

École polytechnique de Louvain

Experimental and numerical characterization of an axisymmetric supersonic ejector

Authors: **Nicolas BAGUET, Guillaume QUERINJEAN**
Supervisors: **Yann BARTOSIEWICZ, Grégoire WINCKELMANS**
Readers: **Philippe CHATELAIN, Romain DEBROYER, Matthieu
DUPONCHEEL**
Academic year 2022–2023
Master [120] in Mechanical Engineering

Acknowledgments

Réaliser un mémoire à deux n'est pas toujours une tâche facile, mais nous avons réussi à nous écouter, concilier les avis parfois divergents et prendre chaque décision ensemble. Notre travail a ainsi toujours été équilibré, et nous sommes très heureux d'avoir pu mener à bien ce projet de longue haleine en binôme. Notre complémentarité a été indispensable dans un mémoire comportant un tel ensemble de problèmes de natures fondamentalement différentes, et a apporté beaucoup de stabilité à notre cheminement intellectuel.

Cependant, nous avons tous les deux pu compter sur nos promoteurs, les Professeurs Yann Bartosiewicz et Grégoire Winckelmans qui, au cours de nos réunions, ont su nous prodiguer de bons conseils et indiquer des pistes de réflexion pour mener à bien nos analyses. Mais surtout, nous les remercions pour leur confiance accordée à notre égard concernant la réalisation du projet expérimental. Malgré les obstacles financiers auxquels nous nous sommes heurtés, le projet a pu se concrétiser et à un mois près, aurait pu se terminer entièrement.

Nous tenons également à remercier grandement Romain pour sa disponibilité, ses réflexions très constructives et sa bienveillance tout au long de l'année. Nous lui souhaitons bonne chance et beaucoup de réussite pour la suite de son doctorat, et espérons que notre travail pourra y apporter un petit quelque chose. Nous remercions également les Professeurs P. Chatelain et M. Papalexandris pour leurs réponses à nos questions.

Le projet expérimental n'aurait jamais pu voir le jour sans l'aide premièrement de Benoît Herman pour sa connaissance détaillée de la conception mécanique ainsi que son support pour terminer le projet malgré les difficultés rencontrées. Mais c'est également toute l'équipe du CREDEM que nous aimerions remercier pour leur participation à notre projet. Merci à Victor, Vincent, Alban, Hervé, Julien et plus spécialement à Frank Hesbois et Arnaud Lion pour leur expertise et disponibilité.

Finalement, être deux aide mais ne suffit pas, c'est pourquoi nous aimerions également remercier individuellement les personnes de notre entourage qui nous ont suivi et supporté tout au long de cette année et plus globalement de nos études.

(Nicolas) Je commencerai par remercier mes parents, pour leur support depuis toujours et leur confiance envers moi. Chacun à sa manière m'a soutenu et la réussite de mes études vient en partie d'eux. Merci Papa pour tes conseils quant à l'écriture du manuscrit. Je remercie également mon frère pour ces moments de détente en sa compagnie. Je finirai par Charlotte, ma copine, qui a été le meilleur support que je puisse avoir tout au long de ces années.

(Guillaume) Je tiens à remercier mes parents, mon frère, ma soeur et Léa pour leur soutien tout au long de mes études, et en particulier cette dernière année ponctuée de joies mais aussi de moments plus difficiles. Merci à eux d'avoir tenu le temps de l'explication du fonctionnement d'un éjecteur supersonique (presque) sans bailler. Enfin, un grand merci à toutes les personnes qui m'ont aidé, encouragé et changé les idées lors du sprint final (quoique plus proche du marathon) qu'était la rédaction de ce travail.

Abstract

Supersonic ejectors are passive devices that use the high stagnation pressure of a driver fluid stream to induce the entrainment of a secondary stream (which is at a lower stagnation pressure), owing to the viscous entrainment effect. In this process, both streams are mixed, compressed and eventually yield a uniform flow at an intermediate stagnation pressure.

Used for more than a century in numerous applications (from aerospace engineering to water desalination cycles and vacuum generators), supersonic ejectors are chiefly envisioned as an alternative to traditional mechanical compressors in air-conditioning systems. The need for renewable and diversified energy sources makes supersonic ejectors a promising technology. Indeed, when used in refrigeration cycles, they offer the advantage of using heat as main energy input, allowing the use of low-grade thermal energy such as waste heat (i.e., heat resulting from industrial processes that would be wasted otherwise), geothermal or solar energy. In addition, as the mixing and compression of the fluid do not require any moving parts, maintenance and operation costs are low.

However, shortcomings linked to their inherent nature have prevented ejectors from becoming an economically attractive technology in the air-conditioning market. Their low coefficient of performance and low versatility regarding their operating conditions are limiting factors to their industrial progress. To tackle this issue, scientific research has been conducted for several decades, but some ejector phenomena still suffer a lack of comprehension. The principal stakes are the blockage (choking) of the entrained mass flow rate and the mixing phenomena.

Following the research conducted in previous Ph.D. and Master's theses at UCLouvain (both experimental and numerical studies), this Master's thesis is devoted to the study of supersonic air ejectors with an axisymmetric geometry. In contrast, previous works used mainly a rectangular geometry. The purpose of this work is twofold:

First, it aims at designing a novel axisymmetric ejector, with a geometry optimizing its performance, and to embed it in a high-quality experimental setup. The mechanical design has to overcome the challenges that arise from the axisymmetric geometry.

Second, it pursues the objective of performing a full numerical characterization of the various phenomena at stake within the designed ejector. Using 2D steady-state RANS numerical methods and a state-of-the-art computational domain, a numerical database composed of simulations spanning a broad range of operating conditions is obtained, and exploited using adequate post-processing tools. An in-depth investigation of the choking phenomenon and other outstanding flow features is then performed, aiming at shedding light on their origin and characteristics. Finally, a quantitative assessment of the mixing phenomenon and associated transfers is carried out.

Contents

Acknowledgments	i
Abstract	ii
List of Abbreviations and Symbols	v
1 Introduction	1
1.1 Context	1
1.2 Ejector phenomena	3
1.2.1 Ejector operation and performance	3
1.2.2 Theories for the secondary choking phenomenon	5
1.2.3 Simplified models	6
1.2.4 CFD for supersonic ejectors	7
1.3 Objectives and structure of the manuscript	8
1.3.1 Experimental investigation	9
1.3.2 Numerical investigation	9
1.3.3 Structure of the manuscript	10
2 Experimental setup and numerical methods	11
2.1 Introduction	11
2.2 Ejector geometry and design	11
2.2.1 Experimental setup	12
2.2.2 Mixing duct length optimization	15
2.2.3 Design process and methods	17
2.2.4 Advancement of the assembly	21
2.3 Numerical methods and setup	22
2.3.1 Compressible RANS equations for axisymmetric flows	22
2.3.2 Fluid properties	24
2.3.3 Flow solver	25
2.3.4 Computational domain	26
2.3.5 Boundary conditions	27
3 Flow topology and choking phenomenon	29
3.1 Introduction	29
3.2 Numerical database: characteristic curves and identification of two working regimes	30
3.2.1 Impact of the operating conditions: characteristic curves	30
3.2.2 Identification of two regimes	32
3.3 General Flow topology features	33
3.3.1 Preliminary remarks	33
3.3.2 Study of the flow fields	34
3.3.3 Impact of p_{01}/p_{02} on the supersonic flow topology	37
3.3.4 Preliminary mixing analysis via velocity and pressure profiles	40
3.3.5 Relative importance of viscous phenomena	42
3.4 The compound choking theory: a universal explanation for the critical operation of supersonic ejectors	44
3.4.1 Set of Mach number fields	44
3.4.2 Analogy to the classical nozzle and definition of the choked region	47

3.4.3	Identification of the choking section	49
3.5	The compound-choking model	51
3.5.1	Original compound-choking model	51
3.5.2	Refining of the compound-choking model	54
3.5.3	Uniform pressure assumption	56
3.6	Detailed insight on Fabri-patterns: onset conditions and impact on the choking phenomenon	57
3.6.1	Numerical evidence of the Fabri-pattern	57
3.6.2	Effect of wall friction	59
3.6.3	Inviscid simulation	62
3.6.4	Origin of the Fabri-pattern: quantitative investigation	63
3.7	Conclusion: A broader interpretation of the choking phenomenon	67
4	Ejector performance: one-dimensional models, transport and mixing phenomena	71
4.1	Introduction	71
4.2	Comparison with 1D thermodynamic models	72
4.2.1	Fabri-choking model	72
4.2.2	Isentropic compound-choking model	73
4.2.3	Non-isentropic compound-choking model	74
4.3	Transport tubes as visualization of transport phenomena	75
4.3.1	Analytical tools	75
4.3.2	Numerical results	78
4.3.3	Exergy footprint	83
4.4	Transfers between the primary and secondary streams	85
4.4.1	Introduction	85
4.4.2	Analytical tools	85
4.4.3	Local and Cumulative transfers	86
4.5	Exergetic losses and balance of a supersonic ejector	88
4.5.1	Exergy losses in the secondary stream	88
4.5.2	Exergy efficiency of a supersonic ejector	91
5	Conclusion and perspectives	93
A	Miscellaneous analytical expressions	95
A.1	Expression of the viscous stress tensor terms	95
A.2	Expression of the fluxes	96
A.3	Expression of the viscous dissipation term	96
B	Mach number fields	97
C	General flow topology	104
C.1	Velocity and pressure profiles	104
C.2	Wall pressure Profile	104
C.3	Criterion for the definition of a unique choking section x_c	108
C.4	Fully inviscid simulations	111
D	Confirmation of the compound choking on a reference case	112
E	Ejector Performance	114
E.1	Fabri model fit	114
E.2	Local flux and cumulative flux transfers between the primary and sec- ondary streams	114
F	Experimental setup: Additional drawings	116

List of Abbreviations and Symbols

Abbreviations

CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulation
LES	Large Eddy Simulation
NXP	Nozzle Exit Position
RANS	Reynolds- Averaged Navier-Stokes equations
VS×2	Two fold Vertical Stretching of the geometry

Alpha numeric symbols

A_i	Cross-section area of the i^{th} stream	$[m^2]$
A_1	Primary nozzle exit area	$[m^2]$
A_t	Primary nozzle throat area	$[m^2]$
c_p, c_v	Specific heat capacity at constant pressure/volume	$[J/(kg\ K)]$
DN	Nominal pipe diameter	$[mm]$
e	Specific internal energy	$[J/kg]$
e_t	Specific total internal energy	$[J/kg]$
$\bar{\mathbf{F}}_\xi$	Mean-flow total exergy flux	$[kg/s^3]$
$\bar{\mathbf{F}}_K$	Mean-flow kinetic energy flux	$[kg/s^3]$
$\bar{\mathbf{F}}_m$	Total linear momentum flux	$[kg/(m\ s^2)]$
$\bar{\mathbf{F}}_q$	Flux of exergy associated with the mean heat flux	$[kg/(m\ s^2)]$
$\bar{f}_\alpha$	Dimensionless local flux of various $\alpha \in [m, K, \xi, q]$	$[-]$
h	Specific static enthalpy	$[J/kg]$
h_t	Specific total enthalpy	$[J/kg]$
I_α	Cumulative dimensionless transfer of various fluxes $\alpha \in [m, K, \xi, q]$	$[-]$
K	Specific mean-flow kinetic energy	$[J/kg]$
k	Turbulence kinetic energy	$[J/kg]$
$\mathbf{I}$	Identity matrix	$[-]$
l_m	Mixing duct length	$[m]$
$\dot{m}$	Mass flow rate	$[kg/s]$
M	Mach number	$[-]$
M_{eq}^∞	Continuous equivalent Mach number	$[-]$
M_{eq}	Equivalent Mach number	$[-]$

M_m	Molar mass	$[kg/mol]$
p	Static pressure	$[Pa]$
p^*	Sonic pressure	$[Pa]$
$p_{0,out}^*$	Critical back pressure	$[Pa]$
$p_{0,out}^b$	Breakdown pressure	$[Pa]$
P_w	Wetted perimeter	$[m]$
Pr	Prandtl number	$[-]$
$\mathbf{q}$	Heat flux vector field	$[J/(m^2s)]$
r	Radial coordinate	$[m]$
R	Specific gas constant	$[J/(K\ kg)]$
R_m	Mixing duct radius	$[m]$
R_1	Primary nozzle outlet radius	$[m]$
Re	Reynolds number	$[-]$
Re_τ	Friction Reynolds number	$[-]$
T	Static temperature	$[K]$
$\mathbf{u}$	Velocity vector field	$[m/s]$
u, v	Axial velocity, Radial velocity	$[m/s]$
x	Axial coordinate - 0 at mixing duct entrance	$[m]$
$x_{\bar{c}}$	Axial position of choking section	$[m]$
y^+	Dimensionless wall distance	$[-]$

Greek letters

α	Slope of off-design curve	$[-]$
β	Compound-flow indicator	$[-]$
γ	Specific heat ratio	$[-]$
δ	Boundary layer thickness	$[m]$
Δ	Dimensionless gain in exergy	$[-]$
Δ	Variation (any variable)	$/$
ϵ	Tolerance criterion for Mach number difference absolute value	$[-]$
ϵ_{p^*}	Back pressure difference indicator of characteristic curve sharpness	$[-]$
η	Efficiency	$[-]$
η_{pol}	Polytropic efficiency	$[-]$
$\theta(x)$	Momentum thickness	$[m]$
θ	Azimuthal coordinate	$[rad]$
$\theta_{s,d}$	Secondary convergent - divergent half angle	$[^\circ]$
κ	Thermal conductivity	$[J/(m\ s\ K)]$
μ	Dynamic viscosity	$[Pa\ s]$
μ_t	Turbulence or eddy viscosity	$[Pa\ s]$
μ	Mach angle	$[^\circ]$
ν	Kinematic viscosity	$[m^2/s]$
π	Local mean-flow total exergy losses due to viscous dissipation	$[-]$
π_θ	Local mean-flow total exergy losses due to heat transfer	$[-]$

Π	Cumulative mean-flow total exergy losses due to viscous dissipation	$[-]$
Π_θ	Cumulative mean-flow total exergy losses due to heat transfer	$[-]$
ρ	Density	$[kg/m^3]$
σ	Entrainment ratio deviation	$[-]$
τ	Viscous stress tensor	$[Pa]$
τ_w	Wall shear stress	$[Pa]$
Φ	Volumetric exergy losses due to viscous dissipation	$[kg\ K/(m\ s^3)]$
Φ_θ	Volumetric exergy losses due to heat transfer	$[kg\ K/(m\ s^3)]$
ξ_t	Specific total exergy	$[J/kg]$
ψ_m	Frictional losses efficiency	$[-]$
ω	Entrainment ratio	$[-]$
ω	Specific dissipation rate	$[1/s]$

Operator

$\overline{\cdot}$	Time average
$\hat{\cdot}$	Density-weighted average
$\sim$	Value computed based on mean-flow quantities

Subscripts

0	Reservoir conditions
t	Stagnation conditions
1	related to primary stream
2	related to secondary stream
out	related to the outlet of ejector
t	related to turbulent variable (superscript also)
w	related to wall conditions
m	related to mixing duct variables

Chapter 1

Introduction

1.1 Context

Supersonic ejectors are passive internal flow devices whose purpose is to transfer momentum and energy from a primary stream towards a secondary — or entrained — stream. The primary flow, at a high stagnation pressure, serves as motive force to induce the entrainment and compression of the secondary flow, which has a lower stagnation pressure. Both streams undergo a mixing process leading to various transfer phenomena, after completion of which they finally are decelerated and compressed, yielding a uniform flow at an intermediate stagnation pressure.

The study and industrial employment of supersonic ejectors date back to the 19th century, with the first use of a low-entrainment ejector in steam locomotives [1, 2] while the first experimental and theoretical results were obtained a century ago [3, 4]. They were, and are used in numerous applications [5], such as thrust augmenters in aerospace propulsion applications [6, 7, 8] (notably in Short-or Vertical Take-off and Landing fighter jets [9]), vacuum generators [10], in chemical, food and process industries (e.g., as pressure boosters in natural gas industry [11]); water desalination cycles [12], fuel cell recirculation systems [13], ejector organic Rankine cycles [14] and in air-conditioning [15]. Specifically, supersonic ejectors are a promising refrigeration technology. In the past decades, the use of air conditioning systems has increased worldwide, and will likely keep on growing in the future — and the energy fraction consumed for thermal comfort consequently as well. The current global energy outlook however requires to adapt our primary energy consumption, both by decreasing the primary energy demand and by promoting renewable energy sources. Using heat as energy source, ejectors can replace compressors that traditionally operate between the evaporator and the condenser of a refrigeration cycle. A classical configuration of an ejector-based refrigeration cycle is illustrated in Figure 1.1.

The basic principle of an ejector refrigeration system is the following: at the exit of the condenser, the liquid refrigerant is separated in two streams. The first one, corresponding to the primary stream, is compressed then evaporated in a boiler, before being expanded in the supersonic nozzle of the ejector, thereby inducing entrainment of the secondary flow. The latter has first extracted heat at the cold source (the evaporator) prior to its suction in the ejector. Both streams end up mixed at the outlet of the ejector.

The main advantage of supersonic ejectors in a refrigeration application lies in the nature of the energy consumption: the heat input used to evaporate the primary flow in the boiler can consist of low-grade thermal energy such as waste heat (e.g., from industrial processes or internal combustion engine exhaust), geothermal energy or solar heat energy. The availability of these energy sources can be leveraged to alleviate the environmental impact of air-conditioning cycles, which traditionally use electricity as main energy source.

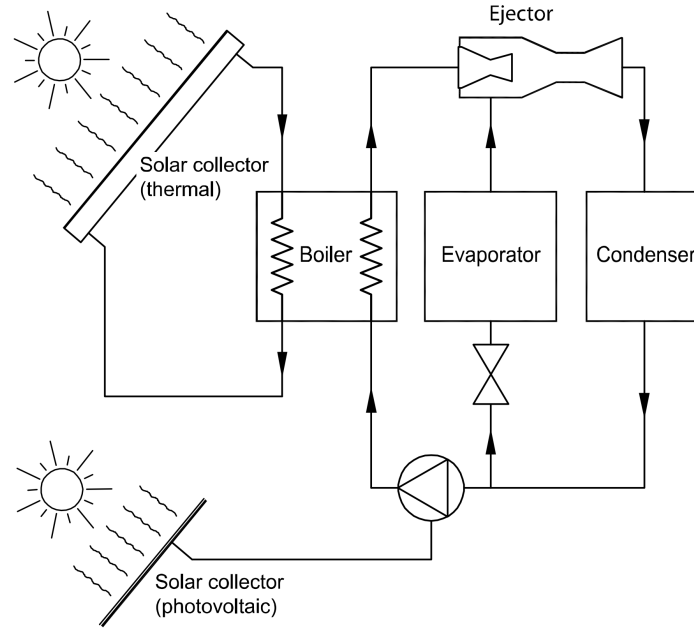


Figure 1.1: Ejector-driven refrigeration cycle, from [16].

Ejectors have additional benefits: they are passive devices, mechanically simple in nature. Indeed, they are simply composed of ducts of particular shapes that are easily manufactured by modern machining technologies. With no moving parts, maintenance and operation costs drop when compared to compressors that are classically used for the compression of the refrigerant. They also allow to work with environmentally friendly refrigerants [17].

Nevertheless, performance and operational stability shortcomings have prevented supersonic ejectors from penetrating the air-conditioning market. They have a low Coefficient Of Performance (COP), i.e., the share of energy input required for a given heat extraction at the cold source (or heat delivered at the hot source in the case of heat pumps) is higher for ejector-driven systems than for existing concepts. Also, ejectors are not very versatile regarding their operating conditions: their performance is highly dependent on the primary, secondary and condenser pressures, and rapidly degrades for operating conditions departing from the design value.

Great effort must still be made to overcome these shortcomings and make this technology industrially ready and reliable. Supersonic ejectors may appear very straightforward in their concept, but their geometrical simplicity contrasts with the complex fluid mechanics and thermodynamics phenomena they enclose, such as transonic flow choking, shockwave-shear layer interaction and transonic turbulent mixing.

Much scientific research has been conducted on supersonic ejectors, with numerous research groups striving to gain insight on ejector phenomena, and to predict and improve their performance. There are two ways of doing so: either by experimental investigation or by models and simulations. The former is the most representative of reality, but is difficult and costly to set up, hindering useful investigations, e.g. geometric parametric studies. The latter are generally much more affordable (depending on the level of complexity of the simulation) in terms of investment and effort, but are limited in fidelity by the very models and assumptions they are based upon. The currently available computational power is indeed far from being able to handle the full complexity of ejector flows.

To this day, despite a partial understanding of some of the phenomena at stake, the prediction of supersonic ejectors performance is still somewhat imprecise. Some paramount concepts such as the origin of the secondary flow choking are still a matter of debate, and require further research.

1.2 Ejector phenomena

1.2.1 Ejector operation and performance

A schematic view of a typical supersonic ejector layout is shown in Figure 1.2.

A supersonic ejector works in presence of a high stagnation pressure flow and a low one, respectively the primary flow and secondary flow. The latter is compressed thanks to momentum and exergy exchanges between both streams.

The primary stream, flowing from a subsonic to a supersonic state in a converging-diverging nozzle, drives the secondary flow. This work of suction of the primary onto the secondary stream is the result of the low static pressure achieved at the exit of the primary nozzle (the primary flow having been expanded) and of the viscous entrainment effect. The latter is due to the shear stresses taking place between both streams. In practice, the viscous shear layer that develops at the interface between both streams is characterized by vortical structures typical of the Kelvin-Helmholtz instability, and rapidly transitions to a turbulent state. Both streams then undergo a mixing process in the so-called mixing duct, thereby exchanging momentum and energy.

Finally, the mixed flow, characterized by the consumption of the original potential flow parts, recovers static pressure through either a normal shock or a series of oblique shocks (designated as second shock train, the first shock train taking place at the outlet of the primary nozzle due to the over-or under-expansion of the primary stream) and a subsequent subsonic divergent, due to the high static pressure imposed downstream (which corresponds to the condenser inlet in refrigeration applications).

Despite a non-moving geometry and a macroscopic performance evaluation based on simple variables, an ejector is hosting complex and diverse local phenomena. The turbulence occurring in the shear layer between is responsible for different flow features that have a strong influence on the macroscopic performance of the ejector.

Previous studies have analyzed two different types of ejectors. The first one is a "constant pressure mixing ejector", where the primary nozzle exit is located within the convergent section of the secondary flow, followed by a constant area section where the mixing continues. This design is said to have the best performance (B.J Huang et al. [18] and Chen.W et al. [19]), and is the one that will be analyzed in this Master's thesis. The other design is called "constant area mixing ejector", characterized by a nozzle exit position situated in a constant area section.

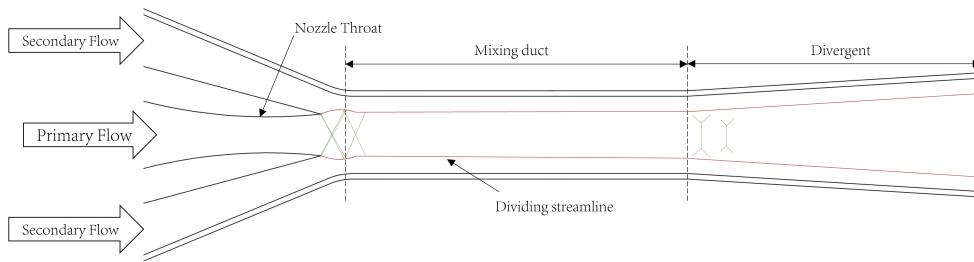


Figure 1.2: Simplified sketch of a supersonic ejector

In the rest of this document, the primary and secondary reservoir pressures are designated respectively by p_{01} and p_{02} . The outlet total pressure, or total back-pressure, is referred to as $p_{0,out}$. Note that in experiments and numerical simulations, it is rather the static back-pressure p_{out} which is monitored/imposed. However, the flow being at very low Mach number at this location, static and total back-pressures can be referred to roughly equivalently.

It is also convenient and meaningful to use dimensionless quantities. Hence, to fully characterize the operating conditions, the variables p_{01}/p_{02} and $p_{0,out}/p_{02}$ will be used.

The purpose of a supersonic ejector for fixed p_{01} is to drive a maximum amount of secondary fluid, at the highest possible back-pressure and with the lowest possible energy losses. Therefore, its performance is primarily determined by the induced secondary mass flow rate $\dot{m}_2$, or more conveniently by the dimensionless entrainment ratio defined as $\omega = \frac{\dot{m}_2}{\dot{m}_1}$.

As they consist of supersonic and transonic ducted flows, ejectors face a performance limitation linked to the inherent nature of such flows: the choking phenomenon. Depending on the operating conditions (which refer to both ratios p_{01}/p_{02} and $p_{0,\text{out}}/p_{02}$), the choking regime (and the associated entrainment ratio) of the ejectors drastically changes, as illustrated in Figure 1.3 with the so-called **characteristic curve**, i.e., the entrainment ratio as a function of $p_{0,\text{out}}/p_{02}$ for fixed p_{01}/p_{02} .

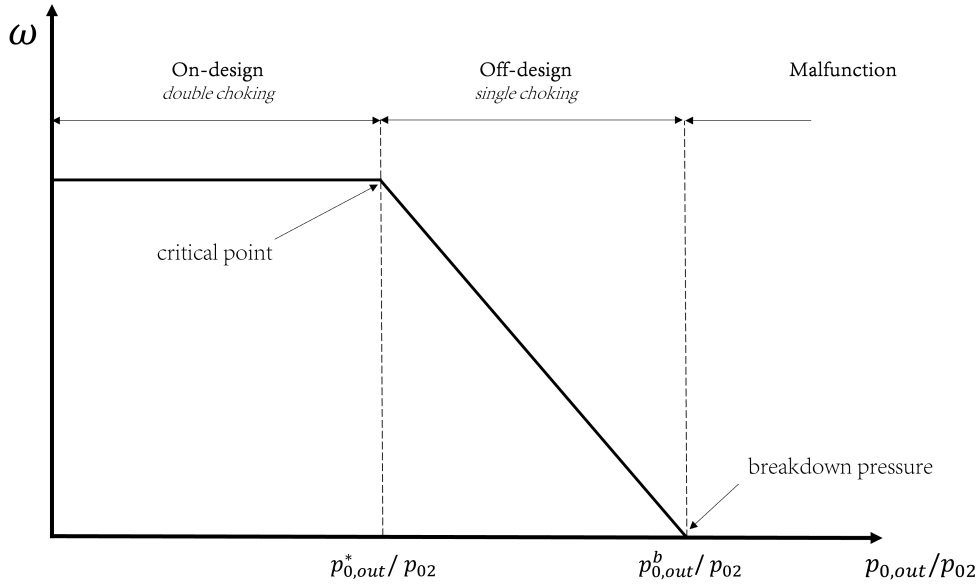


Figure 1.3: Theoretical characteristic curve

Being expanded in a conventional converging-diverging nozzle, the primary flow is choked near the throat position (due to its transition from subsonic to supersonic state), meaning that the primary mass flow rate $\dot{m}_1$ is maximized. The latter only depends on the reservoir conditions and the throat cross-section area. In the operating conditions considered when studying supersonic ejectors, the primary flow is always choked.

As opposed to the primary stream, the secondary stream cannot be choked at a position anterior to the mixing duct inlet, as it is far from reaching sonic conditions. However, depending on the back-pressure ratio $p_{0,\text{out}}/p_{02}$, the secondary mass flow rate $\dot{m}_2$ can also appear to be choked, i.e. to be independent of downstream conditions. When this happens, the operating conditions are referred to as "on-design" or "double-choking" [20], yielding a plateau of the characteristic curve for all the values of $p_{0,\text{out}}$ lower than a certain critical back-pressure $p_{0,\text{out}}^*$.

For values of the back-pressure exceeding this critical value, $\dot{m}_2$ and ω are sensitive to the downstream conditions (i.e., the secondary flow is no longer choked), and decrease in a linear fashion as $p_{0,\text{out}}/p_{02}$ increases. This region of the operating map is called the "off-design" region or "single-choking region".

Finally, if the back-pressure is high enough, $\dot{m}_2$ and ω will take negative values, meaning the flow in the secondary nozzle is reversed. This operating region is labeled as the malfunction region.

1.2.2 Theories for the secondary choking phenomenon

Flow choking in a variable-area supersonic duct is a well-known phenomenon. In a converging-diverging nozzle, as the flow exceeds the speed of sound near the throat, the acoustic characteristics are directed towards the outlet, making the flow insensitive to any perturbation occurring downstream. The analytical prediction of this simple flow choking is straightforward for inviscid flows, and reasonably simple for non-isentropic flows containing boundary layers.

On the other hand, the choking of the secondary flow is a less intuitive feature, for which no scientific agreement has emerged in the last decades. Indeed, radial profiles of pressure and axial velocity are quite complex, due to the different velocities of both streams, and to the viscous interactions leading to the growth of shear and boundary layers. The two recurrent theories for the explanation and modeling of the secondary flow choking are presented hereunder.

The Fabri-choking theory

The first theory, designated as the Fabri-choking theory, relies on the existence of a hypothetical sonic section (y-y section in Figure 1.4) within the secondary stream, i.e. a sort of aerodynamic throat. The attention is focused on the secondary stream "independently" from the primary stream, as the it is already choked. This theory considers that no mixing occurs between both streams (i.e., the interface is a simple contact surface) prior to this y-y section. If the back-pressure is low enough, the secondary flow chokes by becoming sonic at this virtual throat. The mixing phenomena then occur downstream.

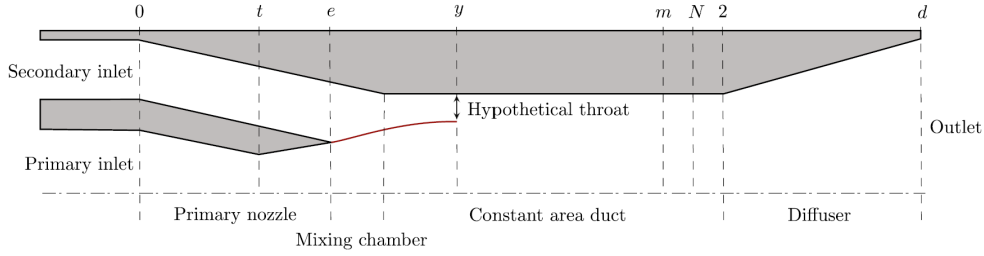


Figure 1.4: Schematic diagram of an ejector and the sections used in the Fabri-choking theory (from [21]).

The main assumptions used by Fabri-choking models [19, 18, 22] are :

1. Steady and one dimensional flow;
2. Static and total pressure are equal at the inlets and the outlet (kinetic energy can be neglected in the total enthalpy).
3. Non-ideal processes (friction, mixing processes,..) are taken into account thanks to isentropic efficiency coefficients.
4. Walls are adiabatic.
5. The primary flow fans out and doesn't mix with the secondary flow until a certain cross section assumed as fictive throat, where the secondary flow becomes sonic in on-design mode operation.

The last assumption is the critical one and is quite debatable. Indeed, a turbulent shear layer rapidly develops at the point of encounter of both streams, leading to mixing and transfer phenomena upstream of the fictive throat.

The compound-choking theory

The second theory was derived by Bernstein et al. [23] and is called the compound-choking theory. The concept is illustrated in Figure 1.5 for an arbitrary number of streams n . As opposed to the Fabri-choking theory which considers each stream individually, the compound-choking theory — as its name suggest — uses a criterion based on the compound flow formed by n streams. Instead of assuming a sonic section in the secondary stream, the theory assumes the blockage of axially propagating acoustic waves when the compound-flow is said to be choked.

To obtain a simplified model and performance predictor based on this theory, an analytical compound-flow indicator β and an equivalent Mach number M_{eq} are defined by deriving the velocity of such acoustic waves, under the assumption of equal static pressure across each section (the static pressure can only vary along the axial direction). The n streams can have different Mach number and total conditions. In the case of a supersonic ejector, $n = 2$. As will be seen later, the compound-choking theory allows a choking of the entire flow without requiring it to be sonic.

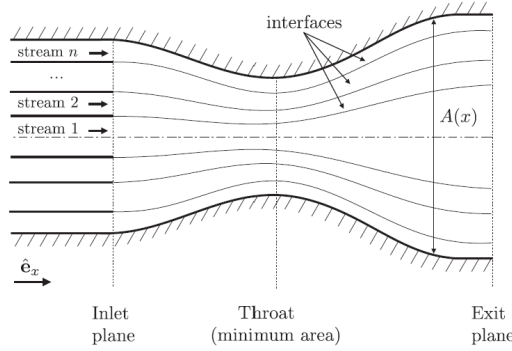


Figure 1.5: Illustration of the streams considered in the compound-choking theory, from [24]

1.2.3 Simplified models

Modern computational resources allow a straightforward, cost-effective and flexible study of supersonic ejectors: either through simplified thermodynamic models or computational fluid dynamics (CFD) simulations, a broad range of tools have been developed in the past 50 years to understand, evaluate and optimize supersonic ejectors.

The most cost-effective but also less accurate models are the so-called thermodynamic models. They are based on strong assumptions, allowing to get rid of the inherent complexity of supersonic ejectors, and still obtaining satisfying results in terms of performance prediction. Their output is very limited: typically, they predict the critical (i.e., on-design) entrainment ratio, and some are able to predict the whole characteristic curve (see Fig. 1.3). Most of those models are one-dimensional (1D), or at most quasi-1D, and rely on strong assumptions such as 1D steady and isentropic flow. They are built upon simple conservation laws for mass, momentum and energy, as well as classical gas dynamics results. A brief state of the art of the existing models is presented hereunder.

The first 1D model for supersonic ejectors, based on simple gas dynamics and conservation principles was obtained by Keenan and Newman [25] in 1942. Keenan et al. [26] later refined this model, proposing the concept of constant pressure mixing. Their model however only provided limited information about the choking phenomenon.

Following the choking theory proposed by Fabri and Siestrunk [22], a strong assumption was then proposed by Munday et al. [27]: according to their theory, the supersonic

primary stream fans out of the primary nozzle, expanding into the secondary stream without mixing with it. Both streams remain separated by a perfect slip surface until a certain section where the secondary flow is choked (at a so-called fictive throat located in the constant area part of the ejector). The mixing process then takes place downstream. Though somewhat unrealistic, this assumption (along with the Fabri choking model) has been subsequently used by a vast majority of thermodynamic models. More recently, Huang et al. [28] developed a widely used 1D model for constant pressure ejectors, using experimentally determined isentropic efficiencies, and capable of predicting the on-design entrainment ratio.

In the past 20 years, numerous simplified 1D models were developed. Li et al. [29] and Chen et al. [19] proposed a model that predicts performance both for the critical and subcritical mode, using the simplifying assumption of ideal gas. Zhu et al. [30] derived a 1D model based on a shock circle choking which takes into account realistic velocity profiles due to the shear and boundary layers. Other models were derived, still based on the Fabri choking assumption [31, 32, 33]. Some consider real gas behaviors [34, 35, 36]. Del Valle et al. [37] derived a model for axisymmetric flows. The interested reader will refer to [38] for a more in-depth review of some of the existing 1D models.

Very recently, a renewed interest for models based on the compound-choking theory [23, 24] emerged. Metsue et al. [21] and Croquer et al. [39] derived thermodynamic models based on the compound-choking assumption, also taking into account real gas behavior.

The aforementioned models are limited in the accuracy of their prediction for the entrainment ratio, also for on-design operating conditions. This limited performance is due to the strong assumptions required to reduce the complexity of the ejector flow (for instance, with irreversibilities being taken into account using constant loss coefficients [40]), and to the lack of understanding of the choking phenomenon.

1.2.4 CFD for supersonic ejectors

1D thermodynamics models provide satisfactory predictions for macroscopic ejector results, such as entrainment ratio prediction. Yet, the limited insight they provide into actual ejector flow phenomena pushed the scientific community to resort to CFD simulations, which offer the advantage of being relatively affordable and easy to set up while offering precise quantitative information about the complex flow physics at stake.

Supersonic ejectors involve high Mach and Reynolds numbers transonic flows, where shockwaves are embedded in turbulent shear-and boundary layers, thus characterized by very small spatial and time scales. Direct numerical simulation (DNS) is thus simply beyond the reach of current computational power. While some Large Eddy Simulations (LES) have been, or are being conducted [41], and promise to deliver valuable information about 3D unsteady phenomena, their computational cost is still prohibitive for simple ejector design or comparison purposes.

Reynolds-Averaged Navier–Stokes (RANS) methods are by far the most popular and affordable CFD method for the study of supersonic ejectors. The use of time-averaged quantities allows to decrease the problem complexity by reducing the number of degrees of freedom, through the modeling of the effect of turbulent structures of all scales with a turbulent viscosity.

Their affordability has made RANS methods a widely used tool to investigate supersonic ejectors. The complexity of the problem is generally further decreased by performing steady-state simulations, and using 2D axisymmetric computational domains instead of 3D domains. Comparison of 2D axisymmetric, steady-state simulations with 3D simulations have shown the relevancy of such simplified simulations [42, 43, 44, 45, 46].

A key parameter in RANS studies is the choice of the turbulence closure model. As highlighted by several works [37, 47, 48, 49, 39], even though the macroscopic (i.e., the entrainment ratio) performance is generally predicted with a relatively low error, the

local flow features are extremely sensitive to the chosen closure model. Hemidi et al. [47, 48] stated that global measurements are not sufficient to compare turbulence models. According to them, there is a strong and erroneous belief in the research community that good prediction of the ejector global performance implies a good prediction of the local flow features.

In the last two decades, several turbulence model comparison studies were conducted, yet no broad agreement for the best-suited closure model emerged among scientists. Initially, turbulence models were assessed by means of the predicted overall performance [49, 43, 44, 50], but studies recently investigated their effect on local flow phenomena [51, 52, 53, 54, 55, 42, 56]. This allowed a better assessment, based for instance on experimental and numerical flow visualization.

Two particular closure schemes seem to stand out: the $k - \epsilon$ *RNG* and $k - \omega$ *SST* schemes, regularly cited as the two best performing models in the case of supersonic ejectors. The choice of the turbulence model for this work is explained in greater detail in section 2.3.

CFD studies of supersonic ejectors are plentiful. Originally, they served the purpose of simply predicting overall performance for varying operating conditions or geometry [57, 11, 47, 58, 59, 60, 61]. The recent emergence of efficient simulation tools has allowed CFD studies to investigate more deeply the local flow features, by means of numerical results such as Mach number and pressure fields visualization, or pressure profiles along the mixing duct wall or centerline [39, 62, 63, 64, 55, 48, 65, 66, 67].

Very recently, Lamberts [68, 69, 70, 24] developed useful methods and obtained significant results concerning the mixing and choking phenomena in a rectangular ejector, by using a state-of-the-art RANS setup, validated with an experimental facility allowing wall-pressure measurements and flow visualization, with shadowgraphy and Schlieren techniques.

He performed a thorough analysis of the momentum, kinetic energy and exergy transfers between the primary and the secondary stream, by assessing the flux of these quantities through the mean-flow dividing streamline. Following the work of Meyers and Meneveau [71], he also developed a numerical post-processing tool allowing to visualize the transport tubes of mean-flow momentum, kinetic energy and exergy, based on the vector fields formed by the local fluxes of these quantities. The developed methods were subsequently used by Soenen [72] for the study of transient phenomena.

In parallel, both CFD results and experimental observations made Lamberts [70] the first to observe numerically and experimentally the Fabri-pattern, which is a basis for the vast majority of existing thermodynamic models (which consider the Fabri-choking as the explanation for the limitation of the secondary mass flow rate). By means of flow variables visualization, his work highlighted the characteristic Y-shockwave pattern, matching closely the experimental observations.

Lamberts also resorted to an alternative explanation for the choking phenomenon [24], namely the compound choking theory proposed by Bernstein et al. [23]. Numerical simulations and their appropriate post-processing showed that the compound choking of the primary and secondary streams was in fact the true origin of the limitation of the secondary mass flow rate.

In the present work, the methods introduced Lamberts [68] will be extensively used.

1.3 Objectives and structure of the manuscript

As its name suggests, the original purpose of this Master's thesis was to conduct both experimental and numerical investigations of a new axisymmetric supersonic ejector (though inspired from the work of Lamberts [68]), envisioned to use air as working fluid. In air-conditioning applications, the working fluid is a refrigerant (such as R134-a). However, for research purposes, air is good choice: it allows to use an open cycle with intake and outlet being the atmosphere, thus permitting a simplified experimental setup while still being representative of general ejector phenomena. The objectives of the experimental and numerical work performed are briefly exposed below.

1.3.1 Experimental investigation

Research on supersonic ejectors at UCLouvain has been going on for several years. Several supersonic ejectors were built during doctoral and Master theses and for research projects. The recent Ph.D. thesis conducted by Olivier Lamberts on the choking and mixing phenomena in supersonic ejectors [68] used a rectangular (i.e., the 3D geometry was a simple extrusion of a 2D geometry). Although very useful for experimental purposes, a rectangular ejector is somewhat different from ejectors that are and could be used in the industry, where the simplest and most straightforward geometry is axisymmetric. This Master's thesis was set up with the objective of designing, manufacturing, calibrating, and validating a new axisymmetric ejector, which would serve as a means of comparison between experimental and numerical data, both in the frame of an ongoing Ph.D thesis (conducted by Romain Debroyer) and potential future Master's theses.

The design of the new ejector was a great part of the workload during the year. The first step was to decide the main dimensions of the new ejector (such as the mixing duct length and diameter). The preliminary design was partially based on the axisymmetric geometry adapted by Soenen [72] from the 2D geometry used by Lamberts.

It is emphasized here that the ejector designed and studied in this Master's thesis is fundamentally different from those used by Lamberts and Soenen. The only identical features are the primary nozzle (optimized by the methods of characteristics) and the ratio of areas between the mixing duct area and the nozzle throat area. In his adaptation of the rectangular geometry to the axisymmetric one, Soenen chose these values so as to obtain identical mass flow rates $\dot{m}_1$ and $\dot{m}_2$ for identical operating conditions. This design choice was kept in this work (and is even not exactly identical due to manufacturing constraints encountered for the mixing duct diameter).

Once the main dimensions of the ejector were fixed, an iterative design process took place during several months. Indeed, many challenges arose from the axisymmetric geometry and associated constraints, and they required multiple versions of the design, each one solving some issues and sometimes creating new ones. In addition to the purely mechanical matters (such as geometric tolerancing and feasibility of the assembly), economical constraints and other specifications (such as modularity of the mixing duct) required to adopt solutions that were not especially the simplest ones.

Additionally to the design of the ejector assembly (i.e., the ejector in itself but also all the pieces required to operate it, e.g. the primary and secondary inlets), the design process embraced all the aspects linked with the experimental setup, such as the data acquisition (which required an important amount of time for equipment testing) but also the layout of the ejector in the experimental facility.

Due to circumstances detailed in section 2.2.4, the complete assembly of the ejector was not feasible within the Master's thesis duration, although most of the designed pieces were manufactured and ready to assemble. Consequently, no experimental run was possible before the finalization of this manuscript.

1.3.2 Numerical investigation

The numerical study of an ejector offers the advantage of not presenting the numerous hurdles that can be encountered for the building and assembly of a real ejector, as long as it is performed with tools such as a commercial software. The computational power available on a modern personal computer indeed allows to conduct relatively complex studies.

Using widely acknowledged numerical simulation tools, this work aims at characterizing the flow inside the designed axisymmetric ejector with high accuracy, using a high-quality numerical setup and adequate post-processing techniques that allow to extract valuable information from raw numerical data computed with the solver.

The objective of the numerical characterization of the designed ejector is to describe the flow in general terms, i.e., to identify the flow features common to all operating condi-

tions. Much effort was made to present results as precise and complete as the available computational power allowed, so as to grasp the essential of the flow phenomena in supersonic ejectors.

The numerical tools are then exploited to investigate the choking phenomenon, and try to answer to the following question: Is the compound-choking theory a satisfying explanation for the secondary flow choking, what are its limitations, and how does the Fabri-pattern relate to these issues?

Finally, the transfer phenomena are thoroughly analyzed with state-of-the-art post-processing techniques, with the objective of accurately characterizing the interaction between the primary and secondary stream.

1.3.3 Structure of the manuscript

The structure of the present work is briefly detailed in this section. It is composed of three chapters, in addition to this introductory part and to the conclusion.

Chapter 2: Experimental setup and numerical methods

This chapter will present the whole design process of the experimental facility, the associated difficulties and the solutions found to overcome these issues. This includes a description of all the designed pieces and used equipment, the presentation of the optimization process that led to a choice of mixing duct length, and a complete explanation of the design of the assembly, e.g. the tolerances.

The numerical setup will then be presented. The equations solved will be briefly recalled, and the flow solver as well as the computational domain will then be described.

Chapter 3: Flow topology and choking phenomenon

This chapter will deal with the flow topology in the axisymmetric ejector in general, but also with more specific flow features. After presenting the global performance of the ejector and the influence of the operating conditions (in the form of characteristic curves), the general flow topology will be set forth.

The choking phenomenon at the origin of the limitation of the secondary mass flow rate will undergo an in-depth investigation. This flow feature will be characterized using visual tools and quantitative indicators, such as one-dimensional equivalent quantities and the concept of choking section location. To the authors knowledge, this type of systematic characterization of the choking phenomenon is not a common result found in the literature.

Furthermore, the authors have identified the Fabri-pattern as playing a role in ejector performance slightly more important than just a particular flow curiosity. It will thus undergo a full characterization process, aiming at identifying the factors that cause its onset.

Finally, based on the variety of results obtained over a large range of parameters, a conclusion on the choking phenomenon and the impact of the Fabri-pattern will be drawn.

Chapter 4: Ejector performance: one-dimensional models, transport and mixing phenomena

The purpose of this chapter will be to assess the performance of the ejector over a wide operating range, both with global and local indicators.

A first part of the chapter will compare the performance prediction of state-of-the-art 1D models with the CFD results.

The mixing and transfer phenomena occurring between the primary and secondary streams will then be evaluated by means of transport tubes, which allow to understand where the momentum and energy contained in the primary stream are transported to. The characterization will then carry on with the presentation of local transfers occurring through the dividing streamlines, and the exergy losses due to heat transfer and viscous dissipation.

Chapter 2

Experimental setup and numerical methods

2.1 Introduction

In this chapter, the experimental setup and design process as well as the numerical methods used throughout this manuscript are presented. The former are detailed in section 2.2, where the ejector layout, measurement devices, geometry optimization and design process are detailed. Section 2.3 is devoted to the numerical methods used in this work, with emphasis being put on the equations that are solved, on the flow solver and its parameters, and finally on the mesh and its quality.

2.2 Ejector geometry and design

Following the work of O. Lamberts: "Choking and mixing phenomena in supersonic ejectors" [68] on a rectangular ejector, the purpose of this thesis is to conduct a similar analysis on an axisymmetric ejector whose area ratios are based on the rectangular one. The main relevant dimensions are presented in this preface, except for the mixing duct length l_m which will undergo an optimization process in section 2.2.2.

As explained in section 1.2.2, the area ratios are critical to define the mass flow rates, choking and associated on and off-design conditions [73].

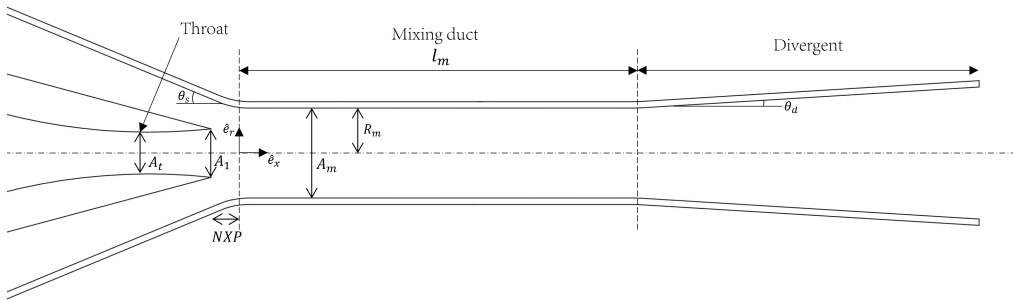


Figure 2.1: Basic geometrical parameters of the ejector

In order to start the design, the area ratios were fixed to values as close as possible to those of Lamberts [68], if the mechanical design constraints allowed it. The geometry of the primary nozzle obtained via the method of characteristics was reused. The nozzle throat has a radius $R_t = 9.75 [mm]$, and the mixing duct $R_m = 21 [mm]$.

The area ratios and the deviation w.r.t. those of Lamberts are shown in Table 2.1. A_t is the primary nozzle throat area, A_1 the nozzle exit area and A_m the mixing duct area. Note that as seen in Figure 2.1, the origin of the coordinate system (i.e. the position where $x = 0$) is the beginning of the mixing duct.

	Rectangular Ejector	Axisymmetric Ejector	Deviation
A_1/A_t	1.33	1.34	0.7 %
A_m/A_t	4.53	4.63	2.16 %

Table 2.1: Geometrical Parameters comparison

The area ratios are almost respected and the primary converging-diverging nozzle is the same as Lamberts design. Besides, the distance between the beginning of the mixing duct and the primary nozzle exit (NXP) is also kept constant : $x_{NXP} = -0.0186 [m]$. However, if this geometrical value is of interest to be modified in the future, it can be done easily thanks to a small adaptor that would fit between the central connection piece (T-connection, explained in the next section) and the convergent of the secondary flow entrance (details on the design can be found in Appendix F).

The angles θ of the inlet secondary stream and the diffuser (diverging section) are respectively $\theta_s = 22.5^\circ$, angle $\theta_d = 3.5^\circ$ (based on typical values for diffuser duct angles [74]).

The macroscopic design will be explained in section 2.2.1, followed by the mixing duct length optimization in section 2.2.2, and finally the design and manufacturing considerations are detailed in section 2.2.3.

2.2.1 Experimental setup

In this section, the overall experimental setup is presented along with the measurement instruments.

The part of interest of a supersonic ejector, i.e. the part where the vast majority of the ejector flow phenomena take place, is composed of the primary converging-diverging nozzle, the secondary converging nozzle, the mixing duct and the divergent, as illustrated in Figure 2.1. Those are the parts that are simulated in the numerical model.

The experimental installation, illustrated with a split view in Figure 2.2, is however more complex. As a matter of fact, the primary and secondary flow require to be properly ducted from their respective intake (high-pressure reservoir for the primary stream and ambient air for the secondary flow) to the inlet of the "strictly-speaking ejector". The flow at the outlet of the diffuser must also be dealt with.

The critical feature of the setup is the piece that hosts the primary and secondary flows from their respective intake at the inlet of the interesting ejector part, referred to as "T-connection". It receives the secondary flow from above and the primary from the left (following the orientation of the view in Figure 2.2). The primary flow is ducted through the T-connection towards the primary nozzle, and the secondary flow is spread around that duct inside the T-connection prior to its injection in the secondary nozzle. The difficulty lies in the fact that due to the axisymmetric geometry, the piece that serves as transport of the primary stream through the T-connection and converging-diverging nozzle (labeled "Primary nozzle" in Figure 2.2) is floating inside the assembly, and is only maintained at its left end.

The whole setup is constituted of four main segments, also identified in Figure 2.2: the primary inlet, the secondary intake, the ejector itself and the outlet. The T-connection is thus the piece that connects the three first main segments. They are briefly detailed below.

The primary flow comes from a buffer tank fed by a piston compressor from Ateliers François (Model CE46B) and controlled by a pressure regulator. It is entering the T-connection at the center and passes through it on its way to the ejector.

