0.47 (12)	0.4790	0.5056	5.56
37	AA	6.44	4.65 (84)	0.47 (12)	0.3398	0.3665	7.85
38	AD	9.41	4.00 (78)	0.47 (12)	0.7412	0.8366	12.87
39	AG	7.73	4.00 (78)	0.47 (12)	0.6132	0.6228	1.57

Table 5.1: Prediction of the entrainment ratio with the compound-choking theory for the 39 experimental ejector configurations of Huang et al. [48]. Error=(model-exp)/exp.

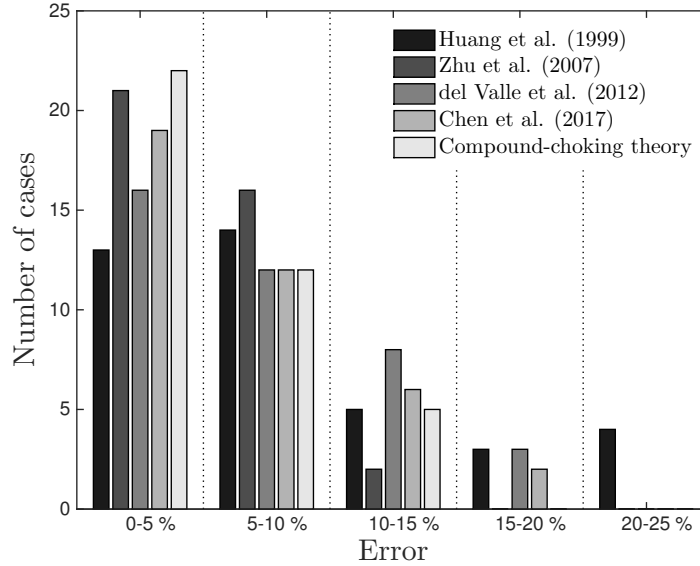


Figure 5.7: Comparison of the error distribution of the compound-choking theory with state-of-the-art 1D models for the prediction of the entrainment ratio based on the experimental data of Huang et al. [48].

The model based on the compound-choking theory is found to yield the largest number of cases with an error lower than 5% (22 cases out of 39). Furthermore, with the model of Zhu et al. [89], the present procedure is the sole whose maximum error is lower than 15%; the other models obtaining errors in the range of 15-20% or more. However, compared to the model developed by Zhu et al. [89], the compound-choking theory is based on more robust physical arguments. Indeed, in [89], the secondary stream is assumed to reach sonic velocities with a static pressure equal to its inlet pressure, which is not consistent with the fact that the secondary stream is significantly accelerated in the suction chamber. Based on Fig. 5.7, it may be concluded that the compound choking theory in its simplest form generally performs better than more sophisticated 1D models (based on the Fabri-choking assumption) to predict the entrainment ratio in the on-design regime.

5.5 Numerical assessment of the limitations of the compound-choking theory

As shown in Section 5.4, most predictions of the compound-choking theory for the on-design entrainment ratio are in the $\pm 10\%$ range when compared to the experimental data of Huang et al. [48]. The following analysis aims at shedding light on the main sources of disagreement between the compound-choking theory and the experiment through a numerical investigation, assuming that CFD satisfactorily models the real flow. However, this study is not performed for the ejector configurations reported in [48]. Indeed, in order to benefit from the numerical setup used throughout this work, the ejector geometry described in Section 2.2.3 serves as the basis for assessing the limitations of the compound-choking theory by means of CFD.

In what follows, the focus is put on the impacts of the operating conditions and of the geometry on the ability of the compound-choking theory to predict the on-design entrainment ratio. Regarding the reservoir conditions, only changes in the primary stagnation pressure, p_{01} , are considered. In addition, the effects of two different geometrical parameters are analyzed, in line with the experimental investigation of Huang et al. [48]: the mixing duct cross-section, A_m , and the inlet converging angle of the secondary nozzle, θ (see Fig. 2.3, p.19). Note that the present analysis essentially serves as a preliminary examination and does not pretend to be an exhaustive parametric study. Hence, when examining the impact of one of the three quantities mentioned above, namely p_{01} , A_m and θ , the two other parameters are fixed and correspond to the configuration of reference, characterized by $p_{01}/p_{02} = 5.13$, $A_m/A_t = 4.53$ and $\theta = 15^\circ$.

5.5.1 Impact of the primary stagnation pressure

First of all, the impact of the primary stagnation pressure, p_{01} , on the capacity of the compound-choking theory to predict the on-design entrainment ratio is investigated. Five cases are considered here, characterized by $p_{01}/p_{02} = 4.11, 5.13$ (reference case), 6.66, 10.0 and 20.0.

The comparison between the compound-choking theory and the CFD regarding the prediction of the entrainment ratio in the on-design regime is shown in Fig. 5.8 as a function of p_{01}/p_{02} . It appears that, for the geometry considered here, the agreement between the compound-choking theory and the numerical simulations varies with the reservoir conditions. The relative difference between both predictions is very small for the lowest values of the primary stagnation pressure, corre-

sponding to $p_{01}/p_{02} = 4.11$ and 5.13 . As p_{01} increases, the entrainment ratio diminishes, and the gap between the theory and the CFD increases.

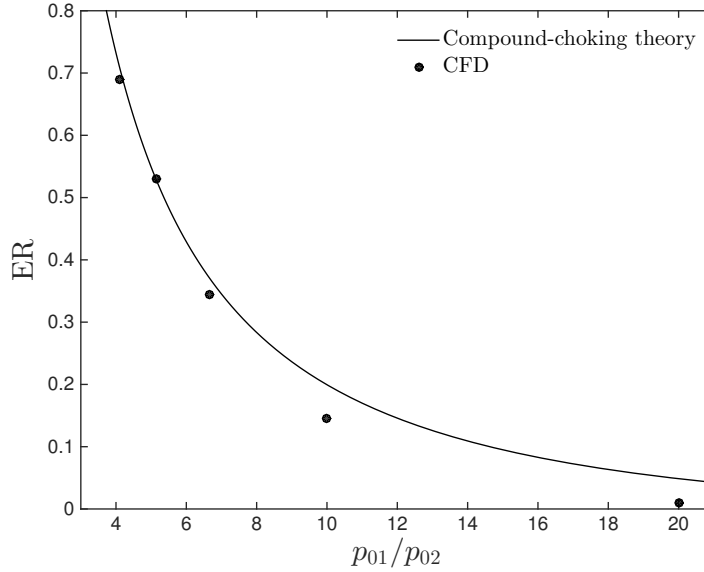


Figure 5.8: Comparison of the compound-choking theory with CFD for the prediction of the on-design entrainment ratio for different values of $p_{01}/p_{02} - A_m/A_t = 4.53 - \theta = 15^\circ$.

In order to understand what causes the compound-choking theory to overestimate the entrainment ratio obtained by means of CFD for certain reservoir conditions, one should consider the differences between both approaches regarding the assumptions that are made. Compared to the numerical methods, the compound-choking theory neglects both the two-dimensional and the non-isentropic aspects of the flow (cfr. Section 5.2.1). Although viscous effects are unavoidable in real applications, it is likely that they do not play a significant role in the choking phenomenon. In other words, it is believed that the entrainment limitation is essentially an isentropic process in common ejector configurations. In that case, the main source of disagreement between the compound-choking theory and the numerical simulations regarding the prediction of the entrainment ratio should lie in the hypothesis of a one-dimensional flow made by the simplified theory. In particular, the compound-choking theory stipulates that the static pressure can vary only along the nozzle and that it cannot change from stream-

to-stream across the nozzle. In order to check this assumption, the profile of the mean static pressure, $\bar{p}$, is examined at the choking position, i.e. at $x = x_c$, by post-processing the numerical results. The reader is reminded that x_c is defined for an on-design operating point as the first axial position where the equivalent Mach number, M_{eq} , reaches unity (cfr. Section 5.3.3).

The profile of the mean static pressure at the choking position is shown in Fig. 5.9 for the different values of the primary stagnation pressures calculated by CFD. The black dot on each curve shows the location of the corresponding dividing streamline at the choking position. Note that, for improved readability, the case corresponding to $p_{01}/p_{02} = 20.0$ is not depicted in the figure. Indeed, as the entrainment ratio obtained by CFD for these reservoir conditions is very close to zero, the pressure profile is significantly different from the other operating points considered here and not really meaningful.

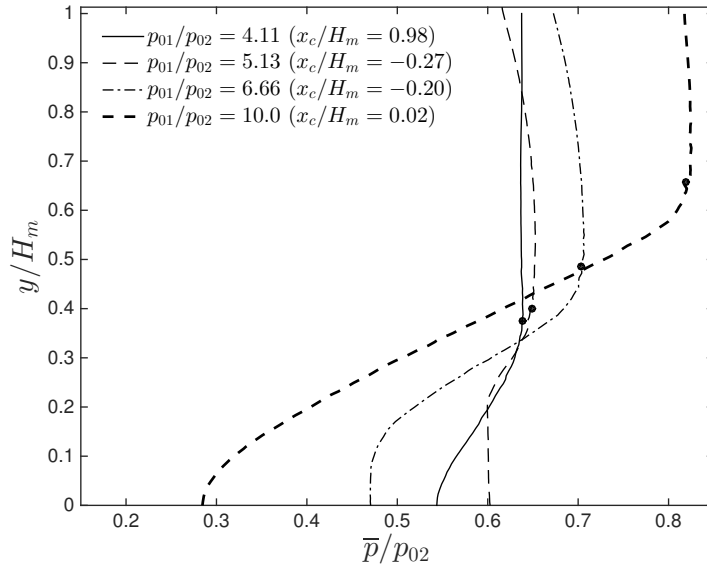


Figure 5.9: Static pressure profile at the choking position for different values of $p_{01}/p_{02} - A_m/A_t = 4.53 - \theta = 15^\circ$. The black dot on each curve shows the location of the corresponding dividing streamline. The centerline and the wall of the mixing duct correspond to $y/H_m = 0$ and $y/H_m = 1$, respectively.

Based on the results shown in Fig. 5.9, it seems that the gap between the compound-choking theory and the numerical simulations regarding the entrainment ratio is closely related to the assumption of a uni-

form static pressure profile made by the compound-choking theory. Indeed, the profile of $\bar{p}$ is far from being flat for the reservoir conditions that yield larger discrepancies regarding the entrainment ratio, namely $p_{01}/p_{02} = 6.66$ and 10.0 .

A close inspection of the curves shown in Fig. 5.9 reveals that departures from a uniform pressure are mainly located below the dividing streamline, thus in the region occupied on average by the primary stream. In contrast, the static pressure is nearly uniform in the secondary stream (region above the dividing streamline). This observation is consistent with the fact that the primary stream is supersonic at the choking position whereas the secondary stream is almost entirely subsonic. To further assess the fact that the pressure may be not uniform for a given axial position, the static pressure field at the entry of the mixing duct is shown in Fig. 5.10 for the case corresponding to $p_{01}/p_{02} = 10.0$. The two-dimensional aspect of the flow in the primary stream due to its supersonic nature is clearly visible. As may be observed, the choking position logically corresponds to an expansion zone in the primary stream, explaining why there is no discontinuity in the pressure profile shown in Fig. 5.9.

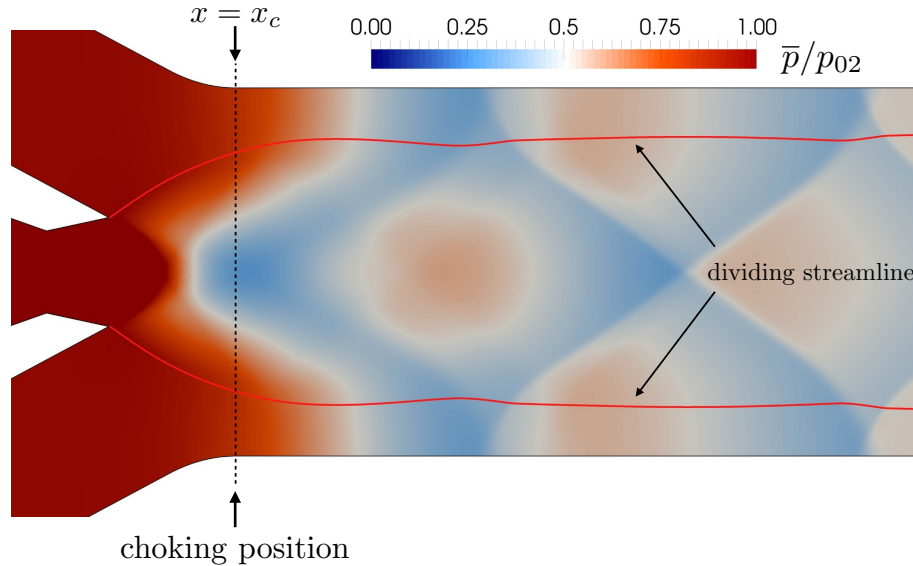


Figure 5.10: Static pressure field in the vicinity of the entry of the mixing duct for $p_{01}/p_{02} = 10.0$.

For the cases corresponding to $p_{01}/p_{02} = 5.13$ and 6.66 , the slight variation of the static pressure in the vertical direction above the di-

viding streamline attests that the secondary stream still experiences a deviation at the choking position. Indeed, for these reservoir conditions, M_{eq} reaches unity a small distance upstream of the entry of the mixing duct (i.e. $x_c/H_m < 0$) and a pressure gradient is still required to deflect the secondary stream.

5.5.2 Impact of the mixing duct cross-section

The focus is now put on the ability of the compound-choking theory to predict the on-design entrainment ratio for different mixing duct cross-sections. For this analysis, A_m is modified by changing the height of the mixing duct, H_m , only. The geometry of the primary nozzle is thus fixed and corresponds to the ejector described in Section 2.2.3 (p.18). Regarding the reservoir conditions and the inlet converging angle, they are identical for all the cases considered here and correspond to the configuration of reference, i.e. $p_{01}/p_{02} = 5.13$ and $\theta = 15^\circ$. Besides the reference geometry for which $A_m/A_t = 4.53$, four additional mixing duct geometries are investigated below, characterized by $A_m/A_t = 3.33$, 5.71, 6.69 and 9.05. Figure 5.11 shows a zoom on the mesh close to the exit of the primary nozzle for the different geometrical configurations.

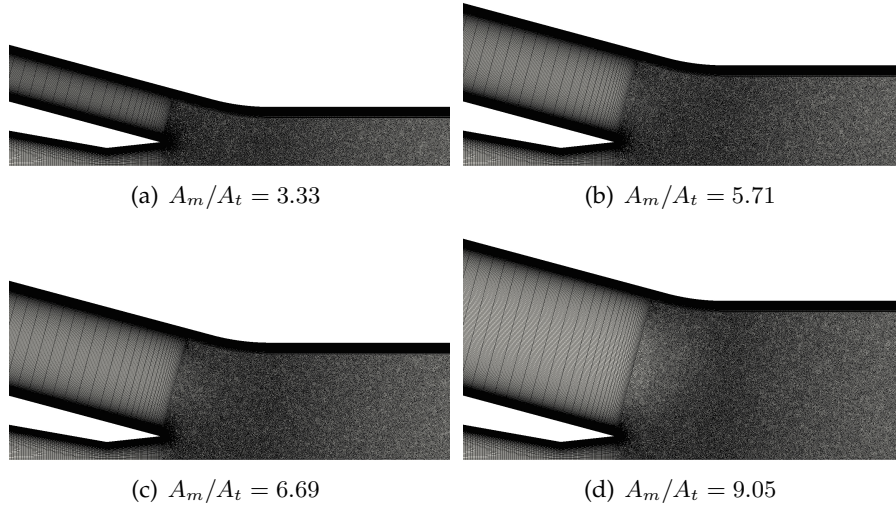


Figure 5.11: Zoom on the mesh close to the exit of the primary nozzle for the additional mixing duct geometries calculated by CFD. The reference case corresponds to $A_m/A_t = 4.53$ (not shown).