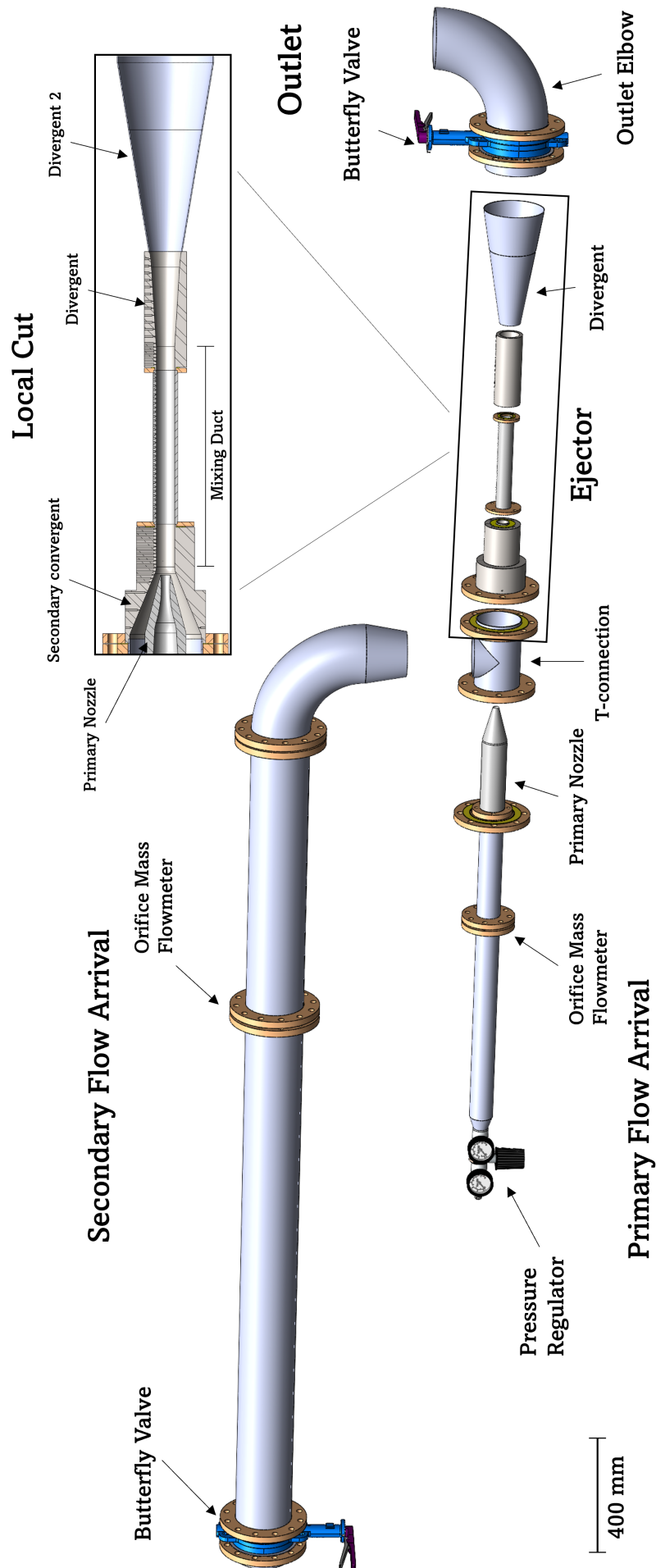


Figure 2.2: Split view of the complete experimental setup, with local cut on the "ejector" part (Solidworks Model). Butterfly valve and pressure regulator CAD models are from [75, 76]

The secondary flow is sucked from ambient air, through a long pipe (whose length is 15 times DN200, this length is required to have a fully-developed flow, needed for mass flow rate calculation using an orifice plate) parallel to the floor. After a shoulder, it is connected to T-connection from the top. Regarding the secondary flow, the T-connection plays the role of a plenum where the flow is assumed to be sufficiently slow so that $p_{02} = p_2$ (the total pressure being used for boundary conditions in CFD simulations while the measured experimental quantity is the static pressure), and nicely spread out around the whole section of the secondary-inlet. The inlet reservoir pressure p_{02} is controlled by a butterfly valve.

The ejector itself is composed of four main parts and divided in several pieces (further detailed in section 2.2.3).

The primary nozzle is floating in the T-connection and ejector (maintained on the left solid flange of the T-connection) and drives the primary stream into the ejector main-body. The latter is divided in three parts: the secondary converging nozzle, the mixing duct and the diffuser (also referred to as divergent).

This last part is composed of a low angle (3.5°) cone followed by a higher angle transition towards 9.5° to smoothly reach the second part of the divergent defined as industrial divergent.

The outlet is composed of the end of a butterfly valve to control $p_{0,out}$ and an elbow to expel the mixed jet towards the roof, used for security reasons in order to prevent any risk of projection in the ejector vicinity where humans could stand.

Concerning the data acquisition, the only measurable quantities in this setup are mass flow rates, static pressures and static temperatures. Both primary and secondary mass flow rates are measured upstream of the T-connection. The outlet mass flow rate, which by redundancy could have indicated potential leaks within the ejector, is not measured for compactness purposes. Indeed, the measurement of a mass flow rate using a diaphragm requires an important pipe length in order to have fully developed flows.

Static pressure ports are used at both inlets (in the region of the T-connection) and at the outlet to measure p_{01} , p_{02} and $p_{0,out}$. At those locations, the low Mach number allows to consider the static and stagnation pressures equal.

Numerous static pressure ports are also drilled all along the ejector, allowing to precisely measure the wall static pressure, particularly in the mixing duct where the ports density is maximum.

Finally, temperature measurements are available at the primary and secondary inlets to measure T_{01} and T_{02} , necessary for the boundary conditions of the numerical simulation.

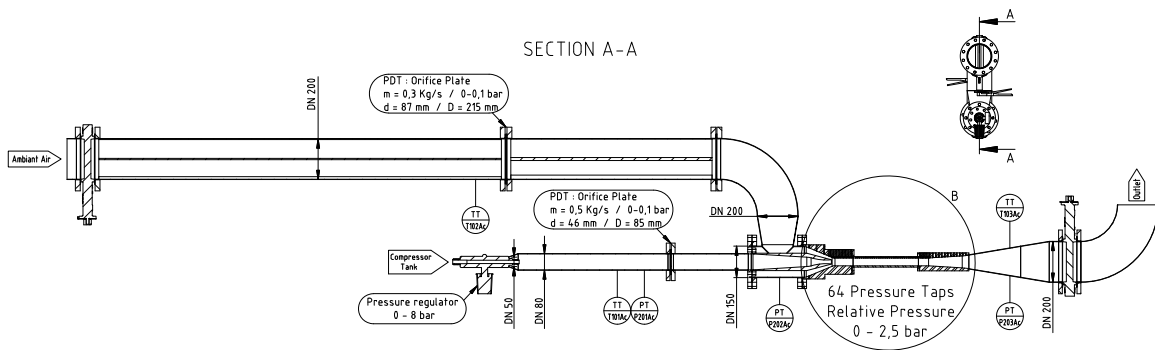


Figure 2.3: Data acquisition - PID of the setup (More detailed pressure information in Appendix 2.3)

The data acquisition setup is shown in Figure 2.3, containing the global view with the different pressure sensors, mass flow meters and temperature sensors. A more detailed view is presented in Appendix F.2 with the allocation of the 64 pressure taps along the ejector walls. Some of them are closed, represented by a "C" but it permits flexibility to measure the pressure at more locations if needed.

Mass flow rates are calculated thanks to orifice plate and pressure taps (1D and 1/2D) that follow the ISO5167 standard (error generally lower than 0.9%). Both differential pressures manometers have a 0-0.1 bar range [68, 19, 77]. Temperatures are measured by PT100 RTD temperature probes (TT on the PID) (uncertainty $\leq \pm 0.5^\circ C$) which are located at the mass flowmeter position to compute precisely the mass flow rates [68, 19].

As already stated, several pressure ports are situated along the ejector geometry to obtain the wall static pressure profile. A Labview program controls, through 2 acquisition cards (National Instruments NI9476 in a rack cDAQ-9184), 64 electronic valves (8x8 sets) from the Clippard brand (Model ET-2M-24-H) that have been tested individually and repaired (leaks free). During an experimental run, these valves allow to switch between the various pressure ports of the ejector, since there is only one pressure manometer. There are also pressure transducers at each inlet and at the outlet (PT). The pressure sensors used are Endress Hauser and Kistler models [68, 19].

2.2.2 Mixing duct length optimization

The chosen geometry is partially inspired from the work of Lamberts [68] concerning the primary converging-diverging nozzle profile and the areas ratios from Table 2.1. However, it remained unclear which mixing duct length should be used. In his work, Soenen [72] first used the same mixing length l_m as the one from [68], leading to poor entrainment ratio performance, before using an upgraded version where the ratio l_m/R_m was instead kept constant.

In this work, it was decided to conduct a more thorough investigation of the mixing duct length and find the value that would lead to optimal performances.

Looking for a similar inquiry in the literature, it was found that P.Gupta et al. [78] have analyzed numerically three different mixing duct length on a rectangular ejector. Yet, no conclusion was drawn on any best geometry or meaningful variable to compare. However, the industrial purpose is to maximize the sucked/compressed mass flow rate coming from the secondary inlet with respect to the input driving mass flow rate. It was thus decided to optimize the entrainment ratio ω .

The **analysis method** consisted in testing different mixing duct lengths and producing their characteristic curves for two values of p_{01}/p_{02} , 3.6 and 5.13. (These primary pressure ratios were only selected in reference to the experimental results of O.Lamberts [68]). The mixing duct lengths considered are $l_m = 200, 260, 300, 430, 500, 550$ [mm]. These simulations were performed on Ansys Fluent, with a numerical setup very similar to that explained in section 2.3.3. The mesh was adapted for each value of l_m (except for the number of elements in the mixing duct, the mesh was identical for all cases). Note that the details of the numerical setup are omitted on purpose as they do not provide relevant information for this section and are detailed in section 2.3.3.

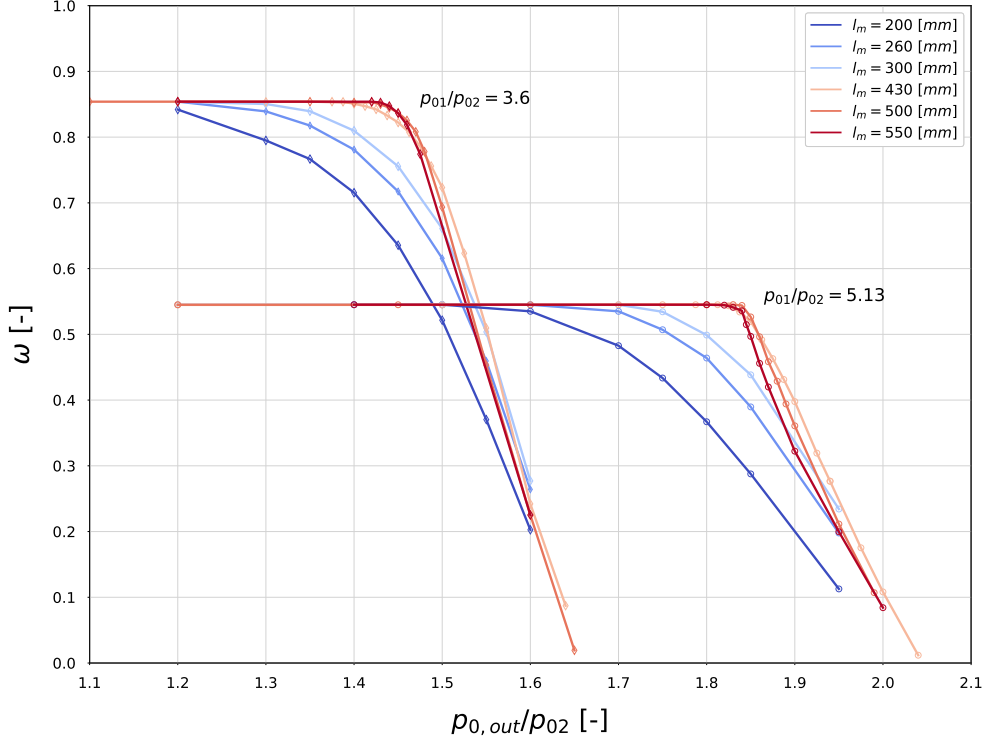


Figure 2.4: Characteristic curves for different mixing duct lengths l_m and two values of p_{01}/p_{02}

Results :

The results observed in Figure 2.4 seem to indicate an optimum, possibly explained by a trade-off between the mixing/exergy transfer increasing with the length, and the loss of total pressure due to wall friction that also increases with the length. First, the on-design entrainment ratio is unaffected by the mixing duct length l_m . This makes sense as it is determined by the area ratios and the value of p_{01}/p_{02} .

On the other hand, the critical point, i.e. the on-design operating conditions with the highest value of $p_{0,out}/p_{02}$, is significantly impacted by l_m : the value of the associated critical back-pressure $p_{0,out}^*/p_{02}$ significantly increases with the mixing duct length. In the case $p_{01}/p_{02} = 5.13$, the critical point of $l_m = 200$ [mm] is located at $p_{0,out}^*/p_{02} \approx 1.55$ whereas for $l_m = 500$ [mm] it is located at $p_{0,out}^*/p_{02} \approx 1.83$, gaining nearly 20%.

A similar observation holds for the curve corresponding to $p_{01}/p_{02} = 3.6$. It shows the direct impact of the mixing duct length on the performance and the importance of a preliminary design in order to maximize the entrainment ratio.

The longer the mixing duct, the "sharper" is the entrainment ratio curve, i.e. the higher the critical back-pressure and the closer the characteristic curve to the theoretical curve (Figure 1.3). A long mixing duct therefore improves the ejector performance (up to a certain extent). Indeed, for fixed operating conditions, it is preferable to compress the maximum mass flow rate at the highest back-pressure.

Zoomed views around the characteristic curves "corners" are shown in Figures 2.5a and 2.5b, where it can be observed that between $l_m = 500$ [mm] and $l_m = 550$ [mm], the actual optimum length is difficult to identify.

As seen in Figure 2.5a, the curve $l_m = 550$ [mm] keeps an "on-design" value slightly further than $l_m = 500$ [mm] but has a steeper off-design slope.

Figure 2.5b somewhat clarifies the situation with a clear optimum for $l_m = 500$ [mm]. Following this investigation, the mixing duct length was set to $l_m = 500$ [mm].

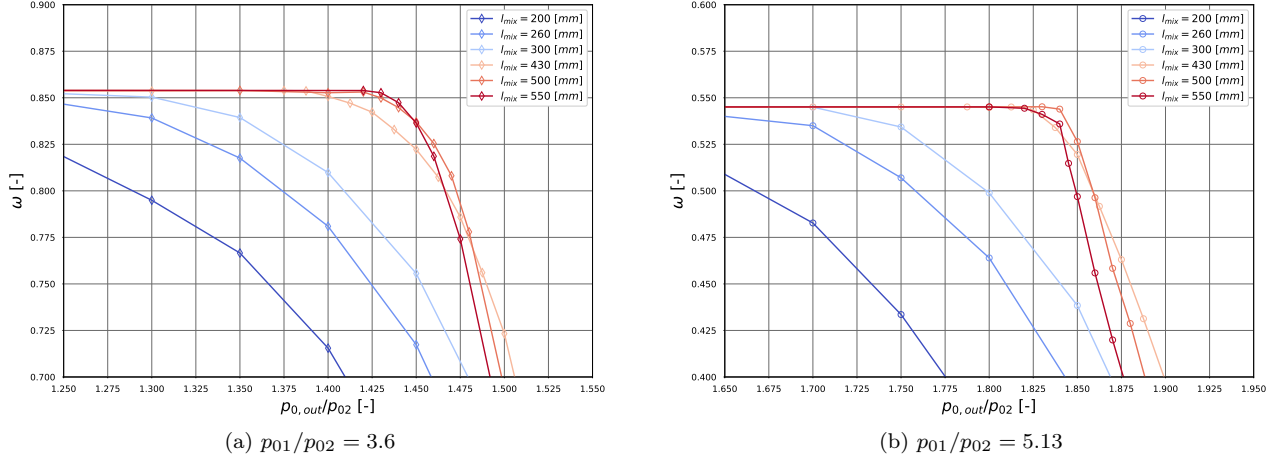


Figure 2.5: Local zoom on the critical point of the characteristic curves for different mixing duct lengths

2.2.3 Design process and methods

Once the mixing duct length chosen, the dimensions of the ejector are known. The domain of CFD analysis is thus entirely defined (Figure 2.6). All the phenomena analyzed in this manuscript are confined in this domain: other parts of the assembly can be considered as auxiliaries (bring the flow to the inlets, eject the flow out, mechanical support, measure the mass flow rates, ...). In order to make the connection between the CFD et and the experimental setup (in the form of boundary conditions at both inlets and at the outlet), pressure ports collars (4 pressure ports all around the duct) are used at the secondary stream inlet and at the outlet. The reservoir pressure of the primary stream is controlled by the pressure regulator.

The challenge is therefore to fit that domain into a complete assembly in the best possible way, while taking into account the various constraints (machinability, price, flexibility, etc.).

This section will present in detail the parts of the assembly that required particular attention in their design: the T-connection, the primary nozzle and the ejector shroud assembly. Their respective design process, the issues encountered and the solutions adopted will be clarified.

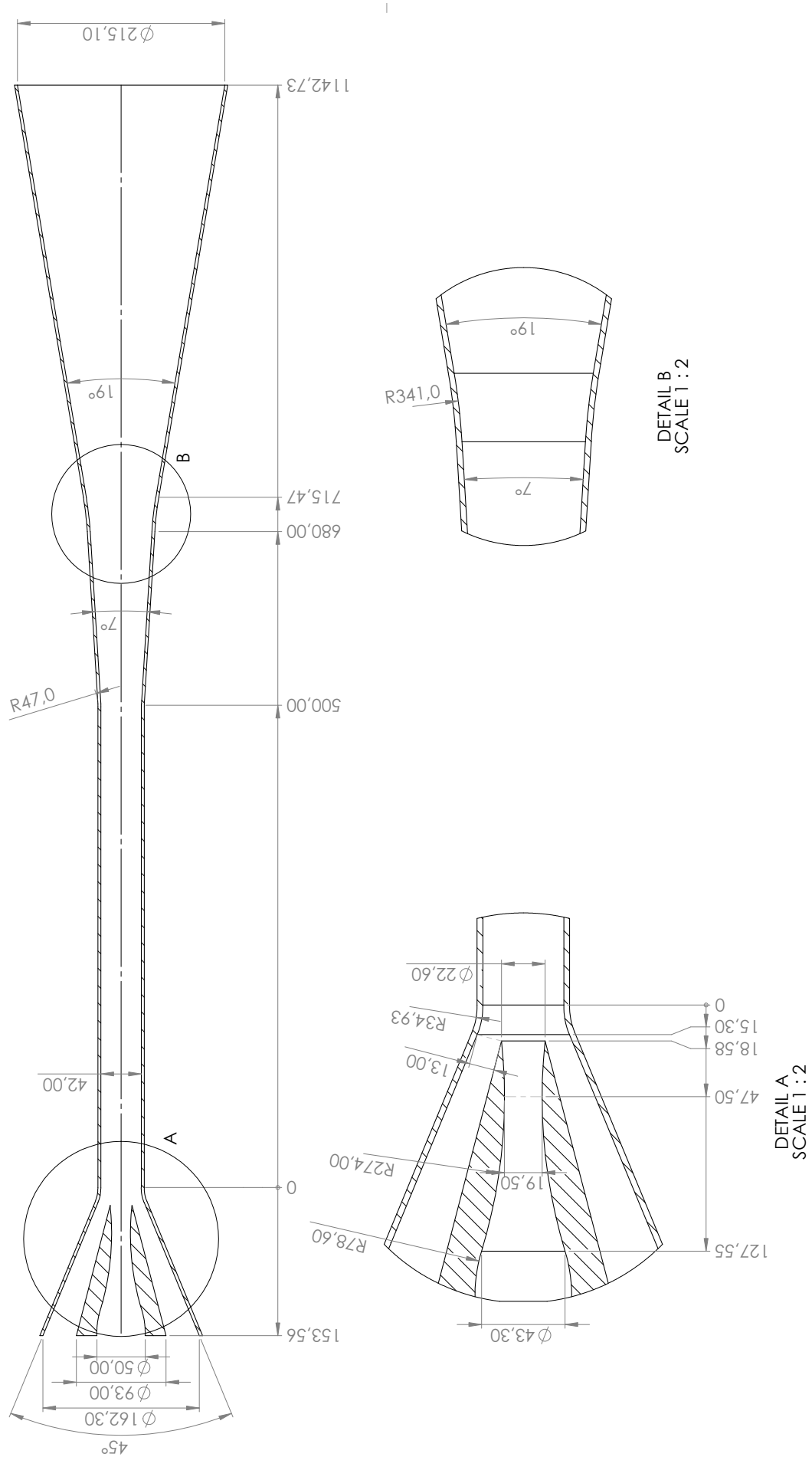


Figure 2.6: Drawing of the ejector modeled as used in the numerical simulations. Dimensions are in millimeters

The T-connection

The T-connection is a key piece for its support function and pressure measurement but has a rather simple design as it consists of a raw industrial piping T-connection. Its function regarding the flow is to be large enough (a plenum) and therefore allow to obtain a static pressure measurement p_2 that is close to the reservoir pressure p_{02} , used in CFD simulations as boundary condition. It is indeed well respected : it was determined that $\frac{p_{02}-p_2}{p_{02}} = 0.2 \%$ (for $\dot{m}_{2,max} = 0.3[kg/s]$).

Regarding its support function, the T-connection determines 2 vital parameters of the ejector: the NXP (i.e. the position of the primary nozzle exit w.r.t mixing duct start), and the co-axiality of the primary nozzle with the ejector shroud. As seen in Figure 2.7, the T-connection is the linkage point between the primary nozzle and the ejector shroud, the latter being fixed to the right-hand side of the T-connection, and the former to the left-hand side.

Due to the important cantilever of the primary nozzle (this piece being floating, i.e. only hung by its left end), the slightest misalignment of the T-connection axis with the nozzle axis can lead to a loss of the axisymmetry. The constraints born from this requirement are paramount to obtain a qualitative axisymmetric ejector, and a number of precautions were consequently taken to ensure compliance with them. For instance, several centering hubs with appropriate adjustment and perpendicularity of contact surfaces (to ensure the positioning of the parts) were incorporated to the design.

Figure 2.7 shows a local cut of the T-connection assembly where the key elements can be identified. Each color is associated to a classification of elements: light grey corresponds to industrial piping, dark grey to the machined parts, yellow to gaskets and orange are the flanges that allow the tightening of the pieces.

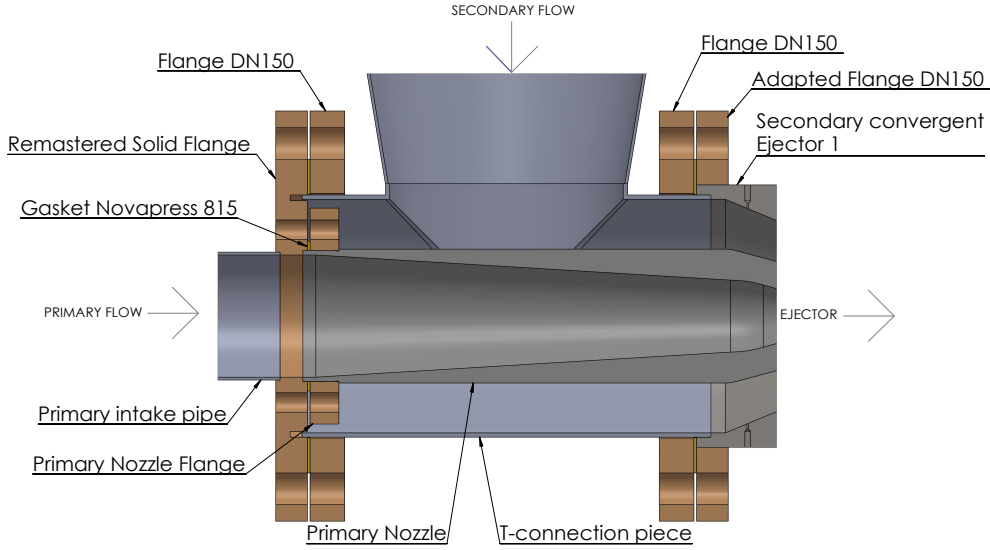


Figure 2.7: Local cut in the T-connection. Solidworks model.

The remastered solid flange, on the left of Figure 2.7, is centered on the T-connection thanks to a groove on the flange that fits with the external diameter of the T-connection piece. The depth of this groove is responsible of the axial position of the shroud w.r.t the primary nozzle. The latter is centered and positioned axially thanks to a groove with tight perpendicularity tolerances and with an adjustment H7g6 recommended by Ricordeau et al. [79] (detailed drawing in Appendix F.4). It is then tightened thanks to another customized flange, positioned on the left-hand side of the converging nozzle with a shoulder.

The three joints used are Novapress 815, 2mm thickness with a compression of 8% (from Eriks supplier [80]). The other flanges are standard DN150 flanges welded on the

T-connection piece, except from the flange ("Adapted Flange DN150") whose internal diameter is adapted.

The piece hosting the convergent section of the shroud is centered on the external diameter of the T-connection thanks to a similar groove with comparable tolerances. The chosen design allows to ensure a metal to metal contact and to avoid a step at the junction. Moreover, an annular removable 3D printed piece is planned to be used to place exactly and center the primary nozzle within the T piece during the assembly process and later withdrawn.

The primary nozzle

The primary nozzle is a really complex piece to manufacture for three main reasons: its length, its circular arc shapes at the throat and a really narrow space to operate. The profile was computed by O. Lamberts [68] using the method of characteristics. The design was motivated by strong compression waves observed in his initial experimental nozzle, which was composed of straight lines.

The matching constraint between stagnation and static pressure must also be applied at the CFD domain start of primary inlet, the variation is higher than the secondary stream but still reasonable ($\frac{p_{01}-p_1}{p_{01}} = 2.7\%$, $\dot{m}_{1,max} = 0.5[kg/s]$). To cope with this issue, the total pressure can be retrieved from the upstream pressure regulator.

Imposed geometrical tolerances are either to center and position the nozzle with respect to the solid flange or either to ensure a correct inner geometry (see Figure F.3 in the Appendix dedicated to drawings).

The assembly of the ejector shroud

This assembly, forming the exterior envelope of the ejector, is represented in Figure 2.8. Its design underwent several iterations in response to a series of constraints, the main ones being cost, machinability and flexibility, all directly linked to the length of the parts. The internal shape, which corresponds to the CFD calculation domain of Figure 2.6, was already fixed at this stage.

The **functional parts** of the ejector shroud are the secondary converging duct, the mixing duct and the diffuser. It is emphasized here that the **physical pieces** split the ejector in a way different of the functional parts. Indeed, the first and third pieces both contain a part of the mixing duct, the latter being too difficult to manufacture in a single piece. The fourth piece is an industrial diffuser, and plays the role of prolongation of the divergent part, which begins in the third piece.

The differentiation between the functional parts and the physical pieces is made on the top and bottom of Figure 2.8, identifying the lengths of the pieces and of the parts, respectively.

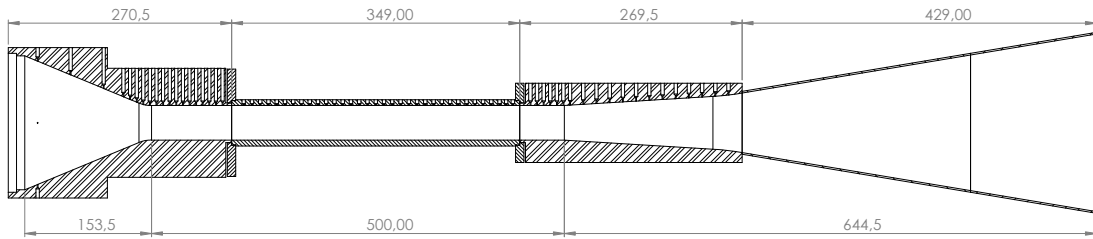


Figure 2.8: Drawing of the ejector assembly. Dimensions are in millimeters, rounded to the tenth.

The split of the ejector into multiple physical pieces was done in order to minimize the machining costs and the impact of the pieces junctions on the measure of static pressure (indeed, those junctions hinder the drilling of pressure ports). The 3 first pieces were machined on a lathe in private workshops. They were also drilled radially along the entire length of the ejector to welcome fittings connected to the pressure transducer. The position of these 64 pressure taps is represented in Appendix F.5. In the regions where critical phenomena are observed (e.g., in the beginning of the mixing duct), there is a maximal gap of 8mm between pressure ports.

Similarly to the previous pieces requiring co-axiality, centering hubs are used to fit in dedicated flanges and ensure perfectly smooth junctions. These flanges are manufactured precisely to clamp the pieces together and are screwed through threads on the first and third pieces. More detailed drawings can be found in Appendix F.6.

The fourth piece of the ejector assembly, i.e. the prolongation of the diffuser part, is composed of three piping cones welded together, slowing down the flow to an acceptable speed for the safety of the operators. The greater angle of this divergent part is gradually being introduced within the last section of the third ejector piece through a circular arc shape. This less precise piece is not critical as the flow is very slow in that region, while it permits to cut-off the price that would be needed for a completely machined divergent.

2.2.4 Advancement of the assembly

The whole (ideal) design being presented, this small section aims to state the advancement of the physical assembly and the main reasons of its unfinished nature when handling in this manuscript.

Despite a start of the design at the beginning of the academic year, the process was quite complicated and marked by several versions. The main difficulty was to conciliate the machining length capacity of UCL lab to decrease the cost of manufactured pieces while keeping a minimal number of junctions.

The ejector was thus split in order to get two pieces of length smaller than the CREDEM limit (first and third piece of Figure 2.8) and two others to be machined in private factories, too complex and/or too long to be done at UCL lab (the primary nozzle and the second piece of the Figure 2.8).

The final design was validated during the first week of February, and the parts to be manufactured in private workshops were ordered after having received several quotations and chosen the less expensive. During the following months, the pressure switchers used by the old ejector were tested and repaired. Moreover, the order containing the raw piping arrived as well as the fittings for the data acquisition.

At the time of beginning the manufacturing of the two last pieces, it turned out they were too long despite the initial validation. It led to a temporary halt of the experimental project, on the basis that the budget was too high and that it would be impossible to meet the deadline if the parts had to be manufactured outside.

But, thanks to the support of B. Herman, and after some negotiations with suppliers, it turned out that these 2 parts could be made quickly and cheaply in an external workshop, only requiring the pressure taps to be drilled at CREDEM.

At the time of handing in this manuscript, the primary nozzle is still missing (though ordered 3 months ago), but all the other separate pieces have arrived. The three ejectors pieces can be seen in Figure 2.9 where they are approximately positioned and maintained. The adapted flanges are currently being machined, and an additional machining of T-connection piece was made as the received piece did not correspond exactly to the nominal dimensions.

There is still little hope to complete the purely mechanical assembly before the end of June. The remaining work for an eventual other Master's thesis will be to connect the system to the compressor using a hose already available at CREDEM, and to lay out the pressure transducers, for which the necessary equipment is already available.

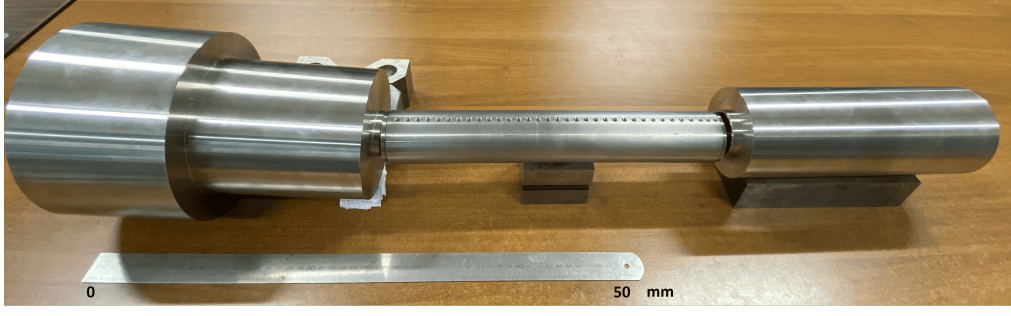


Figure 2.9: Manufactured pieces of the ejector, still missing pressure taps for two pieces. Note that the pieces are positioned approximately, the goal is to simply see the main pieces of the experimental setup.

2.3 Numerical methods and setup

The numerical characterization was achieved using **2D steady-state RANS** numerical methods.

In this section, these methods are introduced. First, the Navier-Stokes and compressible Reynolds-Averaged Navier-Stokes (RANS) equations are briefly recalled. The fluid properties and solver setup are then presented, followed by the computational domain (mesh) used throughout the simulations.

This work is concerned with axisymmetric simulations. In a cylindrical reference frame $(\hat{\mathbf{e}}_x, \hat{\mathbf{e}}_r, \hat{\mathbf{e}}_\theta)$ with coordinates (x, r, θ) , this means that the flow has circular symmetry with respect to the x -axis, i.e.

$$\phi(x, r, \theta) = \phi(x, r, \theta + \lambda) \iff \frac{\partial \phi}{\partial \theta} = 0 \quad \text{for arbitrary } \lambda \in \mathbb{R}$$

A simplified 2D equivalent formulation is therefore assumed, in a reference frame $(\hat{\mathbf{e}}_x, \hat{\mathbf{e}}_r)$ of coordinates (x, r) , x and r being labeled respectively axial and radial coordinates.

The velocity field is denoted as $\mathbf{u}(x, r) = u(x, r) \hat{\mathbf{e}}_x + v(x, r) \hat{\mathbf{e}}_r$.

It is also reminded that in the case of axisymmetric vector fields (such as $\mathbf{u}(x, r)$), the divergence of a vector field takes the following form (refer to Appendix A.1 for more details):

$$\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial r} + \frac{v}{r} \quad (2.1)$$

In this manuscript, the vector and tensor fields are written in a bold font whereas scalar fields are written in normal font.

2.3.1 Compressible RANS equations for axisymmetric flows

Navier-Stokes equations

The general form of the Navier-Stokes equations in the absence of volume forces and heat sources is the following:

$$\text{Continuity: } \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2.2a)$$

$$\text{Momentum: } \rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = -\nabla p + \nabla \cdot \boldsymbol{\tau} \quad (2.2b)$$

$$\text{Energy: } \rho \left(\frac{\partial e_t}{\partial t} + (\mathbf{u} \cdot \nabla) e_t \right) = \nabla \cdot (-\mathbf{q} - p\mathbf{u} + \boldsymbol{\tau} \cdot \mathbf{u}) \quad (2.2c)$$

With e_t the total specific internal energy, defined as

$$e_t = e + \frac{1}{2} |\mathbf{u}|^2 \quad (2.3)$$

The total specific enthalpy is also defined as

$$h_t = h + \frac{1}{2} |\mathbf{u}|^2 \quad (2.4)$$

The viscous stress tensor has the following form for a Newtonian fluid:

$$\boldsymbol{\tau} = \mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T - \frac{2}{3} (\nabla \cdot \mathbf{u}) \mathbf{I} \right) \quad (2.5)$$

With $\mathbf{I}$ the identity matrix. Note that this form is valid under the Stoke's hypothesis, according to which the bulk viscosity of air is negligible. Although this assumption is quite strong in the case of supersonic air flows, the solver used does not allow to take a bulk viscosity into account in the definition of the stress tensor.

The heat flux $\mathbf{q}$ is modeled with the Fourier's law:

$$\mathbf{q} = -\kappa \nabla T \quad (2.6)$$

With κ the thermal conductivity. Both the dynamic viscosity μ and the thermal conductivity κ are functions of the temperature.

Reynolds and Favre average

RANS equations are partial differential equations for the mean-flow variables. Two types of averages are defined hereunder: the conventional time-average and the density-weighted (Favre) average. Note that from now on, any time-dependency will be discarded, since this work is concerned with steady-state simulations.

For a quantity ϕ , the time average is defined as

$$\bar{\phi}(x, r) = \frac{1}{T} \int_{-T/2}^{T/2} \phi(x, r, t + \tau) d\tau \quad (2.7)$$

With T being an adequately chosen averaging period.

To this time-averaged value is associated an instantaneous fluctuation

$$\phi' = \phi - \bar{\phi}$$

The density-weighted (or Favre) average of any quantity ϕ is defined by

$$\hat{\phi} = \frac{\overline{\rho \phi}}{\bar{\rho}}$$

To this density-weighted average is associated another instantaneous fluctuation, denoted as

$$\phi'' = \phi - \hat{\phi}$$

The Favre average allows a simplified notation when applying the time-averaging operator to the Navier-Stokes equations.

Compressible steady-state RANS equations

The compressible RANS equations are obtained by applying the time-average operator to the Navier Stokes equations (2.2a), (2.2b) and (2.2c). Using the notations defined above, the compressible, steady-state RANS equations take the following form:

$$\nabla \cdot (\bar{\rho} \hat{\mathbf{u}}) = 0 \quad (2.8a)$$

$$\nabla \cdot (\bar{\rho} \hat{\mathbf{u}} \otimes \hat{\mathbf{u}}) = -\nabla \bar{p} + \nabla \cdot (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \quad (2.8b)$$

$$\nabla \cdot (\bar{\rho} \hat{\mathbf{u}} \hat{h}_t) = \nabla \cdot \left(-\bar{\mathbf{q}} - \bar{\mathbf{q}}^t + (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}} + \bar{\boldsymbol{\tau}} \cdot \mathbf{u}'' - \frac{1}{2} \left(\overline{\rho (\mathbf{u}'' \otimes \mathbf{u}'') \cdot \mathbf{u}''} \right) \right) \quad (2.8c)$$

With $\bar{\boldsymbol{\tau}}^t$ the Reynolds stress tensor. The symbol $\otimes$ represents the dyadic product.

The Favre-averaged specific total internal energy and enthalpy are given by

$$\begin{aligned} \hat{e}_t &= \hat{e} + \frac{1}{2} |\hat{\mathbf{u}}|^2 + \frac{\overline{\rho |\mathbf{u}''|^2}}{2\bar{\rho}} \\ &\triangleq \hat{e} + K + k \\ \hat{h}_t &= \hat{e} + \frac{\bar{p}}{\bar{\rho}} + K + k \end{aligned}$$

With K the specific mean-flow kinetic energy, and k the specific turbulence kinetic energy (i.e., the kinetic energy of the fluctuating turbulent field).

Closure of the RANS equations

The following approximations are generally made concerning the mean-flow viscous stress tensor and heat flux:

$$\bar{\boldsymbol{\tau}} \approx \mu(\hat{T}) \left(\nabla \hat{\mathbf{u}} + (\nabla \hat{\mathbf{u}})^T - \frac{2}{3} (\nabla \cdot \hat{\mathbf{u}}) \mathbf{I} \right) \quad (2.9)$$

$$\bar{\mathbf{q}} \approx -\kappa(\hat{T}) \nabla \hat{T} \quad (2.10)$$

Following the Boussinesq approximation, the Reynolds stress tensor $\bar{\boldsymbol{\tau}}^t = -\overline{\rho \mathbf{u}'' \otimes \mathbf{u}''}$ is modeled as

$$\bar{\boldsymbol{\tau}}^t \approx \mu_t(\hat{T}) \left(\nabla \hat{\mathbf{u}} + (\nabla \hat{\mathbf{u}})^T - \frac{2}{3} (\nabla \cdot \hat{\mathbf{u}}) \mathbf{I} \right) - \frac{2}{3} \bar{\rho} k \mathbf{I} \quad (2.11)$$

With μ_t the turbulent or eddy viscosity, which models the effect of the eddies of various scale within the turbulent flow.

The terms of $\bar{\boldsymbol{\tau}}$ and $\bar{\boldsymbol{\tau}}^t$ are developed in axisymmetric coordinates in Appendix A.1.

The turbulent heat flux $\bar{\mathbf{q}}^t$ is modeled following a Reynolds analogy:

$$\bar{\mathbf{q}}^t \approx -\frac{c_p \mu_t}{Pr_t} \nabla \hat{T} \quad (2.12)$$

With Pr_t the turbulent Prandtl number. Its value is imposed at $Pr_t = 0.9$, following the work of Lamberts [68]. This simplification has been proven valid for fluids with $Pr \approx 1$, which is the case of air.

The RANS equations still need a closure model, in the form of the eddy viscosity μ_t found in the expression of the Reynolds stress tensor (A.5). Generally, the eddy or turbulent viscosity is modeled using additional evolution equations. For instance, in the $k - \omega$ SST (Shear Stress Transport) turbulence model [81], the eddy kinematic viscosity $\nu_t = \mu_t / \rho$ is defined as the ratio between the turbulent kinetic energy k and of the specific dissipation rate ω . The fields k and ω are then governed by two additional evolution equations which need to be solved along the RANS equations.

As exposed in the introduction, the chosen turbulence closure scheme plays a determining role in the shape of the solution fields. A complete literature survey concerning the use of various turbulence models can be found in [43]. Bartosiewicz et al. [49, 51] conducted a thorough comparison (based on experimental results) of 6 turbulence models, and concluded that the $k - \epsilon$ RNG and $k - \omega$ SST performed best at predicting the local flow, but the $k - \omega$ SST performed better in term of stream mixing prediction. Other studies [53, 54, 43] showed that the $k - \omega$ SST performs better than the $k - \epsilon$ model, presenting the best agreement with experimental data both for the global and local characteristics of supersonic ejectors. Though no agreement exists concerning the actual best suited model, the $k - \omega$ SST model appears as a good closure model choice, and was henceforth used in this work.

2.3.2 Fluid properties

The fluid used throughout the simulations (as for the future experiments) is air, of molar mass $M_m = 28.966$ [kg/kmol].

To close the RANS equations, an equation of state is needed. The ideal gas law is used:

$$p = \rho R T \quad (2.13)$$

With R the specific gas constant, and $R = 287.04$ [J/(kg · K)].

Air is assumed to have constant values of specific heat ratio $\gamma = c_p / c_v = 1.4$, with $c_p = 1006.43$ [J/(kg · K)] (the actual variations of γ with the temperature and the pressure being negligible). The fluid is thus considered as a perfect gas.

The viscosity and thermal conductivity do, on the other hand, quite strongly depend on the temperature. They are assumed to follow the Sutherland law [82]:

$$\mu(\hat{T}) = \mu_0 \left(\frac{\hat{T}}{T_0} \right)^{3/2} \frac{T_0 + S_\mu}{\hat{T} + S_\mu} \quad (2.14)$$

With the reference viscosity $\mu_0 = 1.716 \times 10^{-5} [kg/(m \cdot s)]$, the reference temperature $T_0 = 273.11 [K]$ and the effective temperature $S_\mu = 110.56 [K]$. The thermal conductivity follows the law

$$\kappa(\hat{T}) = \mu(\hat{T}) (1.32c_v + 1.77R) \approx 1457.66 \mu(\hat{T}) \quad (2.15)$$

2.3.3 Flow solver

The compressible steady-state axisymmetric RANS equation are solved using the commercial software ANSYS Fluent (2022-R2, student version release) [83]. This solver uses a finite volume approach for the discretization of the equations and conversion of partial differential equations to algebraic equations. Finite volume solvers work by dividing the computational domain in a number of polyhedrons, or cells (in the present work, they are exclusively triangles and quadrilaterals). In the integral formulation of the RANS equations, the divergence terms become flux terms across the cells walls, approximated with the sum of fluxes crossing the discrete amount of boundaries of each cell. The surface integrals are then discretized on each cell, resulting in a set of algebraic equations for the discrete variables (pressure, velocities, temperature,...) which can then be solved numerically upon linearization of the obtained discretized equations.

Two solver types are available in Fluent: a pressure-based solver, where the pressure field is computed by solving a pressure correction equation based on the continuity and momentum equations (in such a way that the corrected velocity fields satisfies the continuity equation), and a density-based solver, where the density field is computed from the continuity equation and the pressure field from the equation of state (2.13).

Historically, density-base solvers were recommended to solve supersonic flows involving shockwaves [44]. However, pressure-based solvers were recently upgraded and are now able of accurately solving highly compressible flows. Both yield similar results [44, 43], and pressure-based solvers are often used due to their faster convergence [43]. In the present work, convergence was significantly faster with a pressure-based algorithm, which has thus been adopted. The chosen solver is said to be coupled: unlike the so-called segregated algorithm, the pressure-based coupled algorithm solves the momentum and pressure-based continuity equations simultaneously. This has the effect of significantly accelerating the convergence of the solution.

Concerning the turbulence modeling, the $k - \omega$ SST closure model is used, and the viscous heating and compressibility effects are taken into account.

The RANS equation are solved on an axisymmetric domain (which is a 2D numerical domain extending in the axial and radial directions), for steady-state operation (i.e. all the time-dependent derivatives in the RANS equations cancel out, except for the pseudo numerical time used for the convergence of the numerical solution).

Regarding the spatial discretization, Fluent uses co-located schemes, i.e. the pressure and velocity variables are stored at cell centers. Whenever pressure or velocity values are needed to evaluate expressions located at the cell face, interpolation schemes are used.

As they improve solution stability and capture shock in a better manner, second-order upwind schemes are used for the evaluation of density, momentum, turbulent kinetic energy, specific dissipation rate and energy. The pressure discretization scheme is also second order. Gradients are evaluated using a least squares cell-based approach. The solution is initialized with a hybrid approach, and is said to have converged when all the solution scaled residuals are lower than a specified value, in this case 10^{-5} .

2.3.4 Computational domain

The problem being axisymmetric, the computational domain is a 2D slice of ejector. The mesh was generated with Ansys Fluent Meshing. It is displayed in Figure 2.11. It is composed of quadrilaterals and triangles, and is mainly structured (in the structured part, the elements are exclusively quadrilaterals). The only unstructured part lies in the primary nozzle tip region, where both streams meet. This allows a smooth junction between the 3 structured zones (primary and secondary nozzles, mixing duct), and a mesh refinement near the primary nozzle tip. In the unstructured part, the mesh is progressively refined through 10 layers, from a characteristic length of 125 $[\mu m]$ in the bulk to a characteristic length of 20 $[\mu m]$ at the primary nozzle tip. The mesh elements are of quadratic order. The nodes are located at the element's vertices, and at each edge's middle (there are thus 8 nodes per element). The mesh specifications can be found in table 2.2.

Number of elements	493 000
Number of nodes	1 485 000
Radial number of elements in the mixing duct	210
Radial number of elements in the primary nozzle	120
Radial number of elements in the secondary nozzle	176

Table 2.2: Mesh specifications

Efforts have been made to obtain a mesh with local refinement in the areas of interest (i.e., at tip of the primary nozzle and in the boundary layers) while assuring a smooth transition between the various mesh zones. Near the walls, elements are refined in the direction normal to the surface, in order to solve the wall boundary layer as accurately as possible. Given the high friction Reynolds number Re_τ of this type of flow (typically of the order of 10^4), the height of the first cell at the wall must be quite small to achieve $y_{cell}^+ = \mathcal{O}(1)$. In the mixing chamber, the cell height at the wall is around 4 $[\mu m]$. The value of y_{cell}^+ depends on the flow inside the ejector, but does not change significantly between various simulations. Typical values of cells y^+ values can be found in Table 2.3.

Wall	y^+ range [-]
Primary nozzle	5 – 16
Secondary nozzle	1 – 5
Mixing chamber	0.5 – 1.5
Diffuser	0.1 – 1

Table 2.3: Cell y^+ values at the walls

In the $k - \omega$ SST framework, no wall function is used to model the viscous and buffer sub-layers. The flow needs to be wall-resolved, i.e. $y_{cell}^+ < 5$, which is the case nearly everywhere except in the primary nozzle. However, the boundary layers in this nozzle are laminar or weakly turbulent, and the wall-refinement is thus everywhere acceptable. Figure 2.10 shows a typical axial velocity profile at a section located at $x = 380 [mm]$ inside the mixing chamber. The numerical data is compared with the universal turbulent velocity profile. In the laminar sub-layer, the numerical results match closely the linear profile and confirm the good capture of that region by the numerical simulation. In the logarithmic profile zone, the data match the logarithmic profile with the constants $C = 4.1$ and $\kappa = 0.385$.

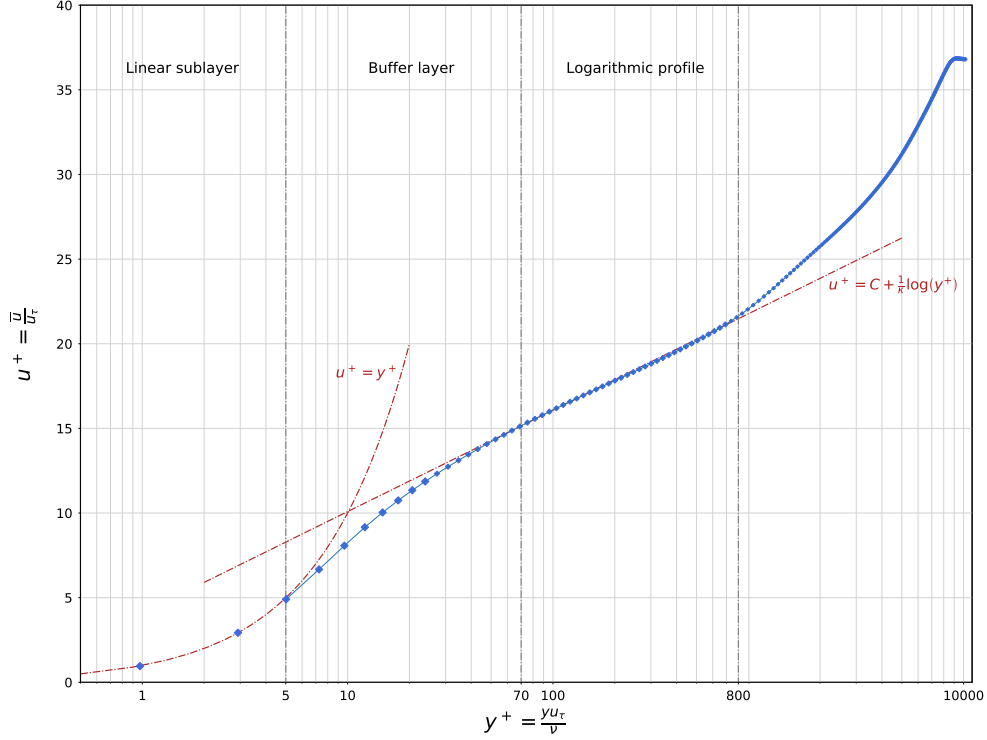


Figure 2.10: Axial velocity profile in the mixing chamber, comparison of numerical data (blue dots and line) with the well-known universal velocity profile (red dash-dotted line).

Due to the limitation on the number of elements (max. 500 000) in Fluent, a cell growth is used in the axial direction in the mixing duct. Elements located at the beginning of the mixing duct have axial length $\Delta x = 0.25$ [mm], and at the end of the mixing duct $\Delta x = 0.5$ [mm]. At the end of the diffuser, they have an axial length $\Delta x = 3$ [mm]. A similar cell growth is used in the primary and secondary nozzles. Except in the boundary layers and near the primary nozzle tip where the mesh is unstructured, elements have the same radial dimension in the mixing duct.

Concerning the mesh quality, 99.6% of the elements have an aspect ratio lower than 100, and the maximum aspect ratio is 400. The high aspect ratio of some cells is due to the cell growth in the boundary layers.

Over 99% of the elements have an orthogonal quality higher than 0.95. The lowest orthogonal quality is 0.59.

2.3.5 Boundary conditions

Boundary conditions need to be applied for the numerical solver.

At the walls, a no-slip condition is applied. They are also implemented as adiabatic.

At both inlets, as the flow is subsonic, two boundary conditions are applied. The reservoir pressures p_{01} , p_{02} and reservoir temperature $T_{01} = T_{02} = 300$ [K] are imposed.

Note that for all the simulations presented in this manuscript, the secondary reservoir condition was set to $p_{02} = 1$ [bar] as this value is close to the atmospheric pressure.

To take into account the weak turbulence present in the flow at both inlets in real experimental flows, the turbulent intensity and a turbulent length scale are specified. The turbulent intensity is set to 5%, and the turbulent length scale l is found with the relation [83]:

$$l = \frac{0.07 D_h}{C_\mu}$$

With D_h the hydraulic diameter, and C_μ an empirical constant set to 0.09.

At the outlet, the flow being subsonic, one condition is imposed. The static pressure p_{out} is specified. Note that although the actual outlet condition is a condition on the static pressure, the outlet condition will subsequently be referred to as a total pressure condition $p_{0,\text{out}}$ since the flow Mach number is very low at this location.

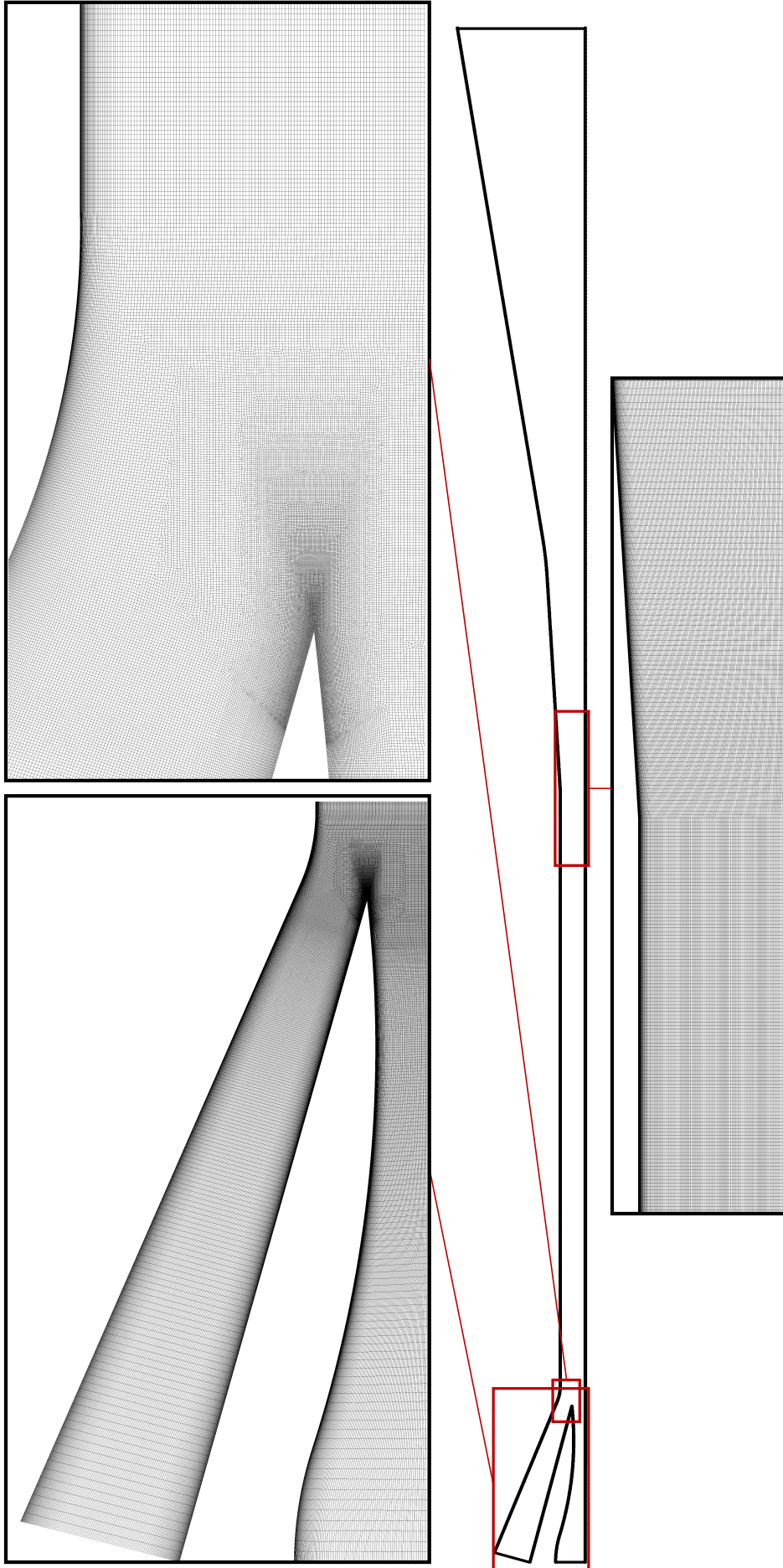


Figure 2.11: Computational domain: meshing of the axisymmetric ejector domain

Chapter 3

Flow topology and choking phenomenon

3.1 Introduction

One of the objectives of this work is to examine thoroughly the flow phenomena at stake in supersonic ejectors, or at least those made visible by a numerical simulation as simple as a 2D steady-state RANS model. In particular, the choking of the secondary flow plays a determining role in the performance of supersonic ejectors, as it constitutes a fundamental limit to the amount of entrained fluid. It is thus crucial to be able to predict the choking regime and its effect on the ejector performance.

The unsatisfactory Fabri-choking theory

As highlighted in section 1.2.2, the Fabri-choking theory, which considers a simple aerodynamic choking of the secondary stream due to the expansion of the primary jet, dominated the explanations for the limitation of the secondary mass flow rate for decades. Yet, despite its wide utilization in simplified theories, the Fabri-choking was only observed by Lamberts et al. [70] (both numerically and experimentally) and Gist et al. [84].

Although the existence of a sonic section within the secondary flow in some particular cases of operating conditions is not questioned, it only seems to be the fortuitous consequence of those particular operating conditions and not a satisfying explanation for the choking phenomenon.

Fabri-choking models assume an under-expansion of the primary stream at the outlet of its nozzle, causing it to expand in the mixing duct, thereby reducing the space available for the secondary stream to flow and causing it to choke. However, this explanation is not suited to cases where the primary flow is not under-expanded at the nozzle exit. Operating conditions with low stagnation pressure ratios p_{01}/p_{02} , which give rise to a severely over-expanded primary flow, can still produce a choked secondary flow, although the latter is not "compressed" by an expanding primary stream. Lamberts et al. [24] emphasized that the secondary stream does not necessarily need to be sonic in order to be choked. The Fabri-choking theory is not satisfactory and not general enough.

The compound-choking theory as alternative explanation

Lamberts used an alternative explanation for the flow choking: the compound choking theory (developed by Bernstein et al. [23]), that considers the combination of the primary and secondary streams as a whole instead of a stream-by-stream handling. Most of all, this theory provides an explanation for the choking of a compound flow that does not contain a fully sonic section. It yields good results concerning the prediction of the ejector performance, although the assumptions the theory is based upon are rather strong and unrealistic (in particular the assumption of 1D flow within each stream).

The authors have identified the need for additional evidence of the choking phenomenon, that either corroborate the existing theories or complete them. One purpose of this chapter is to provide additional insight on that matter.

Structure of this chapter

This chapter is structured in five sections.

- The first will explain the numerical database, through characteristic curves, of the whole set of simulations performed. A quick analysis will separate the set of operating conditions in two regimes, to which the whole manuscript will refer.
- The second will consist of the investigation of the flow topology in general, to illustrate typical flow features predicted by a steady-state RANS simulation of this axisymmetric ejector. It will use mainly visual means to provide the reader with the big picture of supersonic ejector flows.
- The third will be devoted to the choking phenomenon. Using both visual support and quantitative indicators, this section will pursue the objective of extracting information from the flow fields (e.g., the location of the *choking section*).
- The fourth will continue to examine the compound-choking, under the scope of the compound-choking simplified models and some modified indicators (intended to be improved versions of the original ones).
- The fifth section will look into the Fabri-pattern, with the aim of highlighting the origin of occurrence of this particular flow feature.
- Finally, using the large database and the associated numerical evidence, an alternative explanation for the choking phenomenon will be proposed.

3.2 Numerical database: characteristic curves and identification of two working regimes

It is reminded that the secondary reservoir pressure is set to $p_{02} = 1$ [bar] for all cases presented in this manuscript.

3.2.1 Impact of the operating conditions: characteristic curves

One purpose of this work is to cover a wide range of operating conditions to better understand the local scale phenomena but also the global scale impact of the different parameters. In this section, the global performance is evaluated thanks to different variables, the first one being the entrainment ratio ω .

Using the numerical setup introduced in section 2.3, a large number of simulations were carried out. It was chosen to investigate six values of reservoir pressure ratio p_{01}/p_{02} . For each of them, numerous values of $p_{0,out}$ were simulated in order to define as precisely as possible the characteristic curves shown in Figure 3.1, where a little bit more than 90 simulations are reported. This large amount of cases was necessary to obtain curves refined in the region near the critical back-pressure. Indeed, latter analyses require a very precise knowledge of the value of that back-pressure.

It is worth mentioning that many additional simulations with modified numerical setups were carried out. They are not presented here but will be referred to throughout this chapter.

Based on Figure 3.1, several analyses can be made. The first one is the decrease of the on-design entrainment ratio for increasing p_{01}/p_{02} , along with an increase of the critical back-pressure.

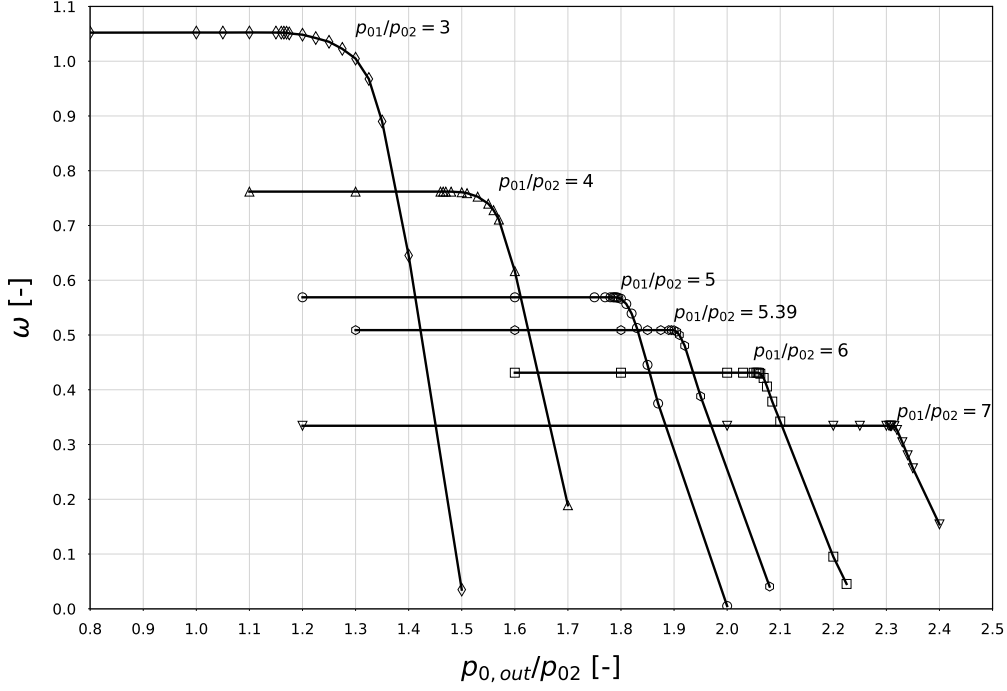
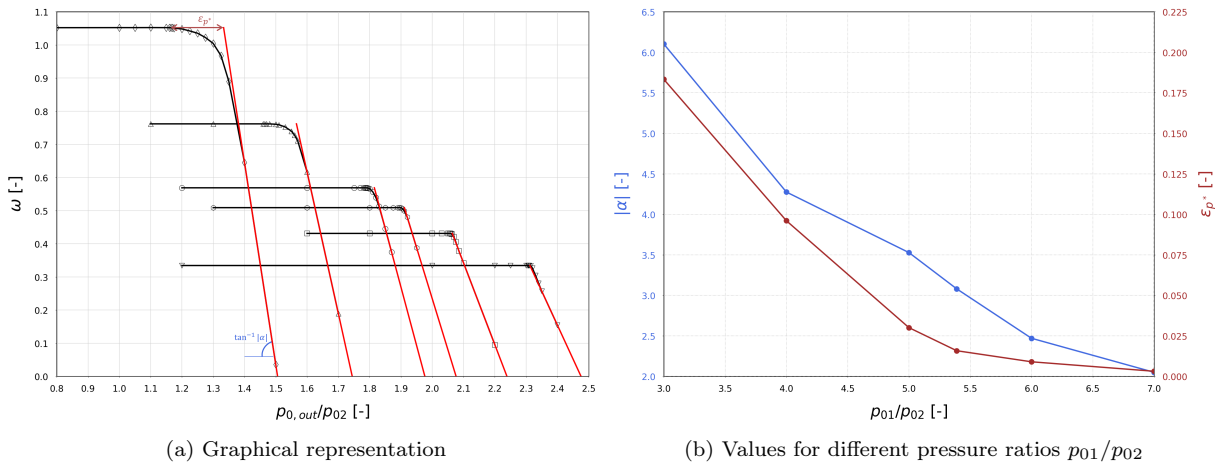


Figure 3.1: Set of numerical characteristic curves

Secondly, quantitative indicators illustrated in Figure 3.2a were associated to each curve. The off-design part of the characteristic curves can be characterized by almost perfect linear profiles (excepted near the breakdown point where a small deviation from the initial slope is observed). These linear off-design profiles have negative slopes labeled α . This slope was found to decay in absolute value with the primary pressure ratio (see Figures 3.2a and 3.2b in blue).

Moreover, the theoretical sharpness — abrupt decrease of ω for back-pressure exceeding the critical value — of the curves (see Figure 1.3) is approached when the pressure ratio p_{01}/p_{02} is increasing. This proximity to the theoretical "sharp corner" is assessed through the value ϵ_{p^*} . The latter is the difference of $p_{0,out}$ between the "theoretical critical point", intersection between the off-design approximated line and the on-design entrainment ratio value, and the CFD critical point approached value as represented on the Figure 3.2a. The evolution of this value with p_{01}/p_{02} is depicted in Figure 3.2b, showing a decrease of this shift (equivalent to an increasing sharpness), that is really close to a zero value starting from $p_{01}/p_{02} = 6$.


 Figure 3.2: Slope and shift values for different pressure ratios p_{01}/p_{02}

Although all the characteristic curves contain valuable information, the results presented in the following section will be confined to a reduced set of operating conditions for most of the analysis.

The cases that are mainly used for this chapter are $p_{01}/p_{02} = 4$ and 6, respectively representatives of *compound-like* and *Fabri-like* regime (explained hereunder). However, if their use is germane, other cases are not excluded.

3.2.2 Identification of two regimes

Following the statement concerning an increase of sharpness of the characteristic curves for increasing p_{01}/p_{02} , and based on observations of the flow topology (which are not exposed here but will be extensively looked into later on), two precisely delimited regimes were identified.

1. The ***compound-like* regime**, occurring for low reservoir pressure ratio p_{01}/p_{02} . This terminology comes from the fact that the numerical simulations performed within this operating range do not display any Fabri-patterns.
2. The ***Fabri-like* regime**, occurring for higher values of p_{01}/p_{02} , where the simulations systematically display the Fabri-patterns.

Emphasis is put here that this terminology of *Fabri-like* and *compound-like* regimes is only a way to classify the various encountered flow topologies, according to a particular criterion (the presence or not of Fabri-patterns). It does not hold as a conclusion concerning the choking phenomenon.

The transition from the *compound-like* to the *Fabri-like* regime occurs at $p_{01}/p_{02} = 5.39$, value for which the first Fabri-pattern is observed.

These *compound-like* and *Fabri-like* regimes are illustrated in Figure 3.3 using two corresponding cases, in the upper and lower half, respectively.

The supersonic region of the flow is highlighted in grey. In the *compound-like* case ($p_{01}/p_{02} = 5$, $p_{0,out}/p_2 = 1.6$), the secondary stream is progressively accelerated to supersonic speeds via the shear layer. For the *Fabri-like* case ($p_{01}/p_{02} = 6$, $p_{0,out}/p_2 = 1.2$), the secondary stream reaches Mach unity in a full sonic section. It is then further accelerated to supersonic speeds before being brought back to a subsonic state via a normal shock. This well delimited supersonic region is referred to as Fabri-pattern.

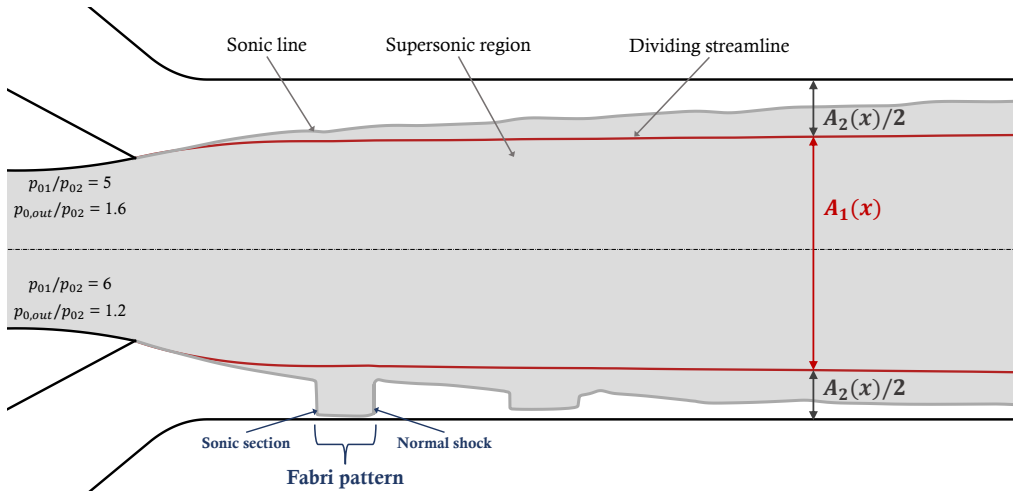


Figure 3.3: Illustration of the two regimes identified by means of the supersonic region. Upper half: *compound-like* regime, lower half: *Fabri-like* regime. Remark: as the cases displayed in both halves are different, the definition of $A_1(x)$ and $A_2(x)$ is not rigorous and serves for illustration purposes only. Twofold vertical stretching for improved readability.

This illustration of both regimes is purposefully limited to basic considerations. Exhaustive results will be presented in sections 3.4 and 3.6.

3.3 General Flow topology features

This section aims to present the flow topology in general, and to familiarize the reader with the post-processed results that will be extensively used afterwards. It is organized in different topics:

- A set of preliminary remarks will be provided concerning some general details of notation that are used during the chapter
- Fields of various variables (Mach number, schlieren,...) will be depicted to introduce them for the following analyses. It will detail the main phenomena and permit to show a first visual representation of these and their location in the ejector.
- Numerical schlieren images for values of p_{01}/p_{02} spanning the whole operating range will identify the various structures of shocks and expansion fans generated at the primary nozzle exit. The distinction between shockwaves and isentropic compression waves will be highlighted.
- Horizontal velocity and pressure profiles along the ejector will be displayed to illustrate the progressive mixing of the two streams.
- The last part to be analyzed on the general topology is linked to the viscosity of the flow. The latter generates shear and boundary layers whose thickness is analyzed thanks to usual definitions of fluid mechanics.

3.3.1 Preliminary remarks

Whenever quantities are plotted as a function of the axial coordinate x in the ejector, the dimensionless variable x/R_m (with $R_m = 21$ [mm] the mixing duct radius) will be preferably used. It might be useful for the reader to bear in mind that the primary nozzle is located at $x/R_m \approx -0.9$, the mixing duct entry at $x/R_m = 0$, the diffuser entry at $x/R_m \approx 23.8$ and the ejector outlet at $x/R_m \approx 54.4$.

In the rest of this document, many figures of numerical results plotted onto the ejector geometry will be used. For improved readability, some of them will be displayed with a twofold stretching in the vertical direction. These cases will be indicated in the legend with the abbreviation "**VS×2**".

The results will also often refer to the so-called "dividing streamline" or "dividing streamtube" (the streamtube being displayed as a line in 2D images). This streamtube is defined as the surface emanating from the primary nozzle tip that is everywhere tangent to the local velocity vector field. It marks the separation between the primary and the secondary streams. By construction, there is no mean-flow mass flux through this tube. Note that this tube only makes sense for mean-flow quantities: in reality, the interaction between both streams results in highly unsteady structures.

As a reminder, since all the results presented in this work originate from a RANS solver, all the flow variables are time-averaged variables. They will be referred to accordingly. However, the Mach number computed by the solver is not a proper time-averaged version of the instantaneous Mach number M , but a computation based on the averaged velocity and temperature fields. For the sake of simplicity, the Mach number calculated by Fluent will be indicated as $\overline{M}$ to indicate that it refers to the time-averaged flow although, it is not a proper time-averaged version of M .

In sections 3.3.2 and 3.3.4, the illustration of the flow topology focuses on the operating conditions $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. This case belongs to the *Fabri-like* regime. However, the general considerations regarding the basic flow topology hold for the other cases.

3.3.2 Study of the flow fields

A 2D image of the entire numerical domain is displayed in Figure 3.4, for $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. In the upper half, the mean-flow Mach number is represented with the sonic line, which is the iso-contour of mean-flow Mach number unity. It can be seen that the flow becomes sonic near the throat of the primary nozzle. The supersonic region extends in the mixing duct and in the first portion of the diffuser, where it progressively vanishes in a shock-train. For this particular case, two Fabri-patterns can be identified at the beginning of the mixing duct. They will be studied in detail in section 3.6.

In the lower half, the turbulence kinetic energy field k (translating the magnitude of the turbulent velocity field fluctuations) is shown with the dividing streamtube (represented as a line in this 2D view of the axisymmetric ejector). In the figure, it is clear that turbulence is mainly located in the shear layer, though it can be perceived that the boundary layer on the mixing duct wall also displays weak turbulence. The shear layer seems to develop approximately symmetrically around the dividing streamtube. According to a LES study of an ejector of similar geometry from R. Debroeyer [85], the shear layer transitions very rapidly to a turbulent state. With a numerical model as simplified as a steady-state RANS model, it is however impossible to obtain more information regarding the vortical structures due to the turbulent flow.

A good portion of the primary stream remains free of any turbulence throughout the mixing duct. On the contrary, the laminar part of the secondary stream is rapidly consumed by the shear and boundary layers, and is fully turbulent at about one third of the mixing duct length.

After the secondary shock train, taking place at the inlet of the diffuser, the flow becomes entirely turbulent, with a high value of k . Section 3.3.5 will be dedicated to the characterization of the shear and boundary layers.

Figure 3.5 displays a numerical schlieren image and a numerical shadowgraphy of the whole ejector, respectively in the upper and lower half, still for the case $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. The former is obtained by computing the axial derivative of the mean-flow density $\frac{\partial \bar{\rho}}{\partial x}$, and the latter the mean-flow density laplacian $\nabla^2 \bar{\rho}$.

The schlieren image allows to visualize gasdynamic discontinuities and waves (compression waves and shockwaves being displayed in dark/black, and rarefaction fans in light/white). As seen in Figure 3.5, the primary flow is under-expanded when it exits its nozzle. A Prandtl-Meyer rarefaction fan then accommodates the primary flow to the secondary stream static pressure by further expanding it. The supersonic flow pattern that follows is then highly two-dimensional.

These operating conditions lie in the previously defined *Fabri-like* regime. A Fabri-pattern is indeed observed in Figures 3.4 and 3.5, i.e. a clear sonic section within the secondary stream followed by a normal shock, forming the so-called Y-shockwave pattern, flow structure that was first highlighted by Lamberts [70]. More comments about the Fabri-pattern will be available in the dedicated section 3.6.

The lower half of Figure 3.5 depicts the mean-flow density laplacian. In addition to showing the gasdynamic features already shown by the schlieren image, it does also highlight the development of the shear and boundary layers.

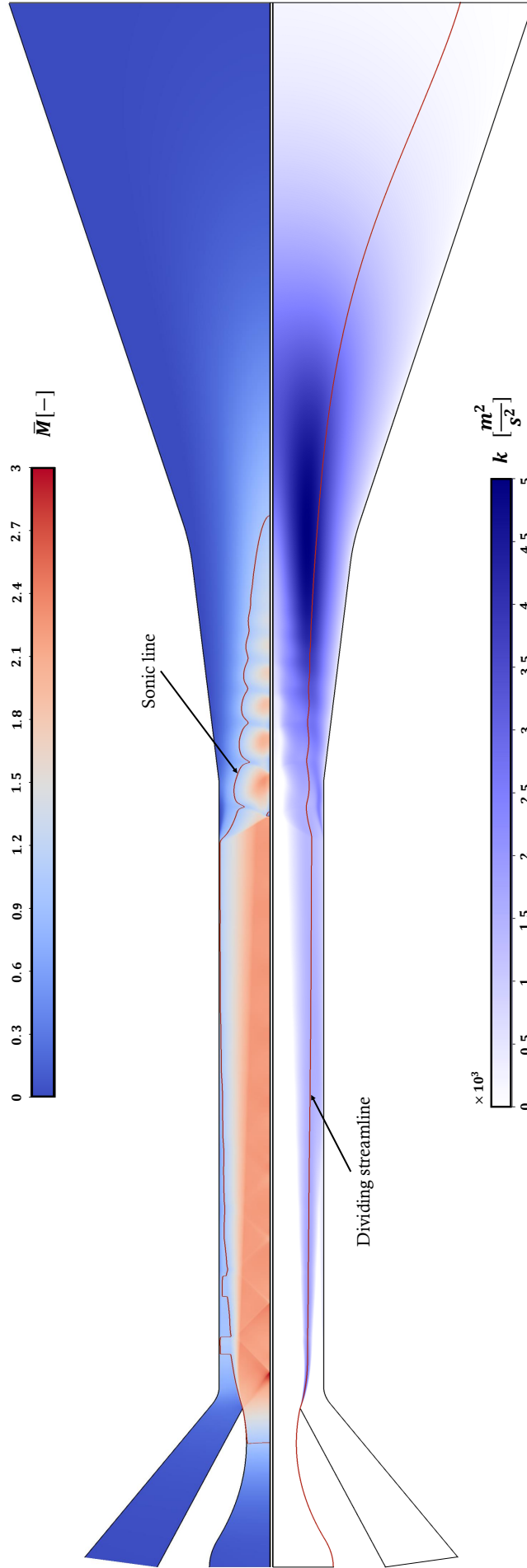


Figure 3.4: Upper half: Mean-flow Mach number field and sonic line, lower half: turbulence kinetic energy field and dividing streamline. The operating conditions are $p_{01}/p_{02} = 6$ and $p_{0,out}/p_{02} = 1.6$ ($VS \times 2$)

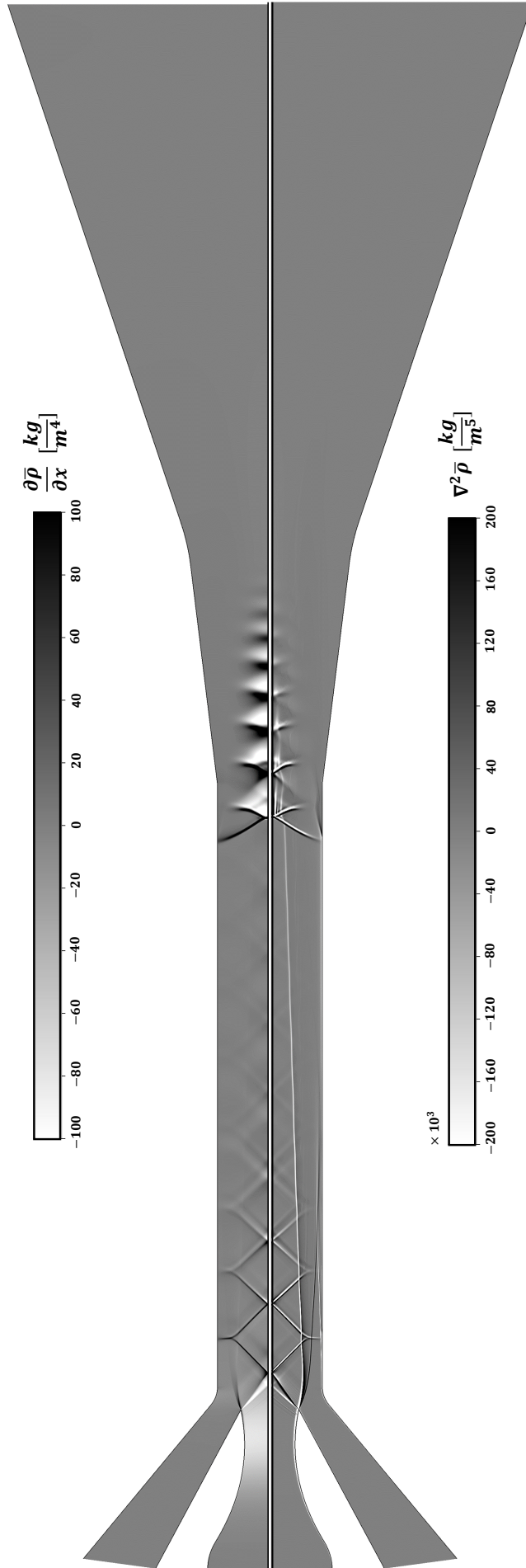


Figure 3.5: Upper half: numerical schlieren (axial derivative of the density field) image of the flow. Shock-wave and compression regions are shown in dark, expansion fans and regions are shown in light. Lower half: numerical shadowgraphy (Laplacian of the density) of the flow. The operating conditions are $p_{01}/p_{02} = 6$ and $p_{0,out}/p_{02} = 1.6$ ($VS \times 2$)

3.3.3 Impact of p_{01}/p_{02} on the supersonic flow topology

Depending on the value of p_{01}/p_{02} , the primary flow can either be over- or underexpanded as it exits the primary nozzle. In the former case, it will be suddenly compressed and slowed down through oblique shockwaves located at the exit nozzle, and in the latter case it will continue its expansion outside the primary nozzle through isentropic expansion waves.

In the case of a perfectly expanded primary flow (which exists for a single value of p_{01}/p_{02}), the static pressure at the outlet of the nozzle perfectly matches that of the secondary stream, and no shockwave or expansion fan are necessary.

This variety of possible cases is illustrated in Figure 3.6 using 5 sets of operating conditions, including a strong and a mild overexpansion, an optimal expansion, and a strong and a mild underexpansion of the primary stream. For each case, a numerical schlieren image is shown in the upper half, and a simplified schematic view of the shock and waves at play is shown in the lower half. The position of the shear layer is also indicated. It corresponds to the iso-contour of turbulence kinetic energy $k = 200 [m^2/s^2]$, which was found to be the most practical criterion to define the shear layer (based on several trials using vorticity contours and velocity profiles).

As opposed to simple converging-diverging nozzles, the complexity of an ejector makes it impossible to predict analytically the ratio p_{01}/p_{02} leading to an optimal expansion. This value was rather obtained by iterative simulations until the computed solution presented a zero expansion or compression at the outlet of the nozzle. The obtained pressure ratio is $p_{01}/p_{02} = 4.93$ (case C of Figure 3.6) with a precision of $\Delta p_{01}/p_{02} = 0.01$.

A first general tendency is the apparent stretching of the 2D structures for increasing p_{01}/p_{02} . This is directly linked to the Mach angle $\mu = \arcsin \frac{1}{M}$ that shrinks as the global Mach number of the primary stream increases with p_{01}/p_{02} .

Cases A and B correspond to a severely and mildly overexpanded primary nozzle, respectively. In the former case, the first oblique shock is stronger, and causes a flow separation region located at the very tip of the nozzle. For case A, the oblique shock converges at a shock triple point, terminating as a circular normal shock at the center of the ejector. As a consequence, the flow behind is abruptly decelerated to a subsonic state, thereby dramatically increasing its pressure and temperature. This forms a small Mach diamond, shown in Figure 3.7 using the temperature field.

For both cases A and B, the oblique shockwave is "reflected" by the axis. As it encounters the shear layer, it is reflected as an expansion fan. Both flow patterns then present a repeat of the initial wave pattern, progressively blurred by viscous effects.

In the figures, it is difficult to discern shockwaves from isentropic compression waves, especially with this kind of saturated plot. The distinction between shockwaves and compression wave is illustrated later on.

Case C corresponds to a perfectly expanded primary nozzle. It can be seen that the primary stream does not encounter any compression or expansion wave at the outlet of the nozzle. However, the perfect expansion is not synonym of absence of 2D flow patterns. Indeed, the information propagates along characteristic directions, which are lines of angle μ w.r.t the local flow direction, i.e. Mach waves. The core of the primary flow only becomes aware of the ending of the nozzle wall some distance downstream, yielding the conical pattern visible in Figure 3.6 C. The flow pattern is then composed of a succession of weak compression and rarefaction waves.

Finally, cases D and E present expansion fans at the outlet of the primary nozzle. The flow pattern is similar in both cases. The expansion fan is "reflected" as a shockwave by the axis. This shockwave is then transmitted through the shear layer and terminates as a normal shock in the secondary stream. It is then reflected back into the primary stream, forming the Y-shock pattern discussed in further detail in section 3.6.

Note that very weak pressure waves exist for all cases near the tip of the primary nozzle. They are likely caused by the mesh. As their impact is clearly negligible, they were not investigated any further.

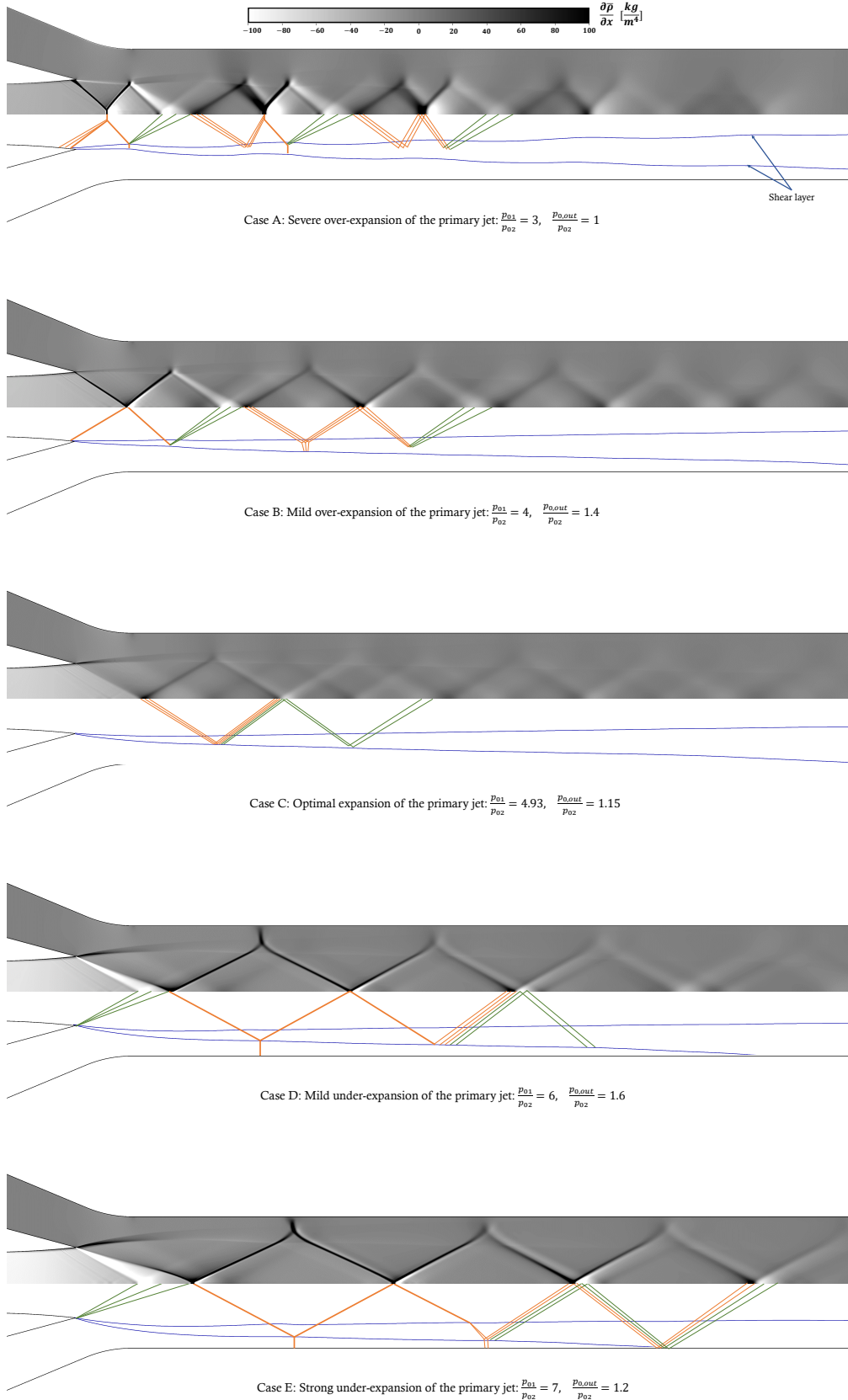


Figure 3.6: Impact of the reservoir pressure ratio p_{01}/p_{02} on the supersonic flow patterns in the ejector. Upper half: numerical schlieren images. Lower half: schematic representation of the waves associated to the flow. Shockwaves are depicted in thick orange, compression waves in thin orange and expansion waves in thin green.

Appearance of a Mach diamond for a severely overexpanded case

The case $p_{01}/p_{02} = 3$, $p_{0,\text{out}}/p_{02} = 1$ is characterized by strong oblique shocks that merge to form a normal shock downstream of the primary nozzle tip. The recompression is associated with an increase of the temperature, giving birth to a Mach diamond.

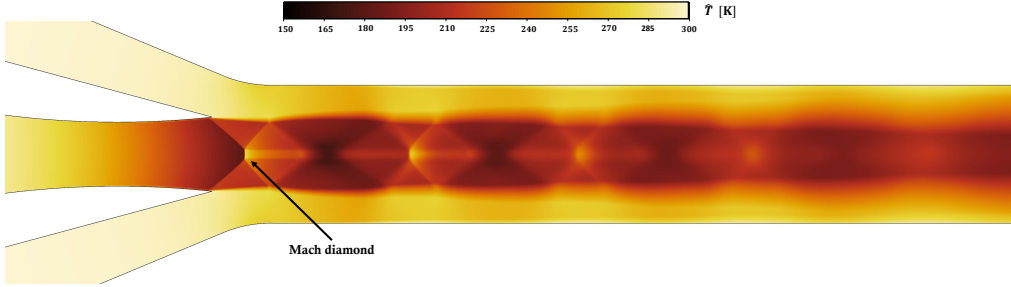


Figure 3.7: Mean-flow temperature field highlighting a Mach diamond. Operating conditions: $p_{01}/p_{02} = 3$, $p_{0,\text{out}}/p_{02} = 1$.

Evaluation of the strength of shockwaves and compression waves

This short parenthesis aims to illustrate quantitatively the strength of compression zones in order to differentiate shockwaves from isentropic compression waves. This analysis was used to obtain the schematic drawings in Figure 3.6.

Figure 3.8 depicts the static pressure (top) and the axial derivative of density (bottom) along a streamline located approximately in the middle of the primary stream, for the case $p_{01}/p_{02} = 4$ and $p_{0,\text{out}}/p_{02} = 1.4$ (whose schlieren image is shown in Figure 3.6 B). Figure 3.8a shows the passage through the 2 first oblique shocks taking place right at the exit of the primary nozzle, and Figure 3.8b through the compression wave that stands right behind the first expansion fan. The difference between both waves is unambiguous: shocks are captured over a significantly smaller amount of cells, typically between 5 and 7. The pressure rise is more abrupt and the density gradient much higher than for the compression wave, where the compression takes place smoothly over a larger amount of computational cells, and with a lower value of $\frac{\partial \bar{\rho}}{\partial x}$.

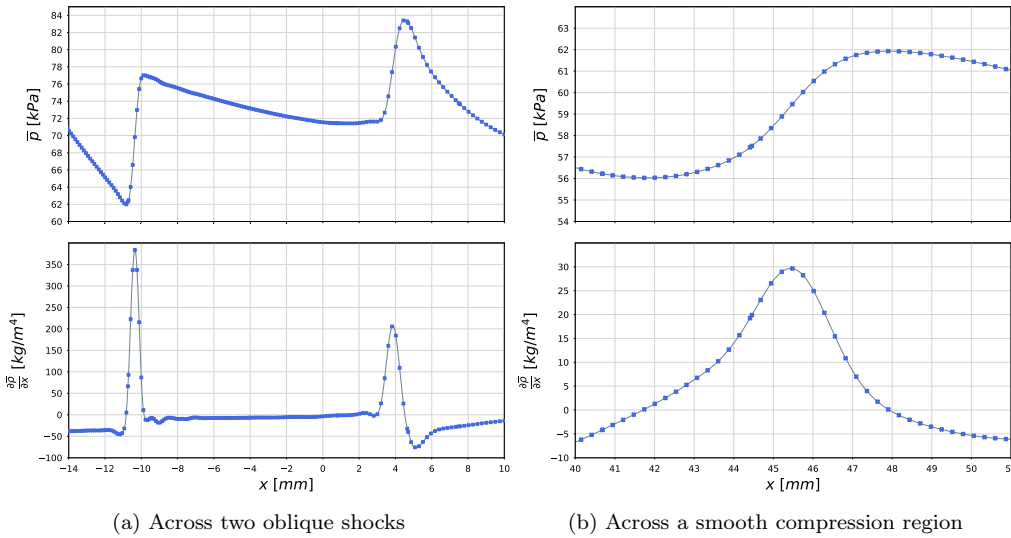


Figure 3.8: Static pressure and density gradient across shockwaves and compression waves. Each cross marker corresponds to a computational cell.

3.3.4 Preliminary mixing analysis via velocity and pressure profiles

A first approach to understand the mixing phenomenon is to have a look at the horizontal velocity and static pressure profiles across sections of the ejector along the axis. Figure 3.10 represents in the upper half several velocity profiles at axial locations corresponding to each vertical black arrow position, normalized by a reference speed $u = 500$ [m/s]. The lower half shows the related Mach number field (supersonic region only), white areas being subsonic regions. Note that the represented case is $p_{0,\text{out}}/p_{02} = 1.6$ and $p_{01}/p_{02} = 6$.

The velocity profiles are, initially, showing a high variation between the primary and secondary streams. The profile then smoothes out along the mixing duct. As soon as the flow enters the divergent, it passes through oblique shocks which converge at a shock triple point that disrupts the almost homogeneous velocity field. Later on, the following shock train decreases in intensity while the flow is progressively slowed down. The velocity even becomes negative locally due to recirculation bubbles.

Similarly, static pressure profiles are also interesting to understand the progressive mixing in the ejector in a parallel analysis with the velocity profiles. Figure 3.9 shows velocity (upper half) and static pressure (lower half) profiles for one off-design operating conditions. The figure speaks for itself by making it easier to get an idea of the mixing process along the ejector, but also the impact of 2D phenomena on the pressure and velocity profiles mostly at the starting of the mixing duct. A general observation of a better mixing performance in the on-design case (Fig. 3.10) with respect to the off-design one can be made.

The comparison was also done for $p_{01}/p_{02} = 4$ but the observed behavior is quite similar excepted for the pressure profiles that seems to be more homogeneous (see Appendix C.1).

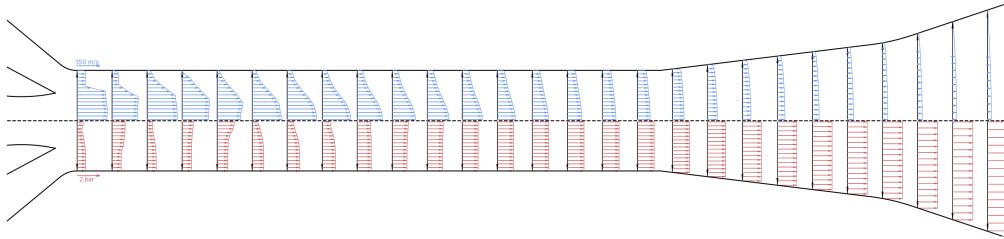


Figure 3.9: Velocity and pressure profiles along the ejector for various back-pressure. The operating conditions are $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 2.1$. (VS $\times 2$)

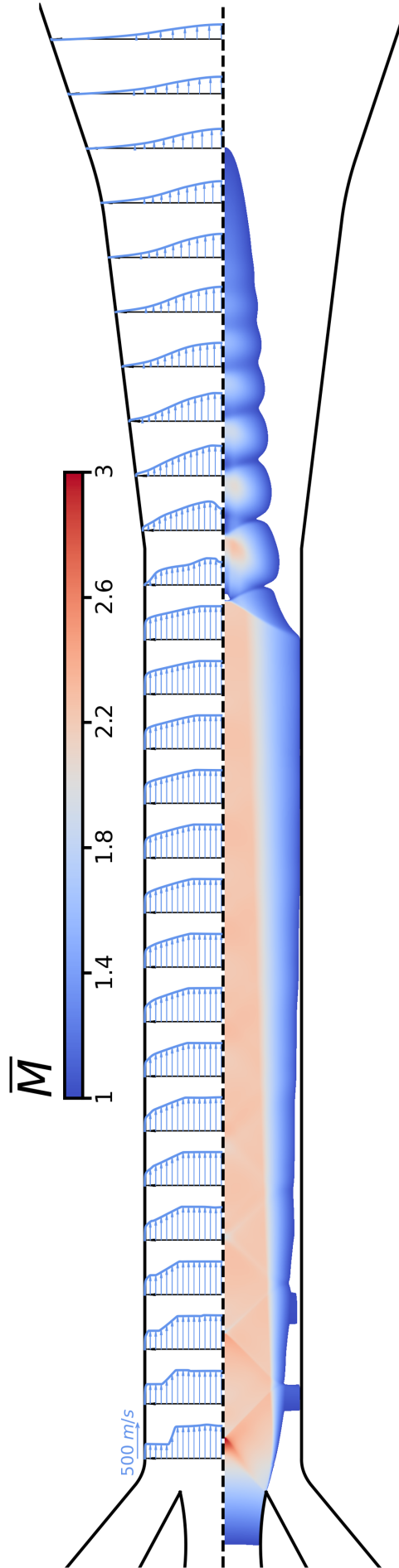


Figure 3.10: Upper half: axial velocity profiles, lower half: mean-flow Mach number field (supersonic region only). The operating conditions are $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. ($\text{VS} \times 2$)

3.3.5 Relative importance of viscous phenomena

Simplified models often make the assumption of inviscid flow, allowing to derive simple indicators to study the performance of the ejector. Others take the irreversible phenomena occurring in the boundary and shear layers into account using isentropic efficiency coefficients. In this section, it is shown that viscous effects are only negligible for a small portion of the mixing duct, and then dominate the flow when all the potential flow initially present has been "consumed".

To quantitatively evaluate the growth of the shear and boundary layers that develop respectively between the two streams and at the wall, their typical characteristic quantities are shown in Figure 3.11 for an on-design perfectly expanded case ($p_{01}/p_{02} = 4.93$, $p_{0,\text{out}}/p_{02} = 1.15$). These quantities are computed from the entrance of the mixing duct ($x/R_m = 0$) up to the point where all the potential flow has disappeared, i.e. where the boundary layer meets the shear layer. For this set of operating conditions, this point is located at $x/R_m \approx 7$.

It is difficult to define a boundary layer thickness (such as the traditional 99% thickness $\delta_{0.99}$) since there is no real external potential velocity $u_e(x)$. Indeed, the velocity profile continuously evolves across the entire radius of the ejector. To cope with this issue, a criterion based on the value of the vorticity was used, allowing to define a thickness for the boundary layer $\delta_{BL}(x)$. A similar criterion was used to define the thickness of the shear layer $\delta_{SL}(x)$.

In classical boundary layer theory, two additional quantities are defined: the displacement thickness $\delta_{BL}^*(x)$ and the momentum thickness $\theta_{BL}(x)$. The flow being compressible and the boundary layer developing on the inner wall of a cylinder, their expressions take the following form using the mean-flow quantities:

$$\delta_{BL}^*(x) = \int_0^\zeta \left(1 - \frac{\bar{\rho}\hat{u}}{\bar{\rho}_e\hat{u}_e}\right) \frac{R_m - y}{R_m} dy \quad (3.1)$$

$$\theta_{BL}(x) = \int_0^\zeta \frac{\bar{\rho}\hat{u}}{\bar{\rho}_e\hat{u}_e} \left(1 - \frac{\hat{u}}{\hat{u}_e}\right) \frac{R_m - y}{R_m} dy \quad (3.2)$$

Where y is the radial distance from the wall, i.e. $y \triangleq R_m - r$.

In boundary layer theory, ζ is chosen such that $\zeta \gg \delta$. In this case, as the boundary layer thickness is unambiguously defined, the value used is $\zeta = \delta$.

It can also be useful to define the momentum thickness of the shear layer. The definition was adapted from [86] for compressible flows.

$$\theta_{SL}(x) = \int_{r_a}^{r_b} \frac{\bar{\rho}\hat{u} - \bar{\rho}_b\hat{u}_b}{\bar{\rho}_a\hat{u}_a - \bar{\rho}_b\hat{u}_b} \left(1 - \frac{\hat{u} - \hat{u}_b}{\hat{u}_a - \hat{u}_b}\right) r dr \quad (3.3)$$

The subscript a refers to the location at the inner boundary of the shear layer, and the subscript b to the outer boundary.

The shear and boundary layer thicknesses are illustrated on the left-hand side of Figure 3.11. The shear layer quickly grows to reach as much as 40% of the ejector radius, while the boundary layers reaches a moderate value of 10% of the radius at $x/R_m \approx 7$. The right-hand side of Figure 3.11 shows the displacement and momentum thicknesses. They both remain low for the boundary layer. The momentum thickness of the shear layer is more important but remains at moderate levels.

A similar analysis (not presented here) was conducted for other values of p_{01}/p_{02} , yielding similar results. The only observed difference was the tendency of the boundary layer of getting slightly thinner with the increase of p_{01}/p_{02} . Also, the axial position of the point where the boundary layer meets the shear layer is located earlier for higher p_{01}/p_{02} , as the shear layer develops at a higher radius when the primary stream keeps expanding in the mixing duct.

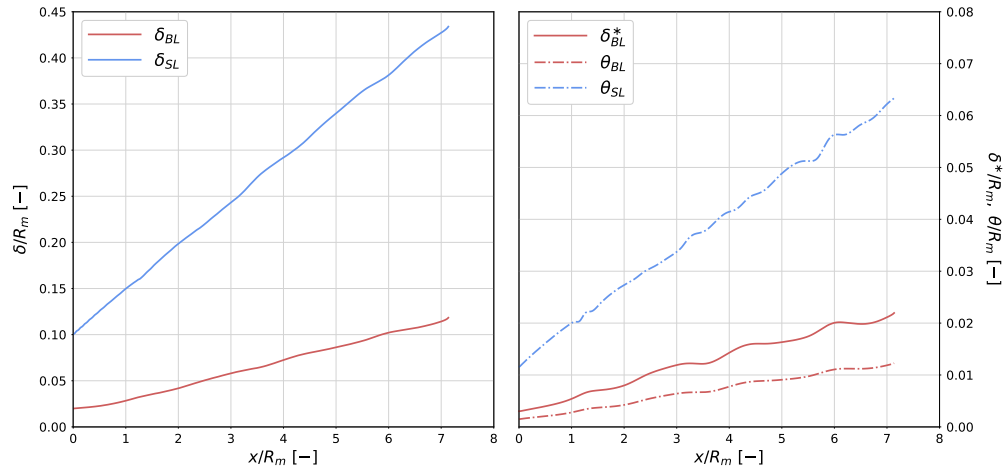


Figure 3.11: Characteristic quantities of the shear and boundary layers along the ejector, $p_{01}/p_{02} = 4.93$, $p_{0,out}/p_{02} = 1.15$

3.4 The compound choking theory: a universal explanation for the critical operation of supersonic ejectors

The general flow topology and its different features being presented, this section aims to validate the compound-choking as the general explanation of the secondary mass flow rate limitation at on-design operating conditions.