The predictions of the on-design entrainment ratio with the compound-choking theory and by means of CFD are shown in Fig. 5.12 as a func-

tion of A_m/A_t . The compound-choking theory is in very good agreement with the numerical simulations for the five geometrical configurations. Hence, for the reservoir conditions considered here, it seems that the mixing duct cross-section does not have a significant impact on the ability of the compound-choking theory to predict the on-design entrainment ratio.

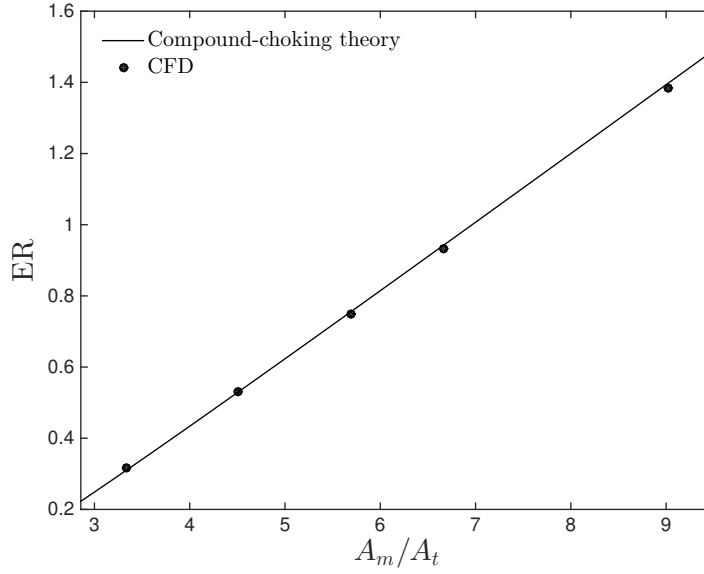


Figure 5.12: Comparison of the compound-choking theory with CFD for different values of $A_m/A_t - p_{01}/p_{02} = 5.13 - \theta = 15^\circ$.

5.5.3 Impact of the inlet converging angle

Finally, the impact of the inlet converging angle, θ , on the agreement between the compound-choking theory and the numerical simulations regarding the on-design entrainment ratio is investigated. For the present analysis, the reservoir conditions and the mixing duct cross-section are kept constant and correspond to the configuration of reference characterized by $p_{01}/p_{02} = 5.13$ and $A_m/A_t = 4.53$. Besides the initial geometry for which $\theta = 15^\circ$, four additional inlet converging angles are simulated by means of CFD : $\theta = 20^\circ, 8^\circ, 4^\circ$ and 0° . The corresponding meshes are shown in Fig. 5.13 with a zoom on the exit of the primary nozzle. The inner wall of the primary nozzle was preferably not impacted by θ in order to keep the primary flow unchanged between the

different simulations. Nevertheless, for the lowest values of θ , the computational domain in the converging part of the primary nozzle had to be shortened. The impact of this modification on the primary mass flow rate has been checked and was found to be negligible.

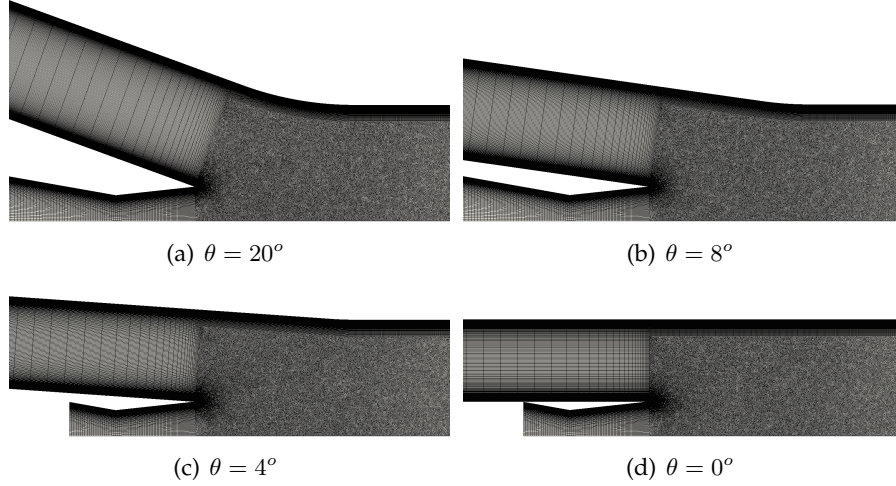


Figure 5.13: Zoom on the mesh close to the exit of the primary nozzle for the additional secondary inlet geometries calculated by CFD. The reference case corresponds to $\theta = 15^\circ$ (not shown).

The comparison between the compound-choking theory and the numerical simulations for the prediction of the on-design entrainment ratio is shown in Fig. 5.14 as a function of θ . As the inlet converging angle is not an input of the simplified model, it has logically no effect on the related prediction of the entrainment ratio. However, as may be seen in Fig. 5.14, θ impacts the entrainment ratio obtained by CFD. The agreement between the compound-choking theory and the numerical simulations is very good for large values of θ but the gap between the theory and the CFD significantly increases as θ decreases. The relative difference between both predictions is maximum for a secondary inlet aligned with the mixing duct, i.e. $\theta = 0$, and reaches almost 13%.

Here again, the assumption of a uniform pressure profile made by the compound-choking theory is the main reason for the disagreement with the CFD. This may be assessed in Fig. 5.15 showing, for the different configurations, the profile of the mean static pressure, $\bar{p}$, extracted from the numerical results at the related choking position, i.e. at $x = x_c$ ($M_{eq} = 1$). The location of the corresponding dividing streamline is indicated by a black dot on each curve. Assuming a uniform pressure

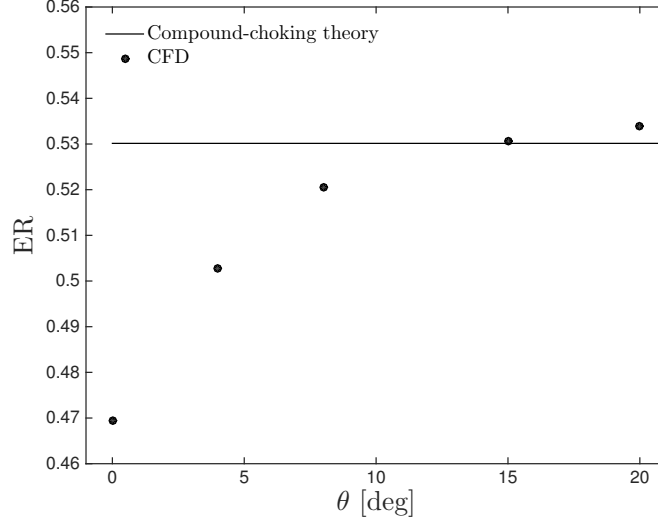


Figure 5.14: Comparison of the compound-choking theory with CFD for different values of $\theta - p_{01}/p_{02} = 5.13 - A_m/A_t = 4.52$.

profile in the entire cross-section becomes less and less valid as θ decreases, at least at the axial position corresponding to $M_{eq} = 1$. As it was the case when varying the primary stagnation pressure (cfr. Section 5.5.1), the profile of $\bar{p}$ is nearly uniform in the secondary stream, lying on average above the black dot, and its two-dimensional aspects are mainly located in the primary stream (region below the dividing streamline).

5.6 Conclusions

In this chapter, an in-depth analysis of the choking mechanism at the basis of the entrainment limitation in supersonic ejectors has been performed. First, for the same ejector geometry than in Chapter 4 but with other reservoir conditions, it was shown that the secondary stream does not reach sonic velocities in a aerodynamic throat as postulated by Fabri and Siestrunk [33]. Moreover, a significant part of the secondary stream remains subsonic within the entire ejector for some operating conditions lying in the on-design regime. Actually, for the ejector geometry and reservoir conditions under consideration, the choking process, i.e. the blockage of the secondary mass flow rate, takes place close to the entry of the mixing duct, at a position that is common to all on-

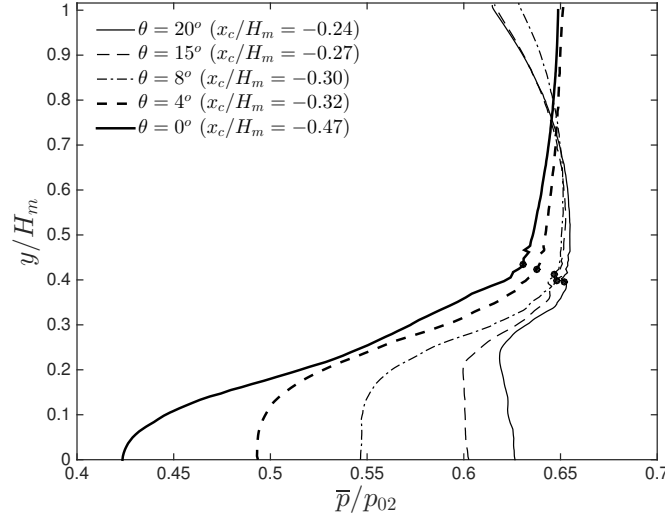


Figure 5.15: Static pressure profile at the choking position for different values of $\theta - p_{01}/p_{02} = 5.13 - A_m/A_t = 4.53$. The black dot on each curve shows the location of the corresponding dividing streamline. The centerline and the wall of the mixing duct correspond to $y/H_m = 0$ and $y/H_m = 1$, respectively.

design cases and where the secondary stream is almost entirely subsonic. The observed choking phenomenology was found to be properly described by a theory developed by Bernstein et al. in the 1960's and known as the compound-choking theory [14]. This theory states that a two-stream compound-compressible nozzle flow may be choked with a subsonic stream if the other one is supersonic, as the combination of the two streams may behave like a sonic stream [47]. The profile of the equivalent Mach number further corroborated the adequacy of the compound-choking theory to yield a physical explanation of the choking phenomenology highlighted with the numerical results. Hence, the sonic line is not a good criterion to assess if the flow is choked within a supersonic ejector.

Then, the ability of the compound-choking theory to predict the on-design entrainment ratio has been assessed against the extensive experimental data of Huang et al. [48]. This theory has been used in its simplest form, for known reservoir conditions and geometrical parameters, according to a methodology proposed by Bernstein et al. [14]. The computational procedure simply consists in simultaneously solving two algebraic equations. Most predictions of the compound-

choking theory are in the $\pm 10\%$ range when compared to the experimental data reported in [48]. Moreover, based on these experimental results, the compound-choking theory is shown to generally perform better than state-of-the-art 1D models relying on the Fabri-choking assumption for the prediction of the on-design entrainment ratio.

Finally, a numerical investigation aiming at shedding light on the limitations of the compound-choking theory has been performed by successively varying the primary stagnation pressure, the mixing duct cross-section and the inlet converging angle. Based on all the considered cases, the main cause of disagreement between the simplified model and the CFD regarding the on-design entrainment ratio seems to be the assumption of a uniform pressure profile made by the compound-choking theory. In general, departures from a uniform pressure profile were found to be mainly located in the supersonic primary stream.

CHAPTER 6

DEVELOPMENT OF AN UPGRADED EXPERIMENTAL SETUP AND COMPARISON WITH 2D AND 3D NUMERICAL SIMULATIONS

6.1 Introduction

As evidenced in the preceding chapters, numerical simulations are a valuable resource to gain insights into the local flow phenomena taking place in supersonic ejectors. In order to pave the way to further numerical and experimental investigations, an upgraded experimental installation aiming at improving the current facility has been designed. This chapter is devoted to a thorough comparison of 2D numerical results with the experimental data obtained with this novel experimental setup and to a preliminary investigation of 3D effects.

This chapter is structured as follows: first, the design of the new ejector is presented in Section 6.2, with special emphasis on the main differences with the current geometry. Experimental results obtained on this upgraded facility are then compared to 2D numerical simulations in Section 6.3. The comparison is performed both at the global and at the local scales and for different operating conditions. The potential sources of disagreement between the CFD and the experiment

are discussed, including 3D effects which are investigated numerically for one operating point in Section 6.4.

6.2 Upgraded experimental setup

The main difference between the upgraded experimental installation and the initial one lies in the primary nozzle geometry. Indeed, in the previous setup (described in Section 2.2.3, p. 18), the primary nozzle had two main disadvantages. The first shortcoming of the primary nozzle geometry in the existing installation is the shape of its converging and diverging sections as they simply consisted in two inclined planes. The resulting throat was sharp, thereby generating compression waves of significant amplitude that propagated downstream and ultimately lead to a non-uniform flow at the exit of the primary nozzle. This was evidenced by the schlieren images shown at the end of Chapter 2 (cfr. Fig. 2.8, p. 32). In the new ejector geometry, the shape of the primary nozzle has been designed by the method of characteristics to avoid these strong compression waves and to ensure a uniform flow at the nozzle exit plane. As may be seen in Fig. 6.1, the adopted geometry for the converging and the diverging parts of the nozzle is a continuous circular arc with a unique radius of curvature. This simple design provides a smooth throat and offers the additional advantage of being easily described. Note that the height of the throat and the height of the exit of the primary nozzle are kept the same as in the initial ejector geometry. However, the length of the diverging part as well as the opening angle at the exit of the primary nozzle are logically different.

The second disadvantage of the primary nozzle used in the initial installation is that it was not fully two-dimensional. Indeed, for sealing purposes, the primary nozzle was made of one single piece with front and back parts progressively vanishing. Hence, in the updated setup, shown in Fig. 6.2, the primary nozzle is made of two symmetrical pieces that are completely independent such that the novel ejector geometry is purely two-dimensional. In this configuration, the sealing of the primary nozzle is ensured with flat seals clamped between the nozzle pieces and the front and back plates. Taking the thickness of these flat seals into account, the width of the ejector in the span-wise direction is 50.0 [mm] (not shown in Fig. 6.2).

Regarding the secondary nozzle of the novel ejector, it has essentially the same geometry as the existing one. In particular, the length and the height of the mixing duct are unchanged, together with the converging angle and the height of the secondary inlet. The distance

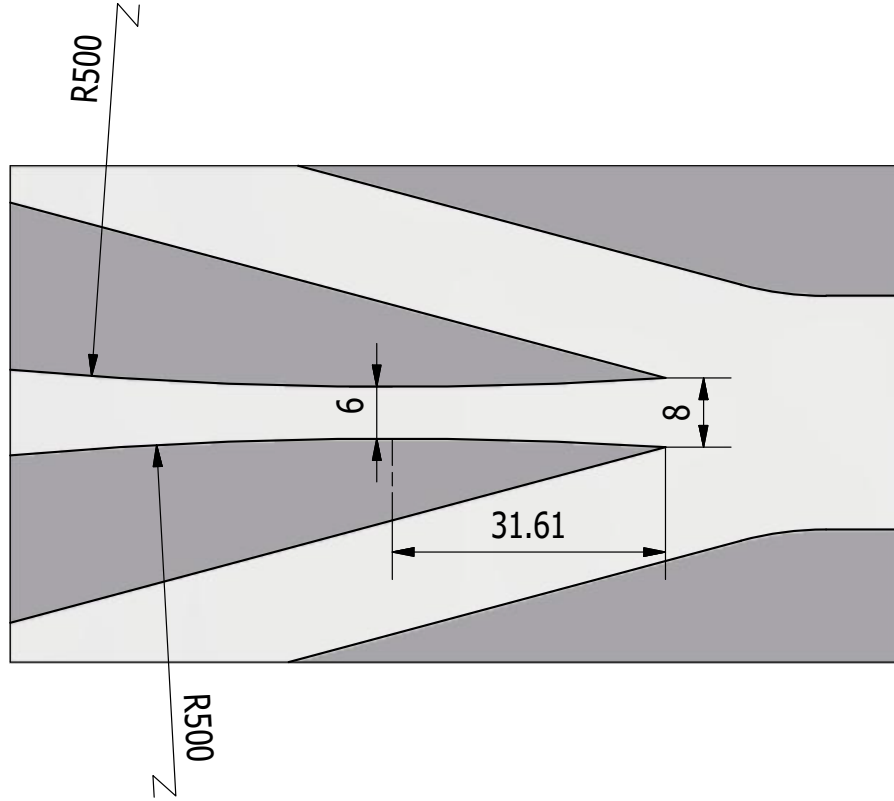


Figure 6.1: Dimensioned drawing of the new primary nozzle geometry. All dimensions are in mm.

between the exit of the primary nozzle and the entry of the mixing duct, the so-called NXP in the literature, is conserved as well. Compared to the initial geometry, the main difference concerning the secondary nozzle is actually the length of the diffuser that is shortened in the new facility for manufacturing purposes. Hence, the piece serving for the connection of the exit of the rectangular ejector to the circular tube located downstream of the diffuser has been replaced.