At this point, the difference between the compound choking theory and the compound-choking simplified models (also referred to as compound choking models) should be clarified. The former refers to the **physical principle** that holds for all compound-flows, i.e., that a flow composed of several streams can prevent the upstream propagation of information even though all the streams do not possess any sonic section. The latter refers to the **simplified models** (such as the one developed by Bernstein et al. [23]) that aim to predict approximately the global performance, using strong assumptions.

In this section, the choking phenomenon will be considered in the broad sense, i.e. as a **physical principle**. The analysis will use visual tools (e.g., of the Mach number field) covering a wide range of operating conditions (various p_{01}/p_{02} and $p_{0,\text{out}}/p_{02}$). Section 3.5 will be then devoted to the use the **simplified compound-choking model** (from Bernstein et al. [23])

3.4.1 Set of Mach number fields

The first step on the path of understanding the choking phenomenon is to browse each characteristic curve by increasing the back-pressure ratio $p_{0,\text{out}}/p_{02}$ and visualizing the Mach number field (in the present, only the supersonic part of the flow) in the mixing duct. This simple visual tool is of deep interest to first understand how the choking phenomenon affects the flow, and in particular how the transition from on to off-design operating conditions is translated in the flow topology.

Figures 3.12 and 3.13 show those supersonic Mach number fields for the reservoir pressure ratios $p_{01}/p_{02} = 4$ and 6, and increasing value of $p_{0,\text{out}}$. Cases A to C are on-design while cases D and E are off-design. Additionally, a more interactive support to really understand the variations between each figure is proposed through this link¹. All the cases for every reservoir pressure ratio p_{01}/p_{02} are also shown in Appendix B.

The main difference between Figures 3.12 and 3.13 is the presence of Fabri-patterns in the latter one, as opposed to the case $p_{01}/p_{02} = 4$ where there is no evidence of sonic section in the secondary flow: the part of the secondary stream that becomes supersonic is solely the effect of the viscous entrainment effect.

Both figures present a shrinking of the supersonic region as $p_{0,\text{out}}/p_{02}$ increases. Also, for low back-pressure, the supersonic region terminates as a strong oblique shock (located in the diffuser for the former case and in the mixing duct for the latter one), followed by a shock train that further compresses the flow. This shock train is composed of successive shocks and expansion regions, and is caused by the 2D nature of this supersonic flow. For higher $p_{0,\text{out}}/p_{02}$, the supersonic region systematically terminates as a secondary shock train, which can be of important length and present numerous "bumps".

An important feature to be observed in these Figures (and which is particularly striking when watching the animations) is that for the on-design cases, a portion of the flow seems to be identical between two adjacent operating conditions. This identical region shrinks as $p_{0,\text{out}}$ increases. It extends from the the ejector inlet to a certain axial position located in the mixing duct. This is due to the fact that the flow is choked, i.e. the information concerning the increasing back-pressure is blocked by the flow and cannot travel upstream. This concept is paramount for the considerations presented in rest of this manuscript, and is illustrated with a simple analogy in the next section.

¹<https://www.youtube.com/playlist?list=PLqErhQDH4L-uCNa13kshhGyFU3Gyeqbsf>

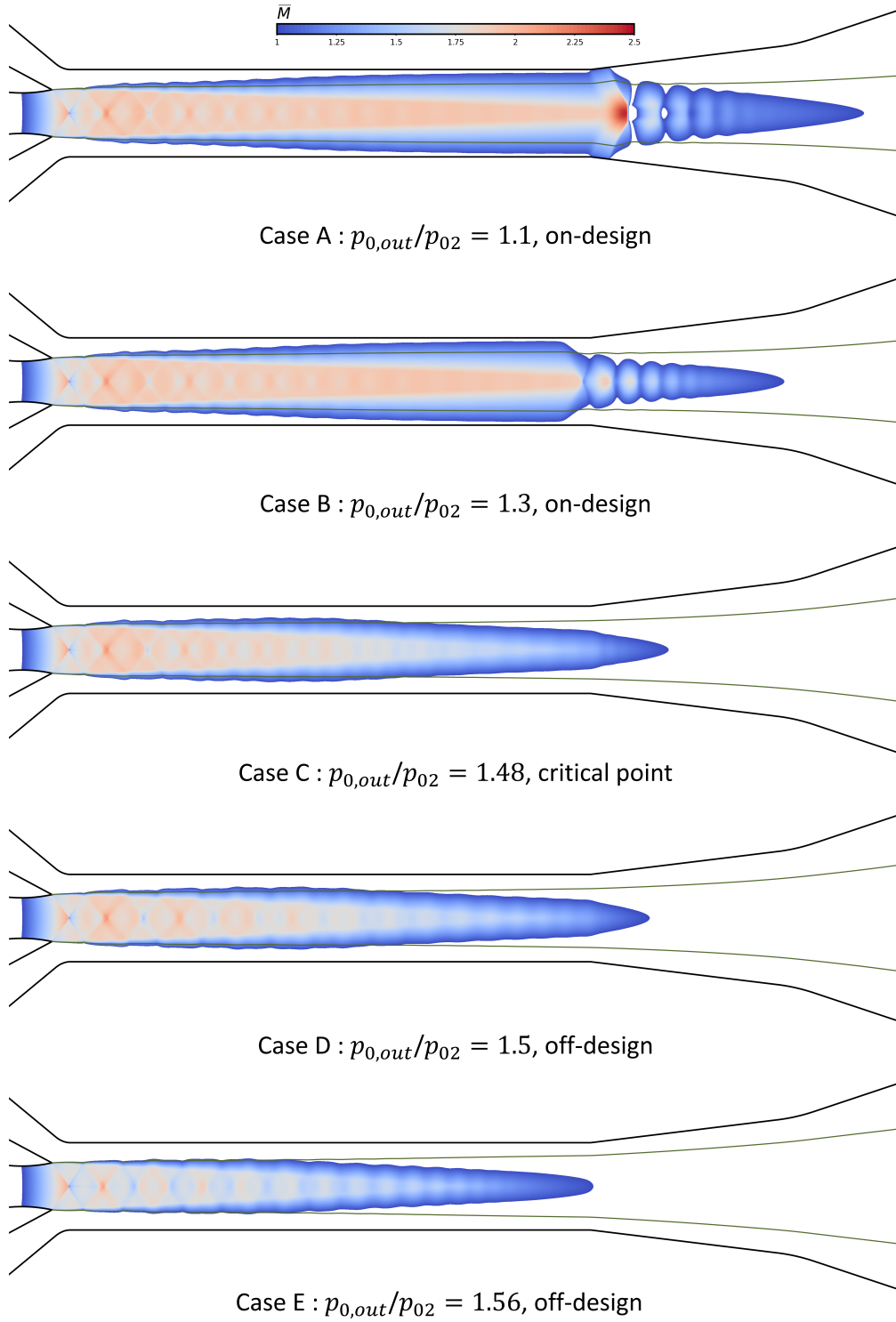


Figure 3.12: Mach number field (supersonic region only) and dividing streamlines - $p_{01}/p_{02} = 4$ (VS $\times 2$)

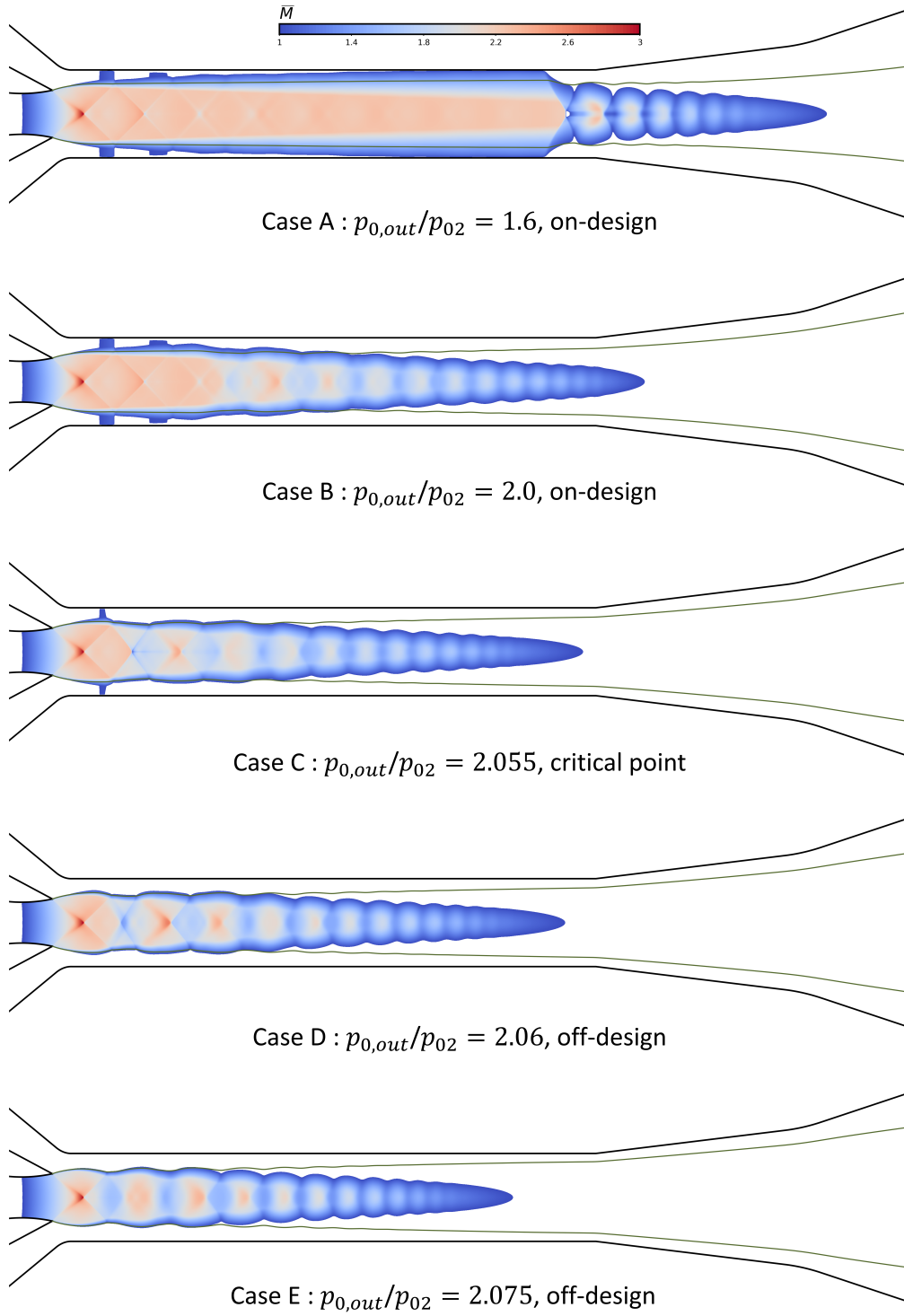


Figure 3.13: Mach number field (supersonic region only) and dividing streamlines - $p_{01}/p_{02} = 6$ (VS $\times 2$)

3.4.2 Analogy to the classical nozzle and definition of the choked region

In order to properly introduce crucial concepts and a terminology that will be widely used throughout the coming analysis, an analogy to a well-known case in compressible flows is made and illustrated in Figure 3.14.

Consider a 1D converging-diverging nozzle that operates in the choked regime (i.e., the flow is sonic at the throat), and assume that the initial back-pressure $p_{out} = p_{out,1}$ is high enough so that a normal shock takes place at the exit of the nozzle (case 1 in Figure 3.14).

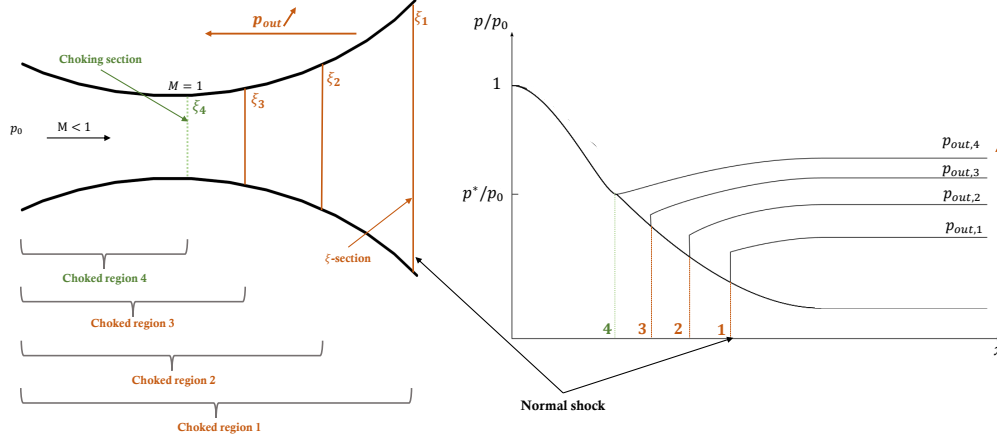


Figure 3.14: Converging-diverging nozzle, with normal shock in the diffuser moving towards the throat as the back-pressure p_{out} increases. The nozzle is represented on the left, and the pressure chart (from [87]) on the right.

If the back-pressure is increased (from $p_{out,1}$ to $p_{out,2}$), the normal shock moves upstream in the divergent part of the nozzle (left-hand side of the figure), so as to achieve a greater compression on the subsonic flow that follows the shock (right-hand side of the figure). If the back-pressure is further increased, the shock will get closer to the throat, and will eventually end up at the throat (case 4 in Figure 3.14). This is the limiting case. Any higher back-pressure will be too great of a "challenge" for this nozzle to expand the flow to supersonic velocities, and the sonic condition will not be reached at the throat.

Terminology: choked region, ξ -section and choking section

For each case, the section upstream of the normal shock (which is composed of the converging part + the supersonic part of the diverging section) is labeled as the **choked region** (this terminology is not 100% rigorous but is quite handy for the considerations of this section). The cross-section containing the normal shock, i.e. the right boundary of that *choked region*, is labeled as the **ξ -section**.

For fixed back-pressure, the *choked region* is insensitive to perturbations coming from downstream, since acoustic waves pile up on the normal shock as they propagate upstream. When considering two adjacent back-pressure cases (e.g., case 2 and 3), the flow is identical for both cases in the *choked region* of case 3. In general, two cases i and $i + n$ (provided that both are sonic at the throat) have the *choked region* of case $i + n$ in common.

A last concept is introduced: the **choking section**, defined as the last ξ -section before the flow becomes fully subsonic. In this simple case, it corresponds to the ξ_4 -section which is the throat. The *choking section* is voluntarily defined in a more complex manner than just as being the sonic section or the throat, since the analogy will be used to illustrate ejector flows where the simple concept of choked throat does not hold any more.

Application to an ejector flow

This simple reasoning can be used to grasp analogous concepts in supersonic ejectors. The behavior of the pressure as a function of the axial distance can indeed be better understood (the only difference between the converging-diverging nozzle and the ejector is that the diverging section is replaced by a constant area zone, the mixing duct). Figure 3.15 shows the mean-flow wall static pressure (for the sake of simplicity, $\bar{p}_{\text{wall}}$ is assumed to represent the static pressure in the whole cross-section) in the mixing duct and part of the diffuser for several back-pressures of the case $p_{01}/p_{02} = 7$.

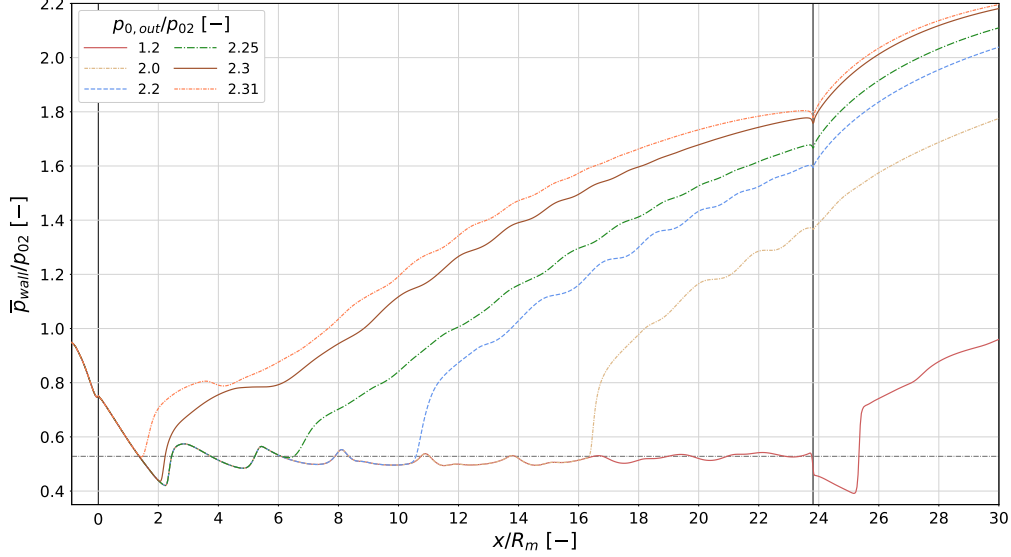


Figure 3.15: Variation of the wall static pressure with the back-pressure ratio $p_{0,\text{out}}/p_{02}$, $p_{01}/p_{02} = 7$.

The parallel with the trivial case of Figure 3.14 is notable. As the back-pressure is increased, the pressure departs that profile ever sooner, allowing a greater compression of the flow. The limiting case, $p_{0,\text{out}}/p_{02} = 2.31$, is the last where the ejector operates in the double-choking regime (analogous to case 4 from Figure 3.14).

In the case of the ejector, the same concepts can be used, i.e., a *choked region*, a ξ -*section* and a *choking section*.

Analysis of the Mach number fields thanks to the analogy

The concept defined thanks to this analogy allows to further investigate the Mach number fields shown in Figures 3.12 and 3.13.

The case $p_{01}/p_{02} = 4$ (Figure 3.12) is the confirmation that the compound-choking theory must be an explanation for the choking phenomenon. Indeed, although there is no fully sonic section, a *choked region* can clearly be identified for cases A, B and C, while the ξ -*section* logically moves upstream as $p_{0,\text{out}}/p_{02}$ increases. However, with the absence of sonic section, it turns out to be difficult to precisely identify the *choking section* only by sight. This will be done rigorously in section 3.4.3.

The case $p_{01}/p_{02} = 6$ (Figure 3.13) lies in the *Fabri-like* regime. In that case, the *choked region* and the *choking section* are more easily identifiable, and the effect of the upstream motion of the ξ -*section* is clearly visible: the Fabri patterns, which are supersonic pockets within the secondary stream, are progressively "eaten out". Case C (last on-design point) presents a reduced Fabri-pattern, which disappears as soon as the back-pressure is slightly increased (case D).

Evidence of the compound-choking on a reference case

As a final evidence of the validity of the compound-choking theory, an additional numerical study was conducted on a modified geometry (an ejector whose mixing duct is composed of circular arcs, hence presenting a clearly defined geometrical throat) and is presented in Appendix D. This result yields an additional proof of the soundness of the compound-choking theory, with a choked secondary mass flow rate despite a vast part of the flow remaining subsonic.

3.4.3 Identification of the choking section

The identification of the exact *choking section* (as defined above) cannot be achieved by simply analyzing visual results such as the Mach number fields. A precise and well-defined criterion to predict the axial location of the *choking section* for each p_{01}/p_{02} is needed for a more in-depth analysis.

Method

The method consisted in using the Mach number field $\bar{M}_i$ of each back pressure $p_{0,out}^i$ and taking the absolute value of the difference with the Mach number field $\bar{M}_A$ of a reference (the lowest) back-pressure ($p_{0,out}^A/p_{02} = 1.6$):

$$\Delta\bar{M} = |\bar{M}_A - \bar{M}_i| \quad (3.4)$$

These comparisons were done for growing back-pressures until the first off-design back pressure was found (see Figure 3.16), the previous case being the critical point.

In Figure 3.16, each half-ejector represents $\Delta\bar{M}$ for 4 different back pressures. The white color shows the respective *choked region* (i.e. the $\Delta\bar{M}$ is null) and the green color shows values higher or equal to $\epsilon = 5 \times 10^{-4}$, taken as saturation value. The boundary of that region corresponds to ξ -*section*, labeled in previous analysis. After this section, the flow differs from the reference case A. As a reminder, one can identify the last ξ -*section* as the *choking section* for the critical pressure, where the flow initially chokes in the ejector (analogy with a simple converging-diverging nozzle).

Due to the difficulty to find a universal criterion that defines precisely this *choking section*, the saturation value ϵ was the topic of much debate. After some comparisons with several tolerances values, it was decided that 5×10^{-4} was the most adapted for the whole range of primary pressure ratios (detailed explanations in Appendix C.3)

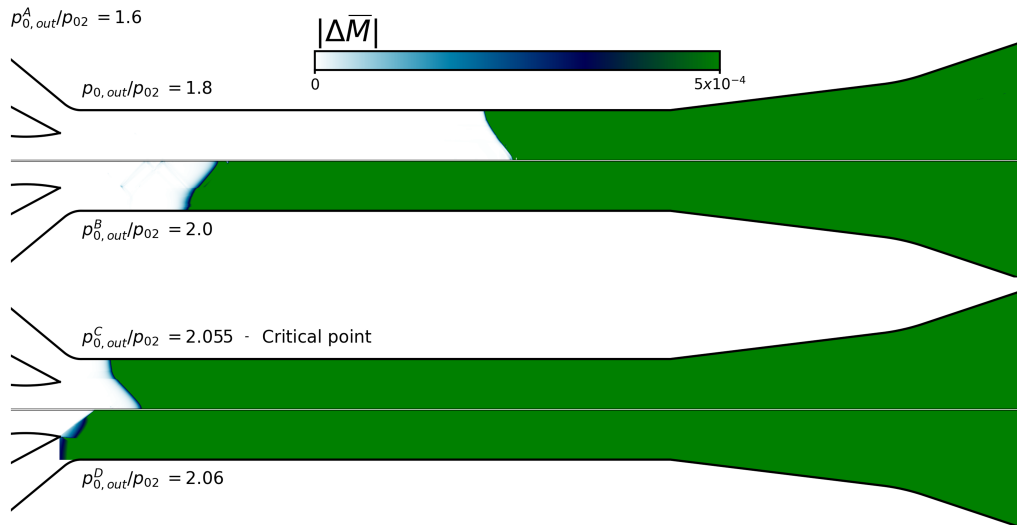


Figure 3.16: Field of $\Delta\bar{M}$ between the cases $p_{0,out}^A/p_{02}$ and $p_{0,out}^i/p_{02}$, $p_{01}/p_{02} = 6$, (VS×2)

Note that it shows the good convergence of the numerical setup as $\Delta\bar{M}$ is close to 0 when the flow is choked although these are results from different simulations.

Conical shape explanation

A zoom on the *choking section* shown in Figure 3.16 (Case C - Critical point) is depicted in Figure 3.17. It shows a characteristic "V" shape — conical in 3D. The *choking section* is thus not completely straight.

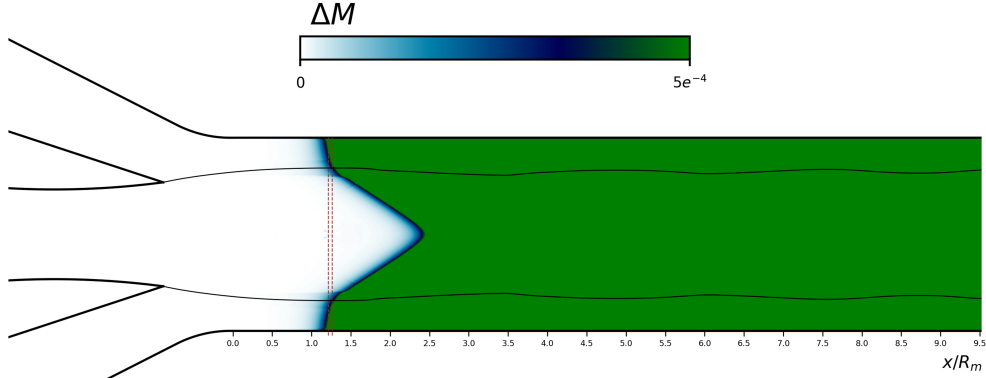


Figure 3.17: $\Delta\bar{M}$ field between the cases $p_{0,out}^A/p_{02}$ and $p_{0,out}^{C,crit}/p_{02}$ - Choking section definitions in dashed red lines (see Appendix C.3)(VS $\times 2$)

It seems that the loss of choking of the region downstream of the ξ -section originates from the secondary stream in the first place. The loss of choking propagates towards the primary stream as a line which can be assimilated to a Mach wave (due to the fact that information propagates along characteristics of angle $\mu = \arcsin \frac{1}{M}$). In this case, the value of the observed angle is found to be 28° , which corresponds perfectly to the local Mach number $M = 2.2$.

This characteristic conical shape was observed for all the cases of the *Fabri-like* regime. On the other hand, the same procedure was applied to the *compound-like* primary pressure ratios $p_{01}/p_{02} = 4$. Figure 3.18 shows two $\Delta\bar{M}$ between the reference case $p_{0,out}/p_{02} = 1.1$ and two on-design back-pressures.

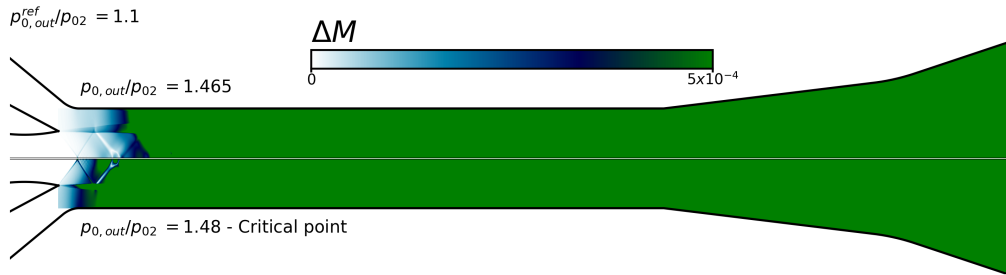


Figure 3.18: Field of $\Delta\bar{M}$ between the cases $p_{0,out}/p_{02} = 1.1$ and $p_{0,out}/p_{02} = 1.465$ (upper half), $p_{01}/p_{02} = 1.48$ (lower half)(VS $\times 2$)

It can be observed that the white region and its boundary (i.e., the *choked region* and the ξ -section) are much less defined compared to the *Fabri-like* case. One might wonder whether this would be associated to a poor convergence of the numerical simulation. This hypothesis is discarded as the level of convergence reached is the same as for the *Fabri-like* cases.

It thus seems as if it was impossible to define an exact back-pressure that would cause the ejector to "un-choke", as well as an exact *choking section*. For on-design back-pressures close to the critical point, there seems to be a continuous transition between the on-design and the off-design regime, highlighted by this $\Delta\bar{M}$ that continuously evolves from white towards green in the supposedly *choked region*.

It might be explained by the fact that in *compound-like* cases, a non-negligible part of the flow remains subsonic. Near the critical back-pressure, *compound-like* on-design cases would then be influenced by the downstream conditions even though they should not be if they were choked in the strict sense. A detailed conclusion on that matter will be given in section 3.7.

Criterion for the definition of a unique choking section x_c

A criterion to define in a precise way the axial position of the *choking section* $x_{\bar{c}_2}$ is defined in Appendix C.3 as well as the obtained values for each primary pressure ratio. The values of the *choking section* are extensively used in the next analysis.

3.5 The compound-choking model

In the previous section, numerical evidence has shown that the compound-choking was indeed (as exposed by Lamberts [24]) a broad explanation of the choking phenomenon. This section aims at assessing the indicators defined in the associated simplified 1D model, criticize the assumptions that lead to it and propose a refined version, in an effort trying to taking into account the intrinsic 2D nature of the flow.

3.5.1 Original compound-choking model

Definition of the compound-flow indicator and 1D equivalent Mach number

The compound-choking model developed by Bernstein et al. [23] and used by Lamberts et al. [24] to predict the entrainment ratio allows to associate a unique compound-flow indicator $\beta(x)$ to a flow formed by the juxtaposition of n one-dimensional streams. In addition, a 1D-equivalent Mach number $M_{eq}(x)$ is assigned to the compound-flow, in an effort to obtain a simple and intuitive (close to 1D gas dynamics) interpretation concerning the choking regime of the compound-flow.

Each stream is characterized by a cross-sectional area $A_i(x)$, a Mach number $M_i(x)$, a temperature T_i , and so on. The static pressure is assumed constant in the radial direction but is allowed to vary axially. The streams do not necessarily need to have the same stagnation pressure. The flow is assumed inviscid and adiabatic.

For more details, the interested reader will refer to [23] and [68].

Bernstein defined the compound-flow indicator as follows:

$$\beta(x) = p(x) \frac{dA}{dp}(x) \quad (3.5)$$

Where

$$A(x) = \sum_i A_i(x) \quad (3.6)$$

Each stream being considered as a 1D flow, $\beta(x)$ takes the following form [23]:

$$\beta(x) = \sum_{i=1}^n \frac{A_i(x)}{\gamma} \left(\frac{1}{M_i^2(x)} - 1 \right) \begin{cases} < 0 & \text{compound-subsonic} \\ = 0 & \text{compound-sonic} \\ > 0 & \text{compound-supersonic} \end{cases} \quad (3.7)$$

Information cannot propagate in a compound-sonic or compound-supersonic flow. In that sense, these cases are said to be compound-choked.

This prediction of the compound-choking regime results from the derivation of the propagation velocity of a weak-pressure disturbance in the compound flow [23].

Following this definition, Hedges and Hill [88] defined a useful equivalent Mach number, assuming uniform γ in the n streams :

$$M_{\text{eq}}(x) = \left(\gamma \frac{\beta(x)}{A(x)} + 1 \right)^{-1/2} \quad (3.8)$$

It has the advantage of offering the same interpretation as for the classical Mach number: $M_{\text{eq}} < 1$, $M_{\text{eq}} = 1$ and $M_{\text{eq}} > 1$ correspond to a compound-subsonic, compound-sonic and compound-supersonic flow, respectively. According to this simplified theory, if the equivalent Mach-number is greater or equal to one at any section of the ejector, the flow is compound choked — meaning that the flow upstream of this section is insensitive to the downstream conditions (i.e., to the back-pressure $p_{0,\text{out}}$).

In supersonic ejectors, 2 streams are considered: the primary and the secondary stream, separated by the dividing streamline. In order to be used in this simplified theory, a single average Mach number must be associated to each of them. Following the method of Lamberts [68], the averaged Mach number of the primary stream M_1 is computed by means of numerical integration as

$$M_1(x) = \frac{U_1(x)}{\sqrt{\gamma R T_1(x)}} \quad (3.9)$$

Where U_1 is computed with

$$U_1(x) = \frac{\dot{m}_1}{\iint_{A_1(x)} \bar{\rho} dA} \quad (3.10)$$

And T_1 with

$$T_1(x) = \frac{1}{A_1(x)} \iint_{A_1(x)} \hat{T} dA \quad (3.11)$$

The averaged Mach number of the secondary stream, $M_2(x)$, is obtained in a similar fashion.

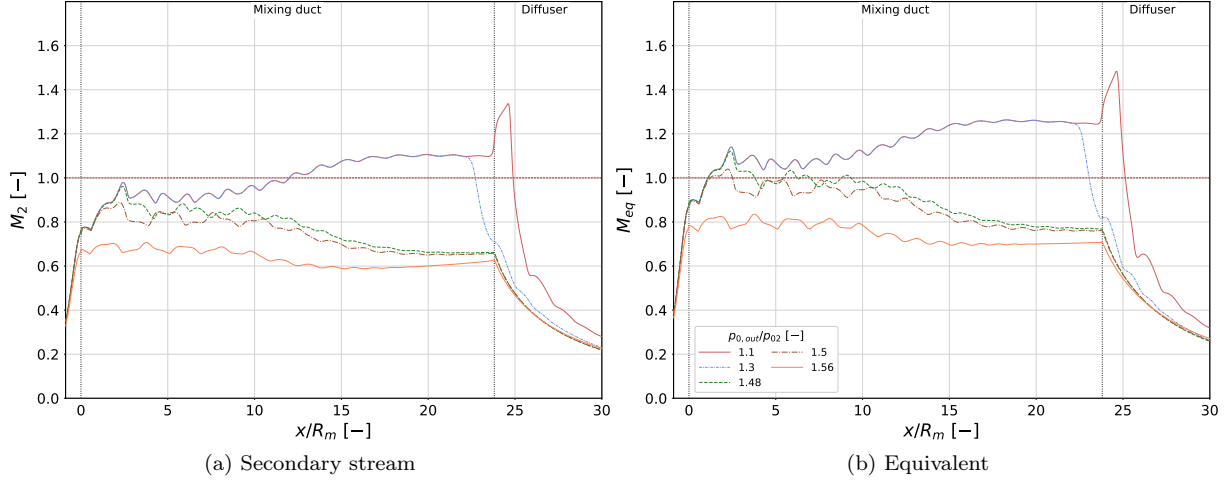
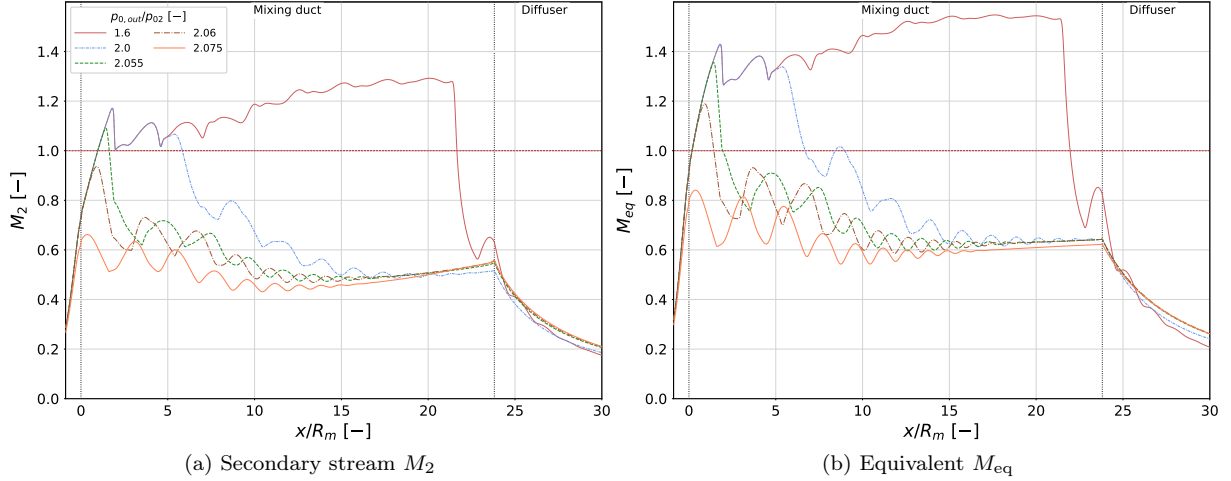
A_1 and A_2 are the cross-sections of the primary and secondary streams at location x , delimited by the dividing streamline and the ejector shroud.

Note that the compound-choking model such as developed by Bernstein et al. [23] is in fact rather a zero-dimensional model than a 1D model, due to the absence of consideration of any length scale. In most of its utilization, it served to predict global performance (i.e., the entrainment ratio) and not local variables. This consideration should be kept in mind as the following results concerning M_{eq} might be of limited accuracy.

Prediction of the choking using the equivalent Mach number

As stated before, M_{eq} is an indicator developed in the frame of a 1D model that aims to predict the choking behavior, for various operating conditions. This indicator can serve as additional evidence of the compound-choking theory, where a classical one based solely on the secondary stream would struggle to predict the correct choking regime. Yet, being the result of a simplified model, it is important to assess its accuracy and criticize the assumptions it is based upon.

The averaged secondary stream and equivalent Mach numbers are therefore shown in Figures 3.19 and 3.20 for the two sets of operating conditions under investigation, as a function of the axial position. The full length is not represented to facilitate the analysis of the phenomena occurring in the mixing duct as the behavior in the divergent is less insightful. The dashed horizontal red line differentiates the appropriate subsonic or supersonic nature.


 Figure 3.19: Equivalent Mach numbers defined by [88, 24] - $p_{01}/p_{02} = 4$

 Figure 3.20: Equivalent Mach numbers definitions used by [88, 24] - $p_{01}/p_{02} = 6$

In Figure 3.20, which is a *Fabri-like* case, the curves corresponding to $p_{0,\text{out}}/p_{02} = 1.6$ and 2.0 present two peaks exceeding $M_2 = 1$, corresponding to the supersonic pockets (Fabri pattern) visible on the Mach number field of Figure 3.13.

In the case of the *compound-like* flow (Figure 3.19), M_2 does not predict the choked nature of the cases correctly (when comparing with Mach number fields and the position of the *choked region*) and must be therefore rejected as a good indicator of the choking regime.

On the other hand, it can be seen in both Figures 3.19b and 3.20b that M_{eq} correctly better predicts the choking regime for the on-design cases (A to C) for both p_{01}/p_{02} . However, the equivalent Mach number fails to predict the correct choking regime of the case $p_{0,\text{out}}/p_{02} = 1.5$ (Figure 3.19). It is known to be an off-design case, yet M_{eq} predicts a compound-sonic section and thus a choked regime.

This analysis (and other conducted on a whole range of operating conditions, not presented here) concludes that M_{eq} seems to be a better indicator than one based on the secondary stream only. Still, some inaccuracies are present for M_{eq} , with a quite important over-prediction observed for the case D. Indeed, the M_{eq} of the first off-design case for both regimes is higher than 1 (almost 1.2 for $p_{01}/p_{02} = 6$), which is contradictory to the fact that these cases are off-design. These inaccuracies are due to the strong assumption of uniform pressure across the section, and therefore motivates to improve this indicator to correctly predict on-design and off-design operating conditions.

3.5.2 Refining of the compound-choking model

As seen in the previous section, the equivalent Mach number M_{eq} derived from the simplified compound-choking model tends to slightly over-predict the true compound Mach number. This section will try to obtain compound-indicators that better predict if the flow is choked at a given location x .

Non-isentropic model

In a first attempt to improve the definition of the equivalent Mach number and based on additional information collected through the previous sections, the aim was to investigate improved models from the literature that use coefficients and take advantage of the wide range of operating conditions at hand. Metsue et al. [21] investigated the effect of non-isentropic nozzles through the definition of polytropic efficiencies η_{pol} for each stream. The theoretical development proposed by Metsue led to a new definition of the compound-flow indicator β :

$$\beta_{\eta_{pol}}(x) = \sum_{i=1}^n \frac{A_i(x)}{\gamma} \left[\eta_{pol,i} \left(\frac{1}{M_i^2(x)} - 1 \right) - \gamma(1 - \eta_{pol,i}) \right] \quad (3.12)$$

This new definition of β behaves similarly to the original one, i.e. $\beta = 0$ indicates a compound-sonic flow.

Based on the knowledge of the critical pressure and the associated flow field information, it is possible to identify A_1 , A_2 , M_1 and M_2 at the precise *choking section* x_{c2} and then make a fit by least squares over 2 parameters, $\eta_{pol,1}$ and $\eta_{pol,2}$.

Unfortunately, the original equivalent Mach number over-predicts the reality and based on the Eq.3.12, polytropic efficiencies lower than 1 lead to an increase of the new equivalent Mach number. Indeed, the fitting results gave values of $\eta_{pol,1} = 1.98$ and $\eta_{pol,1} = 1.12$, discarding the validity of this approach when.

This was to be expected, as the compound-choking models are rather "0-dimensional" models.

Continuous version of the equivalent Mach number

In an second attempt to refine the simple original compound-choking model, and to take into account the local velocity profiles and part of the viscous effects, an upgraded version of the compound-flow indicator β was proposed. Instead of considering two streams corresponding to the primary and secondary streams, a number N of streams is considered. For instance, one could take $N \rightarrow \infty$. In that case, the expression of the compound-flow indicator defined in equation (3.7) becomes an integral:

$$\beta^\infty(x) \triangleq \lim_{N \rightarrow \infty} \frac{1}{\gamma} \sum_{i=0}^N \left(\frac{1}{M_i^2} - 1 \right) A_i = \frac{1}{\gamma} \int_{A(x)} \left(\frac{1}{M^2} - 1 \right) dA \quad (3.13)$$

The equivalent Mach number (defined in equation (3.8)) is then obtained with

$$\begin{aligned} M_{eq}^\infty &= \left(\gamma \frac{\beta^\infty(x)}{A(x)} + 1 \right)^{-1/2} \\ &= \left(\frac{1}{A(x)} \int_{A(x)} \frac{1}{M^2} dA \right)^{-1/2} \end{aligned} \quad (3.14)$$

Note that this expression, which looks like some sort of averaged Mach number, is fundamentally different from a simple mean-flow average $\overline{M}$.

The impact of this new definition of M_{eq} on the accuracy of the prediction of the compound-choking regime is shown in Figures 3.21 and 3.22, where the indicators M_{eq} and M_{eq}^∞ are compared for two values of p_{01}/p_{02} , namely 3 and 7. For each p_{01}/p_{02} , two operating cases are displayed: the last on-design (critical point) and the first off-design case. The choking location computed using the method presented in section 3.4.3 is indicated by a vertical line.

In Figure 3.21, the difference between the discrete and continuous indicators M_{eq} and M_{eq}^∞ is clear. The latter tends to take lower values than the former and to better predict the equivalent Mach number at the choking location. Indeed, the transition between the critical point ($p_{0,out}/p_{02} = 1.17$) and the first-off-design point ($p_{0,out}/p_{02} = 1.175$) almost corresponds to a transition from compound supersonic to compound subsonic (using the continuous equivalent Mach number) at the choking location (from $M_{eq}^\infty = 1.026$ to 1.008).

A similar analysis is done for $p_{01}/p_{02} = 7$ in Figure 3.22. Once again, the continuous version predicts a (locally much) lower compound Mach number than the discrete version. The location of the *choking section* is right upstream of the local maximum of both M_{eq} and M_{eq}^∞ . At that point, both equivalent Mach numbers are significantly greater than one, and there is an abrupt (more than what was observed for $p_{01}/p_{02} = 3$) decrease when reaching off-design operating conditions. As will be explained in section 3.6, this is due to the nature of the choking phenomenon in the *Fabri-like* regime. Once again, the continuous version of the equivalent Mach number seems to better predict the choking regime.

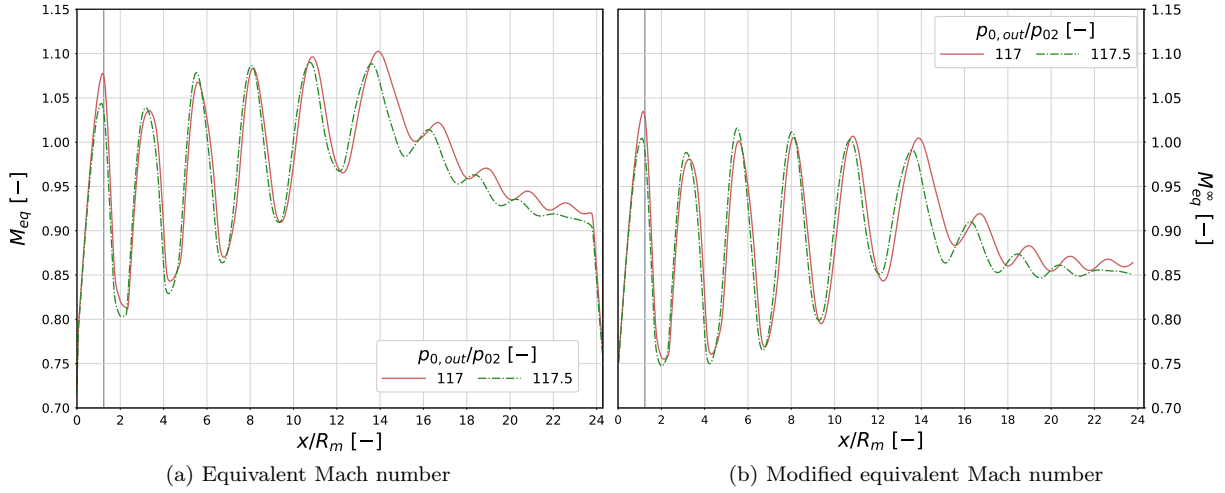


Figure 3.21: Comparison of M_{eq} and M_{eq}^∞ for the critical and an off-design case, $p_{01}/p_{02} = 3$. The computed choking location is indicated by a vertical solid line.

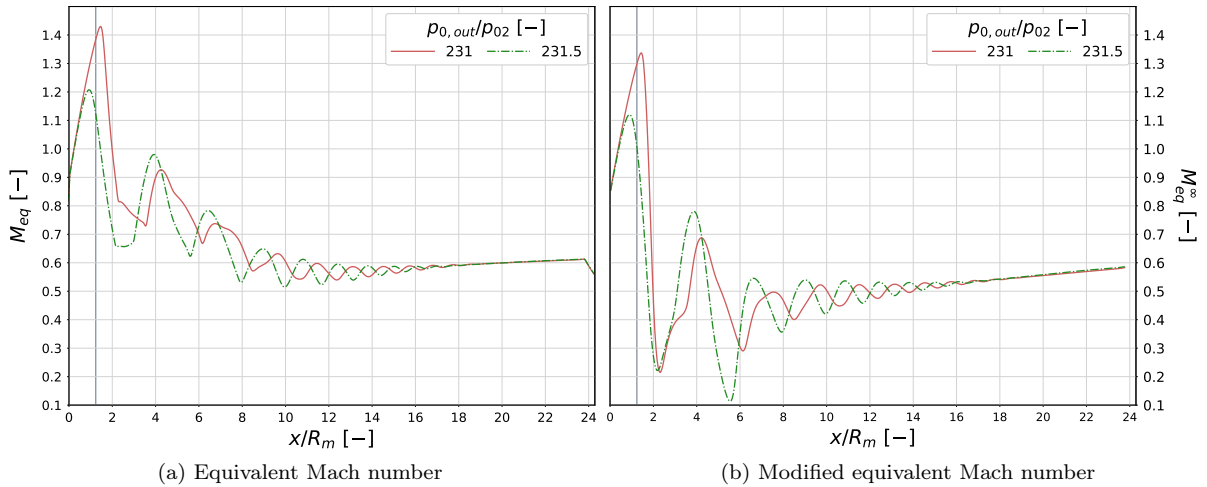


Figure 3.22: Comparison of M_{eq} and M_{eq}^∞ for the critical and an off-design case, $p_{01}/p_{02} = 7$. The computed choking location is indicated by a vertical solid line.

As a final point of comparison between the discrete and the continuous versions of M_{eq} , the values of M_{eq} and M_{eq}^∞ at the choking position are reported in Table 3.1 for all the values of p_{01}/p_{02} , in the form of the excess Mach number w.r.t equivalent Mach number unity: $\Delta M_{\text{eq}} = M_{\text{eq}} - 1$.

It can be observed that the continuous equivalent Mach number always performs better.

p_{01}/p_{02}	ΔM_{eq}	$\Delta M_{\text{eq}}^\infty$
3	0.075	0.026
4	0.039	-0.014
5	0.168	0.107
5.39	0.186	0.137
6	0.320	0.258
7	0.381	0.305

Table 3.1: Difference of predicted Mach number at the choking position for both definitions of M_{eq}

Remark—Exclusion of the boundary layers in the calculation of M_{eq} and M_{eq}^∞

As exposed by Hurtz [89], the calculation of M_{eq}^∞ should exclude the boundary layers. Indeed, from equations (3.13) and (3.14), the integrals are unbounded as $M \rightarrow 0$ when $r \rightarrow R_m$. However, the numerical calculation of these integrals mitigates this issue.

3.5.3 Uniform pressure assumption

The radially uniform pressure assumed by compound models is a really strong hypothesis. In order to assess the validity of this assumption, static pressure profiles $\bar{p}/p_{02}$ at $x/R_m = 0$ are shown for every p_{01}/p_{02} in Figure 3.23. It reveals the variability of the static pressure, far from being negligible, with a maximal factor (between extreme values) of 1.5 on a single profile. The dividing streamlines are also represented on these curves.

As a reminder, the perfectly expanded case is $p_{01}/p_{02} = 4.93$. The case $p_{01}/p_{02} = 5$ is thus near this optimal expansion, and is the closest to uniform in Figure 3.23, yet with still important variation. Under- and over-expanded cases profiles are situated on either side of $p_{01}/p_{02} = 5$ (below and above respectively).

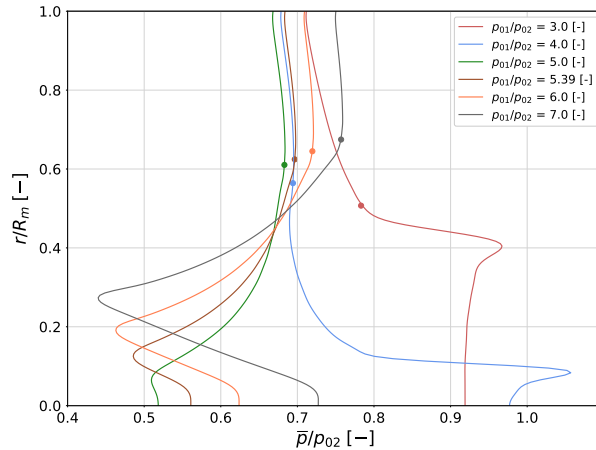


Figure 3.23: Static pressure profiles in the mixing duct of the ejector at $x/R_m = 0$ for various p_{01}/p_{02} at their critical operating condition, • corresponds to the dividing streamline radial position

Another way around is to display the pressure profile for a single operating case, at different positions between the mixing duct start ($x/R_m = 0$) and the choking axial position $x_{\bar{c}_2}/R_m$. It permits to show their level of uniformity, their proximity with the sonic pressure $p^*/p_{02} = \left(\frac{\gamma+1}{2}\right)^{-\frac{\gamma}{\gamma-1}}$ and the differences between a *compound-like* case (Fig. 3.24a) and a *Fabri-like* case (Fig. 3.24b).

First, it can be seen that the primary stream pressure is strongly influenced by the supersonic patterns (expansion fans, shocks,...). The secondary stream is more uniform for both regimes and only reaches p^*/p_{02} at the choking location for the *Fabri-like* primary pressure, as expected since this location is a sonic section.

These figures are the proof of the strong non-uniformity of the static pressure before and at the *choking section*. It explains partially the inaccuracies observed of the compound-choking model.

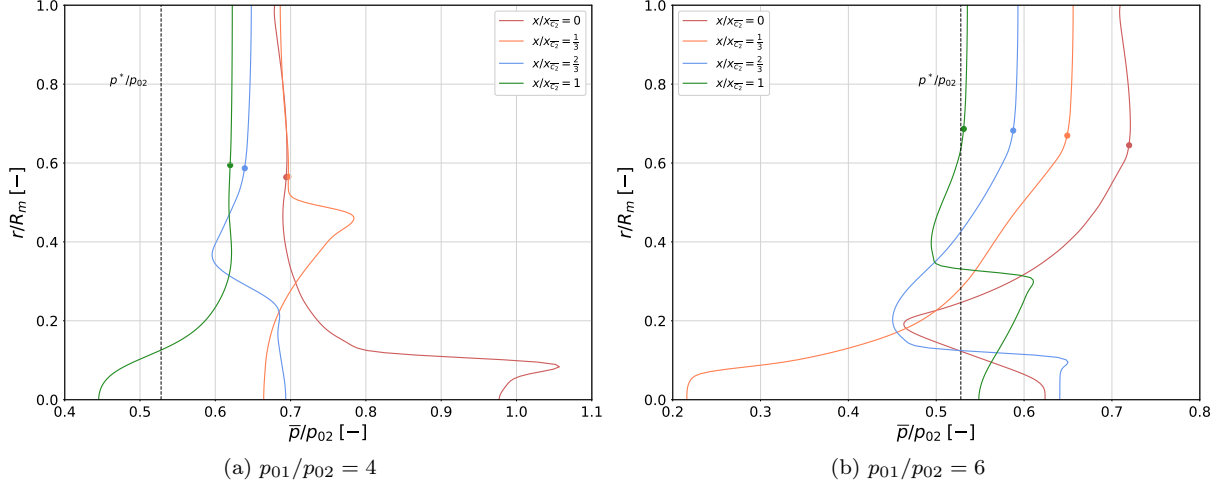


Figure 3.24: Static pressure profiles along the radius of the ejector at various locations in the mixing duct, • corresponds to the dividing streamline radial position

3.6 Detailed insight on Fabri-patterns: onset conditions and impact on the choking phenomenon

In the previous sections, the phenomenon responsible for the limitation of the secondary mass flow rate was explored under the scope of the compound-choking theory, being proved as a general explanation (although the 1D compound model developed by Bernstein et al. [23] showed limited accuracy in the prediction of the location of the sonic section). As a reminder, this explanation is quite novel in the history of the study of supersonic ejectors and aims at supplanting the traditional explanation that used the principle of a sonic section within the secondary stream. However, this Fabri-choking theory is not to be simply thrown away. Lamberts [70] indeed recently observed the so-called Fabri-pattern both numerically and experimentally. The numerical simulations used in his work also made this flow features appear.

This section is concerned with presenting the phenomenology of the Fabri-pattern in greater detail. It will then undergo a thorough investigation in order to isolate the causes of occurrence of this flow feature, by means of additional simulations and numerical tools (e.g. by examining the impact of the wall friction). Adequate conclusions concerning both the pattern in itself and the implications on the choking phenomenon will finally be drawn.

3.6.1 Numerical evidence of the Fabri-pattern

As mentioned earlier, Fabri-patterns appeared systematically in the CFD simulations for operating conditions such that $p_{01}/p_{02} \geq 5.39$, this value being the threshold between the *compound-like* and *Fabri-like* regimes.

In Figure 3.25, a Fabri-pattern occurring at $p_{01}/p_{02} = 6$ and $p_{0,\text{out}} = 1.6$ is depicted by means of a view from the primary nozzle tip and the beginning of the mixing duct. The dividing streamline, the supersonic region (delimited by the iso-contour of $M = 1$) and the iso-contour of sonic pressure $\bar{p} = p_2^*$ where $p_2^* = p_{02} \left(\frac{\gamma+1}{2}\right)^{-\frac{\gamma}{\gamma-1}}$ are shown.

The Fabri-pattern is initiated at the section where the secondary flow becomes sonic (a part of it having already been accelerated to supersonic speeds due to the interaction with the primary stream) and is terminated by a weak normal shock making the secondary flow go subsonic again (except for the part located in the shear layer that remains supersonic). Concerning its static pressure, the secondary stream behaves in accordance with one-dimensional gasdynamics theory. Indeed, it goes through the sonic state exactly where the pressure is equal to the sonic pressure (based on the reservoir pressure p_{02}). It does so in an aerodynamic throat, i.e. a local minimum of the secondary stream cross-sectional area. The evidence of this minimum of $A_2(x)$ is not shown here but will be illustrated in later section. 3.6.4.

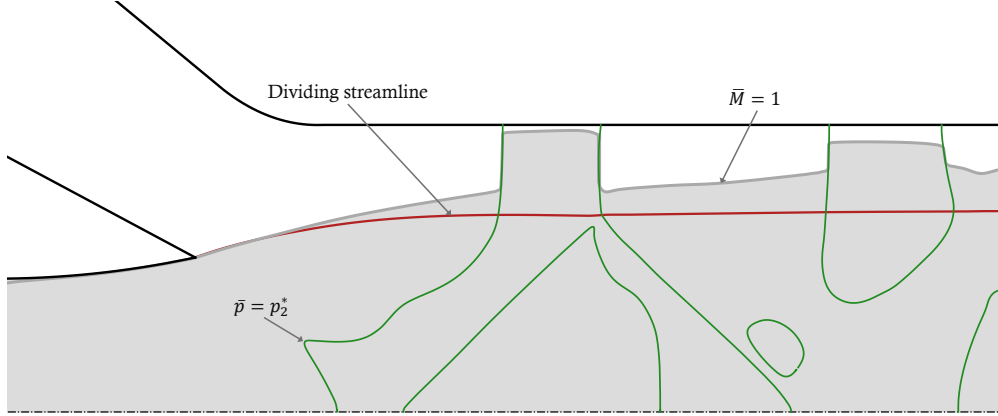


Figure 3.25: Illustration of a Fabri pattern for $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. The supersonic region is shown in grey. The iso-contour of secondary sonic pressure and the dividing streamline are also shown. (VS $\times 2$)

The question of why the secondary flow does not remain supersonic and instead is recompressed via a weak normal shock can be partially answered to by examining Figure 3.26, which allows to visualize the waves at play.

The expansion fan resulting from the underexpansion of the primary flow gives rise to a moderately strong oblique shock ("incident oblique shock" in Figure 3.26) in the primary stream. When this shockwave encounters the secondary stream (which is supersonic), it is transmitted to the latter (transmitted normal shock) and reflected in the primary core. This gives rise to the so-called Y-shockwave pattern, which then repeats several times in the mixing duct.

Note that in Figures 3.25 and 3.26, a second Fabri-pattern is visible, though less striking due to the growth of the shear and boundary layers.

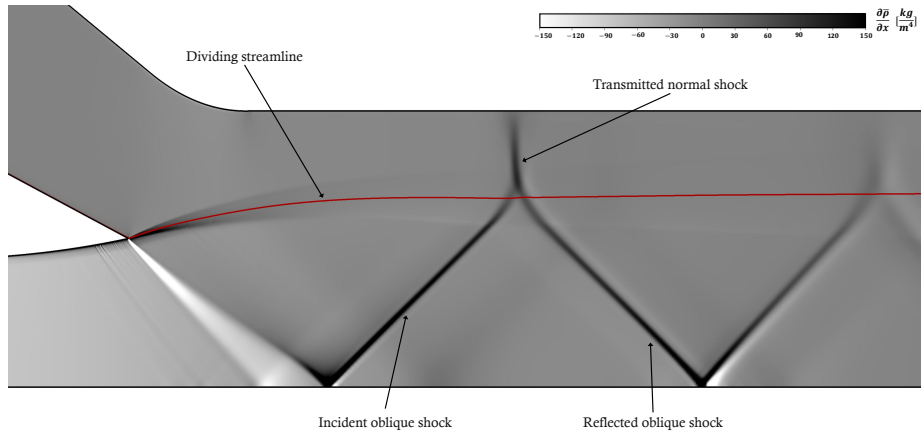


Figure 3.26: Evidence of the Y-shockwave pattern. Numerical schlieren image of the Fabri-pattern (half ejector), $p_{01}/p_{02} = 6$ and $p_{0,\text{out}}/p_{02} = 1.6$. (VS $\times 2$)

3.6.2 Effect of wall friction

When investigating the root cause of phenomena such as secondary flow choking or Fabri-patterns, it is interesting to decompose the complex flow in a set of distinct features, and hope that their individual investigation provides insight for the resultant of all of them. In supersonic ejector flows, those features are for instance the operating conditions, the shockwaves patterns and the shear & boundary layers.

This section aims to study the impact of these features on the flow using the simplest framework: 1D variable-area compressible ducted flows.

Such flows are typically governed by an equation similar to

$$\rho u A du + A dp + \tau_w P_w dx = 0 \quad (3.15)$$

Where τ_w is the wall shear stress, A the local cross-sectional area and P_w the wetted perimeter.

Due to the high level of coupling between all the variables involved, it is desirable to isolate (as much as possible) individual flow features to study their impact. In order to shed light onto the role of the wall friction (and more specifically of the ejector shroud friction), the numerical setup was adapted to suppress the wall shear stress (i.e., $\bar{\tau}_w$ is imposed to zero, which has the effect of suppressing the boundary layers).

Only the shear stress of the ejector shroud was suppressed. It was maintained on the inner walls of the ejector so as not to affect the development of the shear layer. All the other parameters of the simulation were left unchanged.

A complete analysis was made for two reservoir pressure ratios, $p_{01}/p_{02} = 3$ & 5.39, respectively representatives of *compound-like* and *Fabri-like* choking in real conditions.

A set of Mach number fields is shown in Figure 3.28 for $p_{01}/p_{02} = 3$. The result is quite surprising: numerous Fabri-patterns are identifiable, which is fundamentally opposed to what is observed in the no-slip case. It indicates that $\bar{\tau}_w$ has an important influence on the regime (*compound-like* or *Fabri-like*)

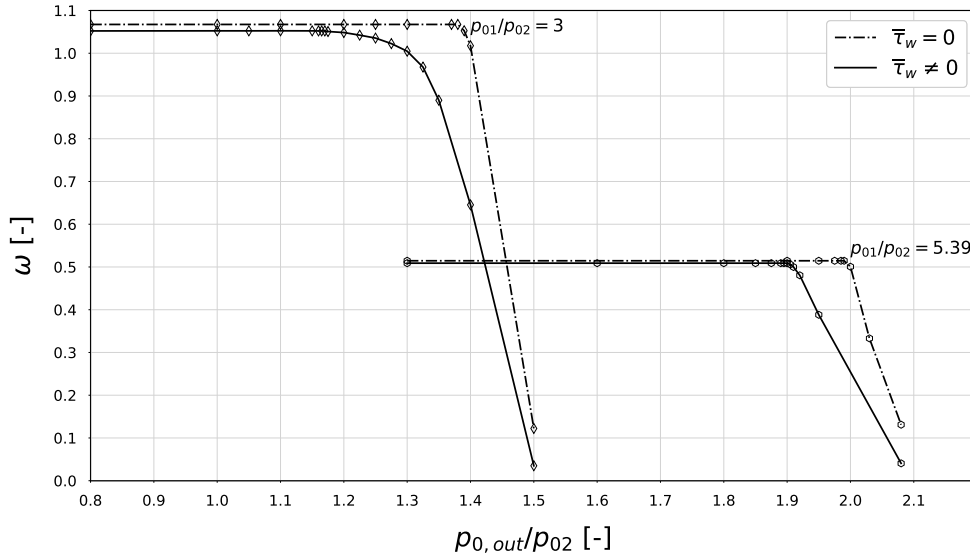


Figure 3.27: Characteristic curves of the ejector with and without friction

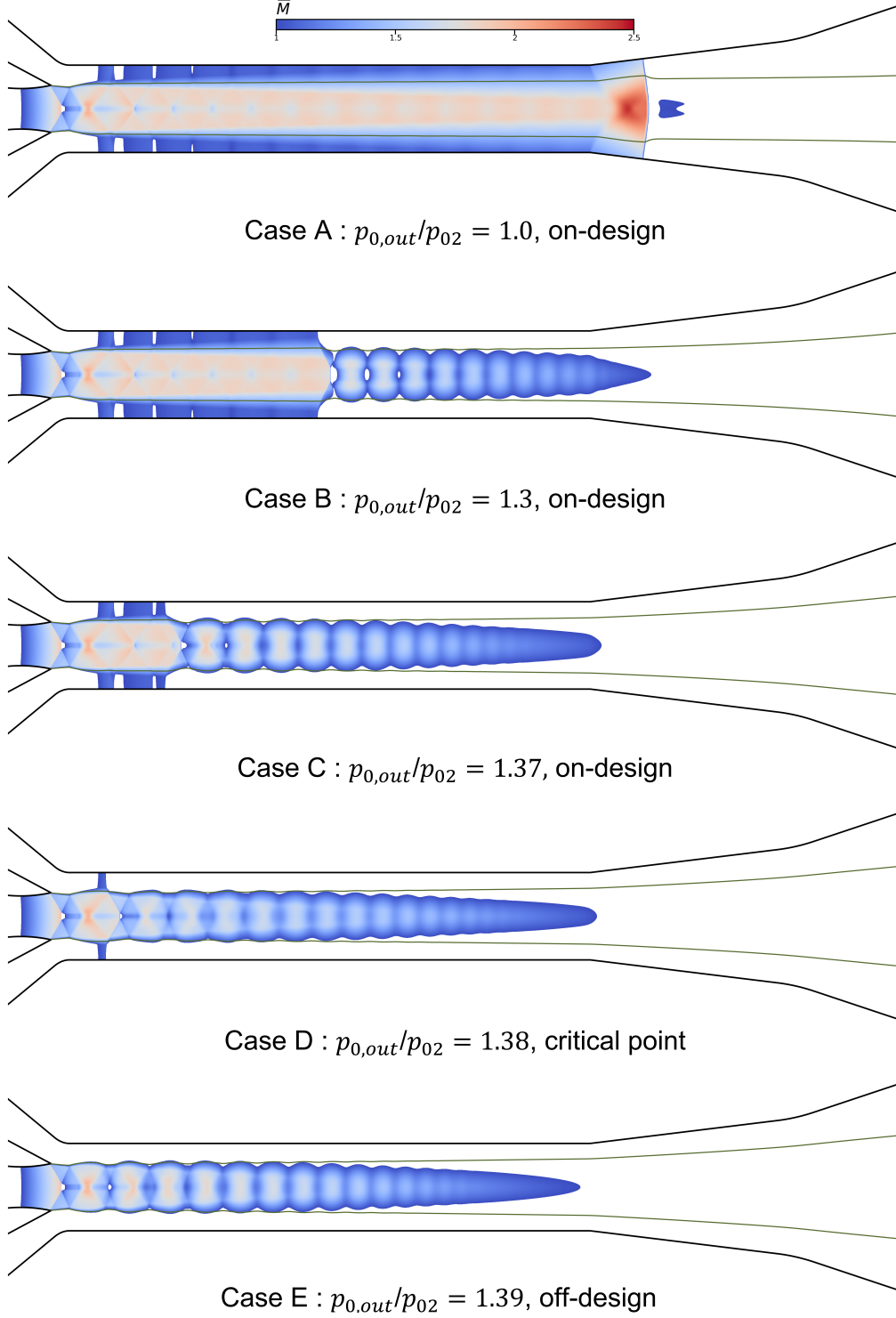


Figure 3.28: Evolution of the Mach number field along the characteristic curve without wall friction at the outer wall, $p_{01}/p_{02} = 3$ [-] (VSx2)

In Figure 3.27, the characteristic curves of $p_{01}/p_{02} = 3$ and 5.39 are displayed. The two different curves associated to each p_{01}/p_{02} corresponds to the no-slip ($\bar{\tau}_w \neq 0$) and slip ($\bar{\tau}_w = 0$) case.

First of all, the initial hypothesis (section 3.2.1) of any correlation between the sharpness of the entrainment curve with the *Fabri-* or *compound-like* regime is confirmed by comparing the curves with and without effect of wall shear stress: the Fabri-pattern effectively has the effect of sharpening the characteristic curve.

The second difference is the on-design entrainment ratio which is slightly higher in the slip case. As the boundary conditions are not modified in the primary nozzle, $\dot{m}_1$ is well conserved. It thus means $\dot{m}_2$ is responsible of the entrainment ratio increase, which is due to the absence of boundary layers.

Finally, for $p_{01}/p_{02} = 5.39$, even though the no-slip case is already in the *Fabri-like* regime, the shape of the characteristic curve is still impacted by the removal of wall shear stress. Indeed, the value of the critical pressure increases when $\bar{\tau}_w = 0$. This is due to the fact that the Fabri-pattern of the no-slip case disappears (causing the loss of choking conditions) for a lower value of $p_{0,\text{out}}$ than for the case with $\bar{\tau}_w = 0$. Indeed, when $\bar{\tau} \neq 0$, the flow loses total pressure, as opposed to the slip case (where the total pressure is conserved all along the mixing duct). The end of the choked region must therefore happen sooner to satisfy the outlet conditions.

Choking section

Similarly to the previous section, the identification of the *choking section* is performed. In the case $p_{01}/p_{02} = 3$, Figure 3.29 shows in the upper half, a field of $\Delta\bar{M}$ between the critical pressure $p_{0,\text{out}}/p_{02} = 1.381$ and the lowest back-pressure $p_{0,\text{out}}/p_{02} = 0.8$ and in the lower half, the Mach number field of the critical pressure. The dotted line represents $x_{\bar{c}_2}/R_m = 1.32$ found with the same criterion as before.

The correspondence of the first sonic section of the Fabri-pattern, the Mach number difference boundary (ξ -section) and the identified $x_{\bar{c}_2}$ proves that they refer to the same phenomenon.

As a reminder, in the no-slip case, p_{01}/p_{02} lies in the *compound-like* regime, and does not present any easily identifiable *choking-section*, as there seems to be a continuous transition between the on-design and the off-design. On the contrary, when $\bar{\tau}_w = 0$, a Fabri-pattern appears and the *choking section* is identified with stunning accuracy. This is a major result: it confirms that *compound-like* cases are not choked in the strict sense, and the presence of a Fabri-pattern (i.e., a sonic section within the secondary stream) clearly defines a true *choking section*.

A final statement can be made concerning the characteristic conical shape first referenced in section 3.4.3, which is again observed in this case, with a higher angle due to the lower Mach number (and the associated higher Mach angle).

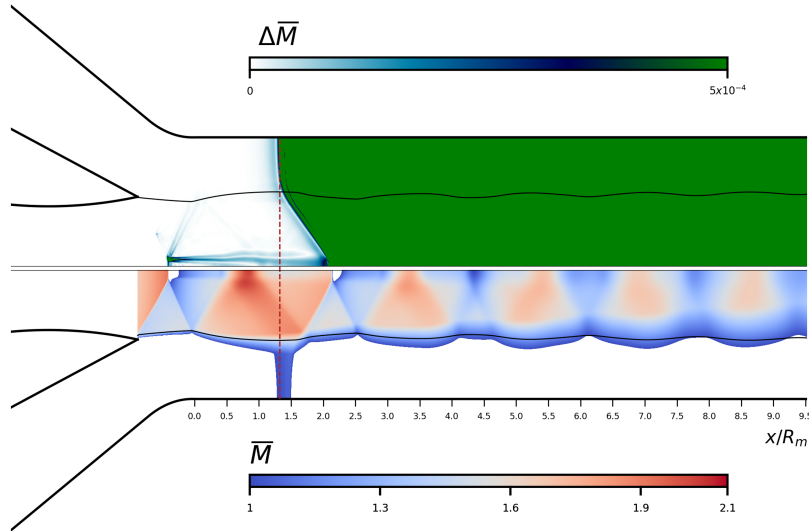


Figure 3.29: Mach number difference ΔM (top) and mach number M (bottom) fields of critical point. $p_{01}/p_{02} = 3$ - $p_{0,\text{out}}/p_{02} = 1.38$ - $\bar{\tau}_w = 0$ ($VS \times 2$)

3.6.3 Inviscid simulation

Another investigation concerns the effect of the shear stress occurring due to the interaction between both streams. As it is impossible to isolate the sole effect of the shear layer, the closest way around is to perform a fully inviscid simulation.

The modified numerical setup differs in the absence of viscosity (i.e. there is no need for a turbulence closure scheme). In order to permit the secondary stream to flow downstream without the viscous entrainment effect, the outlet pressure ratio must be set to $p_{0,\text{out}}/p_{02} < 1$. In this case, the value is set to 0.7.

Two cases are investigated: the severely over-expanded case, $p_{01}/p_{02} = 3$, and the severely under-expanded case, $p_{01}/p_{02} = 7$. Only the former is shown in this section. The latter can be found in Appendix C.4.

In Figures 3.30 and 3.31, the Mach number field and a numerical schlieren image of the flow are illustrated. The absence of viscous interaction between both streams is striking. After a few oscillations due to the over-expanded primary stream, the dividing streamline rapidly becomes a straight line and Fabri-patterns appear clearly. Due to the inviscid flow, the related shocks do not interact with the shear and boundary layers. The Fabri-patterns are thus very well defined, and the associated Y-shockwave patterns even more so (see Figure 3.31). A curiosity appears in the first Fabri-pattern: the normal shock of the classical Y-shockwave pattern is replaced by two oblique shockwaves.

These inviscid simulations will be used for the quantitative study of section 3.6.4.

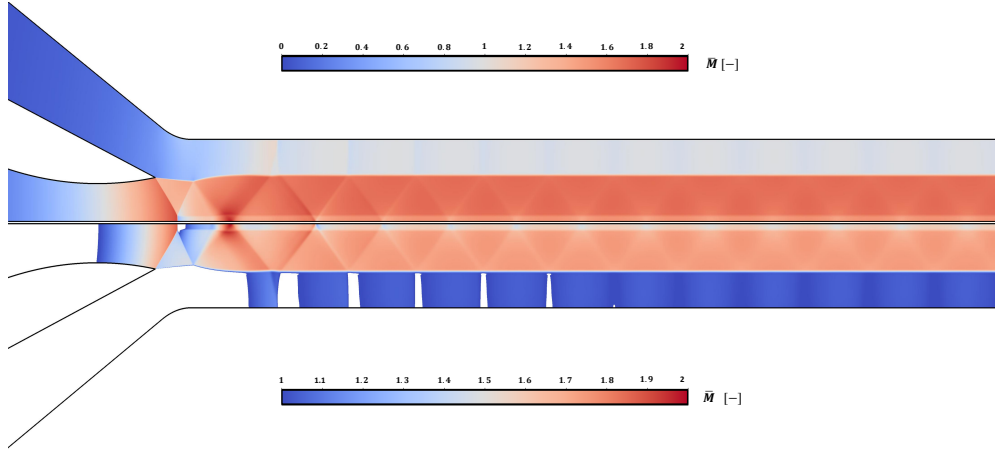


Figure 3.30: Mach number field for the case $p_{01}/p_{02} = 3$, $p_{0,\text{out}}/p_{02} = 0.7$. Upper half: complete Mach number field, lower half: supersonic region only, (VS $\times 2$).

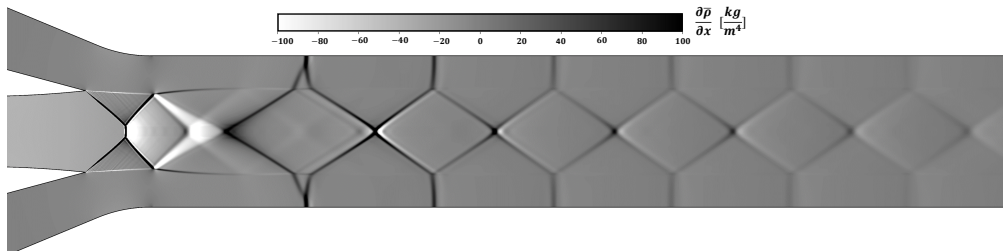


Figure 3.31: Numerical schlieren image for the case $p_{01}/p_{02} = 3$, $p_{0,\text{out}}/p_{02} = 0.7$, (VS $\times 2$).

In Appendix C.4, the same figures for $p_{01}/p_{02} = 7$ are shown. The reader is referred to this section for more details.

3.6.4 Origin of the Fabri-pattern: quantitative investigation

Based on the results presented for the case with $\bar{\tau}_w = 0$ (section 3.6.2) and for the fully inviscid simulation (section 3.6.3), this section will try to shed light on the factors that cause the Fabri-pattern to appear in certain conditions (e.g., at high p_{01}/p_{02} , or low p_{01}/p_{02} but without wall shear stress). The previous sections used mainly visual results. The present section will use quantitative indicators.

First, the shear stress occurring at the wall and at the dividing streamtube will be precisely computed.

In a second step, the study will focus on the most extreme cases, $p_{01}/p_{02} = 3$ and 7, respectively the most extreme *compound-like* and *Fabri-like* cases. For each case, three different regimes will be investigated: the real case, the case with $\bar{\tau}_w = 0$ and the fully inviscid case. They will be examined under the simplified light of variables typical of 1D compressible gas dynamics.

Finally, a last analysis related to the position of the boundary layer initiation will be conducted.

Evaluation of the shear stresses on the secondary stream lateral boundaries

The apparent fundamental role of τ_w on the compound-or-Fabri-like regime might be surprising at first glance and is thus analyzed in this section.

When looking at equation (3.15), it seems that in order to predict whether the secondary stream will reach a sonic section or not, knowledge about the shear stress exerted at the inner and outer surfaces of the secondary stream is required. As a reminder, the shear on its inner surface is positive (it is associated with a positive flux of momentum, i.e. it drives the stream), and on the outer surface negative, causing a loss of total pressure.

To illustrate the effect of these shear stresses in the beginning of the mixing duct, Figure 3.32 shows the cumulative flux of momentum (more precisely the absolute value) at the wall ($\bar{T}_w$) and at the shear layer ($\bar{T}_s$) for three reservoir pressure ratios: $p_{01}/p_{02} = 3, 5$ and 7. In short, the fluxes are computed using the viscous and turbulent stress tensors, and the local surface considered (either the wall or the dividing streamtube). They are then integrated along the axial direction.

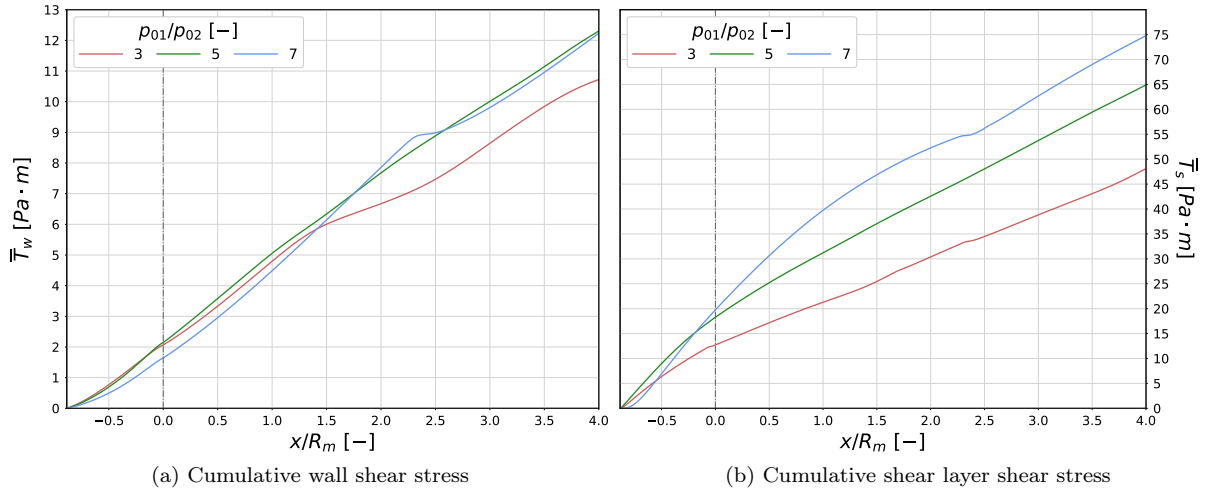


Figure 3.32: Cumulative wall shear stress $\bar{T}_w$ and shear stress between the streamtubes $\bar{T}_s$ in the beginning of the mixing duct for various stagnation pressure ratios p_{01}/p_{02} . The inlet of the mixing duct is indicated by a grey dash-dotted line.

While the flux of momentum at the wall is roughly equal for all three cases, the shear stress at the dividing streamtube and the related momentum flux are significantly impacted by the value of p_{01}/p_{02} , with almost a twofold factor between the cases $p_{01}/p_{02} = 3$ and 7, starting from $x/R_m = 1$.

Impact of the friction regime: 1D-like considerations

As identified in the previous sections, the wall shear stress seems to play an important role in the presence of Fabri-patterns. In this section, the objective will be to study more deeply its effect, using two directions of investigation that are "orthogonal". The first is p_{01}/p_{02} , and the second is the friction regime. The latter term refers to whether the simulation is fully viscous (no-slip condition at the wall), "inviscid" only at the wall ($\bar{\tau}_w = 0$), or fully inviscid.

Six cases will be considered: the three friction regimes for $p_{01}/p_{02} = 3$, and then for $p_{01}/p_{02} = 7$. As a reminder, only the no-slip case with $p_{01}/p_{02} = 3$ is a *compound-like* case, the other 5 cases all presenting Fabri-patterns. The objective of the coming analysis will be to isolate the symptoms of the Fabri-pattern, and to explain why the case no-slip & $p_{01}/p_{02} = 3$ does not contain any sonic section.

A first remark can be made based on the inviscid simulations shown in section 3.6.3 (for $p_{01}/p_{02} = 3$) and Appendix C.4 (for $p_{01}/p_{02} = 7$). The viscous entrainment is not a necessary condition for the Fabri-pattern to appear. It seems to be rather the result of cross-section area variations.

It is also worth mentioning that the mass flow rates are very close for all 3 friction regimes (respectively: real case, no wall friction and fully inviscid), as seen in Table 3.2.

	$p_{01}/p_{02} = 3$		$p_{01}/p_{02} = 7$	
	$\dot{m}_1$ [kg/s]	$\dot{m}_2$ [kg/s]	$\dot{m}_1$ [kg/s]	$\dot{m}_2$ [kg/s]
Real case	0.206271	0.217078	0.48242	0.16125
No wall friction	0.206271	0.220166	0.48242	0.16310
Fully inviscid	0.209014	0.219424	0.48770	0.16045

Table 3.2: Values of $\dot{m}_1$ and $\dot{m}_2$ for the 3 friction regimes and the 2 pressure ratios

The following analyses will focus their attention on the secondary stream. This is a reasonable way of proceeding, since the inviscid simulations clearly indicated that the primary stream does not influence the presence of Fabri-patterns, except for the room it leaves to the secondary stream.

A first insight is given in Figure 3.33, where the dividing streamline is shown for both p_{01}/p_{02} cases, and for the 3 friction regimes.

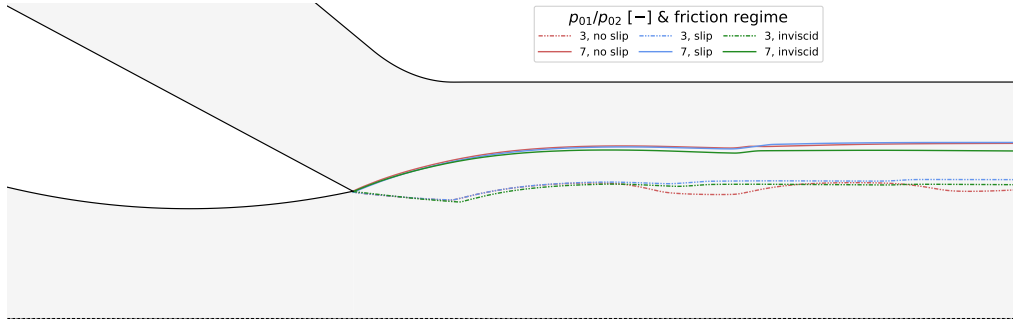


Figure 3.33: Dividing streamlines for three friction regimes: fully viscous (no slip), no friction at the wall and fully inviscid, and for two stagnation pressure ratios: $p_{01}/p_{02} = 3$ and 7. (VSx2)

Due to its further expansion after the nozzle tip, the primary stream takes a sensibly higher volume for $p_{01}/p_{02} = 7$ than for $p_{01}/p_{02} = 3$, leaving a much tighter space for the secondary stream.

The main distinctive feature of this figure is the departure of the no-slip curve from the 2 others in the case $p_{01}/p_{02} = 3$.

Finally, a last interpretation of Figure 3.33 is that the value of the secondary stream cross-sectional area is not a decisive factor on the whether the secondary stream will become sonic or not. The only certain effect is that due to the smaller sonic section, $\dot{m}_2$ will be sensibly smaller for high p_{01}/p_{02} (refer to Table 3.2).

In order to assess quantitatively the impact of that change in secondary cross-sectional area for the no-slip case, the 1D-equivalent Mach number and area of the secondary stream, $M_2(x)$ and $A_2(x)$, are depicted in Figures 3.34 and 3.35 for $p_{01}/p_{02} = 3$ and 7, and for the 3 friction regimes.

These quantities were obtained similarly as for variables used in the compound-choking models (section 3.5). A difficulty arose regarding the numerical computation of these quantities for $x < 0$. As shown in Figure 3.33 for $p_{01}/p_{02} = 7$, the position of the dividing streamline w.r.t the ejector wall makes it difficult to define a section to which the secondary flow would be roughly normal. The method used consisted in using inclined sections of progressive angle, such that the sections are roughly normal to the streamlines. This poor (but inevitable due to the 2D structure of the flow) definition of cross-sections led to erroneous values for $x < 0$, which should be discarded.

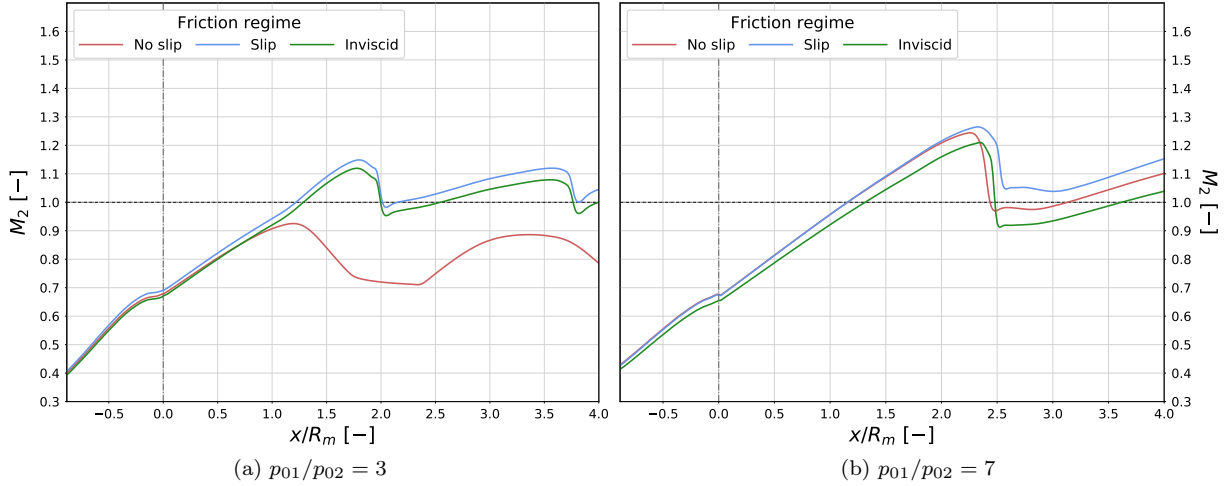


Figure 3.34: Secondary stream Mach number for different friction regimes, for two stagnation pressure ratios. The sonic line is indicated by a horizontal dash-dotted line.

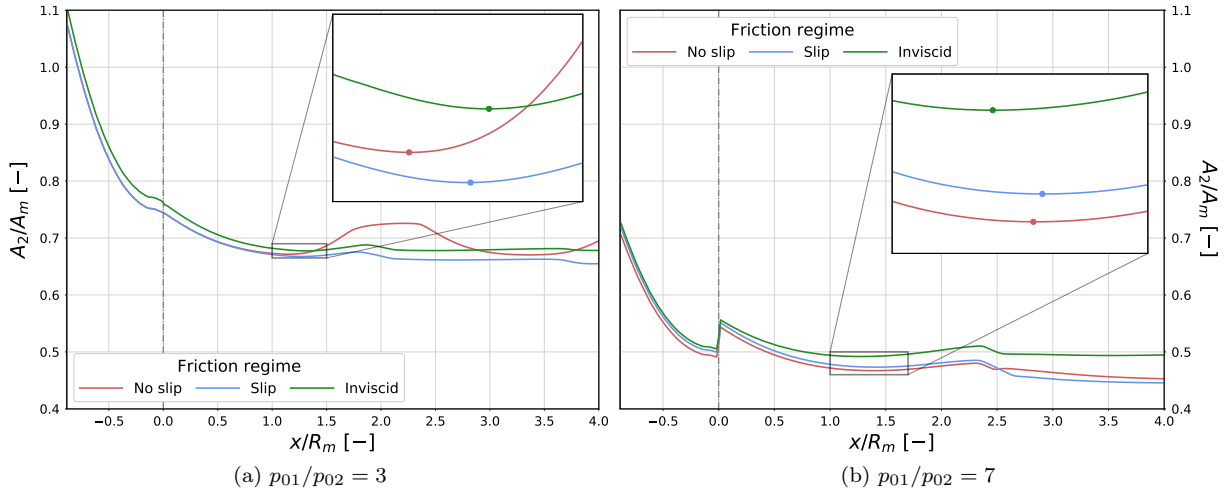


Figure 3.35: Secondary stream cross-sectional area for different friction regimes, for two stagnation pressure ratios. Local minima of A_2 are indicated with dots in the zoomed view.

The investigation taking place below will exclusively refer to the secondary stream as a 1D stream with variables A_2 and M_2 .

In Figures 3.34a and 3.35b, it can be observed that the secondary stream reaches the sonic conditions approximately at $x/R_m = 1.2$, except for the no-slip case of $p_{01}/p_{02} = 3$ which remains subsonic (as a reminder, it is the only case that does not display Fabri-patterns).

As seen in Figures 3.35a and 3.35b, the secondary stream cross-section area A_2 reaches a minimum at the sonic section for all the cases presenting a Fabri-pattern. This is in accordance with classical 1D gas dynamics. Interestingly, the case $p_{01}/p_{02} = 3$ with no-slip condition also presents a minimum of A_2 , but it is not synonym of any sonic section.

With those observations in mind, and the fact that the dividing streamline of the no-slip case departs the 2 other cases starting at $x/R_m \approx 1$, the following partial conclusion can be drawn: the area of the secondary stream reaches a minimum (see Figure 3.35a) too soon compared to the 2 other cases, for which M_2 gets greater than 1. As a result, the secondary stream falls short from becoming sonic, and is slowed down after the minimum of A_2 .

Although it can be pointed out with relative confidence that the non-presence of the Fabri-pattern in the no-slip case is due to the wall friction, there is still no explanation for the physical process that causes the dividing streamline to depart those of the choked cases, and to prevent the existence of a Fabri-pattern for this case.

Impact of the shockwave-boundary layer interaction

As a last analysis of the reason for the absence of Fabri-pattern for the case no-slip & $p_{01}/p_{02} = 3$, an ultimate simulation was performed: only a specified portion of the ejector shroud was imposed with $\bar{\tau}_w = 0$. Specifically, upstream of a position labeled x_{slip} , the wall shear stress $\bar{\tau}_w$ was imposed to zero, while the downstream region had a classical no-slip condition.

This allowed to identify a threshold value of x_{slip} for the Fabri-pattern to appear.

Figure 3.36 displays a numerical schlieren image of the beginning of the mixing duct, for $p_{01}/p_{02} = 3$ and $p_{0,\text{out}}/p_{02} = 1.1$. The upper half shows the case with $x_{\text{slip}} = 37$ [mm], and the lower half the case with $x_{\text{slip}} = 36$ [mm]. In the former case, a Fabri-pattern is present, whose terminating normal shock is clearly visible (precisely between $x = 36$ and 37 [mm]). In the latter case, no Fabri-pattern is visible, i.e. the secondary stream essentially remains subsonic throughout the ejector.

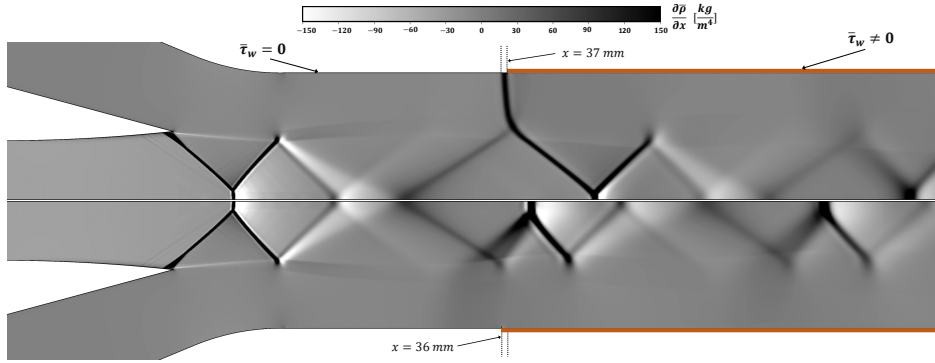


Figure 3.36: Impact of the position of the beginning (black dotted vertical lines) of the no-slip region (dark orange region) on the presence of a Fabri pattern. Numerical schlieren image, $p_{01}/p_{02} = 3$, $p_{0,\text{out}} = 1.1$.

The value of x_{slip} such that the Fabri-pattern appears is thus determined. Furthermore, by analyzing the schlieren image of Figure 3.36, the exact cause of the absence of a Fabri-pattern for the case no-slip & $p_{01}/p_{02} = 3$ is identified with relative confidence: the wall friction and the associated boundary layer prevent the existence of the normal shock that usually terminates the Fabri-pattern, and this even if the boundary layer thickness is very small: in the lower half of Figure 3.36, the boundary layer is initiated only a few tenths of millimeter upstream of the position of the normal shock.

As soon as the boundary layer starting point is located right behind that shockwave, the latter is not compromised and allows the existence of the Fabri-pattern.

This result shows the importance of local phenomena on the broader flow structures. In this case, the interaction between a shockwave and a boundary layer seems to play a determining role in the existence of the Fabri-pattern. Should any further investigation

on the onset of Fabri-patterns take place, they might try to obtain more precise information about the nature of that shockwave-boundary layer information, e.g. by locally refining the mesh in the axial direction in order to capture features such as the famous Lambda foot shockwave.

A relevant question is the following: why do the values of $p_{01}/p_{02} > 5.39$ yield Fabri-patterns, regardless of the friction regime? For these cases, the Fabri-associated shock structures seem to be more resilient to the effect of boundary layers. Part of the answer might lie in the fact that the net momentum flux is higher for high values of p_{01}/p_{02} (Figure 3.32). This matter could also be looked into more deeply.

3.7 Conclusion: A broader interpretation of the choking phenomenon

The prime result of this chapter is the confirmation of the compound-choking theory as a valid explanation of the choking phenomenon. The latter was highlighted using visual aid (i.e., the field of Mach number for various operating conditions along a characteristic curve). Despite a large portion of the secondary flow remaining subsonic throughout the ejector, a choked regime was clearly recognizable, characterized by a maximization of the secondary mass flow rate. In the absence of any sonic section, only an argument close to the compound-choking theory could therefore explain this apparent choking of $\dot{m}_2$.

In the meantime, two distinct working regimes were identified, depending on the reservoir pressure ratio p_{01}/p_{02} : all the on-design cases with $p_{01}/p_{02} \geq 5.39$ exhibit Fabri-patterns, giving rise to the distinction between the *compound-like* and *Fabri-like* regimes.

Moreover, this change of regime for increasing p_{01}/p_{02} was correlated with a modification of the characteristic curve. Higher reservoir pressure ratios (in the *Fabri-like* range) indeed present characteristic curves with sharper corners, with the on-design and the off-design regions very distinctly defined. On the other hand, *compound-like* pressure ratios are characterized by smoother transitions between the on-and off-design regions, i.e., a rounded corner of the characteristic curve. In other words, in *Fabri-like* cases, a critical back-pressure $p_{0,\text{out}}^*$ (which marks the transition between the on-and off-design region) is easier to define than for *compound-like* cases.

Another determining result was obtained by conducting simulations with $\bar{\tau}_w$ imposed to zero at the ejector shroud. This had the effect of turning all the cases into *Fabri-like* cases. This systematic appearance of Fabri-patterns was then correlated to a striking sharpening of the characteristic curve, corroborating the following fact: the Fabri-patterns seem to have the effect of making the transition from on-design to off-design more sudden. They also increase the value of the critical back-pressure.

These observations were the first hints that *something* seemingly changes in the nature of the choking phenomenon when Fabri-patterns are present.

The confirmation of this suspicion came from the study of the evolution of the *choked region* and of the identification of the *choking section*, concepts defined in section 3.4.2. Using the difference between the Mach number fields of cases close to the critical back-pressure and a reference case (i.e., the case with the lowest $p_{0,\text{out}}$), the following observation was made: the identification of the ξ -section was straightforward for *Fabri-like* cases, but turned out to be difficult for the *compound-like* ones. Indeed, for the latter cases, the ΔM presents an ambiguous evolution for operating conditions near the critical pressure, and does not allow to define with accurately a ξ -section or a *choking section*. Conversely, *Fabri-like* cases allowed to define a very precise *choking section*, i.e. the loss of choking between the last on-design and the first off-design points was unambiguously visible in the field ΔM .

Gathering this intriguing observation and the considerations regarding the sharpness of the entrainment curve, an explanation can be formulated. These elements seem to indicate a loss of choking for the *compound-like* cases that appears in a progressive manner, making the definition of a distinct ξ -section impossible for cases near the critical back-pressure, whereas the loss of choking in a *Fabri-like* case is suddenly caused by the disappearing of the Fabri-pattern.

As a reminder, a choked flow can be defined by the saturation of its mass flow rate, but also by the blockage of information-carrying acoustic waves. The compound-choking theory considers that the flow can be choked at a location where the flow is not entirely sonic. In their simplest form, the compound-choking models indeed state that a flow composed of a supersonic primary stream and an entirely subsonic secondary stream can be compound-sonic, and thus choked. In their original argument, Bernstein et al. [23] considered that axially-propagating compound waves propagate at a unique absolute velocity α , which is not the local speed of sound, in order not to violate the assumption of equal pressure at the interface between each stream.

Based on the evidence of progressive "un-choking" in the *compound-like* cases, and sudden "un-choking" in the *Fabri-like* cases, the authors tried to conciliate the compound-choking theory (which proved to be a better explanation than the Fabri-choking theory, the latter being unable to predict correctly the choking observed in *compound-like* cases) and those observations. This procedure was also motivated by the argument on which the compound-choking models are based, i.e. a compound-wave traveling at a uniform speed α , which is difficult to conceive in a case of ejector flows. Indeed, the pressure is not uniform at all along the radial direction, and more generally the flow cannot so easily be summarized by 1D equivalent streams.

An illustrative view (whose sole purpose is to show qualitatively the speculated explanation, the authors lacking knowledge in the field of viscous flow acoustics) is shown in Figure 3.37.

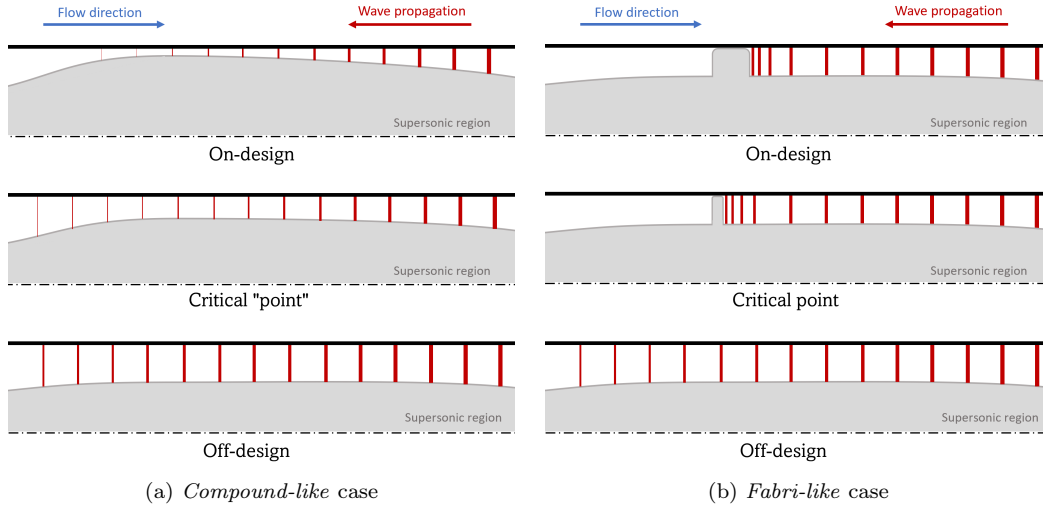


Figure 3.37: Schematic illustration of the partial blockage of information in the *compound-like* case, and the complete blockage in the *Fabri-like* case. Emphasis is put here that this figure only illustrates the concept of information propagation, and does not depict the physical behavior of acoustic wave propagation.

In Figure 3.37a, a *compound-like* case is represented for three operating conditions (the critical "point" refers to the difficulty of identifying a precise critical point in a *compound-like* case) at a given location in the mixing duct. The supersonic region (delimited by the iso-contour $M = 1$) is shown in grey. Figure 3.37b shows a similar situation, for a *Fabri-like* case.

Vertical red lines represent a (very) simplified view of acoustic waves that originate from downstream (more specifically, from the ejector outlet, where the back-pressure is imposed). They represent the information that travels along the local characteristic directions at the speed of sound. Hence, according to this simplified view, no waves can propagate upstream in the supersonic region (hence, no red lines are represented in the grey region).

In the *compound-like* case, pressure disturbances would not be completely blocked at the compound-sonic section. However, they would not be able to propagate any further, notably due to viscous effects (case On-design in Figure 3.37). As the back-pressure closes to the "critical" back-pressure, some information about the downstream conditions would make it past the compound-sonic section, initiating the rounded form of the characteristic curve. In a fully off-design case, acoustic waves would be allowed to travel upstream.

According to the proposed explanation, in the *Fabri-like* case, the Fabri-pattern would completely hinder the propagation of pressure disturbances. At the exact moment the Fabri-pattern disappears (from critical point to off-design), the acoustic waves would be immediately allowed to travel upstream, effectively "un-choking" the secondary stream. This sudden transition would explain the sharp corner observed for the Fabri-pattern case.

The authors emphasize that the explanation proposed above is of very limited impact on the global behavior. From a macroscopic point of view, although the transition from on to off-design is progressive for the *compound-like* cases, it does not fundamentally impact the definition of the on-design and off-design regions. As a matter of fact, to observe this phenomenon of progressive "un-choking", the characteristic curve had to be substantially refined near the critical back-pressure.

The aim of this section is not to propose a truly alternative explanation for the choking phenomenon, but rather to speculate on the observations related to the sharpness of the characteristic curve and the suddenness of the "un-choking". Indeed, the authors do not possess any quantitative evidence concerning the propagation of information in such complex flows, and clearly have limited knowledge on that matter. An interesting perspective would be to effectively investigate the matter of information propagation in supersonic ejectors, using more advanced numerical tools.

Also, it stands out from this chapter that Fabri-patterns (and the associated sonic section in the secondary stream) are only particular flow features, whose onset is governed by intricate parameters. The final conclusion of this chapter could be stated as follows:

The explanation for the choking phenomenon must be based on an argument close to the compound-choking theory. This flow choking can be overridden by a Fabri-pattern, which provides a true sonic section, but is not a necessary condition for the blockage of information coming from the ejector outlet. Its effect is to make the "un-choking" of the secondary flow happen in a more sudden fashion than for the cases without Fabri-patterns.

Chapter 4

Ejector performance: one-dimensional models, transport and mixing phenomena

4.1 Introduction

The purpose of this chapter is to assess the performance of the designed ejector, by means of both global (e.g. the entrainment ratio) and local (e.g. transfers of exergy between the primary and secondary streams) indicators.

In the first section, the global performance will be studied under the scope of simplified 1D thermodynamic models. The next section will then use the tools developed by Lamberts [69], i.e. transport tubes, to visualize the transport of momentum, kinetic energy and exergy from the primary towards the secondary stream. Finally, once again using the quantities defined by Lamberts, the local transfer occurring through the dividing streamline will be investigated along with the sources of exergy losses.

Numerical results exploited in the chapter

Although all the characteristic curves contain valuable information, the results presented in this chapter will be confined to a reduced set of operating conditions, just like in the previous chapter. They were chosen so as to illustrate the two apparent regimes in the ejector, respectively *compound-like* and *Fabri-like*.

As seen in Figure 4.1, the cases that will be mainly used correspond to $p_{01}/p_{02} = 4$ and 6, respectively belonging to the *compound-like* and *Fabri-like* regime.