Aside from this connector, the entire peripheral equipment of the existing facility has been reused in the new experimental installation, including the piping and the measurement systems (described in Section 2.2.1), as well as the visualization setup (presented in Section 2.2.2). The only additional measuring instrument used in the novel test rig is a pressure transducer serving for the measurement of the static pressure at different locations along the wall of the ejector. Indeed, in the new geometry, several holes have been drilled in the top wall of the



secondary nozzle all along the mixing duct and at the entry of the diffuser (see Fig. 6.2). In the other parts of the ejector, the thickness of the wall made the hole drilling process too complicated. For this reason, the holes serving for the measurement of the pressure in the inlets and in the rest of the diffuser have been drilled in the front and back plates of the ejector. The position of these holes is visible in Fig. 6.2. Note that, even though the static pressure can be measured at many locations, a single transducer is used and the pressure to be measured is selected through a multiplexer.

6.3 Comparison of 2D numerical simulations with the experimental data

The numerical results presented in this section have been obtained with the same numerical tools as those used to generate the CFD calculations processed in the previous chapters. As they are thoroughly described in Section 2.3, the numerical methods are not recalled here. These essentially consist in 2D RANS simulations performed with OpenFOAM, an open-source software for CFD which relies on a finite volume approach. The governing equations are solved using an unsteady density-based compressible flow solver together with the wall-resolved $k-\omega$ - SST turbulence model. For each CFD calculation, the values used as boundary conditions are based on the experimental measurements.

Of course, since the primary nozzle geometry of the upgraded experimental setup has been slightly modified compared to the initial installation, the computational domain has been adjusted accordingly (not shown here). It has basically the same structure as the grid presented in Section 2.3.5 and the mesh refinement is also kept unchanged. Note that the geometry that is considered in the CFD model corresponds to the as-designed geometry. The as-built geometry should be very close to the latter, even though there could be small discrepancies.

6.3.1 Selection of the operating conditions

Six characteristic curves obtained with the upgraded ejector geometry are shown in Fig. 6.3. In this section, the performances of the CFD model will first be assessed by comparing a numerical characteristic curve with the experimental data for a certain value of the primary stagnation pressure. The selected curve corresponds to $p_{01} = 3.50$ [bar].

Then, the focus will be put on the ability of the numerical simulations to model the flow in the on-design regime for different reservoir

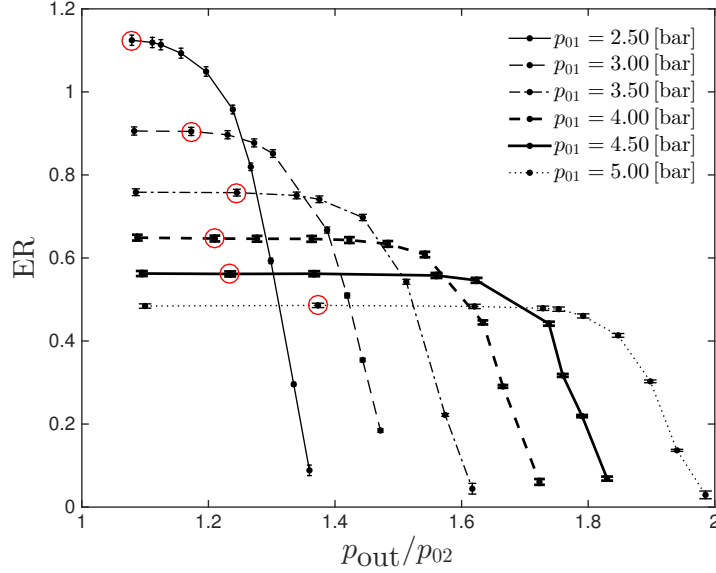


Figure 6.3: Experimental characteristic curves for the upgraded geometry.

conditions. Hence, a single on-design operating point for each characteristic curve will be considered for the following discussion. The selected cases are highlighted with a circle in Fig. 6.3. At the global level, the experimental entrainment ratio is compared to the numerical prediction for the different reservoir conditions. The ability of the compound-choking theory to predict the entrainment ratio when the ejector operates in the on-design regime is also examined for the different operating points. Then, at the local scale, a picture of the flow structure at the exit of the primary nozzle obtained by shadowgraphy is juxtaposed to the corresponding numerical image for the six on-design cases under consideration. Finally, the profile of the static pressure along the wall of the mixing duct is extracted from the numerical results and compared to the experimental measurements.

6.3.2 Characteristic curve

The comparison between the numerical characteristic curve and the experimental data obtained with $p_{01} = 3.50$ [bar] is shown in Fig. 6.4. Logically, as the geometry of the novel ejector is very close to the previous one and since the numerical tools are the same, the performances of the CFD model are essentially unaltered. As it was the case for the initial ejector geometry (cfr. Section 2.4), the numerical simulations slightly

overestimate the on-design entrainment ratio (by approximately 6.5%) as well as the critical back-pressure. As shown by Mazzelli et al. [59], these discrepancies are mainly due to 3D effects.

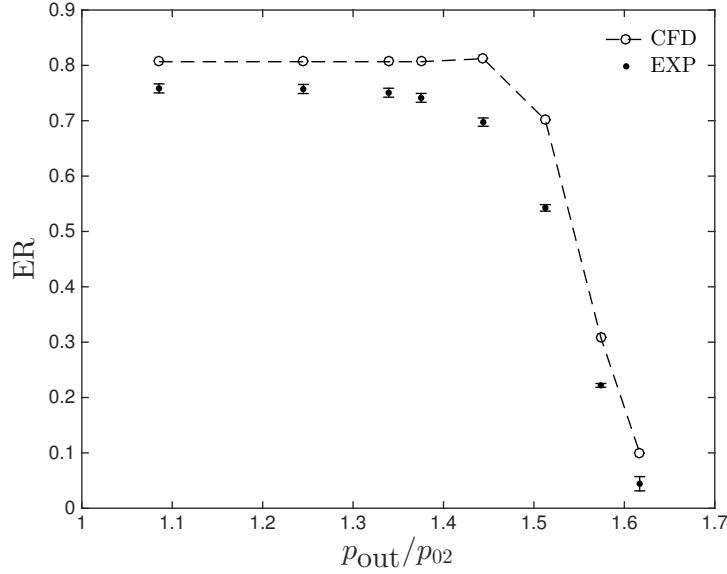


Figure 6.4: Comparison of a numerical characteristic curve with the experimental data obtained on the upgraded setup for $p_{01} = 3.50$ [bar].

6.3.3 On-design entrainment ratio

The performances of the numerical simulations and of the compound-choking theory regarding the prediction of the on-design entrainment ratio for different reservoir conditions may be assessed in Fig. 6.5. The results obtained with both approaches slightly overestimate the experimental entrainment ratios. The error made by the CFD model is approximately the same for the different reservoir conditions and is comprised between 5.6 and 7.2 %. By contrast, the relative gap between the compound-choking theory and the experiment decreases when the primary stagnation pressure, p_{01} , increases. The error of the simplified model is close to 10.2 % for $p_{01}/p_{02} = 2.59$ and falls to 6.8 % for $p_{01}/p_{02} = 5.21$. For the two largest values of the primary stagnation pressure, the simplified model yields almost the same prediction of the entrainment ratio as the 2D RANS simulations.

In order to shed light on the potential sources of error, the prediction of the primary and of the secondary mass flow rates are examined

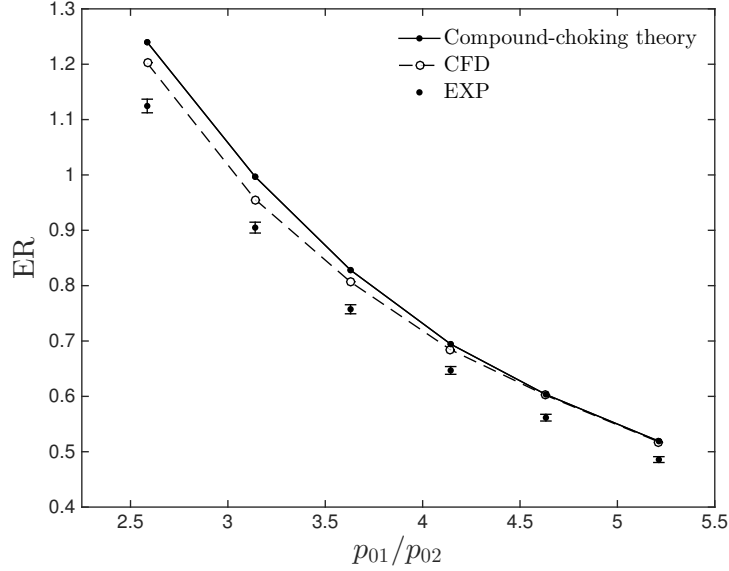
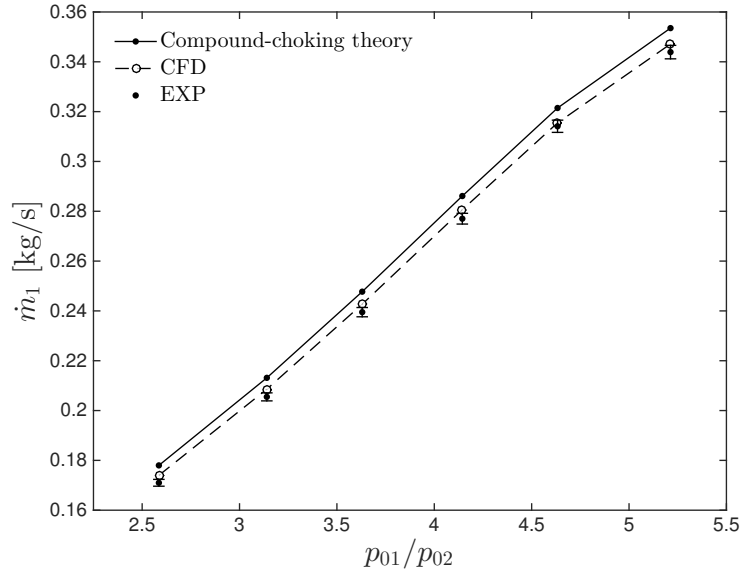


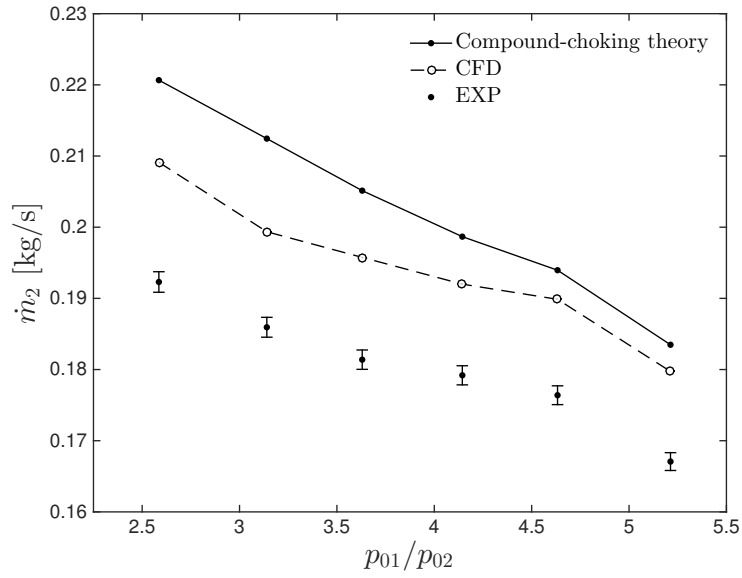
Figure 6.5: Comparison of the on-design entrainment ratio predicted by the CFD model and by the compound-choking theory with the experimental data obtained on the upgraded setup for different reservoir conditions.

separately. As may be seen in Fig. 6.6(a), the primary mass flow rate obtained by CFD is in very good agreement with the experiment for the six operating points. The maximum relative error is less than 2%. The isentropic 1D nozzle flow theory also slightly overestimates $\dot{m}_1$, with an error ranging from 2 to 4 %.

Actually, the disagreement between both models and the experimental data regarding the entrainment ratio is mainly related to the prediction of the secondary mass flow rate, $\dot{m}_2$. This is evidenced in Fig. 6.6(b). The error made by the numerical simulations is comprised between 7 and 9 % for the six cases under consideration. The gap between the CFD and the experiment regarding the secondary mass flow rate is likely to originate from 3D effects that are neglected in the numerical simulations as well as from potential discrepancies between the geometry used in the CFD model and the real ejector. Even though the main geometrical dimensions of the upgraded experimental setup have been checked, slight imponderable differences are unavoidable. Regarding 3D effects, it has been shown by Mazzelli et al. [59] for the previous experimental setup and with certain reservoir conditions, that 3D simulations lead to better predictions of the on-design entrainment



(a) primary mass flow rate



(b) secondary mass flow rate

Figure 6.6: Comparison of the mass flow rates predicted by the CFD model and by the compound-choking theory with the experimental data obtained on the upgraded setup for different reservoir conditions.

ratio. The differences between 3D and 2D simulations on the novel geometry will be investigated in Section 6.4 for the operating point corresponding to $p_{01}/p_{02} = 3.63$.

Finally, based on Fig. 6.6(b) it also appears that the performances of the compound-choking theory are actually lower than the numerical simulations for all the cases as it overestimates $\dot{m}_2$ by 10 to 15 %. As shown in Chapter 5, this is due to the numerous assumptions of the simplified model, notably the one stating that the pressure can only vary along the flow and not across it.

6.3.4 Flow structure at the exit of the primary nozzle

In this section, the focus is put on the flow structure at the exit of the primary nozzle for the six on-design operating points considered above. Experimentally, a Nikon D5000 camera is used at a pixel resolution of 4288x2848 with a focal length of 300 [mm] providing a spatial resolution of approximately 45 [pixel/mm]. The aperture and the exposure time is set to $f/4.0$ and $1/4000$ [s], respectively. In order to reduce the impact of inhomogeneities in the Plexiglas windows, a no-flow picture has been subtracted from the raw images obtained in the presence of the flow. Considering that no knife-edge was used in the experiment (shadowgraphy), the resulting picture is an image of the laplacian of the density field. Hence, the numerical simulations have been processed accordingly. Results are shown in Figs. 6.7 and 6.8 for the different operating conditions under consideration. The reader is reminded that these cases mainly differ from each other with regard to the primary stagnation pressure, p_{01} . The primary nozzle is overexpanded for $p_{01}/p_{02} = 2.59$, $p_{01}/p_{02} = 3.14$ and $p_{01}/p_{02} = 3.63$ (Figs. 6.7(a), (b) and (c), respectively), whereas it seems nearly adapted for the case $p_{01}/p_{02} = 4.14$ (Fig. 6.8(a)). For the two larger values of p_{01} characterized by $p_{01}/p_{02} = 4.63$ and $p_{01}/p_{02} = 5.21$ (Figs. 6.8(b) and (c), respectively), the nozzle is clearly underexpanded.

The saturation settings used in the numerical images are the same for all the cases. The grainy aspect of the shear layer in the numerical images shown in Figs. 6.7 and 6.8 does not reveal small turbulent structures as the CFD calculations consists in RANS simulations. Actually, it is simply related to the computation of the Laplacian of the density on an unstructured mesh. Indeed, as this operator requires the assessment of second order derivatives, the resulting field may look a bit noisy. Regarding the blurry aspect of the experimental image corresponding to $p_{01}/p_{02} = 4.63$, shown in Fig. 6.8(b), it is due to an unintended offset between the no-flow picture and the image taken during operation.



(a) $p_{01}/p_{02} = 2.59$ - $p_{\text{out}}/p_{02} = 1.08$.



(b) $p_{01}/p_{02} = 3.14$ - $p_{\text{out}}/p_{02} = 1.17$.



(c) $p_{01}/p_{02} = 3.63$ - $p_{\text{out}}/p_{02} = 1.24$.

Figure 6.7: Comparison of experimental images (top half) obtained by shadowgraphy (no knife-edge) and numerical images (bottom half) showing the Laplacian of the density field for different values of the primary stagnation pressure.



(a) $p_{01}/p_{02} = 4.14 - p_{out}/p_{02} = 1.21$.



(b) $p_{01}/p_{02} = 4.63 - p_{out}/p_{02} = 1.23$.



(c) $p_{01}/p_{02} = 5.21 - p_{out}/p_{02} = 1.37$.

Figure 6.8: Comparison of experimental images (top half) obtained by shadowgraphy (no knife-edge) and numerical images (bottom half) showing the Laplacian of the density field for different values of the primary stagnation pressure.

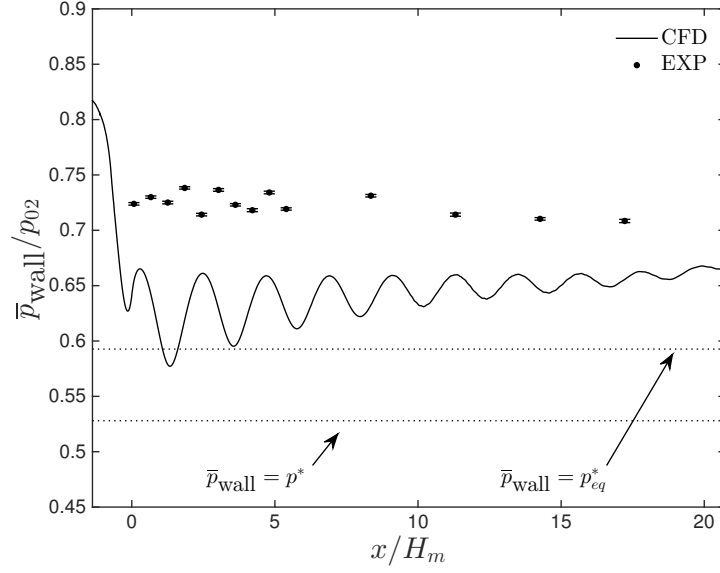
As may be seen in Figs. 6.7 and 6.8, the flow structure predicted by the CFD model is in fairly good agreement with the experimental image for the six operating conditions. In each case, the first shock cells in the core of the primary stream are properly captured by the numerical simulations, together with the shear layer pattern. Nevertheless, small discrepancies may be observed between the CFD model and the experiment. More precisely, in Fig. 6.7(b), the numerical simulation predicts a normal shock wave in the core of the secondary stream, attesting the presence of the Y-shock wave pattern highlighted in Chapter 4. However, this normal shock wave is not visible in the experimental image. This may be due to the exposure time used in the experiment as the picture has been obtained with a common DSLR and not with the high-speed camera that served to evidence the Y-shock wave pattern in Chapter 4. But, as will be shown below, it is more likely that the normal shock wave in the secondary stream is missing in the experiment because of 3D effects. Actually, these are likely to be the main source of discrepancies between the numerical and the experimental results regarding the flow structure downstream of the entry of the mixing duct, observable for different reservoir conditions in Figs. 6.7 and 6.8.

The fact that the shear layer generally fades more rapidly in the experimental pictures compared to the CFD may be due to 3D effects but it may also be impacted by the saturation settings of the numerical images that have been chosen in order to emphasize the supersonic structures in the core of the primary stream for the different operating conditions.

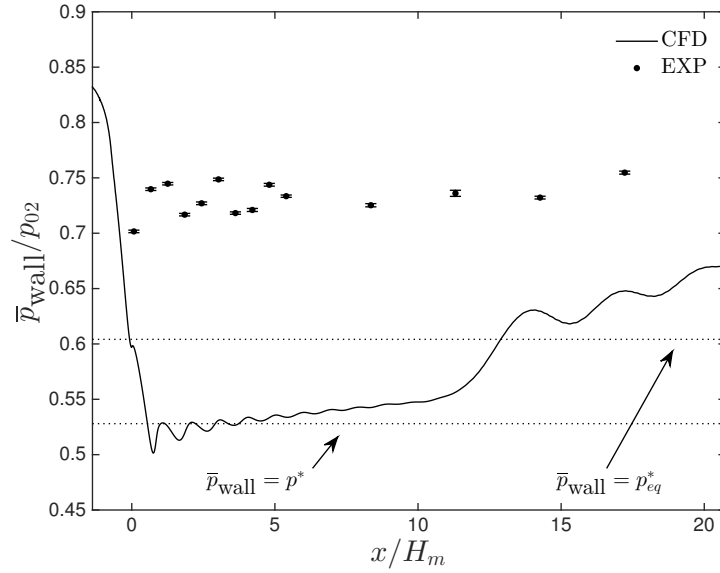
6.3.5 Static pressure profile

As mentioned at the beginning of this section, a second approach – that was not available in the initial experimental setup – is used to compare the CFD model and the experiment at the local scale. It consists in extracting from the numerical results the profile of the static pressure at the wall of the mixing duct and to compare it to experimental measurements. Results are shown in Figs. 6.9, 6.10 and 6.11 for the six cases considered above. As these cases correspond to on-design conditions, the theoretical value of the static pressure corresponding to the compound-choking regime ($M_{eq} = 1$), noted p_{eq}^* and introduced in Section 5.2.3, is also indicated for each operating point.

Although the 2D numerical simulations are in fairly good agreement with the experiment regarding the entrainment ratio (cfr. Section 6.3.3) and the flow structure at the exit of the primary nozzle (cfr. Section 6.3.4), they fail to predict the value of the static pressure acting on

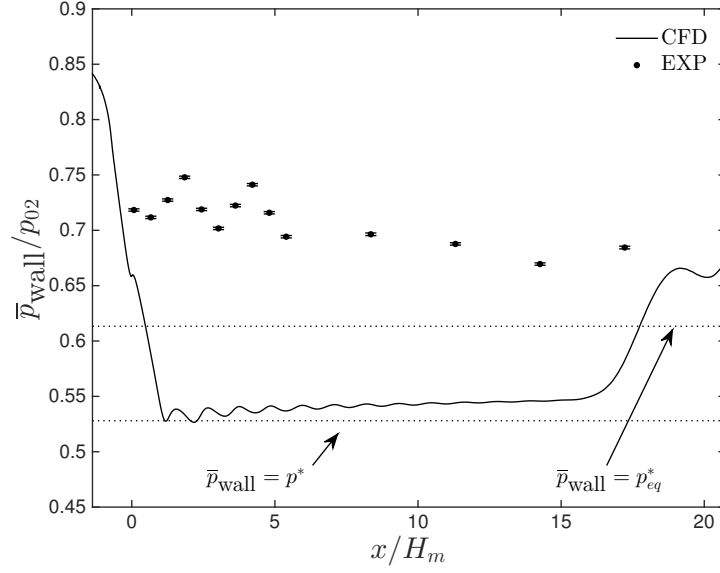


(a) $p_{01}/p_{02} = 2.59$ - $p_{\text{out}}/p_{02} = 1.08$.

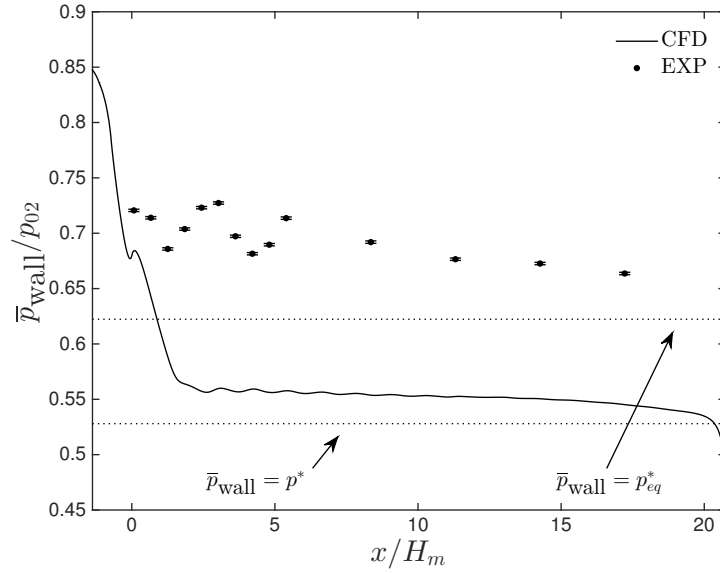


(b) $p_{01}/p_{02} = 3.14$ - $p_{\text{out}}/p_{02} = 1.17$.

Figure 6.9: Comparison of the pressure profile extracted from the numerical results at the wall of the secondary nozzle with the experimental measurements (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

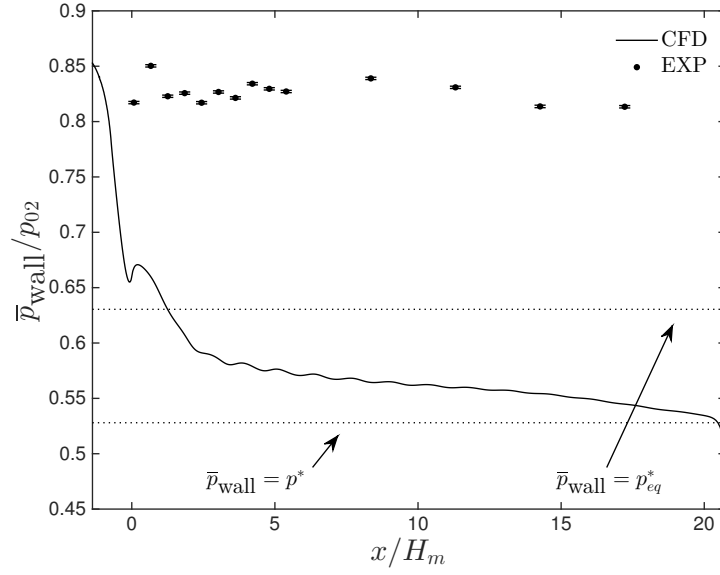


(a) $p_{01}/p_{02} = 3.63 - p_{\text{out}}/p_{02} = 1.24$.

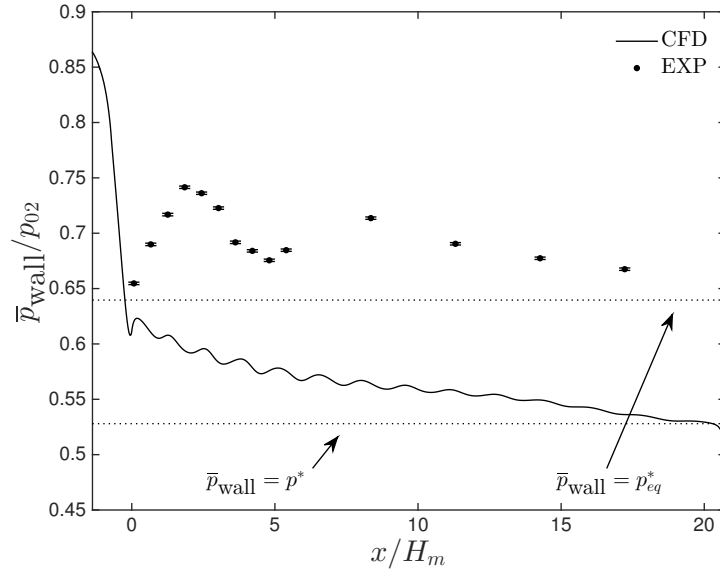


(b) $p_{01}/p_{02} = 4.14 - p_{\text{out}}/p_{02} = 1.21$.

Figure 6.10: Comparison of the pressure profile extracted from the numerical results at the wall of the secondary nozzle with the experimental measurements (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).



(a) $p_{01}/p_{02} = 4.63 - p_{\text{out}}/p_{02} = 1.23$.



(b) $p_{01}/p_{02} = 5.21 - p_{\text{out}}/p_{02} = 1.37$.

Figure 6.11: Comparison of the pressure profile extracted from the numerical results at the wall of the secondary nozzle with the experimental measurements (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

the wall of the mixing duct. Indeed, for the six operating conditions shown in Figs. 6.9, 6.10 and 6.11, the CFD model underestimates – to a greater or a lesser extent – the static pressure on the wall of the ejector all along the mixing duct. The reader is warned that the error bar of each measurement shown in Figs. 6.9, 6.10 and 6.11 only takes into account the uncertainty related to the calibration of the sensor as well as the variability of the signal for a given experiment. Hence, it does not include the variation of the measurement between several reproductions of the same experimental conditions. The reproducibility of these experimental results is thus not guaranteed. Note that the experiment corresponding to $p_{01}/p_{02} = 4.63$ was run at a different moment in the year than the others. It is therefore possible that the experimental setup slightly changed for this operating point in comparison with the other conditions.

Another finding that may be drawn from Figs. 6.9, 6.10 and 6.11 concerns the compound-choking theory. Indeed, the numerical pressure profile falls below the value of the static pressure corresponding to the compound-choking regime, p_{eq}^* , for the six on-design operating points under consideration. In addition, for all the cases, the location where $\bar{p}$ reaches p_{eq}^* is logically close to the entry of the mixing duct ($-0.25 < x/H_m < 1.25$). However, in the experiment, the static pressure was always found to be higher than p_{eq} for all the measuring points located at the wall of the mixing duct, regardless of the reservoir conditions. The origin of this discrepancy will be the subject of future investigations.

In the next section, 3D effects will be shown to be at least partially responsible for the discrepancies between the 2D numerical simulations and the experiment regarding the static pressure profile.

6.4 Numerical investigation of three-dimensional effects

In this section, 3D effects are investigated by means of numerical simulations for an on-design operating point. The selected case is characterized by $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$. Compared to the 2D calculations, the CFD model is exactly the same, except, logically, for the computational domain. The latter is shown in Fig. 6.12. As may be seen, the computational domain used for the 3D simulations only covers a quarter of the entire geometry in order to reduce the computational cost. Symmetry conditions are used on both median planes. Note that the 3D mesh has simply been obtained by extruding the grid

used for 2D simulations in the span-wise direction (60 layers), resulting in a computational domain of approximately 20 million cells. As may be seen in Fig. 6.12, the mesh is refined in the span-wise direction near the side wall in order to properly capture the boundary layer as the numerical simulations rely on the wall-resolved version of the $k-\omega$ - SST turbulence model.