On each curve, different back pressure ratios $p_{0,\text{out}}/p_{02}$ will be studied. They are identified by a number from I to VI to travel along each curve, and were chosen so as to have a sample representative of the whole entrainment ratio curve. The exact values of the corresponding operating conditions are listed in Table 4.1.

	I	II	III	IV	V	VI
$p_{01}/p_{02} = 4$	1.3	1.48	1.53	1.56	1.6	1.7
$p_{01}/p_{02} = 6$	1.6	2.05	2.0625	2.075	2.1	2.2

Table 4.1: Selected numerical simulations - $p_{0,\text{out}}/p_{02}$ values

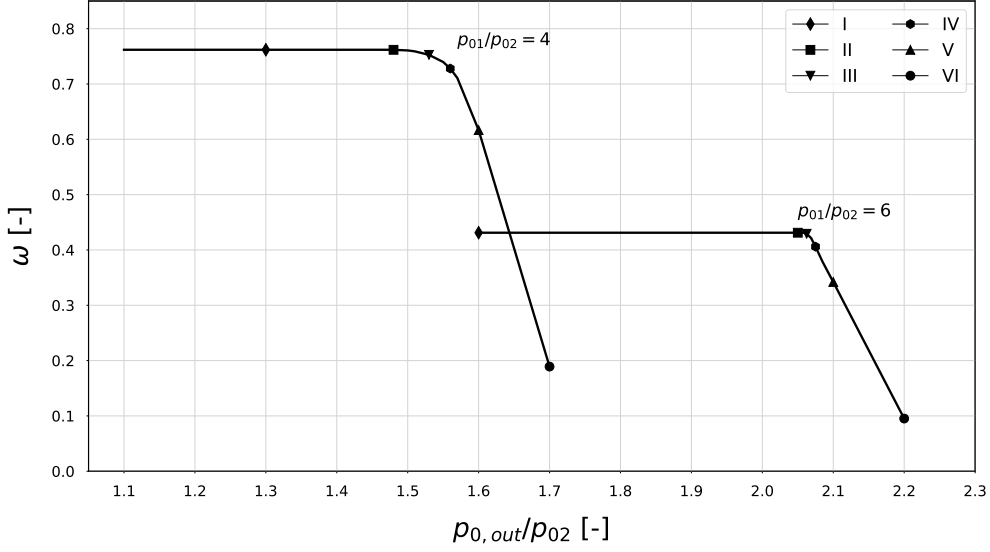


Figure 4.1: Numerical characteristic curves used in this chapter

4.2 Comparison with 1D thermodynamic models

As described in section 1.2.3, one-dimensional thermodynamic models have always been of great interest to monitor an industrial application without performing complex simulations. The great number of numerical data of this work is really interesting to fit the several one dimensional models developed in the literature and assess their accuracy. The analysis will be performed on three different models : a Fabri-choking model (i.e. based on the assumption of a sonic section in the secondary flow), the isentropic compound-choking model and non-isentropic compound-choking model.

These models use an isentropic efficiency for the primary flow which can be determined beforehand using the equation 4.1:

$$\dot{m}_1 = \frac{p_{01} A_t}{\sqrt{T_{01}}} \sqrt{\frac{\gamma}{R} \left(\frac{2}{\gamma + 1} \right)^{\frac{\gamma+1}{\gamma-1}}} \sqrt{\eta_p} \quad (4.1)$$

where η_p can be computed using all the characteristic curves. A mean value $\eta_p = 0.9764$ is obtained with a maximal deviation between the different p_{01}/p_{02} of 0.3%. This value is fixed in the following models as it characterizes the nozzle for every $p_{0,out}$ and isn't sensitive with p_{01} .

4.2.1 Fabri-choking model

The first 1D model that was used comes from the work of WeiXiong Chen et al. [19]. This model uses an empiric parameter ψ_m defined in relation to the geometrical parameter A_m/A_t .

If $\psi_m = 0.84$ (empiric value from [19] adapted to our geometry) is used, the obtained results are far from the CFD numerical results, especially concerning the predicted critical pressure $p_{0,out}^*/p_{02}$.

A further analysis to obtain a better fit by applying weights on the on-design ω and the critical pressure $p_{0,out}^*/p_{02}$ values was performed through three parameters : the secondary stream efficiency η_s , the efficiency reflecting the losses between the nozzle exit and the secondary sonic section η_{py} and finally the frictional losses during the mixing ψ_m (as a reminder, this theory assumes the mixing only starts after the aerodynamic throat).

The fit to the numerical curves obtained with CFD is shown in Figure 4.2.

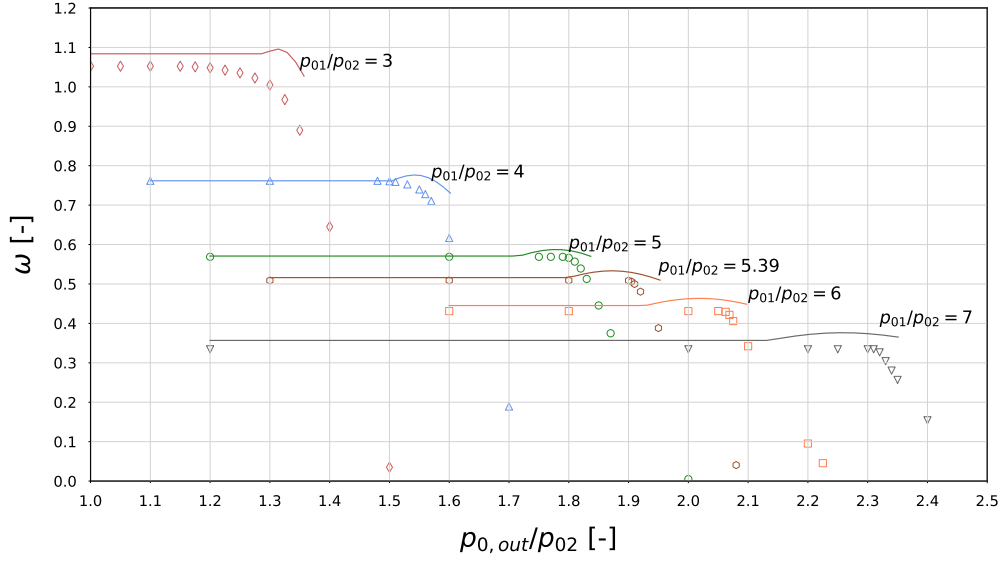


Figure 4.2: Entrainment ratio, comparison of W.Chen model and numerical results
 $\eta_p = 0.9764$, $\eta_s = 0.935$ and $\psi_m = 0.93$ || ($\diamond$): CFD results, (—): 1D Fabri model

These parameters were chosen to fit perfectly the curve $p_{01}/p_{02} = 5$ and one can observe the quite good fit of the entrainment ratio as well as the critical pressure position (excepted the bump already discussed in the literature [21]). However, the behavior of the off-design curve is not accurate and was partially removed from Figure 4.2 for clarity reasons (complete curves can be found in Appendix E.1).

It can be concluded that the model yields limited results especially regarding the off-design curve. Moreover, these results were obtained with a value of ψ_m that is different from the prescribed one.

4.2.2 Isentropic compound-choking model

The second model studied is the isentropic compound choking model developed by O.Lamberts [24], it only computes the on-design entrainment ratio by assuming isentropic processes. Figure 4.3 shows on the left, the entrainment ratio predicted by the model compared to CFD results and on the right, the deviation $\sigma = \frac{\omega_{1D} - \omega_{CFD}}{\omega_{CFD}}$ as a function of the reservoir pressure ratio p_{01}/p_{02} .

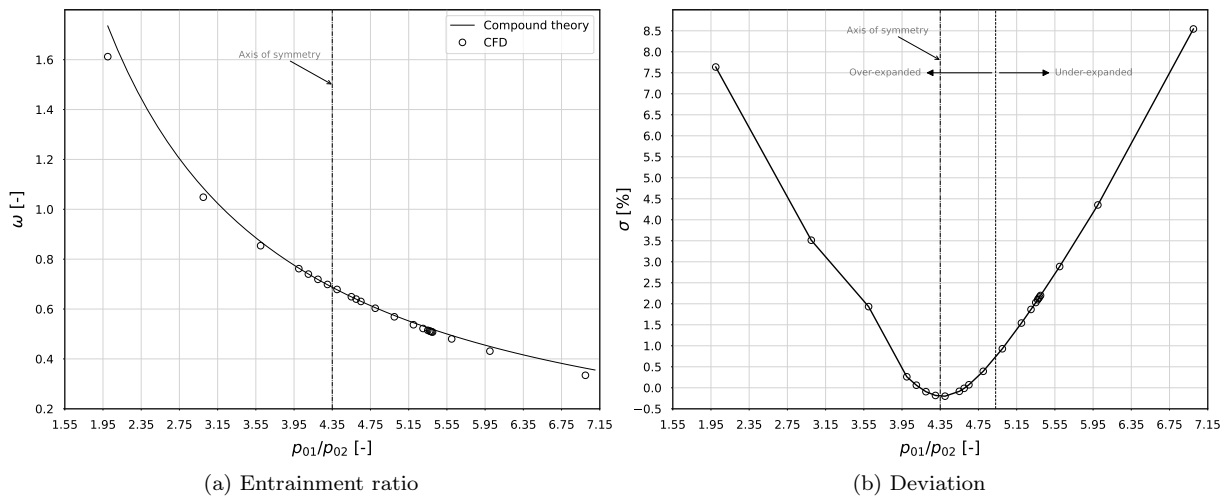


Figure 4.3: On-design entrainment ratio. Comparison of the isentropic compound-choking model with CFD results for various p_{01}/p_{02} . The ratio leading to optimal expansion of the primary flow is indicated by a dotted vertical line.

The deviation follows an atypical pattern with two zeros in $p_{01}/p_{02} = 4.1$ & 4.6. The deviation is symmetrical around the value $p_{01}/p_{02} = 4.35$. The expected behavior was to find a zero situated at the perfectly expanded pressure ($p_{01}/p_{02} = 4.93$) as it is the closest to the uniform pressure assumption of the model but the presumption is not verified according to Figure 4.3b. The behavior of the deviation could not be explained despite further analyses.

4.2.3 Non-isentropic compound-choking model

The last model analyzed is the thermodynamic compound-choking model of Metsue et al.[21], where they initially uses four isentropic efficiencies (η_p , η_s , η_{py} , η_d) and the mixing efficiency η_m but also conclude of the only usefulness of three of them : the primary flow efficiency η_p , the secondary flow efficiency η_s and the mixing efficiency η_m . The main goal was to find the best fittings parameters for the case $p_{01}/p_{02} = 5$ due to its proximity with the perfectly expanded case. The best performing value was found by applying weights on the on-design entrainment ratio and the critical pressure. The result was computed for each curve with these parameters (see Figure 4.4).

The result is remarkably accurate for $p_{01}/p_{02} = 5$, it confirms the accuracy of this model and one remarks the predicted loss of accuracy when moving away from the perfectly expanded case. Nonetheless, the error is still acceptable for the presented cases with a maximal error lower than 7.5% (for $p_{01}/p_{02} = 7$), less than in the isentropic case.

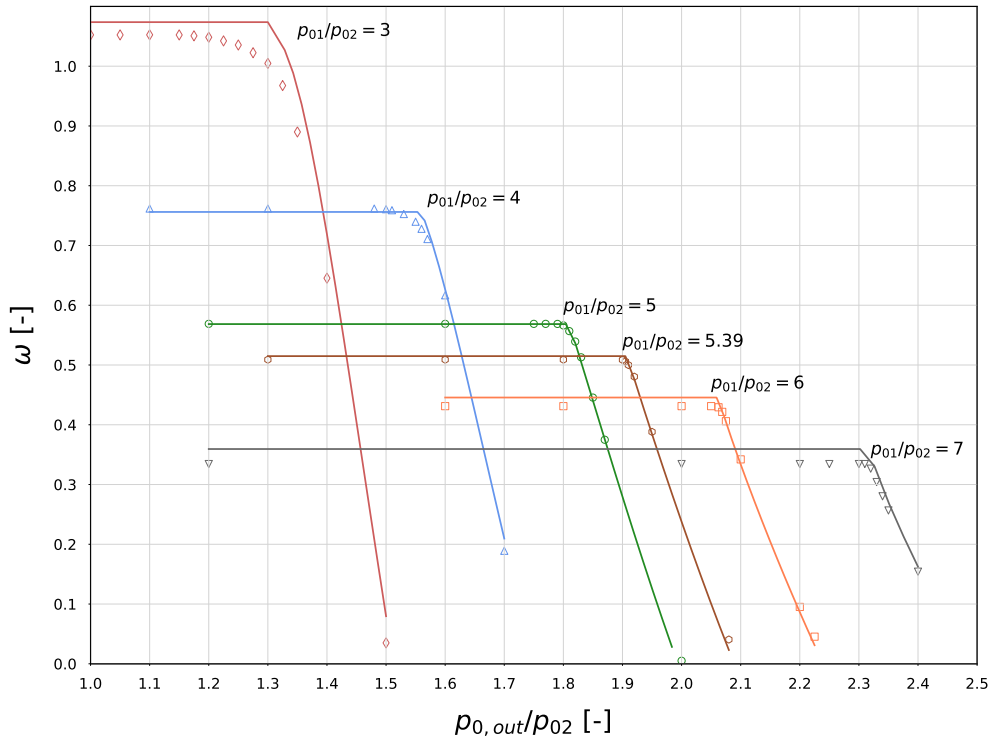


Figure 4.4: Entrainment ratio. Comparison of non-isentropic compound-choking model [21] with numerical results

$\eta_p = 0.9764$, $\eta_s = 0.965$ and $\eta_m = 0.923$ || ($\diamond$): CFD results, (—): compound model

Instead of using a single value for all the characteristic curves, it is also interesting to find the best fitting parameters for each of them (see Fig. 4.5). Concerning the efficiencies, an interesting result is the η_s behavior that matches the Figure 4.3b. The further away from the perfectly expanded case, the lower is the efficiency of the secondary stream (as η_p is constant). The mixing efficiency seems to increase with the primary pressure ratio and we can understand it easily as lowering η_m reduces $p_{0,out}^*/p_{02}$.

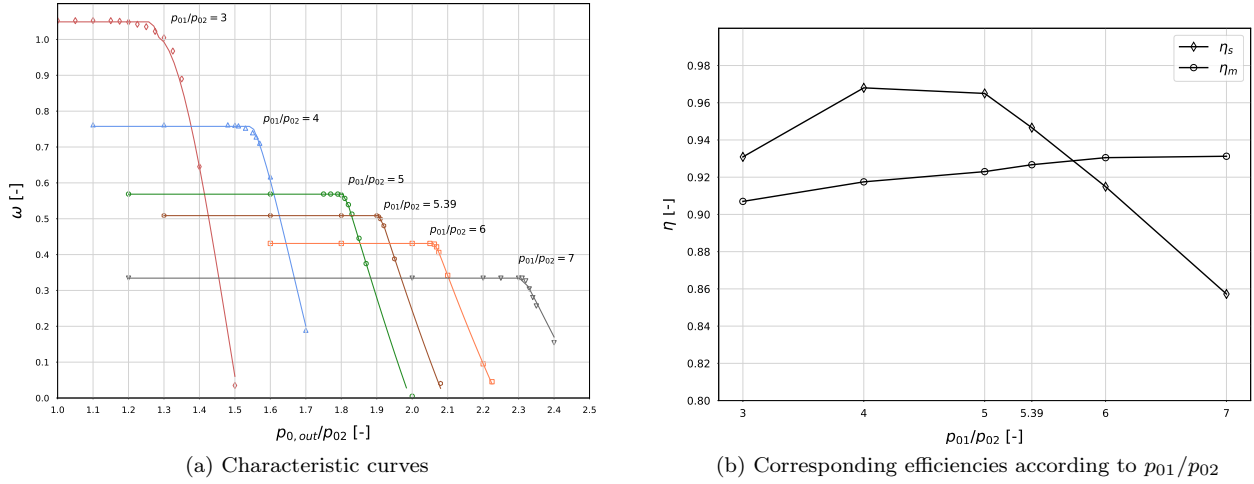


Figure 4.5: Optimal fit of the compound-choking model for each p_{01}/p_{02}

As a conclusion on the accuracy of the compound-choking models, Figure 4.5 shows how well the compound-choking model developed by Bernstein et al. [23] and later improved by Metsue et al. [21] performs at predicting the entrainment ratio of this supersonic ejectors. This contrasts with the fact that the same models struggle to predict the flow locally, e.g. with the value of M_{eq} being clearly overpredicted. This shows that there is an important gap between the prediction of global performance and the actual representation of the local flow phenomena.

However, the reader should bear in mind that this compound-choking model is rather a zero-dimensional model than a 1D model, as it does not really take into account any variation along the axial direction. It is based on very simple assumptions, and for this reason, their performance at predicting the characteristic curves should be saluted.

4.3 Transport tubes as visualization of transport phenomena

In a supersonic ejector, the objective is to transfer momentum from the primary to the secondary stream, using mainly the viscous shear exerted by the former on the latter. This momentum is then "converted" in pressure, which makes ejectors some sort of fluid mechanics-compressors.

In order to evaluate their performance, it is useful to consider the transfer of energy (in the form of kinetic energy and heat) between both streams, as well as the transfer of exergy, which takes into account the second law of thermodynamics and the production of entropy due to irreversible phenomena. Transport tubes allow a qualitative visualization of the transport of these quantities.

4.3.1 Analytical tools

The derivations of this section are based on the work done by Meyers and Meneveau [71], and adapted for compressible flows by Lamberts et al. [69]. All the subsequent developments involve mean-flow quantities.

The concept of transport tubes

One can define fluxes of mean-flow mass, momentum (aligned with either axial or radial direction), kinetic energy and total exergy. They correspond to vector fields, noted in a general fashion $\vec{F}$, i.e. they have an axial and a radial component (in the 3D axisymmetric space, their azimuthal component is zero). For each point in the ejector space, a local surface tangent to this vector field can be defined. By doing so, a so-called transport tube can be constructed, which is simply a two-dimensional surface locally tangent to the considered vector field. An illustration (from [68]) is shown in Figure 4.6.

By construction, a transport tube has no net flux crossing its lateral surfaces: the quantity of interest is transported within the tube, being only affected by the volumetric source and sink terms. In the equivalent 2D space (due to the axisymmetric assumption), a transport tube can be represented by a line, which forms the axisymmetric tube by revolution around the ejector axis. Transport tubes can be thought of as the generalization of the concept of streamtubes, which are the particular case of transport tubes for mass, i.e. locally tangent to the local velocity field.

Based on the local mass, axial momentum, kinetic energy and total exergy fluxes fields, one can construct transport tubes for these quantities. In the case of ejectors, it is of particular interest to construct transport tubes emanating from the primary nozzle tip, which is the circle where the primary and secondary streams first meet. It is then allowed to visualize where the axial momentum, kinetic energy and total exergy contained in the primary stream are transported to.

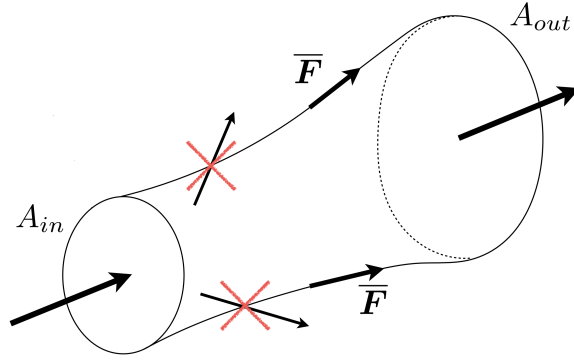


Figure 4.6: Illustration of the concept of an axisymmetric transport tube, from [68]

Transport tubes of momentum and kinetic energy

The transport tube for mass is the classical streamtube, which can easily be obtained using the streamfunction available in the numerical solver, or the velocity vector field. For the other quantities, the expression of the corresponding fluxes (which are vector fields) is required.

The momentum equation of the RANS equations (2.8b) can be rewritten as

$$\nabla \cdot (\bar{\rho} \hat{\mathbf{u}} \otimes \hat{\mathbf{u}} - \bar{\boldsymbol{\tau}} - \bar{\boldsymbol{\tau}}^t) = -\nabla \bar{p} \quad (4.2)$$

To obtain the flux of axial momentum $\bar{\mathbf{F}}_m$, this equation is projected on the axial direction:

$$\nabla \cdot \left(\underbrace{\bar{\rho} \hat{\mathbf{u}} \otimes \hat{\mathbf{u}} - \bar{\boldsymbol{\tau}} \cdot \hat{\mathbf{e}}_x - \bar{\boldsymbol{\tau}}^t \cdot \hat{\mathbf{e}}_x}_{\bar{\mathbf{F}}_m} \right) = -\frac{\partial \bar{p}}{\partial x} \quad (4.3)$$

The axial and radial components of $\bar{\mathbf{F}}_m$ are developed in Appendix A.2.

To determine the kinetic energy flux, the transport equation for the mean-flow kinetic energy $K = \frac{1}{2} |\hat{\mathbf{u}}|^2$ is used. Taking the scalar product of equation (4.2) with the mean-flow velocity vector $\hat{\mathbf{u}}$ yields:

$$(\nabla \cdot (\bar{\rho} \hat{\mathbf{u}} \otimes \hat{\mathbf{u}} - \bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t)) \cdot \hat{\mathbf{u}} = -(\nabla \bar{p}) \cdot \hat{\mathbf{u}} \quad (4.4)$$

$$(\nabla \cdot (\bar{\rho} \hat{\mathbf{u}} \otimes \hat{\mathbf{u}})) \cdot \hat{\mathbf{u}} - \nabla \cdot ((\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}}) + (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) : \nabla \hat{\mathbf{u}} = -(\nabla \bar{p}) \cdot \hat{\mathbf{u}} \quad (4.5)$$

Rearranging the terms, (4.5) becomes

$$\nabla \cdot \left(\underbrace{\frac{1}{2} \bar{\rho} \hat{\mathbf{u}} (\hat{\mathbf{u}} \cdot \hat{\mathbf{u}}) - (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}}}_{\bar{\mathbf{F}}_K} \right) = -(\nabla \bar{p}) \cdot \hat{\mathbf{u}} - (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) : \nabla \hat{\mathbf{u}} \quad (4.6)$$

The axial and radial components of $\bar{\mathbf{F}}_K$ are developed in Appendix A.2.

The concept of total exergy

Specific exergy is an intensive thermodynamic quantity, defined as the 'useful' energy content of a system, or its ability to do or receive work [90] with respect to a reference state. As for enthalpy and internal energy, the concept of total exergy is rather used, allowing to get rid of the necessary knowledge of the fluid celerity to evaluate its exergy content. By doing so, the considered quantity will be insensitive to local fluid acceleration or deceleration.

Total exergy is defined as

$$\xi_t = (h_t - h_{\text{ref}}) - T_{\text{ref}}(s - s_{\text{ref}}) \quad (4.7)$$

Where s is the specific entropy. In the framework of this study, air is considered as a perfect gas. The expression of the total exergy is then

$$\xi_t = c_p(T_t - T_{\text{ref}}) - T_{\text{ref}} \left(c_p \ln \left(\frac{T}{T_{\text{ref}}} \right) - R \ln \left(\frac{p}{p_{\text{ref}}} \right) \right) \quad (4.8)$$

In equations (4.7) and (4.8), the subscript 'ref' refers to a reference thermodynamic state, from which it is considered that no useful work can be extracted. In this case, the reference state is the reservoir state of the secondary stream, as it is considered as only receiving exergy from the primary stream. Hence, $T_{\text{ref}} = T_{02} = 300 \text{ [K]}$ and $p_{\text{ref}} = p_{02} = 1 \text{ [bar]}$. Consequently, the total exergy at the secondary inlet is $\xi_{02} = 0 \text{ [J/kg]}$. As a direct consequence of the second law of thermodynamics, exergy is always destroyed, at least partially, since it can be linked with the entropy production due to irreversible phenomena. Supersonic ejectors being non-ideal engineering devices, one is interested in knowing how well this technology performs at transferring exergy from one stream to another while minimizing these losses. Hence, it is useful to conduct a quantitative exergy analysis in order to evaluate the ejector performance.

Total exergy transport tubes

Mean-flow total exergy is defined from equation (4.8):

$$\tilde{\xi}_t = c_p(\hat{T}_t - T_{\text{ref}}) - T_{\text{ref}} \left(c_p \ln \left(\frac{\hat{T}}{T_{\text{ref}}} \right) - R \ln \left(\frac{\bar{p}}{p_{\text{ref}}} \right) \right) \quad (4.9)$$

The tilde over $\tilde{\xi}_t$ stands for the fact that the total exergy is not an averaged flow quantity (since it is not available in the solver), but is rather computed using averaged quantities. By neglecting the contribution of the turbulence kinetic energy k , Lamberts et al. [69] derived a transport equation for $\tilde{\xi}_t$. This was done by considering the transport equations for the mean-flow total enthalpy and mean flow entropy $\tilde{s}$:

$$\nabla \cdot \left(\underbrace{\bar{\rho} \tilde{\mathbf{u}} \tilde{\xi}_t + (\bar{\mathbf{q}} + \bar{\mathbf{q}}^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right)}_{\bar{\mathbf{F}}_\xi} - (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}} \right) = -T_{\text{ref}} \left(\frac{\Phi}{\hat{T}} + \frac{\Phi_\Theta}{\hat{T}^2} \right) \quad (4.10)$$

With Φ and Φ_Θ being sink terms related to volumetric exergy losses (i.e., entropy generation) due to viscous dissipation and heat transfer. Their definition is the following:

$$\begin{aligned} \Phi &= (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}_d^t) : \nabla \hat{\mathbf{u}} \\ \Phi_\Theta &= -(\bar{\mathbf{q}} + \bar{\mathbf{q}}^t) \cdot \nabla \hat{T} \end{aligned} \quad (4.11)$$

With $\bar{\boldsymbol{\tau}}_d^t$ being the deviatoric part of $\bar{\boldsymbol{\tau}}^t$:

$$\bar{\boldsymbol{\tau}}_d^t = \bar{\boldsymbol{\tau}}^t - \frac{1}{3} \text{tr}(\bar{\boldsymbol{\tau}}^t) \mathbf{I} = \bar{\boldsymbol{\tau}}^t + \frac{2}{3} \bar{\rho} k \mathbf{I} \quad (4.12)$$

The axial and radial components of $\bar{\mathbf{F}}_\xi$ are developed in Appendix A.2. The expression of Φ in axisymmetric coordinates is given in Appendix A.3.

Numerical computation of transport tubes

In an axisymmetric framework, transport tubes can be characterized by curves in a 2D space. Their numerical integration is performed with the fourth order Runge-Kutta method, based on the local fluxes vector fields exported from the flow solver solution. The integration is parametric. Ideally, the integrated transport tube would emanate from the primary nozzle exit circle, but due to viscous effects and possible flow separation at this location, the initial circle has a radius slightly higher than the primary nozzle tip radius: $r_{\text{init}} = R_1 + \epsilon$ where ϵ is small.

As a reminder, the transport tube for mass is the dividing streamtube, also referred as dividing streamline in the axisymmetric framework.

4.3.2 Numerical results

In this section, the transport tubes are shown for $p_{01}/p_{02} = 4$ and $p_{01}/p_{02} = 6$. Four operations conditions are considered, corresponding to cases I, II, IV, VI and I, III, IV, VI, respectively. To properly introduce the visualization tools used, an example (corresponding to $p_{01}/p_{02} = 3$ and $p_{0,\text{out}}/p_{02} = 1.05$) of transport tube figure is shown in Figure 4.7. In the upper half of the ejector section, the momentum, kinetic energy and total exergy transport lines are displayed, and some meaningful streamlines (a large recirculation zone in the diffuser can be observed) within the secondary stream streamlines are shown in the lower half. The dividing streamline is also shown.

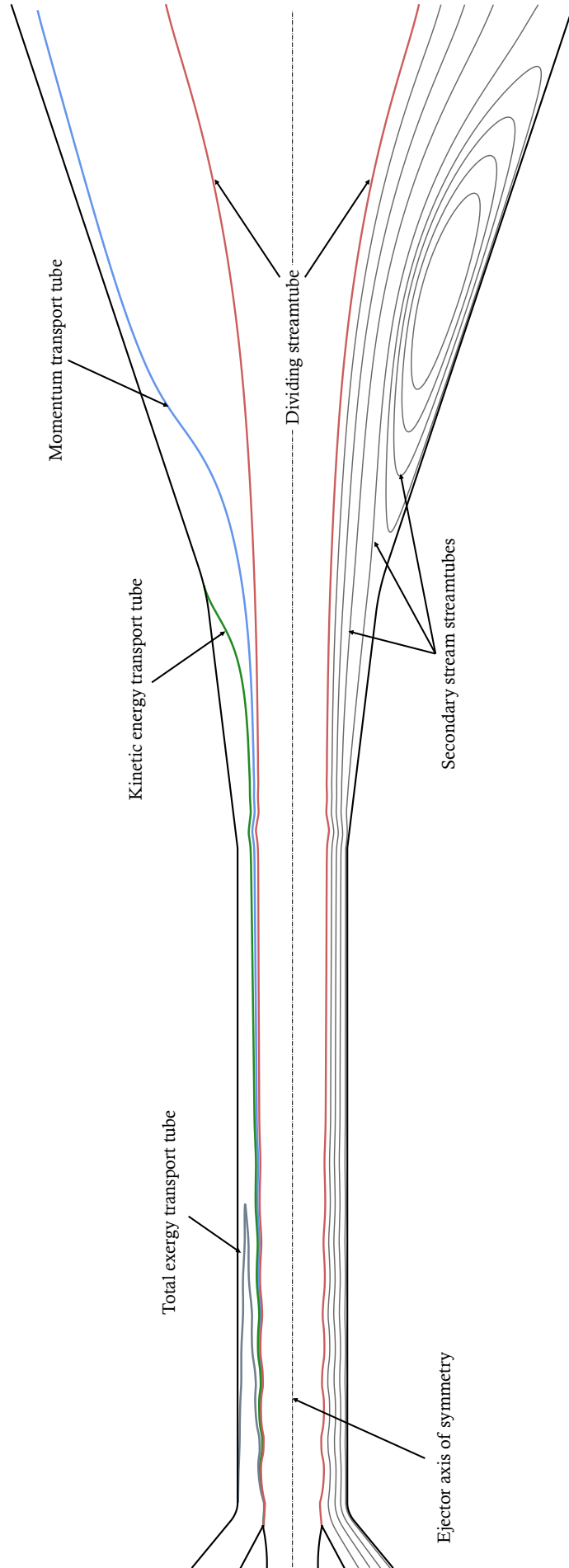


Figure 4.7: Example of cross-section of the ejector displaying the dividing streamtube, the mean-flow transport tubes of momentum, kinetic energy and exergy, as well as streamlines in the secondary stream. The case shown corresponds to $p_{01}/p_{02} = 3$, $p_{0,out}/p_{02} = 1.05$ ($VS \times 2$)

Results for $p_{01}/p_{02} = 4$

The four operating conditions considered are illustrated in Figure 4.8. As $p_{0,\text{out}}$ increases, the space occupied by the primary stream increases. Case I leads to flow separation in the diffuser, giving rise to a recirculation bubble. No major flow separation occurs for case II and IV. For case VI, which is near the breakdown pressure, a recirculation zone appears in the mixing duct, right after both streams enter in contact, and another one takes place in the diffuser.

As seen in equation (4.3), momentum transport tubes depend on the pressure gradient, which acts as a sink term (the same principle is valid for the kinetic energy transport tubes, see equation (4.6)). Furthermore, momentum is absorbed at the ejector walls, in the form of wall shear stress. This is not the case for the kinetic energy transport tube where no net flux exists at the wall.

It can be observed in Figure 4.8 that as $p_{0,\text{out}}$ increases, the momentum and kinetic energy transport tubes depart the dividing streamtube sooner. The same observation holds for the locus of intersection of the tubes with the ejector walls (although the momentum transport tube does not effectively intersect the wall and seems to continue its course in the secondary stream, which can be assumed to happen because of the presence of small recirculation bubbles not displayed in the figure). This has the following interpretation: downstream of this location, all the momentum (and kinetic energy, respectively) of the flow originates from the primary stream, meaning that the momentum and kinetic energy initially contained in the secondary flow have either been converted into static pressure or evacuated from the ejector at the wall.

Concerning the mean-flow total exergy transport tubes, the behavior is slightly more difficult to interpret. They have the advantage of being independent of the pressure gradient when compared to the transport tubes of momentum and kinetic energy, the only sink term in equation (4.10) being due to viscous dissipation and heat transfer. It can be seen for all four cases that the exergy transport tubes rapidly depart the dividing streamtube and even perform an about-turn, indicating a dominance of the turbulent viscous terms over the advection terms. The higher the back-pressure, the sooner this phenomenon happens.

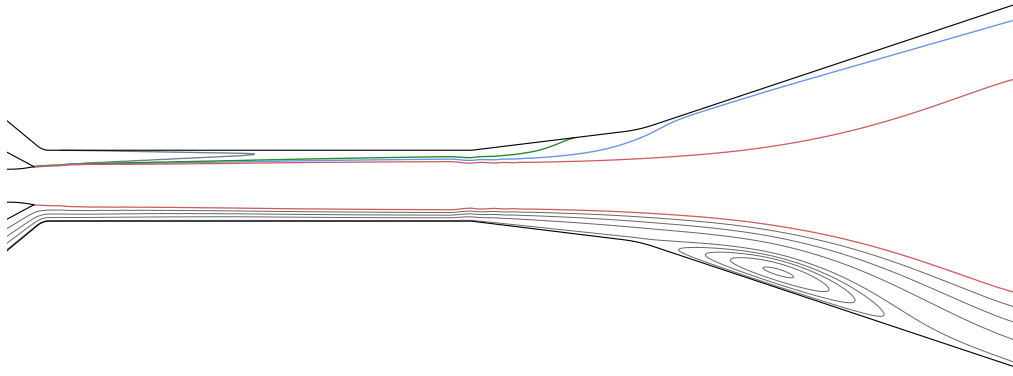
Results for $p_{01}/p_{02} = 6$

Results are illustrated in Figure 4.9. The observations concerning the secondary flow streamlines are similar to the case $p_{01}/p_{02} = 4$, with a large recirculation bubble in the second part of the diffuser due to flow separation for the case of low back-pressure (case IV). This separation is due to the higher flow velocity in the diffuser, which is itself due to the lesser compression to be performed. Note that for case I, the strong oblique shock and its subsequent shock train at the inlet of the diffuser is clearly visible on both streamlines and transport tubes.

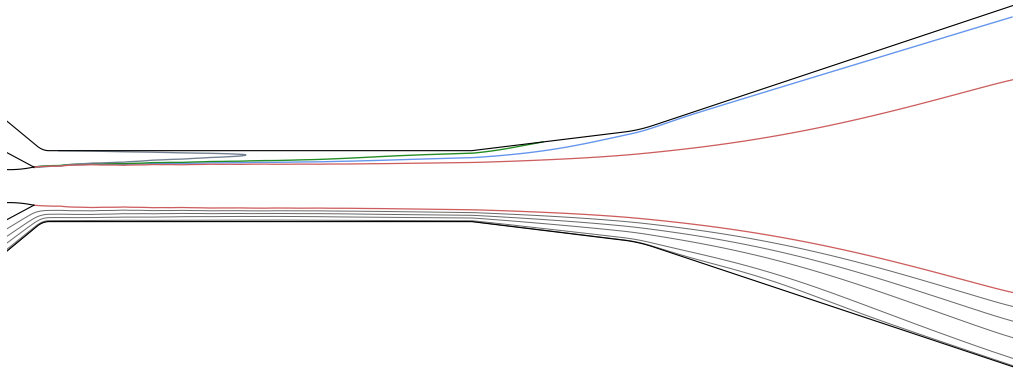
Case VI is very close to the breakdown pressure, where $\dot{m}_2$ is close to 0. The dividing streamtube almost sticks to the ejector wall, leaving almost no room for the secondary flow. Once again, two recirculation zones result from these operating conditions.

It is clear that the momentum and kinetic energy transport tubes departure from the dividing streamline occurs sooner as $p_{0,\text{out}}$ increases, as well as the location of their intersection with the wall. In case IV, due to the recirculation zone in the mixing duct, the numerical integrator was not able to perform the computation of the tubes.

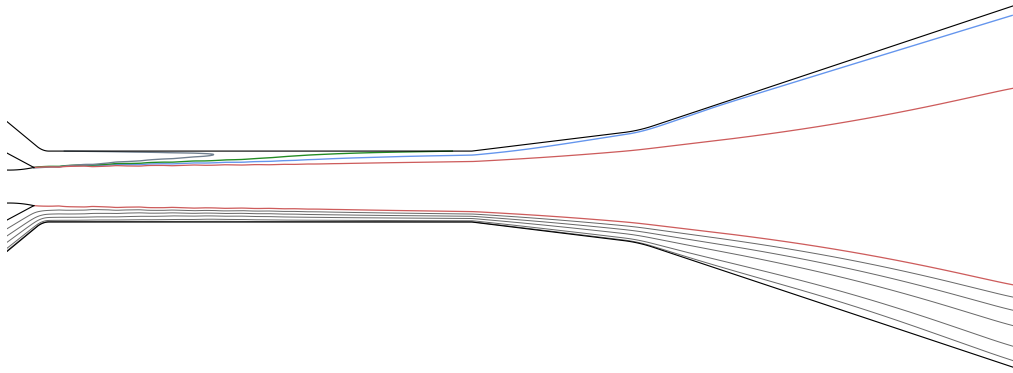
Concerning the exergy transport tube, an about turn takes place for cases I and III, while it adopts a more classical behavior for case IV.



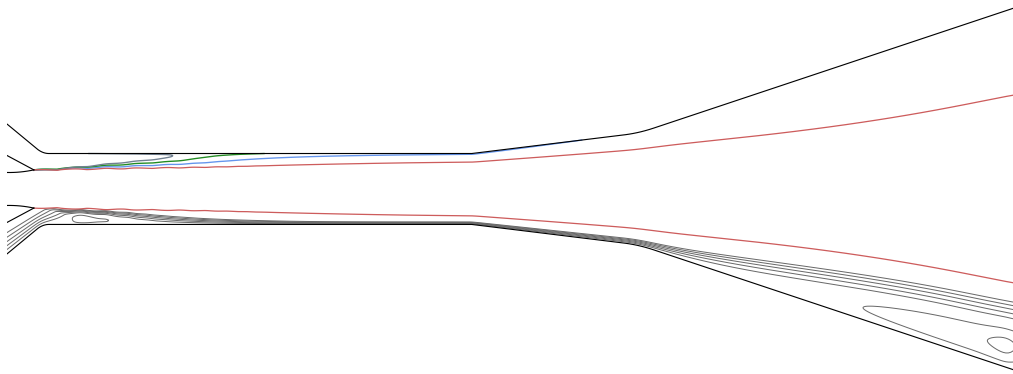
(a) Case I: $p_{0,out}/p_{02} = 1.3$, on-design



(b) Case II: $p_{0,out}/p_{02} = 1.48$, on-design

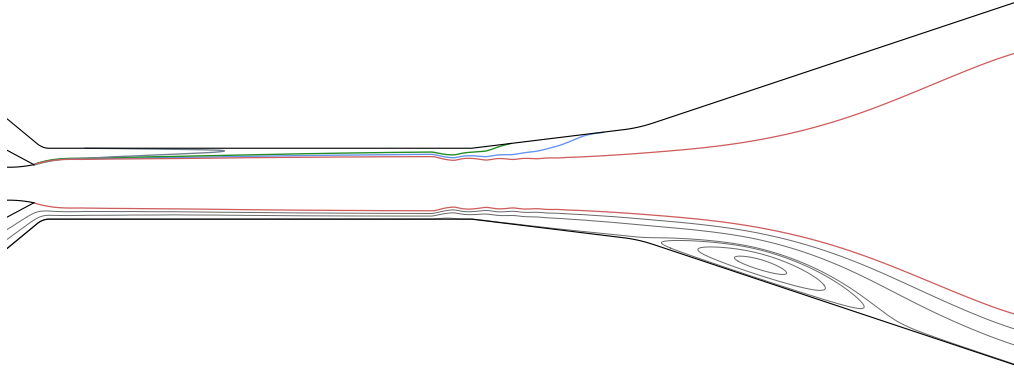


(c) Case IV: $p_{0,out}/p_{02} = 1.56$, off-design

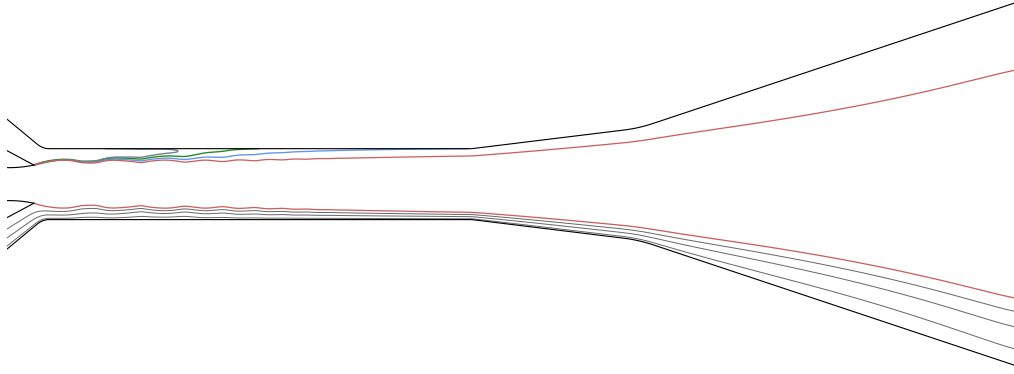


(d) Case VI: $p_{0,out}/p_{02} = 1.7$, near-breakdown

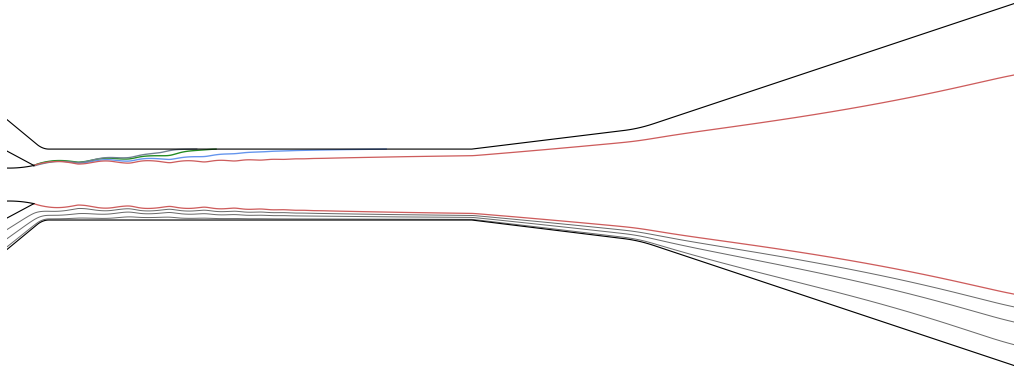
Figure 4.8: Visualization of transport tubes and streamtubes for various values of the back-pressure, $p_{01}/p_{02} = 4$, (VS $\times 2$)



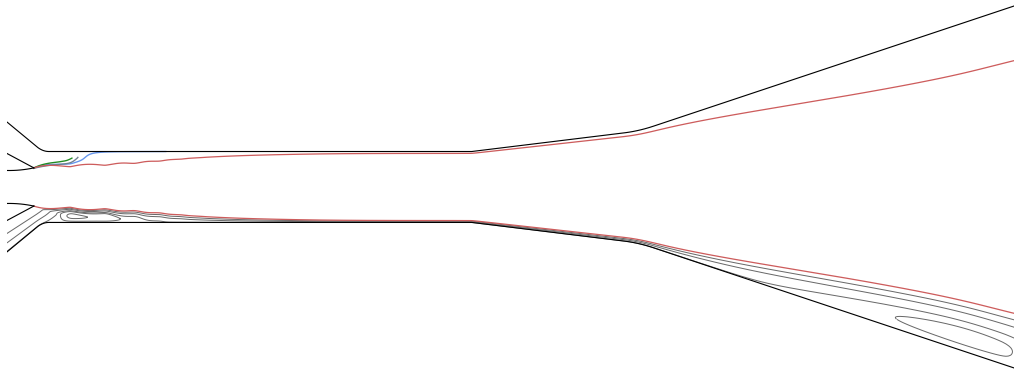
(a) Case I: $p_{0,\text{out}}/p_{02} = 1.6$, on-design



(b) Case III: $p_{0,\text{out}}/p_{02} = 2.0625$, critical point



(c) Case IV: $p_{0,\text{out}}/p_{02} = 2.075$, off-design



(d) Case VI: $p_{0,\text{out}}/p_{02} = 2.2$, near-breakdown

Figure 4.9: Visualization of transport tubes and streamtubes for various values of the back-pressure, $p_{01}/p_{02} = 6$, (VS $\times 2$)

4.3.3 Exergy footprint

As proposed by Lamberts [68], it is interesting to compute the so-called exergy footprint of the primary and secondary streams. The concept is illustrated for an example case in Figure 4.10. The exergy footprint of the primary flow is the area of the exergy transport tube (measured at the primary nozzle tip) that intersects the primary streamtube at a specified downstream location (e.g., the diffuser inlet or outlet). The definition is similar for exergy footprints related to the secondary flow.

Numerically, they are computed by integrating backwards an exergy transport tube from the desired endpoint. The footprint is measured as the cross-sectional area of that tube at the primary nozzle tip location. In this section, two definitions of exergy footprint are used (yielding four different footprints, two for the primary and two for the secondary stream): the first is the exergy footprint originating from ejector outlet (shown in Figure 4.10), and the second from the diffuser inlet. The latter is referred to as case A and the former as case B.

In Figure 4.10, it can be seen that the primary and secondary outlet exergy footprints are rather small. A large part of the exergy contained within the primary stream is dissipated. Graphically, this corresponds to the region above the secondary exergy footprint, shown in grey in Figure 4.10, and denoted as exergy losses footprint. It should be noted that the exergy is not constant in each exergy transport tube cross-section due to the integral sink terms related to entropy production.

Exergy footprints for $p_{01}/p_{02} = 4$

The results are shown in Figures 4.11a and 4.11b. The exergy footprint is adimensionalized with the primary nozzle tip area $A_1 = \pi R_1^2$. The primary, secondary and losses exergy footprints (whose areas sum up to A_1) are shown for various back-pressures. Figure 4.11a displays the footprints computed from the mixing duct outlet (case A, hence the variable of interest is r_A^2/R_1^2), and Figure 4.11b the footprints computed from the ejector outlet, hence the variable r_B^2/R_1^2 .

As a result of the mixing processes continuing in the diffuser, the secondary exergy footprint is higher for case B than case A. The primary outlet exergy footprint (case B) increases with the back-pressure, while the secondary outlet exergy footprint is more or less constant (apart from the extreme case $p_{0,\text{out}}/p_{02} = 1.1$ and $p_{0,\text{out}}/p_{02} = 1.7$). This means that as the back-pressure increases, the amount of exergy transported to useful locations (in this case, at the ejector outlet, in the primary and secondary streams), increases as long as the back-pressure is not too close to the breakdown pressure, where the secondary footprint is greatly reduced. Concerning the mixing duct outlet footprints (case A), shown in Figure 4.11a, the losses footprint does not vary much, but the secondary footprint increases near and after the critical back-pressure (in this case $p_{0,\text{out}}/p_{02} = 1.5$ before decreasing near the breakdown point. For values of the back-pressure lower than the critical one, all footprints are approximately constant.

Exergy footprints for $p_{01}/p_{02} = 6$

Results for this case are shown in Figures 4.12a (case A) and 4.12b (case B). Their interpretation is similar to the case $p_{01}/p_{02} = 4$. It is however worth mentioning that for both cases A and B, secondary footprints are smaller than for the case $p_{01}/p_{02} = 4$, which is due to the fact that the secondary mass flow rate is lower for $p_{01}/p_{02} = 6$ than for $p_{01}/p_{02} = 4$.

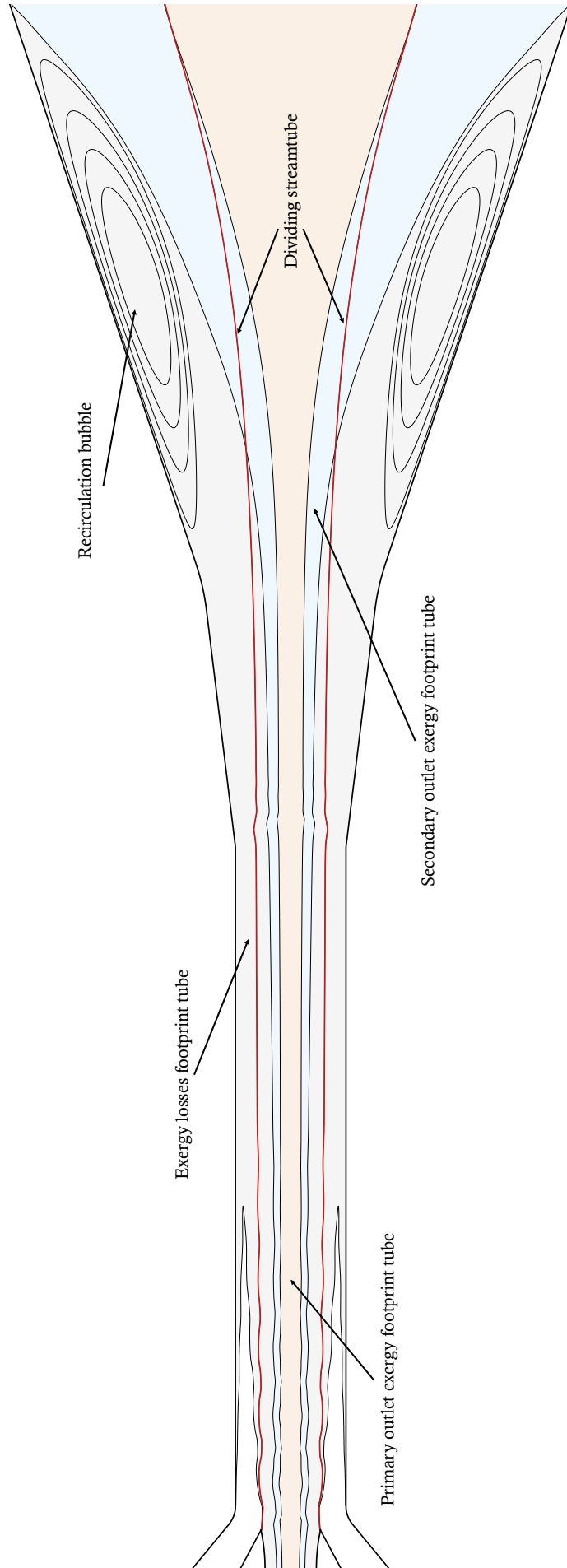


Figure 4.10: Illustration of the concept of exergy footprint. Specifically, the exergy footprint originating from the primary and secondary streams at the ejector outlet are shown, for the case $p_{01}/p_{02} = 3$, $p_{0,\text{out}}/p_{02} = 1$, $VS \times 2$.