The numerical results obtained with the 3D CFD model are compared to the experimental data in the same way as for the 2D calculations. The focus is thus successively put on the prediction of (i) the mass flow rates, (ii) the flow structure at the exit of the primary nozzle, and (iii) the pressure profile along the wall of the mixing duct.

The mass flow rates predicted by the 3D numerical simulation are given in Table 6.1, as well as the corresponding experimental data. For comparison purposes, the performances of the compound-choking theory (referred to as '1D' in Table 6.1) and of the 2D calculation are also included in the table. The agreement between the numerical results and the experiment is slightly better when 3D effects are taken into account in the CFD model. Logically, the improvement mainly concerns $\dot{m}_2$ as $\dot{m}_1$ was already well predicted by the 2D simulation. Nevertheless, the enhancement is limited as the relative gap between the 3D calculation and the experimental measurement regarding the secondary mass flow rate is still 6.10%. This observation tends to corroborate the fact that the choked flow condition is not strongly impacted by 3D effects for the present ejector geometry. The remaining error on $\dot{m}_2$ may probably be attributed to small differences between the real and the modeled geometry as well as to slight discrepancies between the boundary conditions used in the simulation and the experimental conditions.

	$\dot{m}_1$ [g/s]	error [%]	$\dot{m}_2$ [g/s]	error [%]	ER [%]	error [%]
1D	247.72	3.43	205.14	13.10	82.81	9.34
CFD 2D	242.69	1.33	195.76	7.93	80.66	6.50
CFD 3D	241.95	1.02	192.44	6.10	79.54	5.02
EXP	239.50	-	181.39	-	75.74	-

Table 6.1: Comparison of the different models regarding the prediction of the mass flow rates for the operating point characterized by $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$. The 1D approach refers to the compound-choking theory. Error=(model-exp)/exp.

The focus is now put on the ability of the 3D numerical simulation to predict the flow structure at the exit of the primary nozzle. The comparison between the numerical results and the experiment is shown in

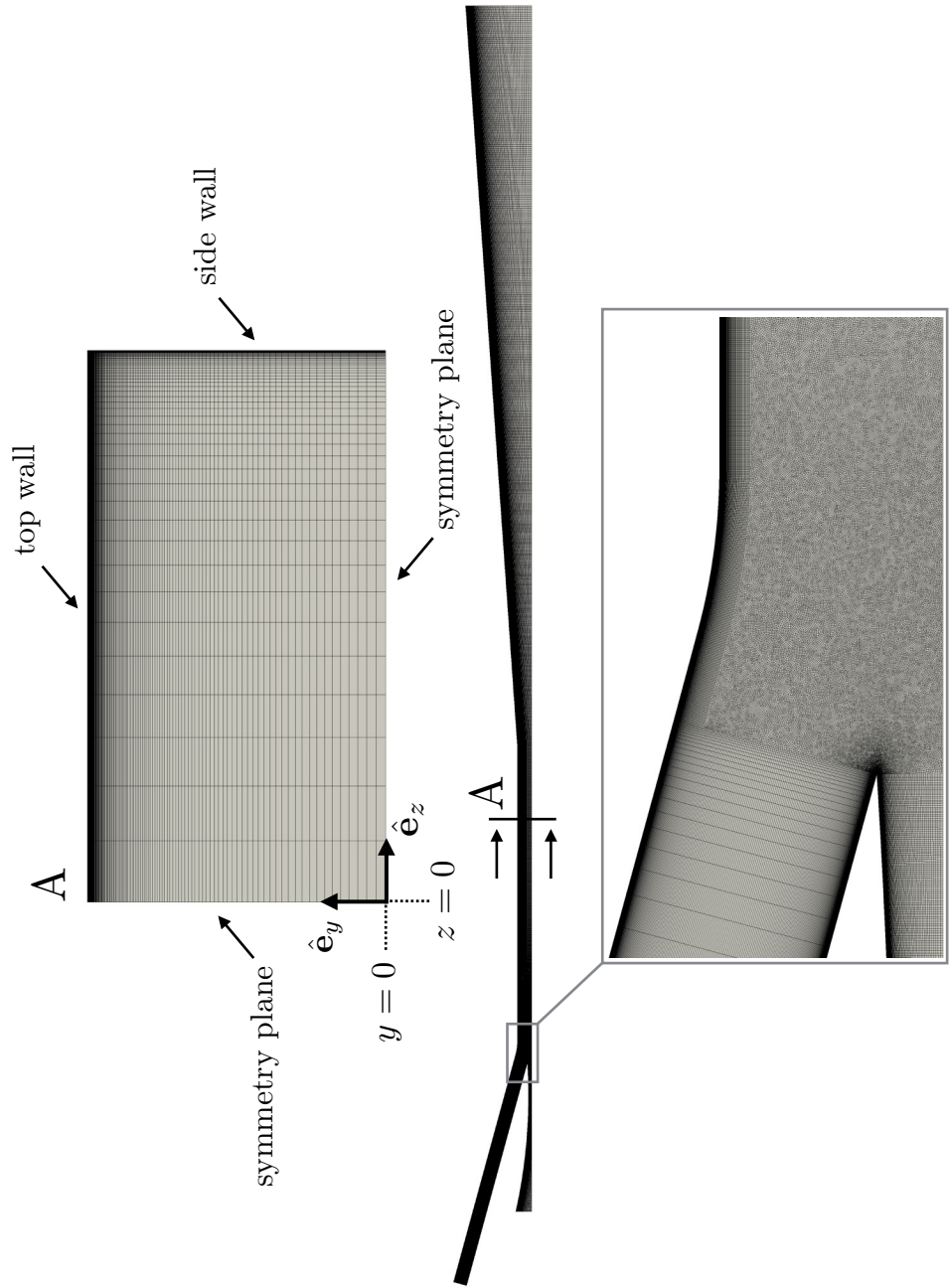


Figure 6.12: Grid structure of the computational domain for 3D numerical simulations on the novel ejector geometry.

Fig. 6.13. The experimental picture is logically the same as the one used in Fig. 6.7(c) for the validation of the 2D calculation. As mentioned above, it has been obtained by shadowgraphy and it is thus an image of the Laplacian of the density field. Hence, the latter has been calculated in each cell of the computational domain in the post-processing phase of the CFD calculation. Even though the numerical image to be compared with the experimental picture should ideally be the average in the span-wise direction of the Laplacian of the density, the one that is shown in Fig. 6.13 corresponds to the median plane of the ejector ($z = 0$). Nevertheless, the variation of the density field in the span-wise direction has been checked and is relatively small for the region of the flow under consideration (not shown here). Hence, the numerical image resulting from the average process would be essentially the same as the one extracted from the median plane and shown in the bottom half of Fig. 6.13.

Although the ability of the 2D and the 3D simulations to predict the experimental mass flow rates is almost equivalent, the 3D CFD model performs much better regarding the prediction of the flow structure. Indeed, as may be seen in Fig. 6.13, the different supersonic structures visible in the 3D calculation are in very good agreement with the experiment, including those located further downstream in the mixing duct. As a reminder, the 2D numerical simulation properly captures the first shock cells only.

The discrepancies between the 2D and the 3D CFD calculations regarding the structure of the flow at the exit of the primary nozzle are evidenced in Fig. 6.14 where both numerical images are juxtaposed. It appears that the flow on the median plane of the ejector starts being impacted by 3D effects at a small distance downstream of the entry of the mixing duct. Compared to the 2D simulation, the 3D model predicts stronger oblique shock waves in the mixing duct. The analysis of the pressure profile along the wall of the ejector will shed light on this difference.

Figure 6.15 shows the profile of the static pressure along the top wall of the mixing duct obtained with the 2D and the 3D simulations, together with the experimental measurements. In the experiment, the holes for measuring the pressure are located in the median plane of the ejector ($z = 0$). Hence, the pressure profile of the 3D simulation shown in Fig. 6.15 has been extracted from the numerical results accordingly. The inability of the 2D calculation to predict the pressure acting on the wall of the mixing duct is mainly due to the role played by 3D effects since the pressure profile obtained with the 3D model is in much better agreement with the experimental data.



Figure 6.13: Comparison of experimental (top half) and numerical (bottom half) shadowgraph images for $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$. The numerical results correspond to the median plane ($z=0$) of the 3D simulation.

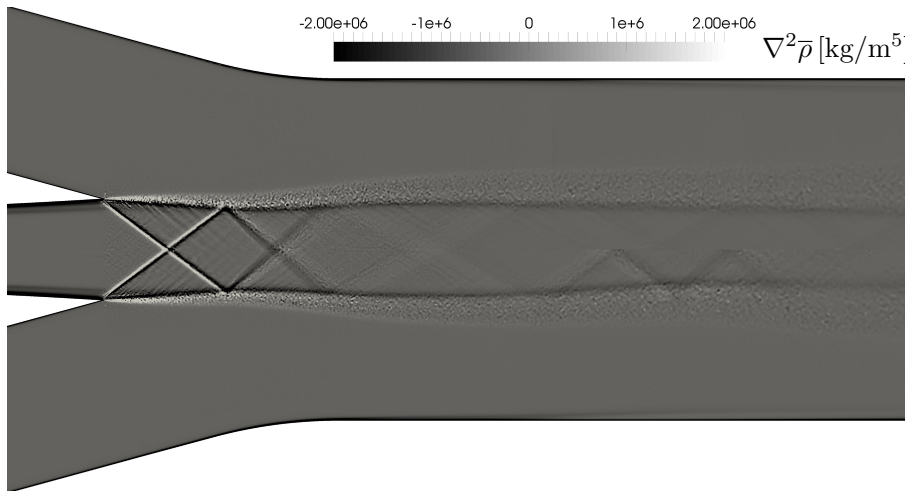


Figure 6.14: Comparison of the Laplacian of the density field extracted from the 2D (top half) and from the median plane ($z=0$) of the 3D (bottom half) numerical simulations for $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$.

As previously observed, the 3D simulation starts to differ from the 2D calculation at a small distance downstream of the entry of the mixing duct. Actually, this position nearly corresponds to the location

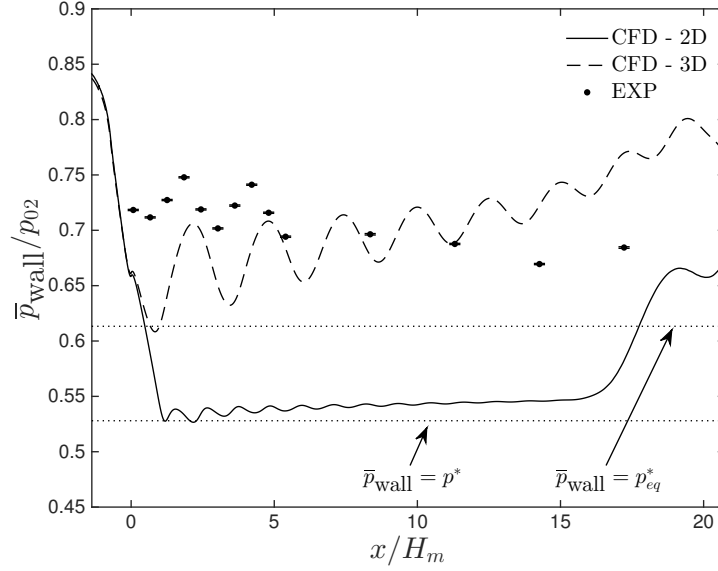


Figure 6.15: Comparison of the pressure profile extracted from the numerical results at the wall of the secondary nozzle (median plane) with the experimental measurements for $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

where $\bar{p}_{wall}$ reaches p_{eq}^* . While the static pressure keeps falling down to the critical pressure, p^* , in the 2D numerical results, $\bar{p}_{wall}/p_{02}$ goes up and oscillates around 0.7 in the 3D calculation. Therefore, it is likely that the 2D simulation predicts a sonic section in the core of the secondary stream, whereas the 3D model should not. This is effectively the case, as evidenced in Fig. 6.16 showing the supersonic region of the flow at the exit of the primary nozzle obtained with the 2D (top half) and the 3D (bottom half) simulations. Consistently with the analysis of the pressure profile, the Mach number field illustrating the 3D results is taken on the median plane of the ejector ($z = 0$). A sonic section in the core of the secondary stream clearly exists in the 2D numerical simulation shown in the top half of Fig. 6.16, but this flow feature is eradicated by taking 3D effects into account (bottom half). Thus, for the present ejector geometry and operating conditions, 3D effects seem to prevent the formation of a sonic section in the core of the secondary stream and the subsequent Y-shock wave pattern. However, as shown above, the mass flow rates predicted by the 3D simulation are very close to those obtained with the 2D CFD model. These observations thus give an additional argument in favor of the compound-choking theory to model

the choking mechanism at the basis of the entrainment limitation in supersonic ejectors, even when a Fabri-pattern is present.

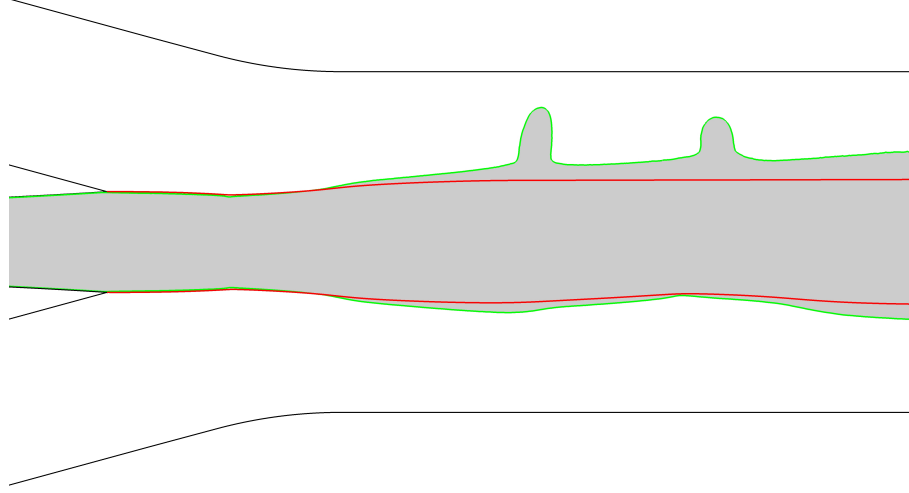


Figure 6.16: Comparison of the supersonic region in the 2D (top half) and in the median plane of the 3D (bottom half) numerical results for $p_{01}/p_{02} = 3.63$ and $p_{\text{out}}/p_{02} = 1.24$. The corresponding dividing streamline is shown in red.

So, discrepancies between 2D and 3D simulations are relatively small with regard to the mass flow rates but may however seem significant when considering the flow structure in the mixing duct. In order to gain insights into the differences observed between the 2D and the 3D numerical results, some theoretical considerations may be helpful. Indeed, for a 1D isentropic flow of a calorically perfect gas, the static pressure is actually highly sensitive to the variations of the flow area when the latter is close to the critical section, noted A^* in common textbooks [73, 92], and given by

$$A^* = \frac{\dot{m}\sqrt{T_0}}{p_0} \sqrt{\frac{R}{\gamma} \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma+1}{\gamma-1}}}. \quad (6.1)$$

Figure 6.17 shows the variation of the static pressure with the flow area for a 1D isentropic flow in the subsonic regime when the fluid is a calorically perfect gas characterized by $\gamma = 1.4$. As may be seen, p/p_0 goes from 0.528, corresponding to the critical pressure ratio p^*/p_0 , to more than 0.7 when A/A^* only moves from 1 to 1.1. This means that the relatively large gap between the 2D and the 3D simulations regarding the static pressure acting on the top wall of the mixing duct could be related to rather small variations of the secondary flow area.

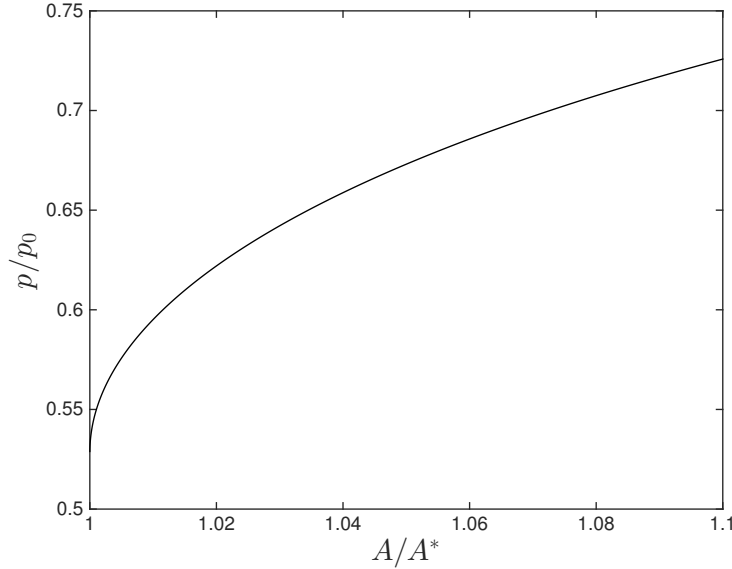


Figure 6.17: Variation of the static pressure with the flow area for a 1D isentropic subsonic flow of a calorically perfect gas with $\gamma = 1.4$.

In order to confirm this, both simulations are further compared on the basis of the dividing surfaces. The concept of the dividing surface was introduced in Section 3.4.1 (p. 46). In brief, dividing streamlines are constructed from points located in the nozzle exit plane, just above the tip of the primary nozzle. The resulting dividing surface may be interpreted as a virtual separation between the primary and the secondary streams all along the ejector.

The dividing surfaces obtained with the 3D CFD model are shown in Fig. 6.18. Even though the simulation has been performed on a quarter of the real geometry, the entire ejector is depicted in this figure for improved readability. As illustrated in Fig. 6.18, the dividing surface allows to define the cross-sections through which the primary and the secondary stream virtually flow, noted $A_1(x)$ and $A_2(x)$, respectively.

The comparison between the secondary flow area, $A_2(x)$, obtained by processing the 2D and the 3D numerical results is shown in Fig. 6.19, through the ratio A_2/A_2^* . Note that A_2^* has been calculated with Eq. 6.1 for each simulation, using the corresponding secondary mass flow rate, $\dot{m}_2$. The order of magnitude of the difference between both simulations in terms of A_2/A_2^* is consistent with the gap between both pressure profiles observed in Fig. 6.15. Hence, the latter is mainly due to discrepan-

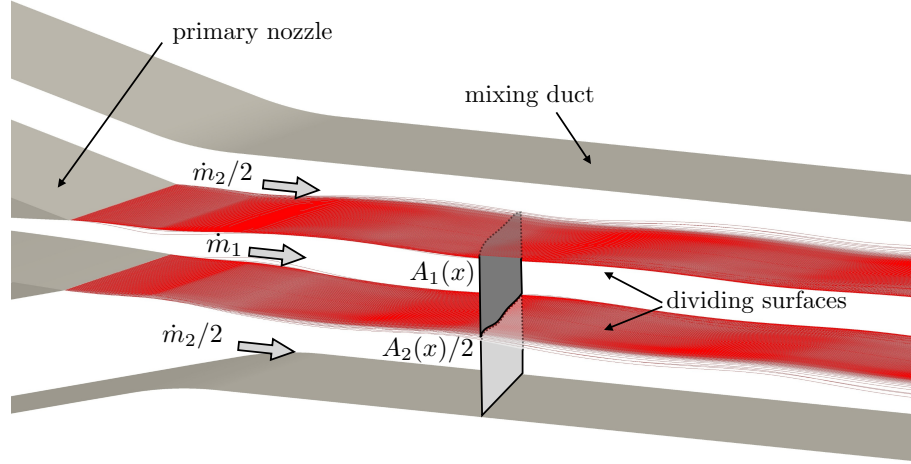


Figure 6.18: Dividing surfaces predicted by the 3D CFD model for $p_{01}/p_{02} = 3.63$ and $p_{\text{out}}/p_{02} = 1.24$.

cies in terms of the secondary flow area, at least close to the entry of the mixing duct.

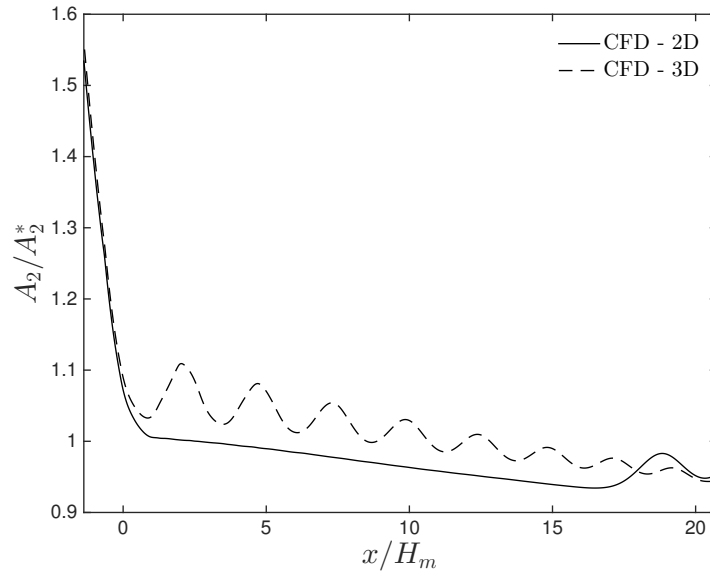


Figure 6.19: Comparison of the 2D and the 3D numerical simulations based on $A_2(x)/A_2^*$ for $p_{01}/p_{02} = 3.63$ and $p_{\text{out}}/p_{02} = 1.24$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

The fact that $A_2(x)$ is larger in the 3D simulation than in the 2D calculation means that the primary mass flow rate, which is almost the same in both models, leaves more space for the secondary mass flow rate when 3D effects are taken into account. Figure 6.20 shows the static pressure profile on the ejector centerline (core of the primary stream) predicted by the 2D and the 3D simulations. It clearly appears that the 3D simulation starts departing from the 2D model from the first shock wave located in the mixing duct ($x/H_m \simeq 1$). Hence, discrepancies between 2D and 3D simulations regarding the structure of the flow could be related to shock wave boundary layer interactions (SWBLI) on the side wall of the ejector. The investigation of such interactions would require additional analyses that fall beyond the scope of the present thesis.

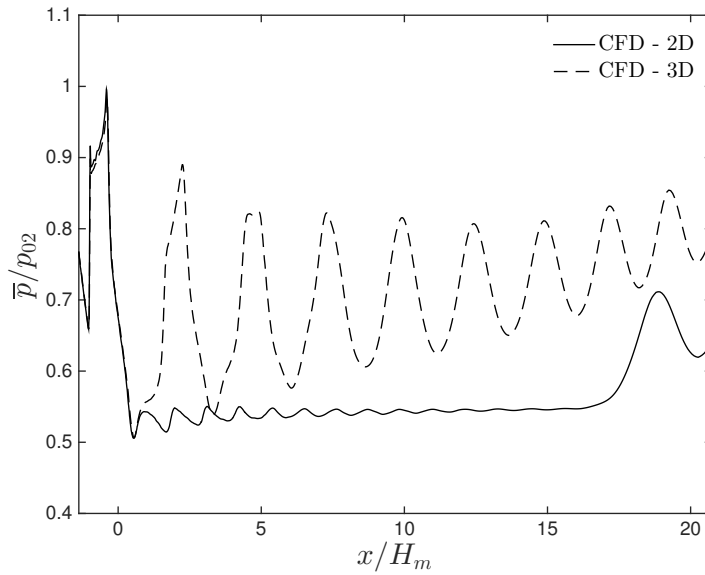


Figure 6.20: Comparison of the pressure profile on the ejector centerline predicted by 2D and 3D simulations for $p_{01}/p_{02} = 3.63$ and $p_{out}/p_{02} = 1.24$ (the mixing duct extends from $x/H_m = 0$ to $x/H_m \simeq 20.75$).

6.5 Conclusions

In this chapter, the focus was put on the development of an upgraded experimental setup which primarily aims at validating numerical sim-

ulations. The main improvement of the novel installation concerns the geometry of the primary nozzle but the height at its throat and at its exit were unchanged compared to the previous geometry. In addition, the static pressure may be measured at several locations along the walls of the ejector in the new facility, which was not the case before.

The experimental data acquired on this upgraded setup have been compared to the results of 2D numerical simulations obtained with the same numerical tools as those used to model the flow in the previous geometry. The first comparison dealt with the prediction of a characteristic curve. The performances of the 2D CFD model are essentially unaltered compared to the initial installation. More precisely, the numerical results slightly overestimate both the entrainment ratio in the on-design regime as well as the critical back-pressure.

Then, a special emphasis was put on the ability of the 2D numerical simulations to model the flow in on-design conditions for different primary stagnation pressures. Hence, the CFD model has been run for one on-design operating point of each six different characteristic curves. The focus was successively put on the prediction of (i) the mass flow rates, (ii) the flow structure at the exit of the primary nozzle and (iii) the top wall static pressure profile.

The mass flow rates obtained in the 2D numerical simulations are in good agreement with the experiment. For the six cases under consideration, the error made by the CFD model regarding the primary mass flow rate is always found to be less than 2%, while it is comprised between 7 and 9 % for the secondary mass flow rate. The slight overestimation of $\dot{m}_2$ may be related to 3D effects and to imponderable discrepancies between the real and the modeled geometries. The use of the RANS approach may also play a role in the remaining error.

The agreement between the 2D results and the experimental visualizations regarding the structure of the flow at the exit of the primary nozzle is also fairly good for the different reservoir conditions. In particular, the first shock cells seem to be properly captured by the CFD model. However, further downstream, slight differences are noticeable between the numerical and the experimental images for the six operating points.

In addition, the 2D numerical simulations deeply underestimate the static pressure acting on the top wall of the mixing duct, regardless of the reservoir conditions. A 3D calculation has shown that it was mainly related to 3D effects. Although the secondary mass flow rates predicted by the 2D and the 3D simulations are almost the same, the secondary flow passage in the mixing duct is comparatively larger in the CFD model that incorporates 3D effects. As the static pressure is very sen-

sitive to the flow area when the latter is close to its critical value, the small discrepancies regarding the secondary flow passage result in significant differences in terms of static pressure.

CHAPTER 7

CONCLUSIONS AND PERSPECTIVES

The limitation on the performance of supersonic ejectors can be principally traced to the choking and the mixing mechanisms at play in such devices. Hence, a better understanding of these complex flow phenomena is necessary in order to identify potential improvements of the performances of ejector systems.

In this context, this work has aimed at gaining insights into the flow physics taking place in supersonic ejectors, with special emphasis on choking and mixing phenomena. The adopted approach has mainly consisted in thoroughly processing the results of 2D RANS simulations of the flow in a rectangular ejector using air as working fluid. The calculations have been performed with OpenFOAM, an open source software for CFD relying on a finite volume method. Experimental investigations have also been carried out in order to validate the numerical results both at the global and at the local scales. Finally, theoretical considerations have been referred to throughout this thesis to support the different analyses.

Even though the number of studies related to supersonic ejectors and resorting to the RANS approach was already considerable when this thesis started, the potential of such calculations was thought to be underexploited. As a matter of fact, the present work has shown that processing the results of such simulations could effectively lead to a better understanding of the choking and the mixing phenomena taking place in supersonic ejectors. In the following section, the main con-

tributions of this thesis are exposed. The limitations of the proposed analyses are outlined in Section 7.2, together with some research perspectives.

7.1 Main achievements

The present work distinguishes itself by considering the ejector from the perspectives of thermodynamic cycles and of compressible flow phenomena; this combined approach and the novel post-processing tools have allowed to highlight the relations between local flow phenomena and device-scale performance. In particular, the following achievements are believed to constitute significant contributions for the ejector community:

- To gain insights into the mixing processes occurring within supersonic ejectors, new post-processing methods have been developed in Chapter 3. First, the momentum and energy tube analysis tools of Meyers and Meneveau has been extended to the concept of exergy in order to construct exergy transport tubes. These exergy tubes constitute powerful visualization tools that are particularly suited to supersonic ejectors to identify the region of the flow where transfers are important. For the present ejector configuration, the mixing between both streams is not yet completed at the entry of the diffuser for some cases lying in the on-design regime.

In addition to this, a post-processing method aiming at quantifying the local transfers and losses of exergy within the ejector has been proposed. This approach offers some novel phenomenological insights but also leads to the definition of new quantitative indicators to assess these transfers and losses. The operating conditions resulting in the highest global gain in exergy of the secondary stream actually corresponds to the optimization of two effects: minimized losses and maximized exergy transfer. Logically, this operating point nearly matches with the critical point of the characteristic curve.

- The analyses performed in Chapters 4 and 5 have undoubtedly contributed to a better understanding of the choking phenomena in supersonic ejectors. In state-of-the art simplified models, the secondary stream is generally assumed to reach sonic conditions in a sort of aerodynamic throat when the ejector operates in the on-design regime. This choking mechanism is often referred to as

the Fabri-choking, named after the scientist who first proposed it in the late 1950's. Even though the presence of a sonic section in the core of the secondary stream was postulated more than seventy years ago, to the author's knowledge, it had never been observed numerically nor experimentally. This work is the first one that brings numerical and experimental evidences attesting the acceleration of the secondary stream to supersonic velocities through an aerodynamic throat in a supersonic ejector. However, contrary to the idea of Fabri and Siestrunck, it seems that this flow feature is not the reason for the entrainment limitation, but rather a specific flow pattern. Hence, the sonic section in the core of the secondary stream is proposed to be referred to as the Fabri-pattern rather than the Fabri-choking.

The choking mechanism, at the basis of the entrainment limitation, has been highlighted by processing the numerical results obtained on the same ejector geometry but with a different primary stagnation pressure. The flow in the ejector appears to be choked at a location where the secondary stream is almost entirely subsonic. Hence, the sonic line, defined as the iso-contour of Mach number unity, is not a good criterion to assess if the flow is choked within a supersonic ejector. Interestingly, the observed choking process has been found to be properly described by a theory developed in the 1960's and known as the compound-choking theory. For an unknown reason, recent studies referring to this particular theory could not be found.

Yet, the compound-choking theory seems to be more adequate than the Fabri-choking model to describe the choking process occurring in supersonic ejectors. More precisely, three arguments have been presented in favor of the compound-choking theory as the explanation of the entrainment limitation in most ejector configurations. Firstly, in many numerical studies where the sonic line is shown for on-design operating conditions, the latter is found to progressively expand from the exit of the primary nozzle towards the wall of the ejector, with no apparent sonic section in the core of the secondary stream. Secondly, based on a comparison with comprehensive experimental data, the compound-choking theory has been shown to generally perform better than state-of-the-art 1D models – relying on the Fabri-choking assumption – for the prediction of the on-design entrainment ratio. Finally, as shown in Chapter 6, 3D effects may prevent the formation of a

sonic section in the core of the secondary stream without significantly impacting the secondary mass flow rate.