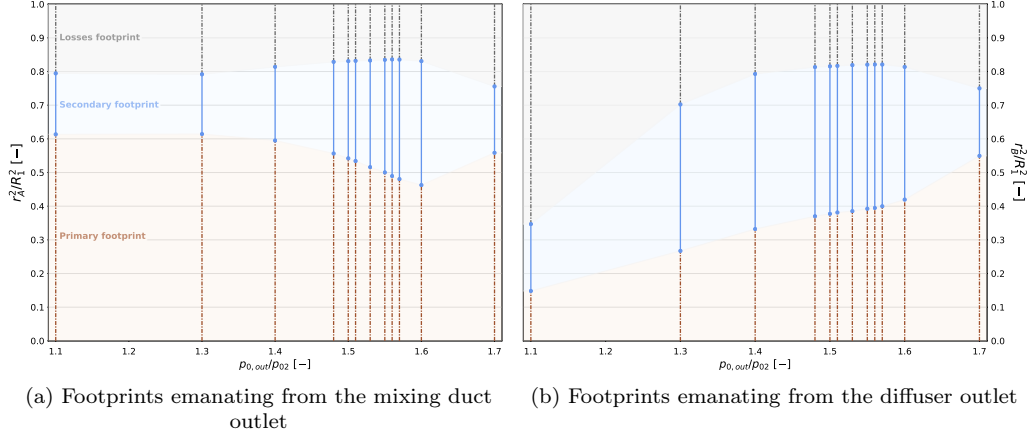


Figure 4.11: Variation of exergy footprints at the primary nozzle exit with the back-pressure, $p_{01}/p_{02} = 4$

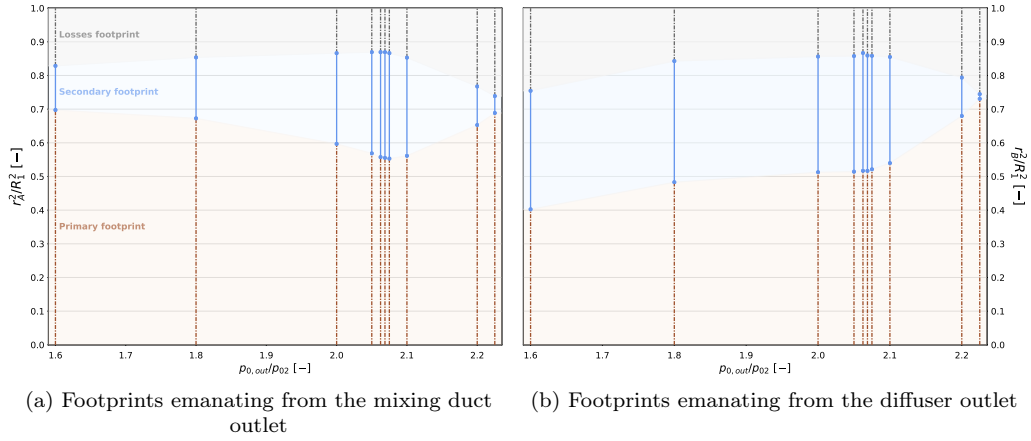


Figure 4.12: Variation of exergy footprints at the primary nozzle exit with the back-pressure, $p_{01}/p_{02} = 6$

4.4 Transfers between the primary and secondary streams

4.4.1 Introduction

Transport tubes allow to qualitatively visualize the transport of momentum, kinetic energy and total exergy. Further insight on the transfer phenomena can be gained by analyzing the transfers between the primary and secondary streams, i.e. through the dividing surface, which essentially occur because of the turbulent shear stresses in the shear layer.

The local transfers of axial momentum, kinetic energy and total exergy are simply the fluxes going out of the control volume whose boundary is defined by the dividing streamtube. The only contribution to the transfer between the primary and secondary streams is due to viscous turbulence effects, the advection terms being null by construction.

4.4.2 Analytical tools

Local transfers through the dividing streamtube

The local flux of a mean-flow vector field $\bar{\mathbf{F}}$ at a given point through the dividing surface is given by

$$\bar{f} = \bar{\mathbf{F}} \cdot \hat{\mathbf{n}} \quad (4.13)$$

With $\hat{\mathbf{n}}$ the normal pointing outwards the streamtube.

Lamberts [68] introduced the following dimensionless local fluxes, respectively for the axial momentum, kinetic energy and total exergy:

$$\bar{f}_m = \frac{A_m}{\dot{m}_1 \sqrt{2\xi_{01}}} \bar{\mathbf{F}}_m \cdot \hat{\mathbf{n}} \quad (4.14)$$

$$\bar{f}_K = \frac{A_m}{\dot{m}_1 \xi_{01}} \bar{\mathbf{F}}_K \cdot \hat{\mathbf{n}} \quad (4.15)$$

$$\bar{f}_\xi = \frac{A_m}{\dot{m}_1 \xi_{01}} \bar{\mathbf{F}}_\xi \cdot \hat{\mathbf{n}} \quad (4.16)$$

With A_m the mixing duct cross-section area.

Lamberts [68] also defined a flux of exergy associated with the mean heat flux and turbulent heat transport:

$$\bar{\mathbf{F}}_q = (\bar{\mathbf{q}} + \bar{\mathbf{q}}^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) \quad (4.17)$$

It should be noted that

$$\bar{\mathbf{F}}_\xi = \bar{\mathbf{F}}_K + \bar{\mathbf{F}}_q \quad (4.18)$$

As for the momentum, kinetic energy and exergy, a dimensionless local flux through the dividing surface is defined for $\bar{\mathbf{F}}_q$:

$$\bar{f}_q = \frac{A_m}{\dot{m}_1 \xi_{01}} \bar{\mathbf{F}}_q \cdot \hat{\mathbf{n}} \quad (4.19)$$

Cumulative transfers

The local transfers obtained above can be integrated along the axial distance to obtain the cumulative transfers of momentum, kinetic energy, exergy and heat exergy content. For a general dimensionless flux through the dividing streamtube $\bar{f}_\alpha$, $\alpha \in \{m, K, \xi, q\}$, the cumulative transfer for the axisymmetric case is given by

$$I_\alpha(x) = \frac{1}{A_m} \int_{x_c}^x \bar{f}_\alpha(\chi, r_s(\chi)) 2\pi r_s(\chi) d\chi \quad (4.20)$$

With $r_s(x)$ the radius of the dividing streamtube at axial location x , and x_c the axial coordinate of the point where both streams initially meet, i.e. the primary nozzle tip.[69]

4.4.3 Local and Cumulative transfers

The local and cumulative transfers of the different fluxes defined previously are depicted in Figures 4.13 to 4.16 for the case $p_{01}/p_{02} = 6$ where the 6 operating conditions are represented. (The other transfers curves for $p_{01}/p_{02} = 4$ are available in Appendix E.2) Considering the global aspect of different fluxes, one can first identify a similar behavior and order of magnitude of the momentum, mean kinetic energy and the total exergy.. The cumulative transfers of every flux are decreasing towards high back-pressure, leading to a less good transfer towards secondary stream.

On all transfer curves, peaks and drops are observed. They are linked to the inherent 2D structure of the flow (e.g. interaction of shockwaves with the shear layer).

As explained in the previous chapter, low back-pressure operating conditions have a part of the flow that is identical due to the flow being either compound-supersonic or entirely supersonic through Fabri-patterns. For those parts of the flow (which mainly correspond to case I), the momentum, kinetic energy and exergy transfers are significantly lower in the mixing duct. The higher transfer for regions upstream the ξ -section of each on-design operating condition can be explained by the presence of the secondary shock-train, which enhances the mixing process by the turbulence it induces.

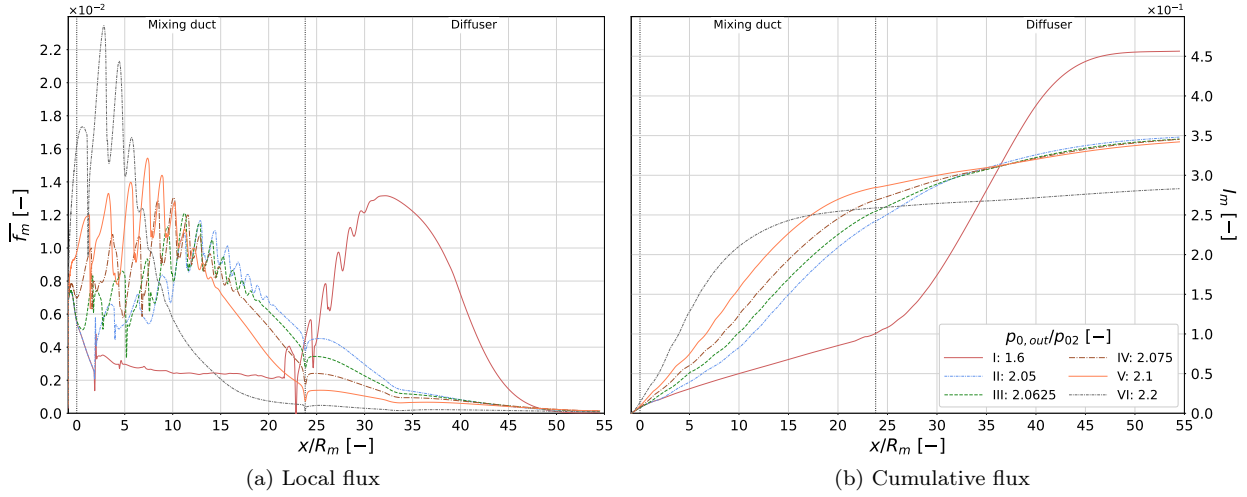


Figure 4.13: Dimensionless momentum flux - $p_{01}/p_{02} = 6$

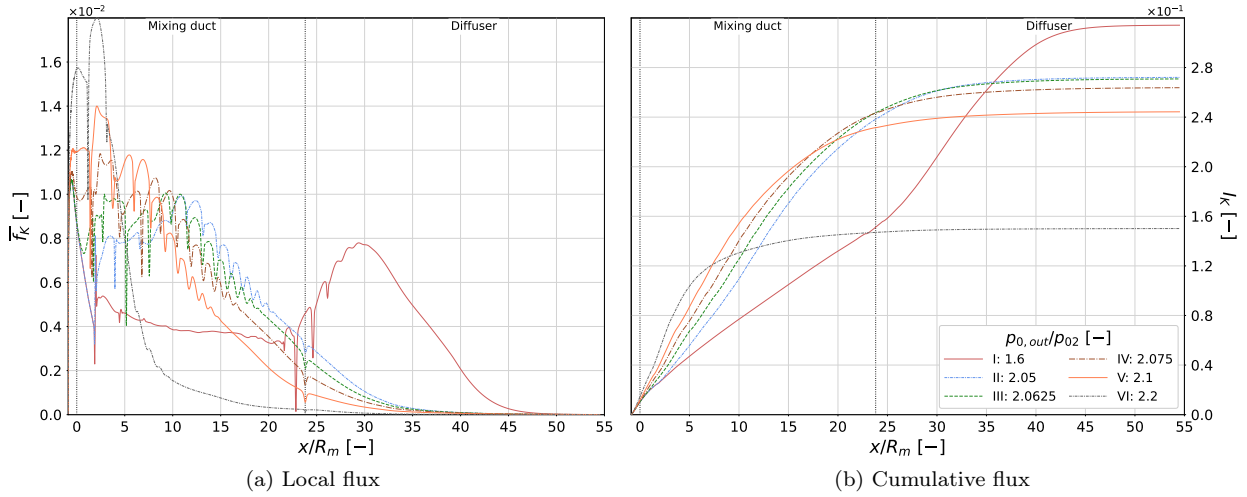


Figure 4.14: Dimensionless mean-flow kinetic energy flux - $p_{01}/p_{02} = 6$

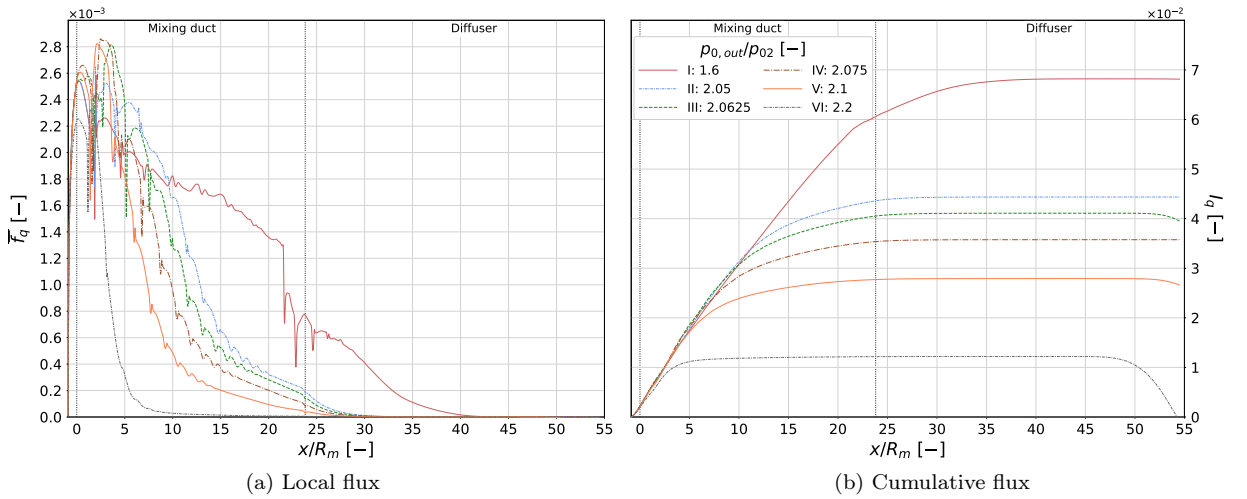


Figure 4.15: Dimensionless mean-flow heat flux (exergy content) - $p_{01}/p_{02} = 6$

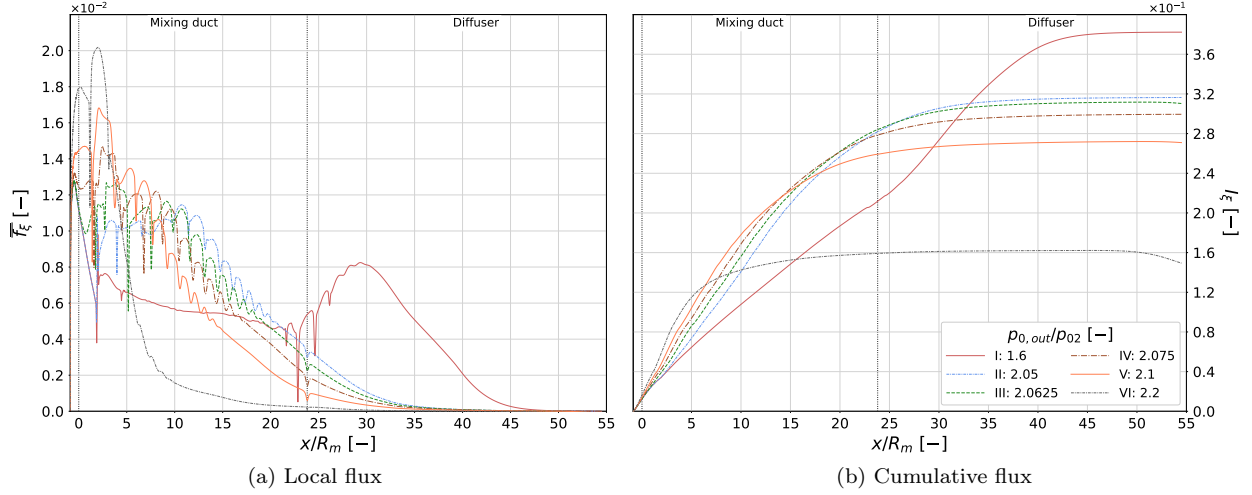


Figure 4.16: Dimensionless mean-flow total exergy flux - $p_{01}/p_{02} = 6$

4.5 Exergetic losses and balance of a supersonic ejector

Besides total exergy transport tubes, transfers through the dividing streamline and exergy footprint, a further quantitative analysis of the exergetic efficiency of a supersonic ejector can be conducted by analyzing the exergy losses in the ejector. This section aims at performing such an analysis. The introduced quantities and the associated methodology are from the work of Lamberts [68].

4.5.1 Exergy losses in the secondary stream

Total exergy is lost in the flow where entropy is produced via irreversible phenomena and heat transfer. The sink terms are governed by Φ and Φ_Θ as seen in the transport equation for total entropy (4.10), the former representing the viscous dissipation and the latter heat transfer. Lamberts [68] introduced the following integral quantities to assess the axial distribution of these sink terms in the secondary stream:

$$\pi_2(x) = \frac{R_m T_0}{\dot{m}_1 \xi_{01}} \iint_{A_2(x)} \frac{\Phi}{\hat{T}} dA \quad (4.21)$$

$$\pi_{\Theta,2}(x) = \frac{R_m T_0}{\dot{m}_1 \xi_{01}} \iint_{A_2(x)} \frac{\Phi_\Theta}{\hat{T}^2} dA \quad (4.22)$$

Where $A_2(x)$ represents the cross-section area of the secondary streamtube at axial location x .

Cumulative quantities are also defined for π_2 and $\pi_{\Theta,2}$, so as to illustrate the total exergy loss that the secondary stream underwent up to axial location x . They are noted $\Pi_2(x)$ and $\Pi_{\Theta,2}(x)$, the former being defined as

$$\Pi_2(x) = \frac{1}{R_m} \int_{x_c}^x \pi_2(\chi) d\chi \quad (4.23)$$

$\Pi_{\Theta,2}(x)$ is defined in an analogous manner.

Figures 4.17 and 4.18 show the axial distribution of these losses for the case $p_{01}/p_{02} = 6$ and for the 6 operating conditions I to VI.

As seen in Figures 4.17a and 4.17b, for the cases II to VI, the main exergy losses due to viscous dissipation occur in the mixing duct. Comparatively, the losses in the diffuser are negligible. However, for case I (i.e., lowest back-pressure), a vast amount of viscous losses occur at the end of the mixing duct and in the diffuser. This is due to the strong

oblique shock that strongly compresses the flow and to the flow separation taking place in the diffuser. Case VI, which is very near to the breakdown point, shows a peak of viscous dissipation at the beginning of the mixing duct.

Except for the first case for which they are much higher, cumulative viscous losses take roughly the same value at the outlet of the ejector. It should be pointed out that locally, the π_2 pattern is closely correlated with that of the transfers presented in section 4.4.3, since both are linked to the turbulent shear stresses $\bar{\tau}^t$.

Concerning the exergy losses in the secondary stream due to heat transfer (Figure 4.18, which uses the same legend as Figure 4.17), they mainly occur in the mixing duct. In Figure 4.18b, it can be seen that the total amount of exergy destroyed due to heat transfer decreases as $p_{0,\text{out}}/p_{02}$ increases. A sudden peak in losses near the ejector outlet is also observed for some cases.

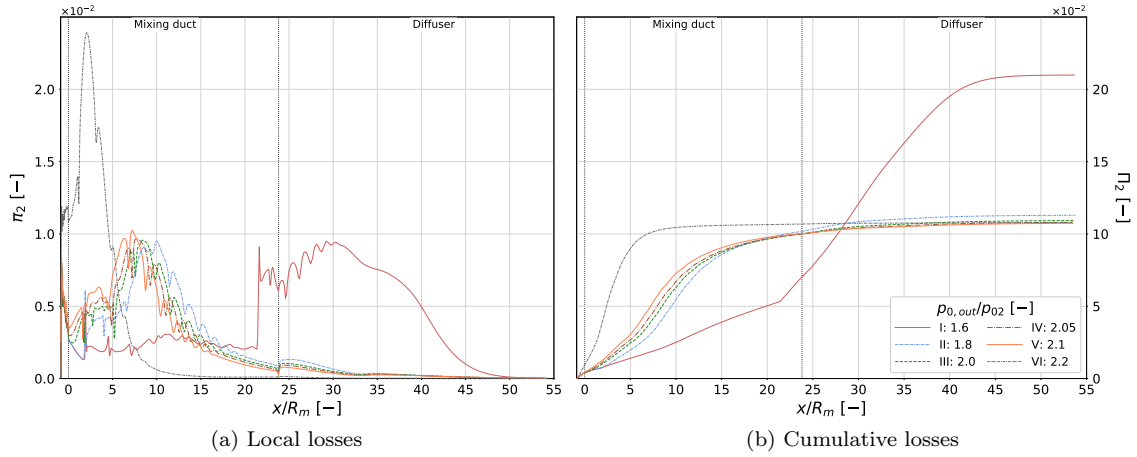


Figure 4.17: Secondary stream exergy losses due to viscous dissipation, $p_{01}/p_{02} = 6$

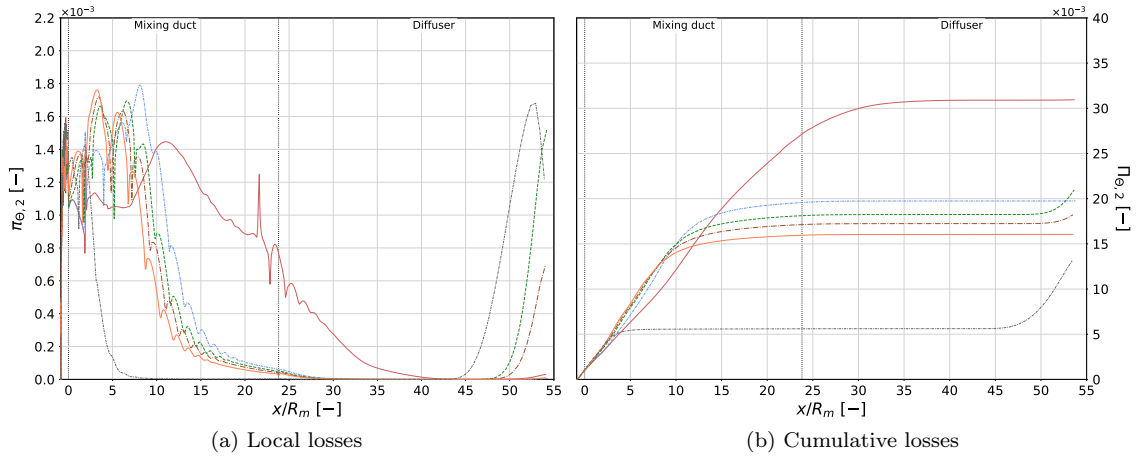


Figure 4.18: Secondary stream exergy losses due to heat transfer, $p_{01}/p_{02} = 6$

To gain more insight on where the exergy is destroyed, i.e. where the irreversible phenomena take place, Figure 4.19 shows the value of the viscous dissipation term field Φ . The top figure represents an on-design case, and the bottom one a near-critical off-design case.

As expected, the viscous dissipation is localized mainly in the shear layers and shock train (which are quite turbulent), though the strong oblique shock for the case $p_{0,\text{out}}/p_{02} = 1.6$ also visibly generates entropy.

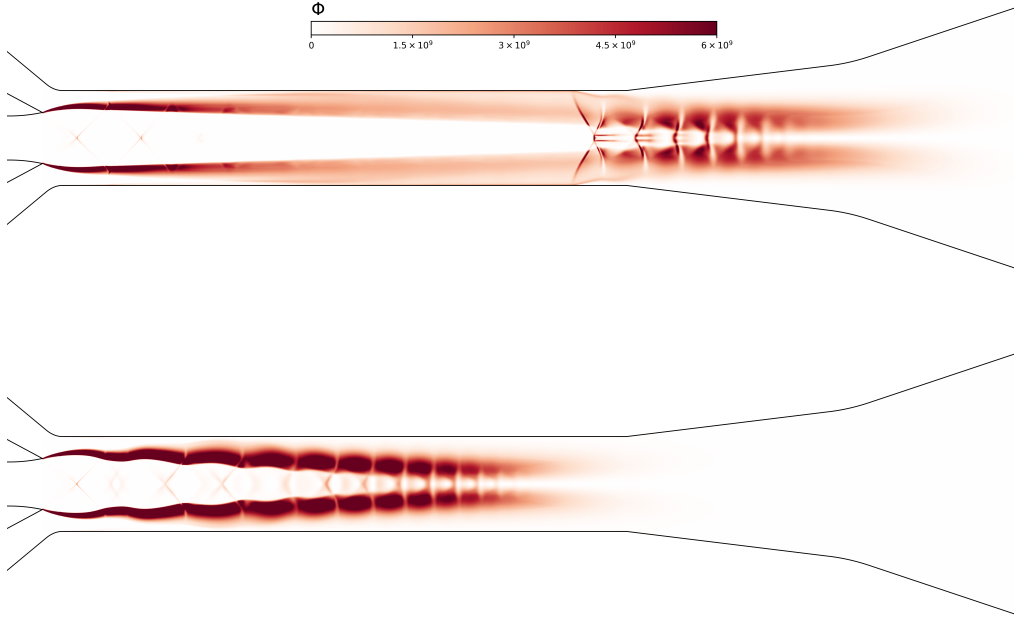


Figure 4.19: Location of viscous dissipation: Field of Φ , $p_{01}/p_{02} = 6$. Top: $p_{0,\text{out}}/p_{02} = 1.6$, bottom: $p_{0,\text{out}}/p_{02} = 2.0625$ (VS×2).

Finally, the net gain in exergy of the secondary stream is analyzed through the dimensionless quantity introduced by Lamberts [68]:

$$\Delta_2(x) = \frac{\dot{m}_2 \xi_{t2}(x)}{\dot{m}_1 \xi_{01}} \quad (4.24)$$

Where

$$\xi_{t2}(x) = \frac{1}{\dot{m}_2} \iint_{A_2(x)} \bar{\rho} \tilde{\xi}_t u \, dA \quad (4.25)$$

At a location x , Δ_2 represents how much the secondary stream has gained exergy until that point. The profile of $\Delta_2(x)$ is shown for $p_{01}/p_{02} = 6$ for operating points I to VI in Figure 4.20.

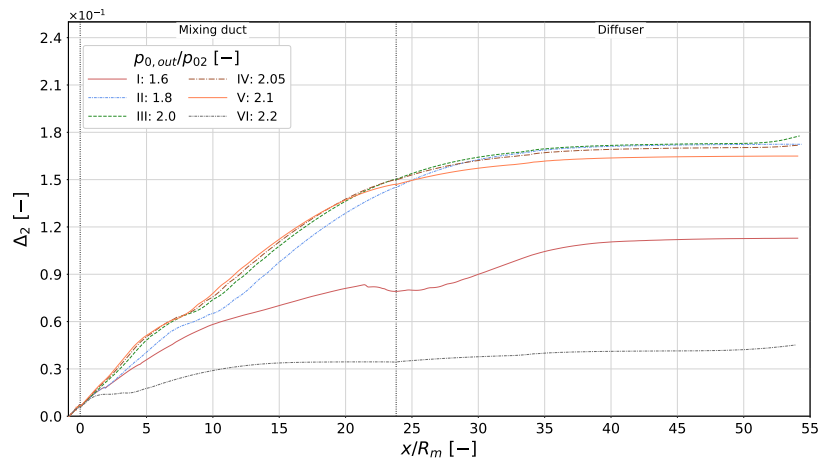


Figure 4.20: Dimensionless gain in exergy of the secondary stream for various operating points, $p_{01}/p_{02} = 6$

For all 6 cases, it can be seen that a majority of exergy is gained by the secondary stream in the mixing duct. The net exergy gain is more or less equal for cases II to V, while case I shows a reduced gain, and case VI even more so.

4.5.2 Exergy efficiency of a supersonic ejector

As a final performance evaluation tool for the exergy efficiency of an ejector, Lamberts introduced [68] two exergy efficiencies. They are defined based on the first and second law of thermodynamics. In terms of ejector quantities, these laws translate to [68]

$$(\dot{m}_1 + \dot{m}_2) h_{0,\text{out}} = \dot{m}_1 h_{01} + \dot{m}_2 h_{02} \quad (4.26)$$

$$(\dot{m}_1 + \dot{m}_2) \xi_{\text{out}} \leq \dot{m}_1 \xi_{01} + \dot{m}_2 \xi_{02} \quad (4.27)$$

With ξ_{out} being the specific total exergy at the outlet of the ejector.

Lamberts [68] defined two hypothetical quantities that would apply for a reversible ejector, i.e. an ejector which does not produce entropy.

The first one is the reversible outlet specific total exergy, obtained from (4.27):

$$\xi_{\text{out}}^{\text{rev}} = \frac{\xi_{01} + \omega \xi_{02}}{1 + \omega} = \frac{\xi_{01}}{1 + \omega} \quad (4.28)$$

Since by definition $\xi_{02} = 0$. From (4.27), one can also obtain an expression for a reversible entrainment ratio:

$$\omega^{\text{rev}} = \frac{\xi_{01} - \xi_{\text{out}}}{\xi_{\text{out}} - \xi_{02}} = \frac{\xi_{01} - \xi_{\text{out}}}{\xi_{\text{out}}} \quad (4.29)$$

Based on the two definitions (4.28) and (4.29), two exergy efficiencies can be defined, depending on whether the ejector is perceived as a flow mixer of a mechanical compressor. The former point of view considers the ejector as a device that mixes two streams of different exergetic content, resulting in a mixed stream with intermediate exergy. The inputs to the ejector are the primary and secondary streams, and the output the mixed stream. This leads to the following definition of exergy efficiency:

$$\eta_m = \frac{(\dot{m}_1 + \dot{m}_2) \xi_{\text{out}}}{\dot{m}_1 \xi_{01} + \dot{m}_2 \xi_{02}} = \frac{\xi_{\text{out}}}{\xi_{\text{out}}^{\text{rev}}} \quad (4.30)$$

The other point of view consists in considering the supersonic as a device compressing the secondary flow using the exergy content of the primary flow, i.e. transferring the exergy from one stream to another. The input is then the primary stream, and the output the compressed secondary stream. This leads to another definition of the exergy efficiency:

$$\eta_c = \frac{\dot{m}_2 (\xi_{\text{out}} - \xi_{02})}{\dot{m}_1 (\xi_{01} - \xi_{02})} = \frac{\omega}{\omega^{\text{rev}}} \quad (4.31)$$

These exergy efficiencies are illustrated for two operating curves, namely $p_{01}/p_{02} = 4$ and $p_{01}/p_{02} = 6$.

A further distinction was added in the form of two points of view *A* and *B* (yielding a total of four efficiencies: $\eta_{m,A}$, $\eta_{m,B}$, $\eta_{c,A}$ and $\eta_{c,B}$). Point of view *A* consists in considering the outlet of the mixing duct as "out" state in equations (4.30) and (4.31), while *B* refers to the efficiencies at the true outlet of the ejector. The *A* point of view allows to assess the efficiency of the single mixing duct without the losses occurring in the diffuser.

For $p_{01}/p_{02} = 4$ (Figure 4.21), the first observation is that the pattern of η_m and η_c (both for points of view *A* and *B*) are very similar, the latter being significantly smaller than the former. This is due to the very definition of η_c , which only considers the exergy contained in the secondary stream as output of the machine whereas η_m considers the whole exergy content as output.

There is a major difference between the *A* and the *B* point of view. The former has a sort of plateau for $\eta_{m,A}$ and $\eta_{c,A}$ similar to that of the entrainment ratio curve. Their values drop when the back-pressure closes to the critical back-pressure, and finally adopt low values for the cases near the break-down point.

On the other hand, the *B* point of view yields increasing values of $\eta_{m,B}$ and $\eta_{c,B}$, then reaching a maximum for a value of $p_{0,\text{out}}/p_{02}$ slightly higher than the critical back-pressure before dropping for further off-design conditions. The reason for this growing of η with $p_{0,\text{out}}$ in the on-design region is the following: at low $p_{0,\text{out}}/p_{02}$, only a small compression work is done on the secondary stream, meaning that the exergy contained in the primary stream is "wasted" as it is not properly used to compress the secondary flow.

The maximum efficiency (following the B point of view) should be achieved when the maximum entrainment and compression is done on the secondary stream. This is not really the case in Figure 4.21, where the maximum efficiency occurs slightly after the critical point, but it is for the case $p_{01}/p_{02} = 6$ (Figure 4.22).

This final result does quite remarkably conclude this study by corroborating the observations regarding the choking phenomenon made in Chapter 3. The case $p_{01}/p_{02} = 4$ lies indeed in the *compound-like* regime, and presents a characteristic curve that is not sharp. Yet, the efficiencies $\eta_{m,B}$ and $\eta_{c,B}$ indicate an optimum for higher $p_{0,out}/p_{02}$, as if they were not sensitive to the smoothing of the corner caused by the progressive loss of choking.

On the other hand, the case $p_{01}/p_{02} = 6$ is in the *Fabri-like* regime, where the characteristic curve has a sharper corner (resulting from the sudden loss of choking inherent to the Fabri-patterns). It does present a maximum of the "B" efficiencies at the critical back-pressure. The observation made in Figure 4.21 thus indicates that the exergy efficiency is independent from the regime (either Fabri or compound).

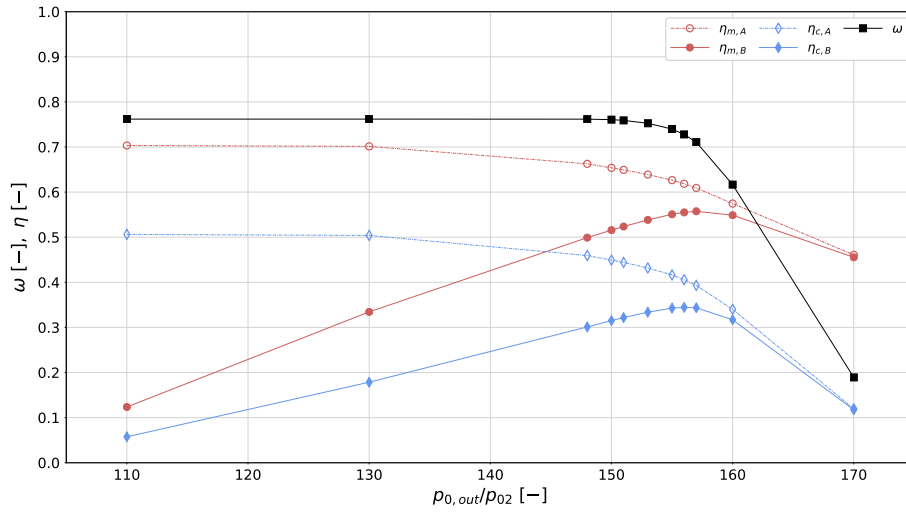


Figure 4.21: Exergy efficiencies measured at the mixing duct outlet and ejector outlet, $p_{01}/p_{02} = 4$.

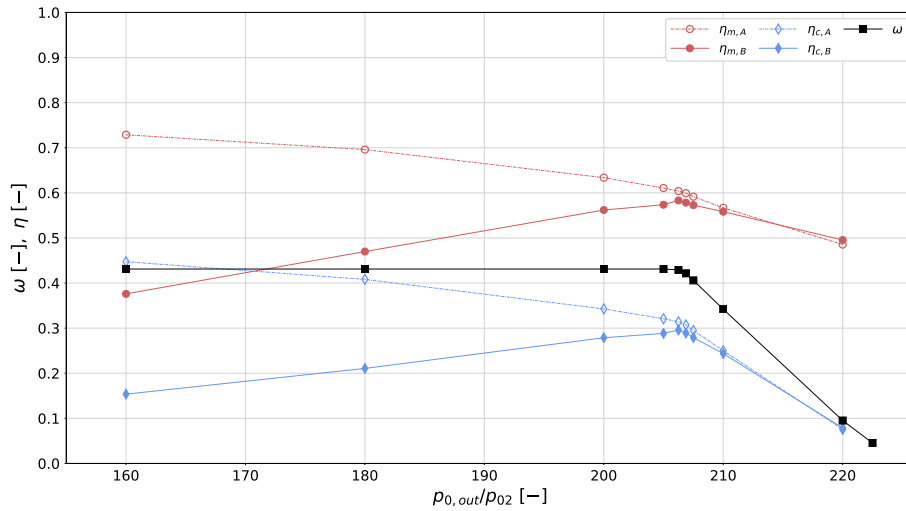


Figure 4.22: Exergy efficiencies measured at the mixing duct outlet and ejector outlet, $p_{01}/p_{02} = 6$.

Chapter 5

Conclusion and perspectives

The limited performance of supersonic ejectors, which results from complex and variable phenomena, drives the development of studies to understand them numerically and experimentally. This was the subject of this Master's thesis. The objectives were to perform a preliminary numerical analysis to define the ejector geometry yielding the best performance, carry out the mechanical design and construction stages in order to finally corroborate the experimental and numerical results and perform exhaustive numerical investigations to improve the understanding of ejector phenomena.

Due to experimental difficulties beyond the author's control, the final assembly of the ejector could not be finished, and the collection of experimental data was not possible. However, although the experimental objective was not completely achieved, the complete design was. All the required components were ordered, but delivery delays made final assembly impossible before handing in this manuscript. It is therefore reasonable to conclude that the design work is completed, leaving only the purely mechanical assembly and the calibration of the sensors to be carried out before the experimental data can be gathered.

In order to conduct the numerical investigation, a high-quality mesh was used to perform 2D steady-state RANS simulations using the commercial software Fluent. A versatile post-processing package was also developed, enabling the raw flow variables to be extracted and used to carry out advanced analyses at all levels. Some of the tools used were first introduced by Lamberts [68], and numerous others were first developed in this work.

This numerical setup allowed to engender numerical data covering a wide range of operating conditions. An in-depth examination of these cases supported by a quantitative analysis led to a better understanding and broader definition of the choking phenomenon. It confirmed the compound-choking as the general explanation of limitation of the secondary flow rate, though some additional nuances concerning the transition from the off-design to the on-design regime were brought forth, supported by a back-of-the-envelope speculative acoustic argument. Numerical evidence also showed that this compound explanation could be overridden, in specific conditions, by the so-called Fabri pattern. The onset conditions for the appearance of Fabri pattern were identified and proven to be the result of several highly interacting parameters, through a quantitative study not yet performed in the literature.

Various one-dimensional models were considered in terms of their overall performance and local prediction of phenomena. The compound-choking model developed by Bernstein et al. [23] was validated but an improvement proposal was made, yielding results that performed better locally.

Finally, the transport and transfers phenomena were investigated using the various tools developed mainly by Lamberts et al. [24]. This study provided a performance analysis through notably the visualization of transport tubes and the computation of transfers of momentum and energy through the dividing streamline. An exhaustive exergy analysis concluded the characterization of the ejector.

Limitations and perspectives

The authors would like to conclude this manuscript by presenting the limitations encountered through this work, and a set of improvements and other research studies to continue to gain knowledge on ejector phenomena.

- The first perspective, aborted in this Master's thesis due to previously mentioned reasons, would be to corroborate the CFD results with experimental results using the new ejector design, for various configurations. The information brought by wall pressure profiles can first and most of all serve the role of (in)validating the numerical setup. It can then additionally provide an interesting insight on the various features presented in this manuscript, such as the *choking section* or the onset of Fabri-patterns in an axisymmetric geometry. Lamberts [68] observed an important discrepancy of the pressure profiles between 2D CFD and experimental results for a rectangular geometry, reduced when performing 3D computations. It is of great hope to obtain a better matching between the 2D axisymmetric CFD results and the axisymmetric experimental results.
- Other experimental work can be done easily thanks to the flexibility of the new design, and this without too much cost involved. Important geometrical parameters such as the NXP and the mixing duct length l_m can be modified and their effect studied.
- In the continuity of the work on Fabri patterns, a more precise quantification of the parameters responsible of their onset could be of great interest, e.g. by using a refined mesh at the locations of shockwave-boundary layer interaction.
- Although the detailed analysis on the choking phenomenon was only performed based on the interpretation of results, a theoretical knowledge on the propagation of information in 2D viscous flows is sorely needed to understand properly the phenomena at stake. For instance, an acoustic simulation would be of great interest to differentiate the behavior of acoustic waves for the two regimes defined (*Compound-like* and *Fabri-like*). The lack of expertise of the authors in this specific field is therefore a limitation of this work, and the conclusions drawn are only intended to serve as mere speculation.
- Finally, the very numerical model at the origin of the results presented in this manuscript is of substantial simplicity when compared to the actual flow structures taking place in supersonic ejectors. Mixing phenomena are paramount and are the result of the turbulence taking place in the mixing duct. By definition, a 2D steady-state RANS study can only provide limited insight and accuracy on the study of turbulent flow features. Furthermore, numerous studies have shown the great sensitivity of numerical simulations to the chosen turbulence closure schemes. Consequently, 3D unsteady simulations are envisioned to shed light on the vortical structures at hand and their impact on the mixing and choking phenomena. In a trade-off between computational cost and fidelity, LES simulations promise to deliver valuable information that pass under the radar of simplified numerical methods such as the one used in the present work. This is the subject of the ongoing Ph.D. thesis of Romain Debroeyer.

Appendix A

Miscellaneous analytical expressions

A.1 Expression of the viscous stress tensor terms

Under the Stokes hypothesis, the time-averaged viscous stress tensor is given by

$$\bar{\boldsymbol{\tau}} = \mu \left(\nabla \hat{\mathbf{u}} + (\nabla \hat{\mathbf{u}})^T - \frac{2}{3} (\nabla \cdot \hat{\mathbf{u}}) \mathbf{I} \right) \quad (\text{A.1})$$

The mean-flow velocity gradient is given by

$$\nabla \hat{\mathbf{u}} = \begin{pmatrix} \frac{\partial \hat{u}}{\partial x} & \frac{\partial \hat{v}}{\partial x} & 0 \\ \frac{\partial \hat{u}}{\partial r} & \frac{\partial \hat{v}}{\partial r} & 0 \\ 0 & 0 & \frac{\hat{v}}{r} \end{pmatrix} \quad (\text{A.2})$$

The divergence of the velocity thus has the expression

$$\nabla \cdot \hat{\mathbf{u}} = \frac{\partial \hat{u}}{\partial x} + \frac{\partial \hat{v}}{\partial r} + \frac{\hat{v}}{r} \quad (\text{A.3})$$

The axial and radial components of $\bar{\boldsymbol{\tau}}$ are then

$$\begin{aligned} \bar{\tau}_{xx} &= \mu \left(\frac{4}{3} \frac{\partial \hat{u}}{\partial x} - \frac{2}{3} \left(\frac{\partial \hat{v}}{\partial r} + \frac{\hat{v}}{r} \right) \right) \\ \bar{\tau}_{xr} &= \bar{\tau}_{rx} = \mu \left(\frac{\partial \hat{u}}{\partial r} + \frac{\partial \hat{v}}{\partial x} \right) \\ \bar{\tau}_{rr} &= \mu \left(\frac{4}{3} \frac{\partial \hat{v}}{\partial r} - \frac{2}{3} \left(\frac{\partial \hat{u}}{\partial x} + \frac{\hat{v}}{r} \right) \right) \end{aligned} \quad (\text{A.4})$$

The Reynolds tensor reads

$$\bar{\boldsymbol{\tau}}^t = \mu_t \left(\nabla \hat{\mathbf{u}} + (\nabla \hat{\mathbf{u}})^T - \frac{2}{3} (\nabla \cdot \hat{\mathbf{u}}) \mathbf{I} \right) - \frac{2}{3} \bar{\rho} k \mathbf{I} \quad (\text{A.5})$$

Its axial and radial components are then

$$\begin{aligned} \bar{\tau}_{xx}^t &= \mu_t \left(\frac{4}{3} \frac{\partial \hat{u}}{\partial x} - \frac{2}{3} \left(\frac{\partial \hat{v}}{\partial r} + \frac{\hat{v}}{r} \right) \right) - \frac{2}{3} \bar{\rho} k \\ \bar{\tau}_{xr}^t &= \bar{\tau}_{rx}^t = \mu_t \left(\frac{\partial \hat{u}}{\partial r} + \frac{\partial \hat{v}}{\partial x} \right) \\ \bar{\tau}_{rr}^t &= \mu_t \left(\frac{4}{3} \frac{\partial \hat{v}}{\partial r} - \frac{2}{3} \left(\frac{\partial \hat{u}}{\partial x} + \frac{\hat{v}}{r} \right) \right) - \frac{2}{3} \bar{\rho} k \end{aligned} \quad (\text{A.6})$$

A.2 Expression of the fluxes

The axial momentum flux is defined by

$$\bar{\mathbf{F}}_m = \bar{\rho} \hat{\mathbf{u}} \hat{\mathbf{u}} - \bar{\boldsymbol{\tau}} \cdot \hat{\mathbf{e}}_x - \bar{\boldsymbol{\tau}}^t \cdot \hat{\mathbf{e}}_x \quad (\text{A.7})$$

The axial and radial components of the axial momentum flux are then

$$\begin{aligned} \bar{F}_{m,x} &= \bar{\rho} \hat{u}^2 - \bar{\tau}_{xx} - \bar{\tau}_{xx}^t \\ \bar{F}_{m,r} &= \bar{\rho} \hat{u} \hat{v} - \bar{\tau}_{xr} - \bar{\tau}_{xr}^t \end{aligned} \quad (\text{A.8})$$

The kinetic energy flux is defined by

$$\bar{\mathbf{F}}_K = \frac{1}{2} \bar{\rho} \hat{\mathbf{u}} (\hat{\mathbf{u}} \cdot \hat{\mathbf{u}}) - (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}} \quad (\text{A.9})$$

The axial and radial components of the mean-flow kinetic energy flux are respectively given by

$$\begin{aligned} \bar{F}_{K,x} &= K \hat{u} - \hat{u} (\bar{\tau}_{xx} + \bar{\tau}_{xx}^t) - \hat{v} (\bar{\tau}_{xr} + \bar{\tau}_{xr}^t) \\ \bar{F}_{K,r} &= K \hat{v} - \hat{u} (\bar{\tau}_{xr} + \bar{\tau}_{xr}^t) - \hat{v} (\bar{\tau}_{rr} + \bar{\tau}_{rr}^t) \end{aligned} \quad (\text{A.10})$$

The mean-flow total exergy flux is defined by

$$\bar{\mathbf{F}}_\xi = \bar{\rho} \hat{\mathbf{u}} \tilde{\xi}_t + (\bar{\mathbf{q}} + \bar{\mathbf{q}}^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) - (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}^t) \cdot \hat{\mathbf{u}} \quad (\text{A.11})$$

The axial and radial components of the mean-flow total exergy flux are

$$\bar{F}_{\xi,x} = \bar{\rho} \hat{u} \tilde{\xi}_t + (\bar{q}_x + \bar{q}_x^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) - \hat{u} (\bar{\tau}_{xx} + \bar{\tau}_{xx}^t) - \hat{v} (\bar{\tau}_{xr} + \bar{\tau}_{xr}^t) \quad (\text{A.12})$$

$$\bar{F}_{\xi,r} = \bar{\rho} \hat{v} \tilde{\xi}_t + (\bar{q}_r + \bar{q}_r^t) \left(1 - \frac{T_{\text{ref}}}{\hat{T}} \right) - \hat{u} (\bar{\tau}_{xr} + \bar{\tau}_{xr}^t) - \hat{v} (\bar{\tau}_{rr} + \bar{\tau}_{rr}^t) \quad (\text{A.13})$$

A.3 Expression of the viscous dissipation term

The expression of the term Φ is developed herein. Φ was defined in (4.11) as

$$\Phi = (\bar{\boldsymbol{\tau}} + \bar{\boldsymbol{\tau}}_d^t) : \nabla \hat{\mathbf{u}} \quad (\text{A.14})$$

Where $\bar{\boldsymbol{\tau}}_d^t$ is the deviatoric part of the Reynolds stress tensor.

The notations related to the mean-flow variables will be subsequently omitted for simplicity and clarity purposes. Although the problem is axisymmetric, the tensors considered need to be used in their 3D tensorial form.

The viscous stress tensor is given by

$$\boldsymbol{\tau} = \begin{pmatrix} \tau_{xx} & \tau_{xr} & 0 \\ \tau_{rx} & \tau_{rr} & 0 \\ 0 & 0 & \tau_{\theta\theta} \end{pmatrix} \quad (\text{A.15})$$

The form of $\boldsymbol{\tau}^t$ is similar.

Hence, the developed expression for Φ is

$$\Phi = (\tau_{xx} + \tau_{d,xx}^t) \frac{\partial u}{\partial x} + (\tau_{xr} + \tau_{d,xr}^t) \left(\frac{\partial u}{\partial r} + \frac{\partial v}{\partial x} \right) + (\tau_{rr} + \tau_{d,rr}^t) \frac{\partial v}{\partial r} + (\tau_{\theta\theta} + \tau_{d,\theta\theta}^t) \frac{v}{r} \quad (\text{A.16})$$

Appendix B

Mach number fields

This Appendix shows an additional set of Mach numbers definition from the compound-choking model of used by Lamberts [24], and the entire set of Mach number fields obtained with the numerical setup. Each page presents a primary pressure ratio. These summary files contain much information about the choking, and were an important source of understanding of the choking phenomenon.

Additionally, a more interactive support to really understand the variations between each figure is proposed through this link¹.

Compound choking model

In order to present a case of p_{01}/p_{02} with the full characteristic curve, Figure B.1 shows M_2 and M_{eq} (from the compound-choking model [24]) inside part of the ejector, for $p_{01}/p_{02} = 7$ (*Fabri-like* regime).

The motion of the ξ -section towards the inlets as the back-pressure increases is well visible, with the curves departing sooner the curve from $p_{0,out}/p_{02} = 1.2$.