- In order to pave the way to further numerical and experimental investigations, an upgraded experimental setup has been developed. The main purpose of this novel installation is to improve the shortcomings of the previous geometry. The experimental data obtained on this facility have been compared to numerical simulations. The 2D CFD model used in this work allowed the prediction of the on-design entrainment ratio measured in the experiment with a relative error ranging from 5.6 to 7.2 %. These discrepancies are mainly related to the secondary mass flow rate as the error made on the primary mass flow rate is less than 2 % for all the cases that have been considered. The numerical results were also found to be in fairly good agreement with the experiment regarding the flow structure at the exit of the primary nozzle. However, the 2D simulations failed to predict the supersonic structures in the mixing duct. Consequently, large discrepancies have been found between the 2D CFD model and the experimental measurements regarding the static pressure profile on the top wall of the ejector.

An additional numerical investigation has demonstrated that this gap was essentially due to 3D effects. Indeed, even though the 2D and the 3D simulations essentially predict the same mass flow rates for on-design operating conditions, 3D effects lead to small differences regarding the secondary flow passage in the mixing duct. The pressure being highly sensitive to the flow area for a transonic stream, these discrepancies are associated to relatively large variations of the static pressure. The numerical simulation that incorporates 3D effects also properly captures the flow structure in the mixing duct, unlike the 2D CFD model. The comparison between the experimental picture obtained by shadowgraphy and the corresponding numerical image extracted from the 3D numerical results, shown in Chapter 6 (cfr. Fig. 6.13), is repeated below. To the author's knowledge, such a comparison at the local level between experimental and numerical investigations has never been presented in the literature.



Figure 7.1: Comparison of an experimental picture obtained by shadowgraphy (top half) with the corresponding image extracted from the 3D numerical results (bottom half).

7.2 Limitations and research perspectives

The final discussion of this document is devoted to the presentation of the limitations and the research perspectives of this thesis.

- Regarding the understanding of the mixing processes in supersonic ejectors, the main contribution of this work consists in the development of quantitative post-processing methods, which were clearly lacking in the literature. However, it should be kept in mind that the ultimate goal of gaining insights into the flow physics at play in supersonic ejectors is to enhance their performances, primarily by improving their design. Hence, it would be interesting to apply the proposed methodology to a large number of ejector configurations in order to highlight the influence of the different geometrical parameters on the local and cumulative transfers and losses of exergy.
- Even though the numerical tools used in this work have been shown to be well suited to model the turbulent compressible flow in a supersonic ejector, the present numerical setup is not adapted for large parametric studies. Indeed, the selected solver is unsteady and relies on the first order Euler explicit time integration method. Hence, the CFL condition requires very low time steps

to ensure the stability of the calculations. For the present simulations, the time step used in the CFD model was close to 5.10^{-9} [s]. Combined with a relatively large computational domain, the resulting computational cost of a 2D simulation was quite high, typically ranging from 5000 to 6000 [CPU.hours]. Hence, in order to facilitate future numerical investigations, it could be useful to spend some time developing a steady-state version of the compressible flow solver *rhoCentralFoam* available in OpenFOAM and used in this work.

- Another limitation of the analysis proposed in this thesis to investigate the mixing phenomena stems from the fact that it was applied to the results of RANS simulations. Yet, these tools may be easily transposed to the context of 3D Large Eddy Simulation. Within this framework, the proposed methodology could be applied on the averaged fields but also on the fluctuating parts of the corresponding quantities, thereby providing a more accurate insight into the phenomenology of the transfers and the losses of exergy in supersonic ejectors.
- With regard to the development of simplified models of supersonic ejectors, this thesis offers some interesting perspectives. As mentioned above, the compound-choking theory seems to be more suited than the Fabri-choking assumption to model the choking mechanism in such devices. However, almost all state-of-the-art 1D models are based on the Fabri-choking theory. Hence, an appealing research perspective consists in modifying the choking condition used in simplified models. In particular, if real gas effects and 2D features in the primary stream are taken into account, the resulting model should outperform all of its competitors regarding the prediction of the entrainment ratio in the on-design regime. Moreover, the use of the compound-choking theory – even in its simplest form – in existing 1D models should also improve their ability to predict the critical back-pressure.
- Finally, with regard to the experimental installation, some further improvements of the setup could be made. As mentioned in Chapter 6, in the upgraded facility, the primary nozzle is made of two symmetrical pieces that are completely independent. Even though this design offers the advantage of yielding a purely two-dimensional geometry, it may raise a problem regarding the reproducibility of the experimental conditions. Indeed, the relative position of both pieces may slightly vary when disassembling and

reassembling the experimental installation because of some play in the fixings. This results in slight modifications of the primary nozzle geometry. As the latter has a significant impact on the flow within the ejector, this may lead to relatively large variations in the experimental measurements between two assembling phases. Hence, it could be interesting to enhance the experimental setup in order to fix this issue. Regarding research perspectives for experimental investigations, performing Particle Image Velocimetry (PIV) in the ejector would provide valuable information at the local scale, notably for the validation of the numerical results of LES.

Acknowledgements

The present work benefited from the computational resources provided by the supercomputing facilities of the Université catholique de Louvain (CISM/UCL) and the Consortium des Équipements de Calcul Intensif en Fédération Wallonie Bruxelles (CÉCI) funded by the Fond de la Recherche Scientifique de Belgique (F.R.S.-FNRS) under Convention No. 2.5020.11.

APPENDIX

COMPOUND WAVES

A.1 Compound Waves

In this appendix, an analytical expression for the velocity of a weak pressure disturbance in a compound-compressible nozzle flow is derived. This analysis may be found in the work of Bernstein et al. [14] for the case of perfect gases with different thermodynamic properties. In what follows, all the streams are assumed to have identical thermodynamic properties.

According to Bernstein et al., the compound-choking phenomena can be explained by examining the effects of small pressure disturbances on the flow [14]. Figure A.1 shows the diagram of such a disturbance. Consistently with one-dimensional theory, the flow area is taken as constant in wave calculations. To respect the condition of equal static pressures at the stream interfaces, this pressure disturbance cannot propagate at different absolute velocities in each stream. As a consequence, the wave must be continuous and must travel as a single compound wave. As mentioned in [14], the pressure rise across the wave cannot vary from stream to stream, although the wave is not necessarily plane.

The absolute terminal velocity in the upstream direction of the compound wave is designated by α . Hence, $\alpha > 0$ corresponds to *compound-*

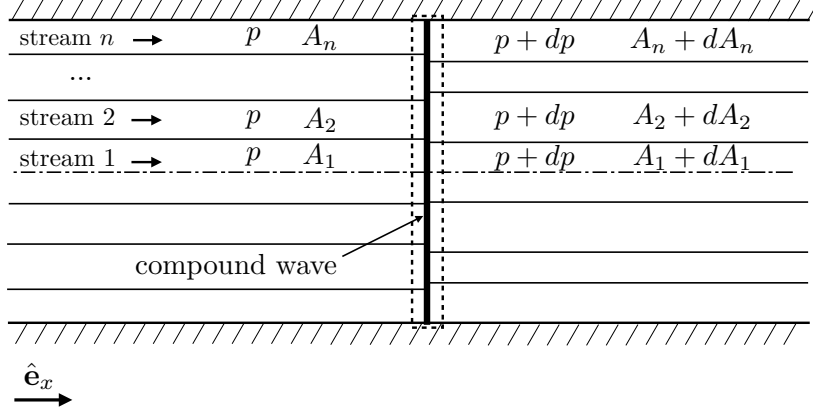


Figure A.1: Upstream moving compound wave in terminal state (adapted from [14]).

subsonic flow; $\alpha = 0$ corresponds to *compound-sonic* flow and $\alpha < 0$ corresponds to *compound-supersonic* flow.

A way of deriving an analytical expression for the compound wave velocity, α , consists in treating each stream separately as a flexible tube and to write the conservation of mass, momentum and entropy across the compound wave. In what follows, it is proposed to write these equations in the frame of reference of the compound wave. If V_i is the absolute velocity of the i th stream, its relative velocity in the frame of reference of the compound wave, noted V'_i , will be given by $V'_i = \alpha + V_i$. Note that the Mach number of the i th stream, M_i , is still based on the absolute velocity, i.e.

$$M_i = \frac{V_i}{\sqrt{\gamma RT_i}}. \quad (\text{A.1})$$

such that

$$V'_i = \alpha + M_i \sqrt{\gamma RT_i}. \quad (\text{A.2})$$

The conservation of mass may be written, for the i th stream and in the frame of reference of the compound wave, as

$$\rho_i V'_i A_i = (\rho_i + d\rho_i)(V'_i - dV_i)(A_i + dA_i), \quad (\text{A.3})$$

where the variation of the velocity across the wave is taken as $-dV_i$, consistently with wave calculation of common textbooks (see for example [73]). This leads to

$$\frac{dV_i}{V'_i} = \frac{dA_i}{A_i} + \frac{d\rho_i}{\rho_i}. \quad (\text{A.4})$$

The conservation of momentum of the i th stream may be expressed, in the frame of reference of the compound wave, as

$$pA_i - (p + dp)(A_i + dA_i) + pdA_i = \rho_i V_i' A_i [(V_i' - dV_i) - V_i'] . \quad (\text{A.5})$$

Note that one must not forget the contribution, in the x-direction, of the pressure force acting on the side surface, and which is obtained as pdA_i . This equation may be simplified as follows

$$dpA_i = \rho_i V_i' A_i dV_i , \quad (\text{A.6})$$

and finally

$$\frac{dp}{p} = \frac{\rho_i V_i'^2}{p} \frac{dV_i}{V_i'} . \quad (\text{A.7})$$

For a perfect gas, the following equation must also hold in each stream

$$p = \rho_i R T_i , \quad (\text{A.8})$$

which leads to

$$\frac{d\rho_i}{\rho_i} = \frac{dp}{p} - \frac{dT_i}{T_i} . \quad (\text{A.9})$$

Finally, the conservation of entropy in the i th stream may be written, under the perfect gas assumption, as

$$ds_i = c_p \frac{dT_i}{T_i} - R \frac{dp}{p} = 0 , \quad (\text{A.10})$$

resulting in the following equation

$$\frac{dp}{p} = \frac{\gamma}{\gamma - 1} \frac{dT_i}{T_i} . \quad (\text{A.11})$$

By combining Eqs. (A.9) and (A.11), one may obtain

$$\frac{d\rho_i}{\rho_i} = \frac{1}{\gamma} \frac{dp}{p} . \quad (\text{A.12})$$

Inserting this equation into the mass equation, Eq. (A.4), leads to

$$\frac{dV_i}{V_i'} = \frac{dA_i}{A_i} + \frac{1}{\gamma} \frac{dp}{p} . \quad (\text{A.13})$$

This equation may be inserted into the momentum equation, Eq. (A.7) to give

$$\frac{\rho_i V_i'^2}{p} \left(\frac{dA_i}{A_i} + \frac{1}{\gamma} \frac{dp}{p} \right) = \frac{dp}{p} . \quad (\text{A.14})$$

Noting that

$$\frac{\rho_i}{p} = \frac{1}{RT_i}, \quad (\text{A.15})$$

and inserting the expression of the relative velocity, V_i' , given in Eq. (A.2), one finally obtains the following equation

$$\gamma \left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2 \left(\frac{dA_i}{A_i} + \frac{1}{\gamma} \frac{dp}{p} \right) = \frac{dp}{p}, \quad (\text{A.16})$$

which is valid for each stream. By regrouping the terms involving the ratio dp/p , this equation may be rewritten as follows

$$\gamma \left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2 \frac{dA_i}{A_i} = \frac{dp}{p} \left[1 - \left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2 \right], \quad (\text{A.17})$$

or

$$dA_i = \frac{A_i}{\gamma} \frac{dp}{p} \left(\frac{1}{\left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2} - 1 \right) \quad (\text{A.18})$$

By summing over all streams, one obtains (since $\sum_{i=1}^n dA_i = 0$)

$$0 = \sum_{i=1}^n \frac{A_i}{\gamma} \left(\frac{1}{\left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2} - 1 \right). \quad (\text{A.19})$$

This last equation may be combined with the definition of the compound-flow indicator, β , given in Eq. (5.1), to yield

$$\beta = \sum_{i=1}^n \frac{A_i}{\gamma} \left(\frac{1}{M_i^2} - \frac{1}{\left(\frac{\alpha}{\sqrt{\gamma RT_i}} + M_i \right)^2} \right). \quad (\text{A.20})$$

As mentioned by Bernstein et al., examination of this equation shows that the compound wave velocity, α , and the compound flow indicator, β , must always have the same sign [14].

BIBLIOGRAPHY

- [1] A.L. Addy, J.C. Dutton, and C.D. Mikkelsen. *Supersonic Ejector-Diffuser Theory and Experiments*. Department of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign, 1981.
- [2] O. B. Adeyinka and G. F. Naterer. Modeling of entropy production in turbulent flows. *Journal of Fluids Engineering*, 126(6):893–899, 03 2005.
- [3] N. M. Al-Najem, M.A. Darwish, and F.A. Youssef. Thermovapor compression desalters: energy and availability analysis of single- and multi-effect systems. *Desalination*, 110(3):223 – 238, 1997.
- [4] M. Alperin and J. J. Wu. Thrust augmenting ejectors, Part I. *AIAA Journal*, 21(10):1428–1436, 1983.
- [5] M. Alperin and J. J. Wu. Thrust augmenting ejectors, Part II. *AIAA Journal*, 21(12):1698–1706, 1983.
- [6] I. Asproulas. *RANS Modelling for Compressible Turbulent Flows Involving Shock Wave Boundary Layer Interactions*. PhD thesis, University of Manchester, 2014.
- [7] K. Banasiak, M. Palacz, A. Hafner, Z. Buliski, J. Smoka, A. J. Nowak, and A. Fic. A CFD-based investigation of the energy performance of two-phase R744 ejectors to recover the expansion work in refrigeration systems: An irreversibility analysis. *International Journal of Refrigeration*, 40:328 – 337, 2014.
- [8] S. Barre and J.P. Bonnet. Detailed experimental study of a highly compressible supersonic turbulent plane mixing layer and comparison with most recent DNS results: towards an accurate description of compressibility effects in supersonic free shear flows. *International Journal of Heat and Fluid Flow*, 51:324 – 334, 2015.

- [9] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier. CFD-experiments integration in the evaluation of six turbulence models for supersonic ejectors modeling. *Integrating CFD and Experiments Conference, Glasgow, UK*, 2003.
- [10] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier. Numerical and experimental investigations on supersonic ejectors. *International Journal of Heat and Fluid Flow*, 26(1):56 – 70, 2005.
- [11] Y. Bartosiewicz, Z. Aidoun, and Y. Mercadier. Numerical assessment of ejector operation for refrigeration applications based on cfd. *Applied Thermal Engineering*, 26:604–612, 04 2006.
- [12] A. Bejan. Fundamentals of exergy analysis, entropy generation minimization, and the generation of flow architecture. *International Journal of Energy Research*, 26(7):0–43, 2002.
- [13] F. Duncan Berkeley. Ejectors have a wide range of uses. *Petroleum Refiner*, 37:95–100, December 1958.
- [14] A. Bernstein, W. H. Heiser, and C. Hevenor. Compound-compressible nozzle flow. *Journal of Applied Mechanics*, 34(3):548–554, 09 1967.
- [15] G. Besagni and F. Inzoli. Computational fluid-dynamics modeling of supersonic ejectors: Screening of turbulence modeling approaches. *Applied Thermal Engineering*, 117:122 – 144, 2017.
- [16] G. Besagni, R. Mereu, and F. Inzoli. Ejector refrigeration: A comprehensive review. *Renewable and Sustainable Energy Reviews*, 53:373 – 407, 2016.
- [17] A. Bouhanguel, P. Desevaux, and E. Gavignet. Flow visualization in supersonic ejectors using laser tomography techniques. *International Journal of Refrigeration*, 34(7):1633 – 1640, 2011. Ejector Technology.
- [18] A. Bouhanguel, P. Desevaux, and E. Gavignet. Visualization of flow instabilities in supersonic ejectors using large eddy simulation. *Journal of Visualization*, 18(1):17–19, Feb 2015.
- [19] W. Chen, D. Chong, J. Yan, and J. Liu. Numerical optimization on the geometrical factors of natural gas ejectors. *International Journal of Thermal Sciences*, 50(8):1554 – 1561, 2011.

- [20] W. Chen, M. Liu, D. Chong, J. Yan, A. B. Little, and Y. Bartosiewicz. A 1d model to predict ejector performance at critical and sub-critical operational regimes. *International Journal of Refrigeration*, 36(6):1750 – 1761, 2013.
- [21] W. Chen, C. Shi, S. Zhang, H. Chen, D. Chong, and J. Yan. Theoretical analysis of ejector refrigeration system performance under overall modes. *Applied Energy*, 185:2074 – 2084, 2017. Clean, Efficient and Affordable Energy for a Sustainable Future.
- [22] D. Chong, M. Hu, W. Chen, J. Wang, J. Liu, and J. Yan. Experimental and numerical analysis of supersonic air ejector. *Applied Energy*, 130(C):679–684, 2014.
- [23] D. Chong, J. Yan, G. Wu, and J. Liu. Structural optimization and experimental investigation of supersonic ejectors for boosting low pressure natural gas. *Applied Thermal Engineering*, 29(14):2799 – 2807, 2009.
- [24] S.K. Chou, P.R. Yang, and C. Yap. Maximum mass flow ratio due to secondary flow choking in an ejector refrigeration system. *International Journal of Refrigeration*, 24(6):486 – 499, 2001.
- [25] K. Chunnanond and S. Aphornratana. Ejectors: applications in refrigeration technology. *Renewable and Sustainable Energy Reviews*, 8(2):129–155, 2004.
- [26] K. Chunnanond and S. Aphornratana. An experimental investigation of a steam ejector refrigerator: the analysis of the pressure profile along the ejector. *Applied Thermal Engineering*, 24(23):311 – 322, 2004.
- [27] S. Croquer, S. Poncet, and Z. Aidoun. Turbulence modeling of a single-phase r134a supersonic ejector. part 1: Numerical benchmark. *International Journal of Refrigeration*, 61:140 – 152, 2016.
- [28] S. Croquer, S. Poncet, and Z. Aidoun. Turbulence modeling of a single-phase R134a supersonic ejector. part 2: Local flow structure and exergy analysis. *International Journal of Refrigeration*, 61:153 – 165, 2016.
- [29] S. Croquer, S. Poncet, and N. Galanis. Comparison of ejector predicted performance by thermodynamic and cfd models. *International Journal of Refrigeration*, 68:28 – 36, 2016.

- [30] J. Garcia del Valle, J. M. Siz Jabardo, F. Castro Ruiz, and J. San Jos Alonso. A one dimensional model for the determination of an ejector entrainment ratio. *International Journal of Refrigeration*, 35(4):772 – 784, 2012.
- [31] J. Garcia del Valle, J. Sierra-Pallares, P. Garcia Carrascal, and F. Castro Ruiz. An experimental and computational study of the flow pattern in a refrigerant ejector. validation of turbulence models and real-gas effects. *Applied Thermal Engineering*, 89:795 – 811, 2015.
- [32] P. Desevaux. A method for visualizing the mixing zone between two co-axial flows in an ejector. *Optics and Lasers in Engineering*, 35(5):317 – 323, 2001.
- [33] J. Fabri and R. Siestrunk. Supersonic air ejectors. *Advances in Applied Mechanics*, 5:1–34, 1958.
- [34] Fluent Inc. Determining turbulence parameters, 2016. Accessed: 2016-03-01.
- [35] J. Gagan, K. Smierciew, D. Butrymowicz, and J. Karwacki. Comparative study of turbulence models in application to gas ejectors. *International Journal of Thermal Sciences*, 78:9 – 15, 2014.
- [36] N. Galanis and M. Sorin. Ejector design and performance prediction. *International Journal of Thermal Sciences*, 104:315 – 329, 2016.
- [37] C. Geuzaine and J.-F. Remacle. Gmsh: A 3-D finite element mesh generator with built-in pre-and post-processing facilities. *International Journal for Numerical Methods in Engineering*, 79(11):1309–1331, 2009.
- [38] M. Ghazikhani, I. Khazaee, and S. Vahidifar. Exergy analysis of two humidification process methods in air-conditioning systems. *Energy and Buildings*, 124:129 – 140, 2016.
- [39] G. Grazzini, A. Milazzo, and D. Paganini. Design of an ejector cycle refrigeration system. *Energy Conversion and Management*, 54(1):38 – 46, 2012.
- [40] J.E. Green. Interactions between shock waves and turbulent boundary layers. *Progress in Aerospace Sciences*, 11:235 – 340, 1970.

- [41] C. J. Greenshields, H. G. Weller, L. Gasparini, and J. M. Reese. Implementation of semi-discrete, non-staggered central schemes in a colocated, polyhedral, finite volume framework, for high-speed viscous flows. *International Journal for Numerical Methods in Fluids*, 63:1–21, 2010.
- [42] S. He, Y. Li, and R.Z. Wang. Progress of mathematical modeling on ejectors. *Renewable and Sustainable Energy Reviews*, 13(8):1760 – 1780, 2009.
- [43] K. R. Hedges and P. G. Hill. Compressible flow ejectors: Part ii—flow field measurements and analysis. *Journal of Fluids Engineering*, 96(3):282–288, 09 1974.
- [44] A. Hemidi, F. Henry, S. Leclaire, J.-M. Seynhaeve, and Y. Bartosiewicz. CFD analysis of a supersonic air ejector. part I: Experimental validation of single-phase and two-phase operation. *Applied Thermal Engineering*, 29(89):1523 – 1531, 2009.
- [45] A. Hemidi, F. Henry, S. Leclaire, J.-M. Seynhaeve, and Y. Bartosiewicz. CFD analysis of a supersonic air ejector. part II: Relation between global operation and local flow features. *Applied Thermal Engineering*, 29(1415):2990 – 2998, 2009.
- [46] C. Hirsch. *Numerical computation of internal and external flows*, volume 2. John Wiley & Sons, Ltd., 2006.
- [47] H. J. Hoge and R. A. Segars. Choked flow - a generalization of the concept and some experimental data. *AIAA Journal*, 3(12):2177–2183, 2017/12/14 1965.
- [48] B.J. Huang, J.M. Chang, C.P. Wang, and V.A. Petrenko. A 1-D analysis of ejector performance. *International Journal of Refrigeration*, 22(5):354 – 364, 1999.
- [49] M. Ji, T. Utomo, J. Woo, Y. Lee, H. Jeong, and H. Chung. CFD investigation on the flow structure inside thermo vapor compressor. *Energy*, 35(6):2694 – 2702, 2010.
- [50] J.H. Keenan, E.P. Newman, and F. Lustwerk. An investigation of ejector design by analysis and experiment. *Journal of Applied Mechanics*, Trans ASME 72:209–309, 1950.
- [51] F. Kong and H.D. Kim. Analytical and computational studies on the performance of a two-stage ejectordiffuser system. *International Journal of Heat and Mass Transfer*, 85:71 – 87, 2015.

- [52] A. Kurganov, S. Noelle, and G. Petrova. Semidiscrete central-upwind schemes for hyperbolic conservation laws and Hamilton–Jacobi equations. *SIAM J. Sci. Comput.*, 23(3):707–740, March 2001.
- [53] E. F. Kurtz, Jr. Theoretical model for predicting steam-ejector performance. *Journal of Engineering for Industry*, 98(2):645–651, 05 1976.
- [54] O. Lamberts, P. Chatelain, and Y. Bartosiewicz. Numerical and experimental evidence of the Fabri-choking in a supersonic ejector. *International Journal of Heat and Fluid Flow*, 69:194 – 209, 2018.
- [55] W. E. Lear, G. M. Parker, and S. A. Sherif. Analysis of two-phase ejectors with Fabri choking. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 216(5):607–621, 2002.
- [56] W. E. Lear, S. A. Sherif, and G. M. Parker. Effect of Fabri choking on the performance of two-phase jet pumps. In *Collection of Technical Papers. 35th Intersociety Energy Conversion Engineering Conference and Exhibit (IECEC) (Cat. No.00CH37022)*, volume 2, pages 1190–1197 vol.2, 2000.
- [57] Z. Ma, H. Bao, and A. P. Roskilly. Thermodynamic modelling and parameter determination of ejector for ejection refrigeration systems. *International Journal of Refrigeration*, 75:117 – 128, 2017.
- [58] S. Manservigi and F. Menghini. A CFD four parameter heat transfer turbulence model for engineering applications in heavy liquid metals. *International Journal of Heat and Mass Transfer*, 69:312 – 326, 2014.
- [59] F. Mazzelli, A. B. Little, S. Garimella, and Y. Bartosiewicz. Computational and experimental analysis of supersonic air ejector: Turbulence modeling and assessment of 3D effects. *International Journal of Heat and Fluid Flow*, 56:305 – 316, 2015.
- [60] F. Mazzelli and A. Milazzo. Performance analysis of a supersonic ejector cycle working with r245fa. *International Journal of Refrigeration*, 49:79 – 92, 2015.
- [61] R. K. McGovern, G. Prakash Narayan, and J. H. Lienhard. Analysis of reversible ejectors and definition of an ejector efficiency. *International Journal of Thermal Sciences*, 54:153 – 166, 2012.

- [62] J. Meyers and C. Meneveau. Flow visualization using momentum and energy transport tubes and applications to turbulent flow in wind farms. *Journal of Fluid Mechanics*, 715:335–358, 2013.
- [63] J. T. Munday and D. F. Bagster. A new ejector theory applied to steam jet refrigeration. *Industrial & Engineering Chemistry Process Design and Development*, 16(4):442–449, 1977.
- [64] Langley Research Center NASA. Implementing turbulence models into the compressible RANS equations, 2014. Accessed: 2016-11-01.
- [65] H. Pearson, J. B. Holliday, and S. F. Smith. A theory of the cylindrical ejector supersonic propelling nozzle. *The Journal of the Royal Aeronautical Society*, 62(574):746751, 1958.
- [66] K. Pianthong, W. Seehanam, M. Behnia, T. Sriveerakul, and S. Aphornratana. Investigation and improvement of ejector refrigeration system using computational fluid dynamics technique. *Energy Conversion and Management*, 48(9):2556 – 2564, 2007.
- [67] S. M.V. Rao and G. Jagadeesh. Novel supersonic nozzles for mixing enhancement in supersonic ejectors. *Applied Thermal Engineering*, 71(1):62 – 71, 2014.
- [68] S. M.V. Rao and G. Jagadeesh. Observations on the non-mixed length and unsteady shock motion in a two dimensional supersonic ejector. *Physics of Fluids*, 26(3):036103, 2014.
- [69] N. Ruangtrakoon, T. Thongtip, S. Aphornratana, and T. Sriveerakul. CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle. *International Journal of Thermal Sciences*, 63:133 – 145, 2013.
- [70] E. Rusly, L. Aye, W.W.S. Charters, and A. Ooi. CFD analysis of ejector in a combined ejector cooling system. *International Journal of Refrigeration*, 28:1092–1101, 11 2005.
- [71] N. Bilir Sag, H.K. Ersoy, A. Hepbasli, and H.S. Halkaci. Energetic and exergetic comparison of basic and ejector expander refrigeration systems operating under the same external conditions and cooling capacities. *Energy Conversion and Management*, 90:184 – 194, 2015.
- [72] G.S. Settles. *Schlieren and Shadowgraph Techniques: Visualizing Phenomena in Transparent Media*. Springer Berlin Heidelberg, 2001.

- [73] A.H. Shapiro. *The dynamics and thermodynamics of compressible fluid flow*, volume 1. Ronald Press Co., 1953.
- [74] N. Sharifi and M. Boroomand. An investigation of thermo-compressor design by analysis and experiment: Part 1. validation of the numerical method. *Energy Conversion and Management*, 69:217 – 227, 2013.
- [75] J. Sierra-Pallares, J. Garcia del Valle, P. Garcia Carrascal, and F. Castro Ruiz. A computational study about the types of entropy generation in three different R134a ejector mixing chambers. *International Journal of Refrigeration*, 63:199 – 213, 2016.
- [76] T. Sriveerakul, S. Aphornratana, and K. Chunnanond. Performance prediction of steam ejector using computational fluid dynamics: Part 1. validation of the CFD results. *International Journal of Thermal Sciences*, 46(8):812 – 822, 2007.
- [77] T. Sriveerakul, S. Aphornratana, and K. Chunnanond. Performance prediction of steam ejector using computational fluid dynamics: Part 2. flow structure of a steam ejector influenced by operating pressures and geometries. *International Journal of Thermal Sciences*, 46(8):823 – 833, 2007.
- [78] D.-W. Sun and I. Eames. Recent developments in the design theories and applications of ejectors. *Journal of the Institute of Energy*, 68:65–79, 1995.
- [79] W. Sutherland. The viscosity of gases and molecular force. *Philosophical Magazine*, (5):507–531, 1893.
- [80] T. G. Tillman and W. M. Precz, Jr. Thrust characteristics of a supersonic mixer ejector. *Journal of Propulsion and Power*, 11(5):931–937, 1995.
- [81] S. Varga, A. C. Oliveira, and B. Diaconu. Numerical assessment of steam ejector efficiencies using CFD. *International Journal of Refrigeration*, 32(6):1203 – 1211, 2009.
- [82] H. G. Weller, G. Tabor, H. Jasak, and C. Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Comput. Phys.*, 12(6):620–631, November 1998.
- [83] D.C. Wilcox. *Turbulence Modeling for CFD (Third Edition)*. DCW Industries, Incorporated, 2006.

- [84] R. L. Yadav and A. W. Patwardhan. Design aspects of ejectors: Effects of suction chamber geometry. *Chemical Engineering Science*, 63(15):3886 – 3897, 2008.
- [85] J. Yan, W. Cai, and Y. Li. Geometry parameters effect for air-cooled ejector cooling systems with r134a refrigerant. *Renewable Energy*, 46(Supplement C):155 – 163, 2012.
- [86] X. Yang, X. Long, and X. Yao. Numerical investigation on the mixing process in a steam ejector with different nozzle structures. *International Journal of Thermal Sciences*, 56:95 – 106, 2012.
- [87] R. Yapc and H.K. Ersoy. Performance characteristics of the ejector refrigeration system based on the constant area ejector flow model. *Energy Conversion and Management*, 46(18):3117 – 3135, 2005.
- [88] M.T. Zegenhagen and F. Ziegler. A one-dimensional model of a jet-ejector in critical double choking operation with R134a as a refrigerant including real gas effects. *International Journal of Refrigeration*, 55:72 – 84, 2015.
- [89] Y. Zhu, C. Cai, C. Wen, and Y. Li. Shock circle model for ejector performance evaluation. *Energy Conversion and Management*, 48(9):2533 – 2541, 2007.
- [90] Y. Zhu, W. Cai, C. Wen, and Y. Li. Numerical investigation of geometry parameters for design of high performance ejectors. *Applied Thermal Engineering*, 29(5):898 – 905, 2009.
- [91] Y. Zhu and P. Jiang. Experimental and numerical investigation of the effect of shock wave characteristics on the ejector performance. *International Journal of Refrigeration*, 40:31 – 42, 2014.
- [92] R.D. Zucker and O. Biblarz. *Fundamentals of Gas Dynamics*. Wiley, 2002.