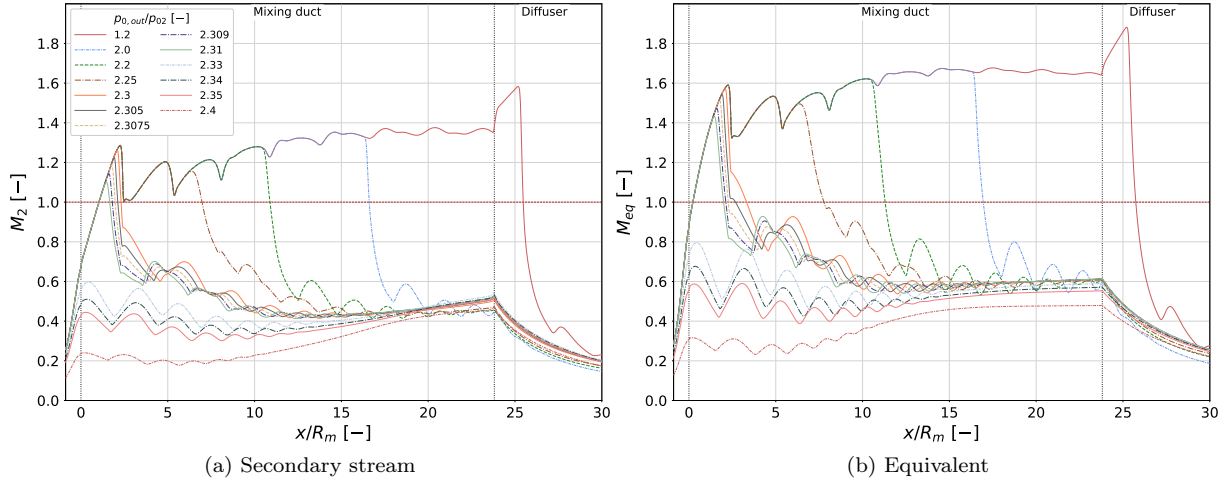
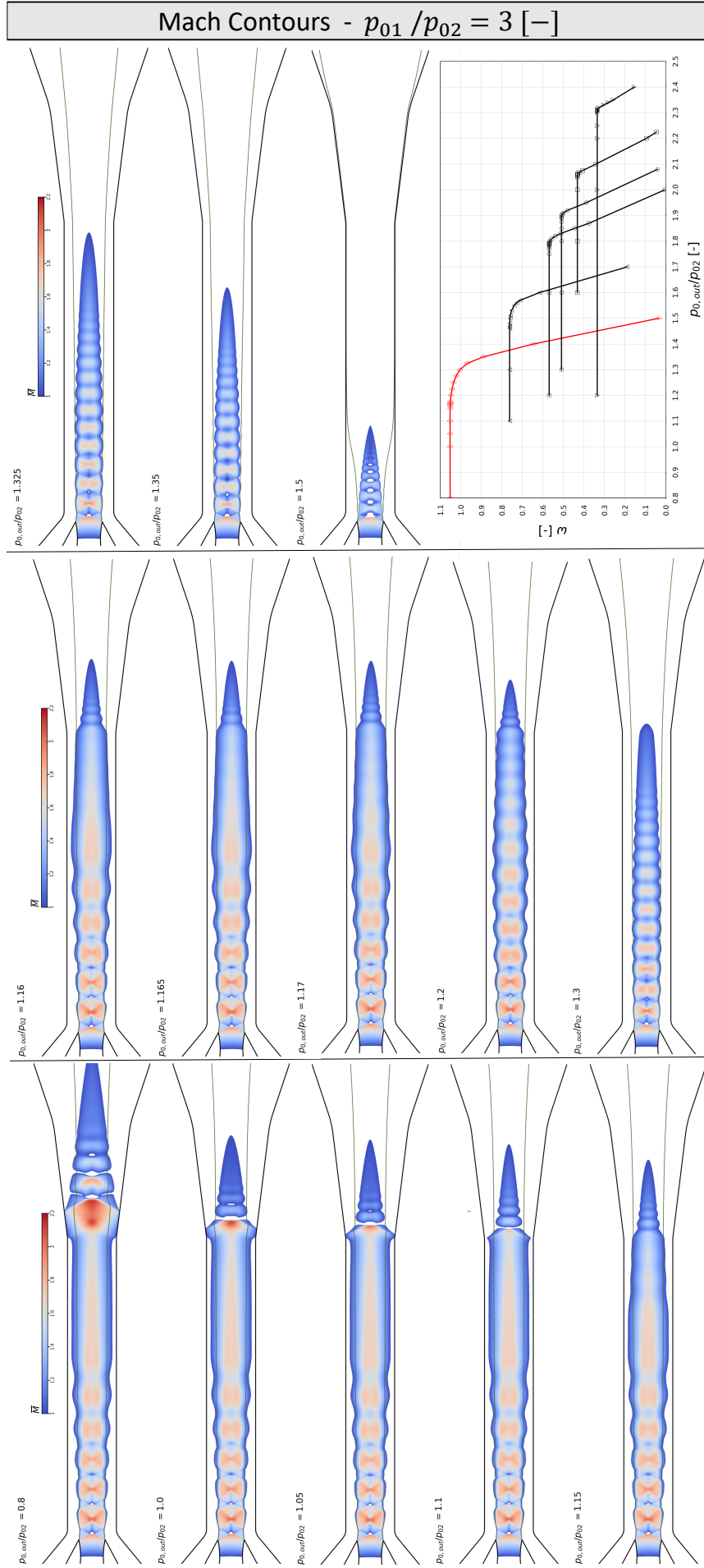
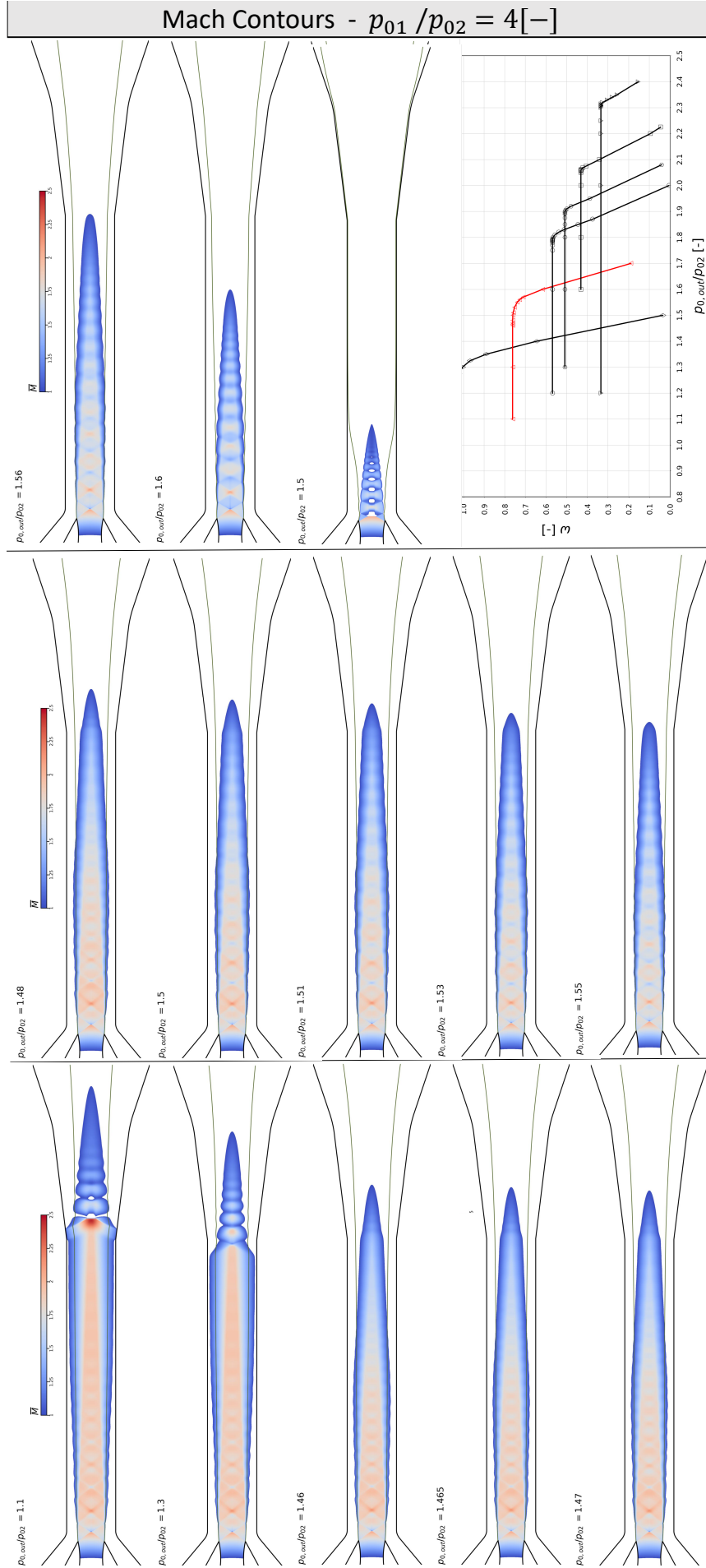
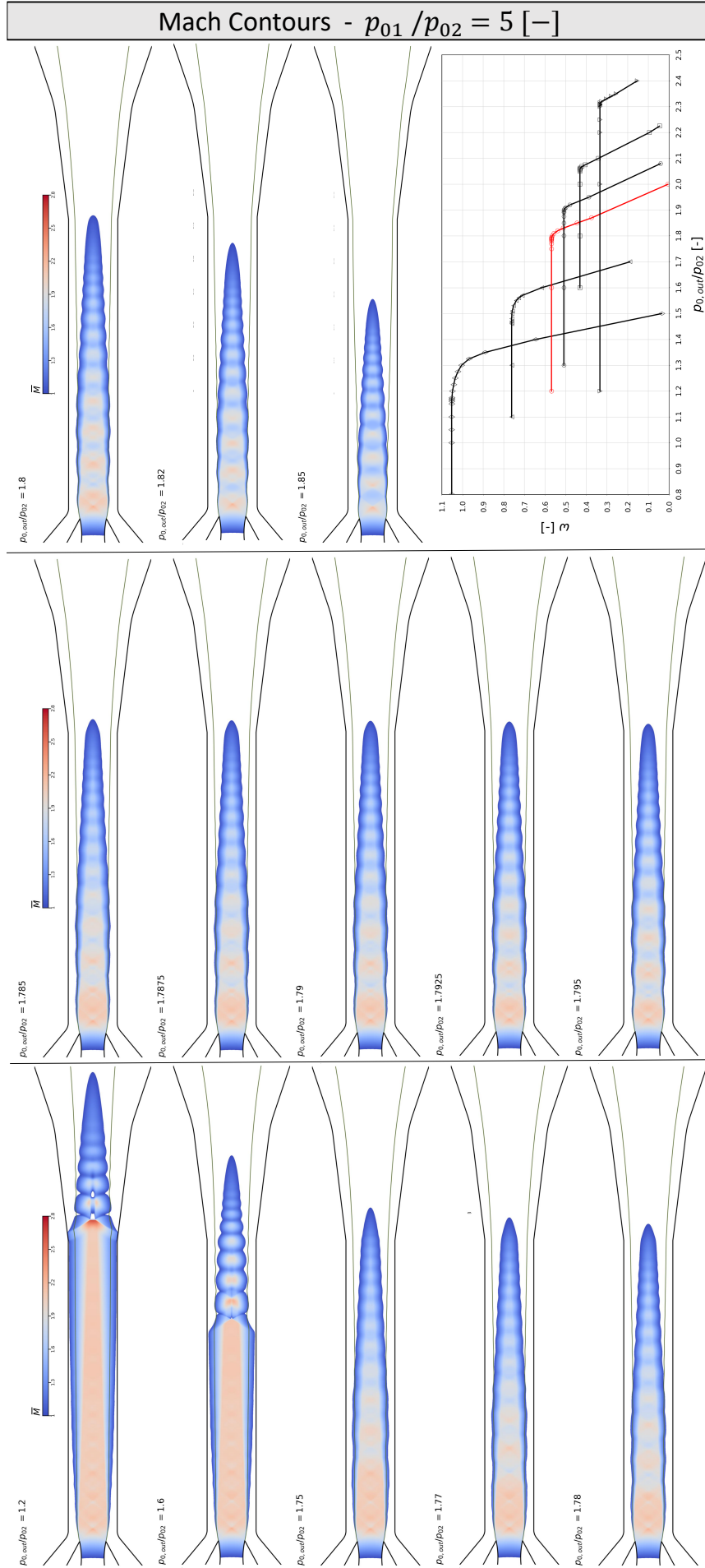


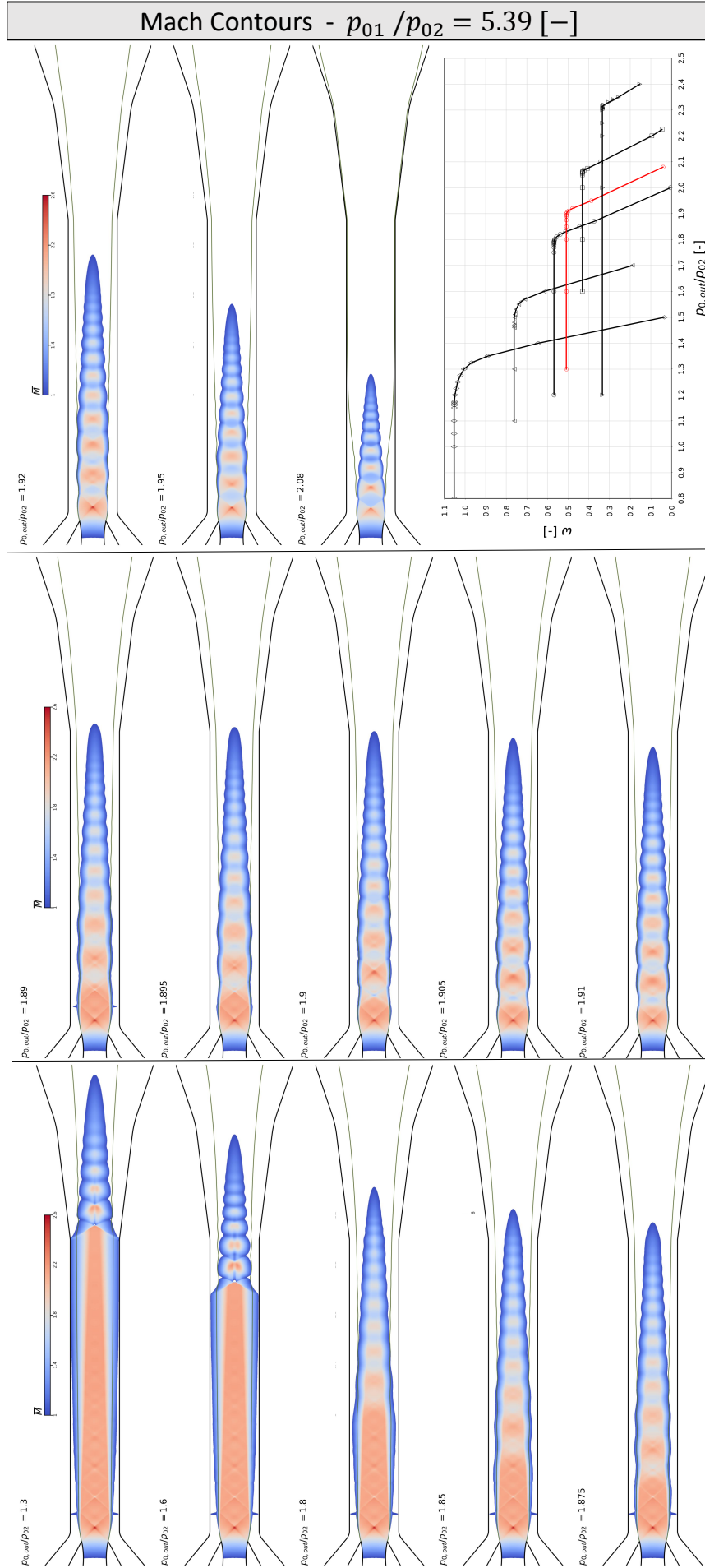
Figure B.1: M_2 and M_{eq} - $p_{01}/p_{02} = 7$

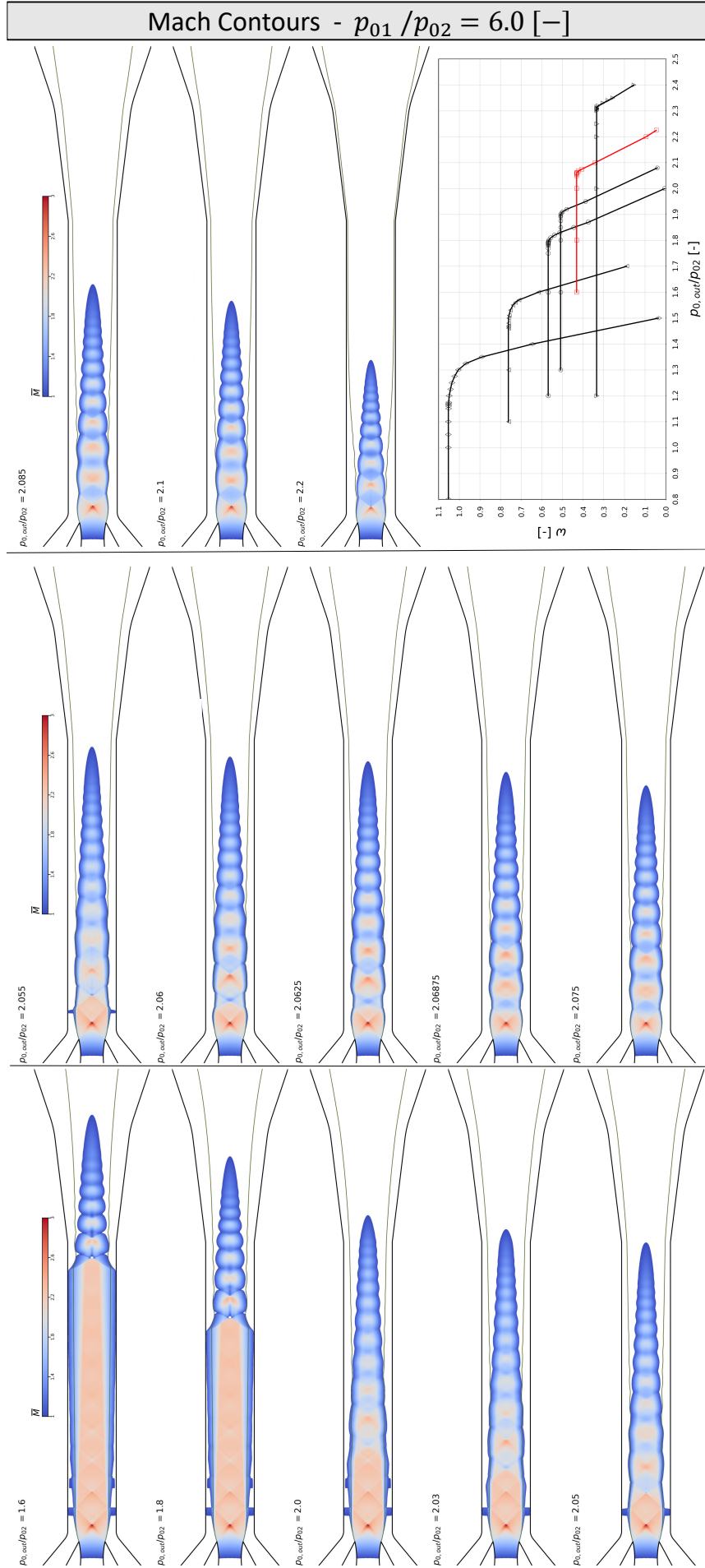
¹<https://www.youtube.com/playlist?list=PLqErhQDH4L-uCNa13kshhGyFU3Gyeqbsf>

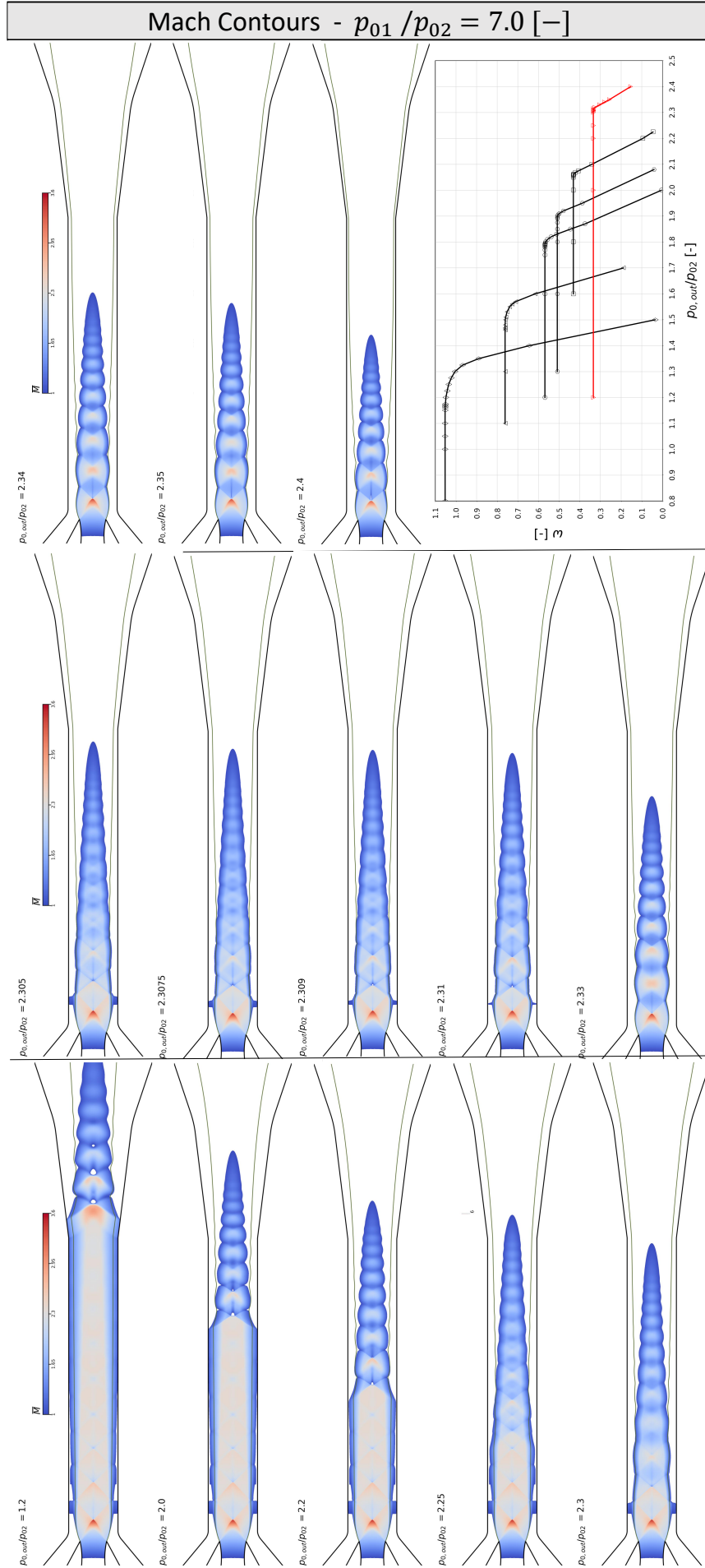

 Figure B.2: Mach number fields - $p_{01}/p_{02} = 3$ (VS $\times 2$)


 Figure B.3: Mach number fields - $p_{01}/p_{02} = 4$ (VS $\times 2$)


 Figure B.4: Mach number fields - $p_{01}/p_{02} = 5$ (VS $\times 2$)


 Figure B.5: Mach number fields - $p_{01}/p_{02} = 5.39$ (VS $\times 2$)


 Figure B.6: Mach number fields - $p_{01}/p_{02} = 6$ (VS $\times 2$)


 Figure B.7: Mach number fields - $p_{01}/p_{02} = 7$ (VS $\times 2$)

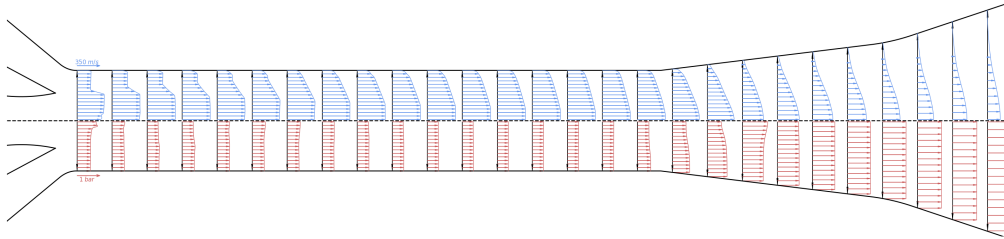
Appendix C

General flow topology

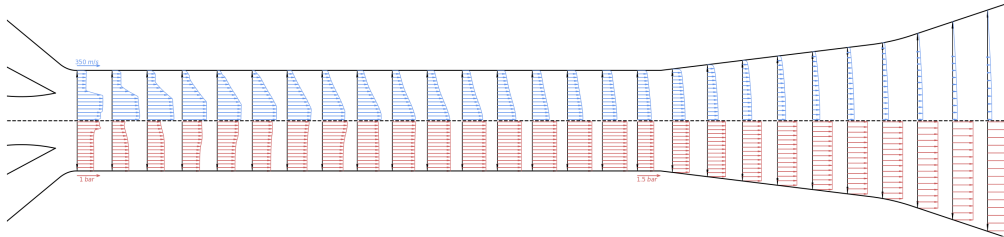
This appendix shows various figures that constitute an extension to the work presented in this manuscript.

C.1 Velocity and pressure profiles

Figures C.1 show the velocity and pressure profiles for $p_{01}/p_{02} = 4$ with one on-design and one off-design case (done for $p_{01}/p_{02} = 6$ in the manuscript).



(a) $p_{0,out}/p_{02} = 1.3$



(b) $p_{0,out}/p_{02} = 1.6$

Figure C.1: Velocity and pressure profiles comparison
 $p_{01}/p_{02} = 4$ (VS $\times 2$)

C.2 Wall pressure Profile

This appendix section presents the wall static pressure p_{wall} along the mixing duct. Originally, wall pressure values were used to compare numerical and experimental results [68, 72], as this is the only local flow variable retrievable in this type of experimental setup. However, as explained above, no experimental data were collected during this Master's thesis, making the use of numerical wall pressure analysis less valuable. Despite this lesser interest, numerical results of p_{wall} are presented anyway, as they illustrate quite well the flow phenomena and could be of use for future works.

In the first place, results for 3 values of p_{01}/p_{02} , namely 4, 5.39 and 6 (corresponding to the *compound-like* regime, the transition between the *compound-like* to *Fabri-like* regime, and the *Fabri-like* regime, respectively) are shown in Figures C.2, C.3 and C.4. For each p_{01}/p_{02} , 6 cases of back-pressure corresponding to the operating conditions I to VI (referring to the cases used in fourth chapter) are reported (except for $p_{01}/p_{02} = 6$ where an additional back-pressure case was added to yield further insight). For improved readability, the values in the diffuser are not reported, since the behavior of p_{wall} is very simple in that region.

The first observation for all three cases is that in the *choked regions*, the pressure profiles have more or less a constant behavior (especially visible for case of lowest back-pressure). In Figures C.2, C.3 and C.4, cases I and II are in critical modes, and the added case $p_{0,\text{out}} = 1.8$ in figure C.4 is also in critical operation. For those cases, the pressure profile is logically identical up to a certain point where the higher $p_{0,\text{out}}$ case departs the lower one as it is no longer choked, allowing to achieve the necessary compression. This is especially visible in figure C.4. When the flow is no longer double-choked, the wall static pressure increases with x .

The line corresponding to $p_{\text{wall}}/p_{02} = p^*/p_{02}$ (whose value is approximately 0.528) is also indicated with a horizontal dash-dotted line. It allows to shed light on the transition between the compound-like regime to the Fabri-like regime: it can clearly be observed that for $p_{01}/p_{02} = 4$, the wall static pressure is always above p^* , indicating the absence of any sonic section (when approximating the secondary stream as a simple 1D isentropic stream). For $p_{01}/p_{02} = 5.39$, i.e. the case of the first Fabri-pattern, the wall pressure just reaches p^* which is in accordance with the presence of a sonic section. Finally, for $p_{01}/p_{02} = 6$, the static pressure takes values well below p^* , indicating a large Fabri-pattern where the secondary flow goes significantly above Mach 1.

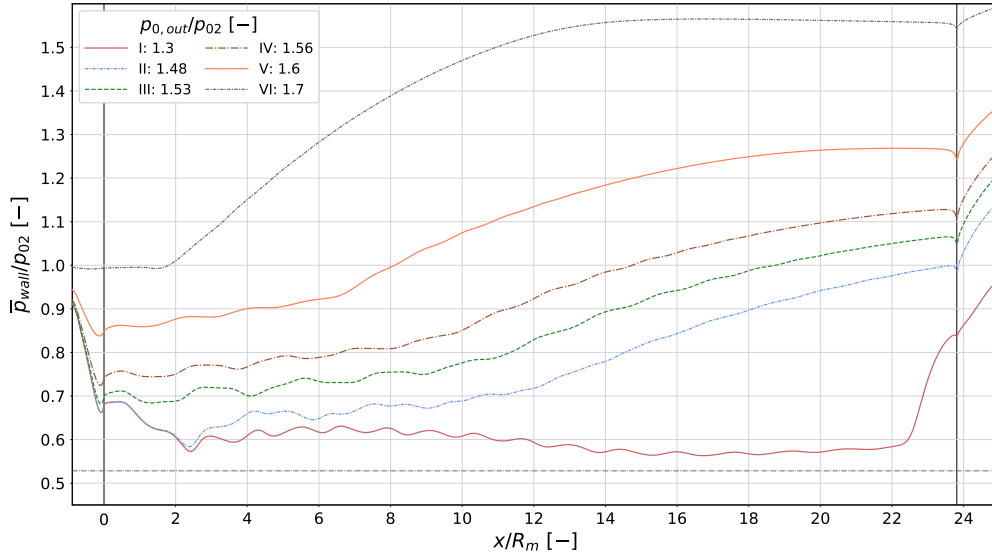
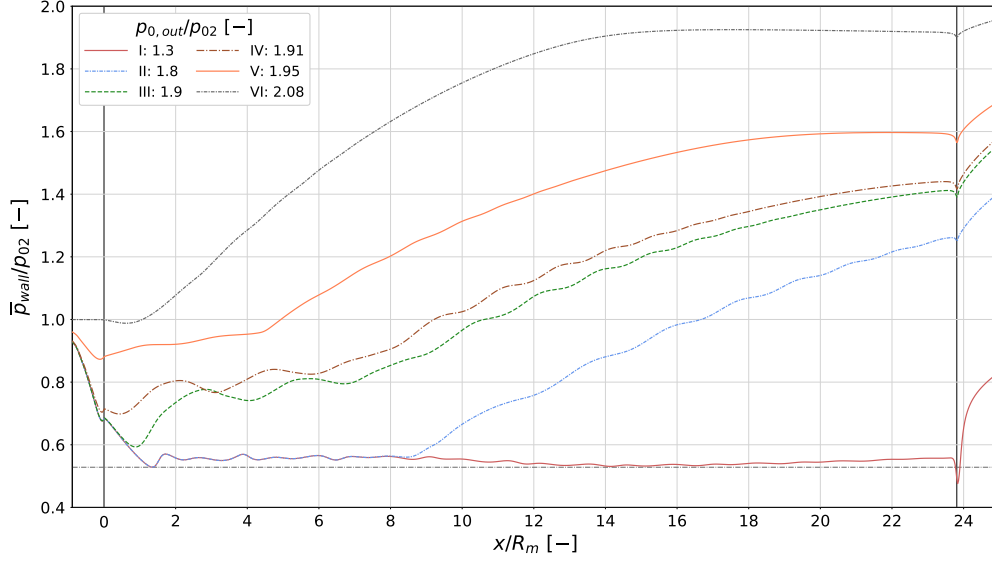
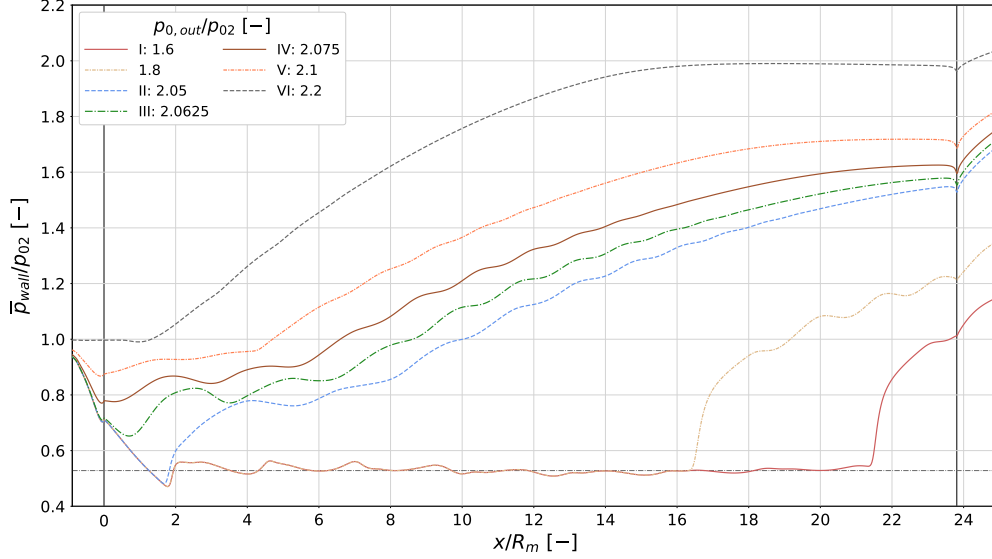
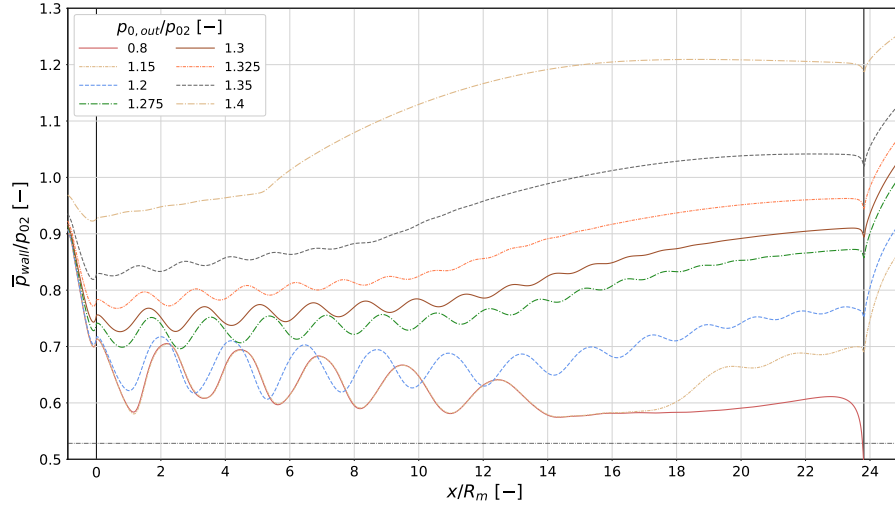
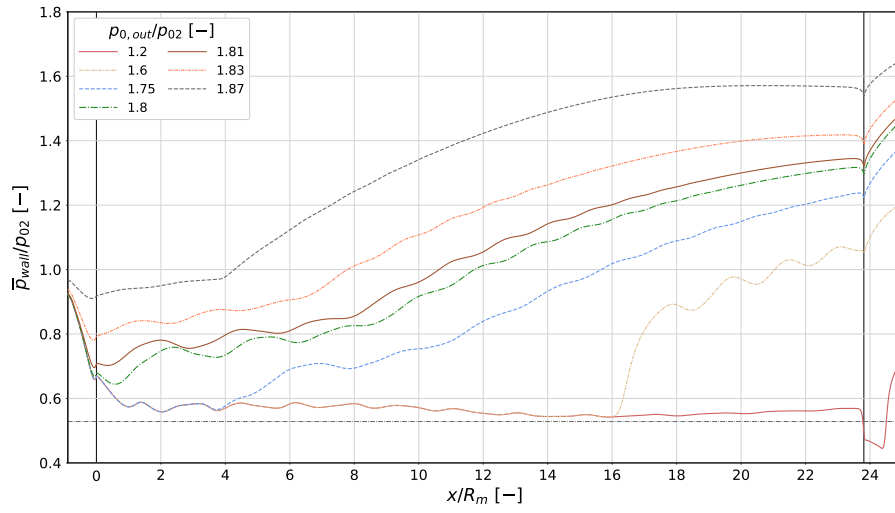
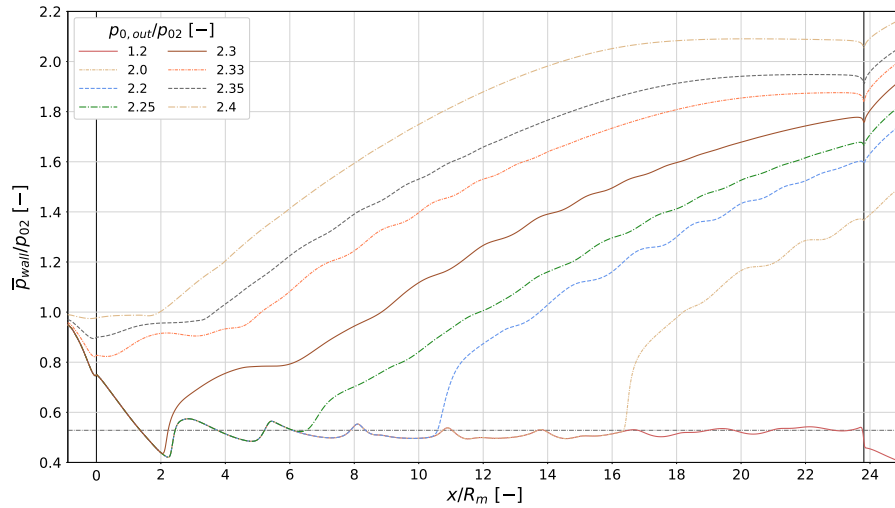


Figure C.2: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 4$


 Figure C.3: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 5.39$

 Figure C.4: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 6$

There is a true potential to corroborate these results using experimental measurements. Although Lamberts [68] showed important discrepancies between the numerical and experimental wall static pressure, he also observed that these are partly due to 3D effects. The axisymmetric geometry might allow to get rid of these 3D effects and promise encouraging results for the experimental and numerical study of the compound-and Fabri regimes.

Finally, pressure profiles for additional values of p_{01}/p_{02} are shown on the next page.

Wall static pressure for additional p_{01}/p_{02} Figure C.5: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 3$ Figure C.6: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 5$ Figure C.7: Wall static pressure in the mixing duct, $p_{01}/p_{02} = 7$

C.3 Criterion for the definition of a unique choking section x_c

This Appendix presents the criterion used to define the axial position of the *choking section* x_c for each p_{01}/p_{02} . The results for each pressure and finally a small comparison of the impact of the chosen tolerances on that definition are depicted through several figures.

Defining a unique value of x_c as a straight section is interesting to perform further analysis (e.g., to assess the value of M_{eq} at the location where it is supposed to be unity). This therefore raises a question : "How should the choking section be defined ?" There are two ways, relatively close to each other. The first consists in taking the whole cross-section and compute the average value of the Mach number difference $\overline{\Delta M}$ on the discrete points of the mesh :

$$\overline{\Delta M}_{\bar{c}} = \frac{1}{N} \sum_{i \in A(x)}^N \Delta \overline{M}_i \quad (C.1)$$

where N is the number of mesh elements of the whole section $A(x)$.

The second definition is based on the observation that the secondary stream is the first to "un-choke" (this was better observed for *Fabri-like* cases, see section 3.4.3). It leads to a similar definition of the value $\overline{\Delta M}$ that only takes the secondary stream into account.

$$\overline{\Delta M}_{\bar{c}_2} = \frac{1}{N_2} \sum_{i \in A_2(x)}^{N_2} \Delta \overline{M}_i \quad (C.2)$$

where N_2 is the number of cells spanning the cross-section of the secondary stream area $A_2(x)$ defined thanks to the dividing streamline.

This analysis is then applied to the critical back-pressure case, and the first axial position x where $\overline{\Delta M}$ exceeds ϵ (which was set to $\epsilon = 5 \times 10^{-4}$) is therefore considered as the choking section :

- $x_{\bar{c}} = x$ if $\overline{\Delta M}_{\bar{c}}(x) \geq \epsilon$
- $x_{\bar{c}_2} = x$ if $\overline{\Delta M}_{\bar{c}_2}(x) \geq \epsilon$

The second definition was used instead of the one considering the whole section.

In Figure 3.16, both definitions are identified for $p_{01}/p_{02} = 6$ where $x_{\bar{c}_2}/R_m = 1.2$ is closer to the nozzle exit and almost perfectly aligned with the vertical boundary of the secondary stream. Comparatively, $x_{\bar{c}}/R_m = 1.29$ is slightly shifted to the right due to the still choked primary stream that lowers the average.

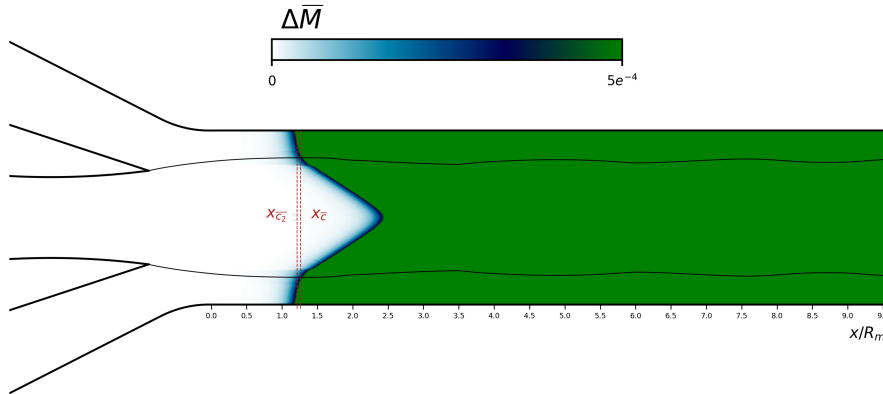


Figure C.8: Field of $\overline{\Delta M}$ between the cases $p_{0,out}^A/p_{02}$ and $p_{0,out}^i/p_{02}$, $p_{01}/p_{02} = 6$, (VS $\times 2$)

Influence of p_{01}/p_{02} on the axial location of the *choking section*

Figure C.9 shows the identified sections $x_{\bar{c}_2}$ for each primary pressure ratio p_{01}/p_{02} . They are all represented except for the lowest one (3) due to the lack of precision of the computed $\Delta\bar{M}$. In that case, it was found difficult to apply the previously defined criterion. Nonetheless, the field $\Delta\bar{M}$ was still interesting but had to be analyzed separately.

The values are first of all of the same order of magnitude for all cases, as they belong to a range from $x_{\bar{c}_2}/R_m = 0.8$ to 1.7. Regardless of the *compound-like* or *Fabri-like* regime, the *choking section* is thus located early in the mixing duct.

A minimum of $x_{\bar{c}_2}$ is observed for $p_{01}/p_{02} = 5.39$, which as reminder corresponds to the first *Fabri-like* case.

The high pressure cases (6 & 7) are almost equal and by having a look at the Mach fields, we observe a strong similarity of their first Fabri pattern.

The compound-like operating conditions are choked farther downstream..

Note that $x_{\bar{c}}$ is very close to $x_{\bar{c}_2}$ but is shifted upwards by an average of $x/R_m = 0.2$ [-] with respect to $x_{\bar{c}_2}$.

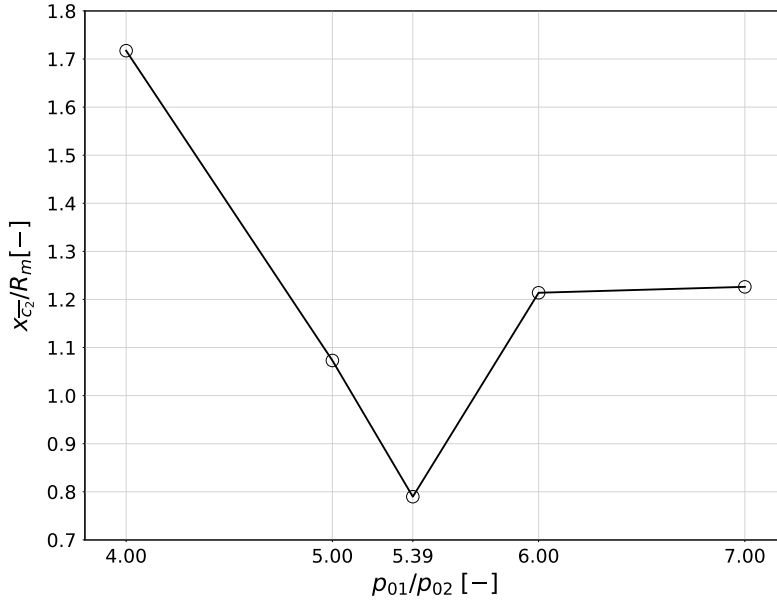


Figure C.9: Location of choking sections $x_{\bar{c}_2}$ depending on p_{01}/p_{02}

Comparison of various tolerances

Finally, in this section, various values of the tolerance ϵ are displayed to show their impact of the precise definition of the *choking section*. This is done for the case $p_{01}/p_{02} = 6$ (a *Fabri-like* case). Each Figure displays on top an on-design case compared to the reference case, and at the bottom the critical point (last *choked region*) compared to the reference case.

It is comforting to observe that the global shape of the white and green areas is not fundamentally impacted by the value of the chosen tolerance. This shows that the *ξ-section* is very well defined, as there is an abrupt gap of $\Delta\bar{M}$ at this location. Note that the same does not hold for *compound-like* cases.

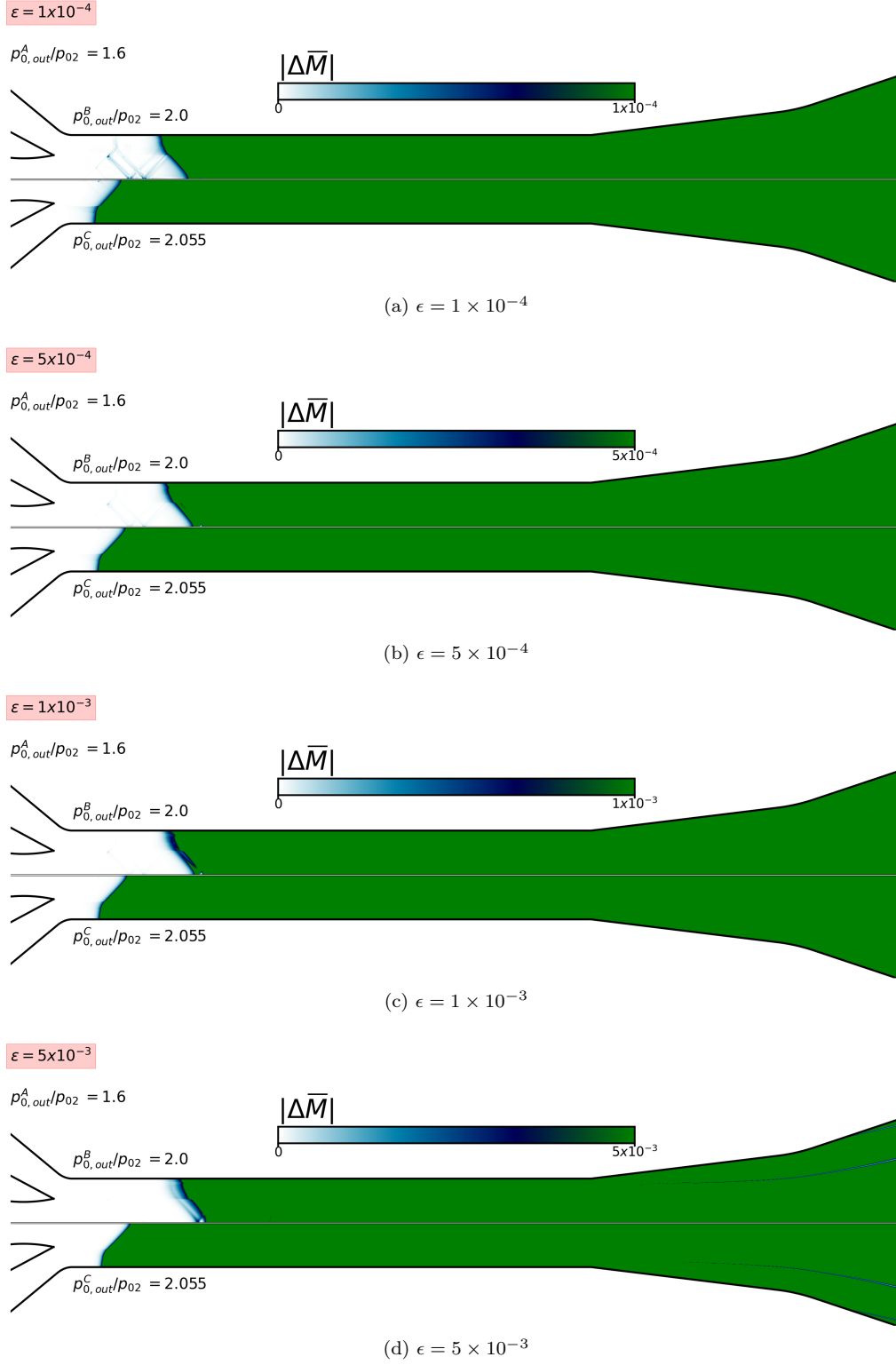


Figure C.10: Absolute value of Mach number difference $\Delta \bar{M}$ - Various tolerances comparison - $p_{01}/p_{02} = 6$ [-] (VS $\times 2$)

C.4 Fully inviscid simulations

In addition to the fully inviscid simulation for $p_{01}/p_{02} = 3$ presented in section 3.6.3, a simulation at the other extremum of the p_{01}/p_{02} range was computed with an inviscid model: $p_{01}/p_{02} = 7$.

Figures C.11 and C.12 show the Mach number field and a numerical schlieren.

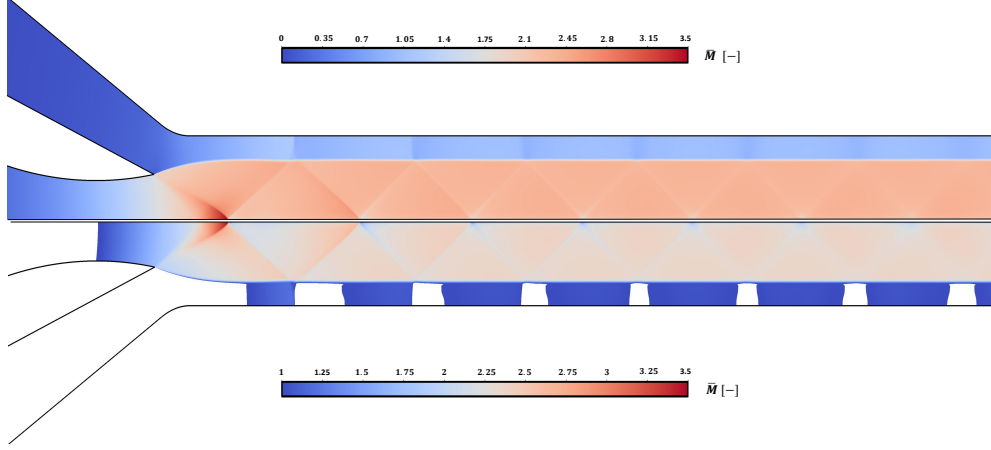


Figure C.11: Mach number field for the case $p_{01}/p_{02} = 7$, fully inviscid. Upper half: entire M range, lower half: supersonic region only, (VS $\times 2$).

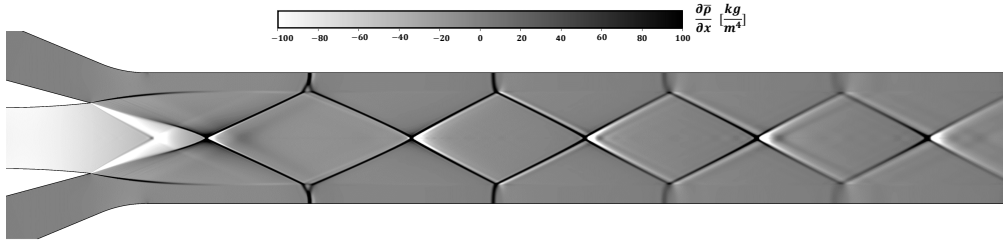


Figure C.12: Numerical schlieren image for the case $p_{01}/p_{02} = 7$, fully inviscid (VS $\times 2$).

The Fabri-patterns are clearly defined, and the dividing streamline adopts a straight behavior in the mixing duct very rapidly.

Appendix D

Confirmation of the compound choking on a reference case

As a final investigation of the origin of the choking phenomenon in a compound-compressible flow, and as a validation of the analytical tools developed and/or utilized in sections 3.4 and 3.5, the case of a modified ejector is studied. The purpose of this simple study is not to thoroughly analyze the flow in this ejector, but to qualitatively illustrate the choking phenomenon at play in a geometry where one gets rid of the ambiguous "throat" of a supersonic ejector (i.e., its mixing duct).

The new ejector consists in the same primary nozzle as the one used throughout this work, but is followed by a circular mixing duct presenting a minimal cross-section area at a single point, the throat. It is then followed by a diffuser.

The computational domain is composed of approximately 450 000 elements, and the other features of the solver setup are the same as those presented in section 2.3. The main relevant dimension is the mixing duct throat radius: $R_t = 30 [mm]$.

Only the case $p_{01}/p_{02} = 7$ was investigated. The computation yielded an on-design entrainment ratio $\omega = 1.07088$. On the basis of several simulations, it was found that the ejector operates in the double choked mode up to a critical back-pressure ratio of $p_{0,out}^*/p_{02} = 0.975$.

The Mach number field of the supersonic region within the ejector is shown for an on-design case, the critical case and an off-design case in Figure D.1.

From the computation of ω for several back-pressure ratios (not presented here), it is clear that the secondary flow is choked. It is also visible in Figure D.1 that a large portion of the secondary stream remains subsonic throughout the ejector. There is no sonic section at the throat. The mechanism responsible for the limitation of secondary mass flow rate is thus not the Fabri-choking. The only explanation for the secondary choking is the compound choking argument.

This reference geometry allows to define more precisely the *choking section*, which should be located around the shroud throat [23]. The values of the equivalent Mach numbers M_{eq} and M_{eq}^∞ defined previously are reported in Table D.1 at the throat for the three cases of Figure D.1. The results are quite convincing, both cases A and B being identified as compound-choked whereas case C is clearly compound-subsonic at the throat.

	M_{eq}	M_{eq}^∞
Case A	1.148	1.114
Case B	1.036	0.981
Case C	0.837	0.758

Table D.1: Values of both equivalent Mach numbers at the throat for 3 cases

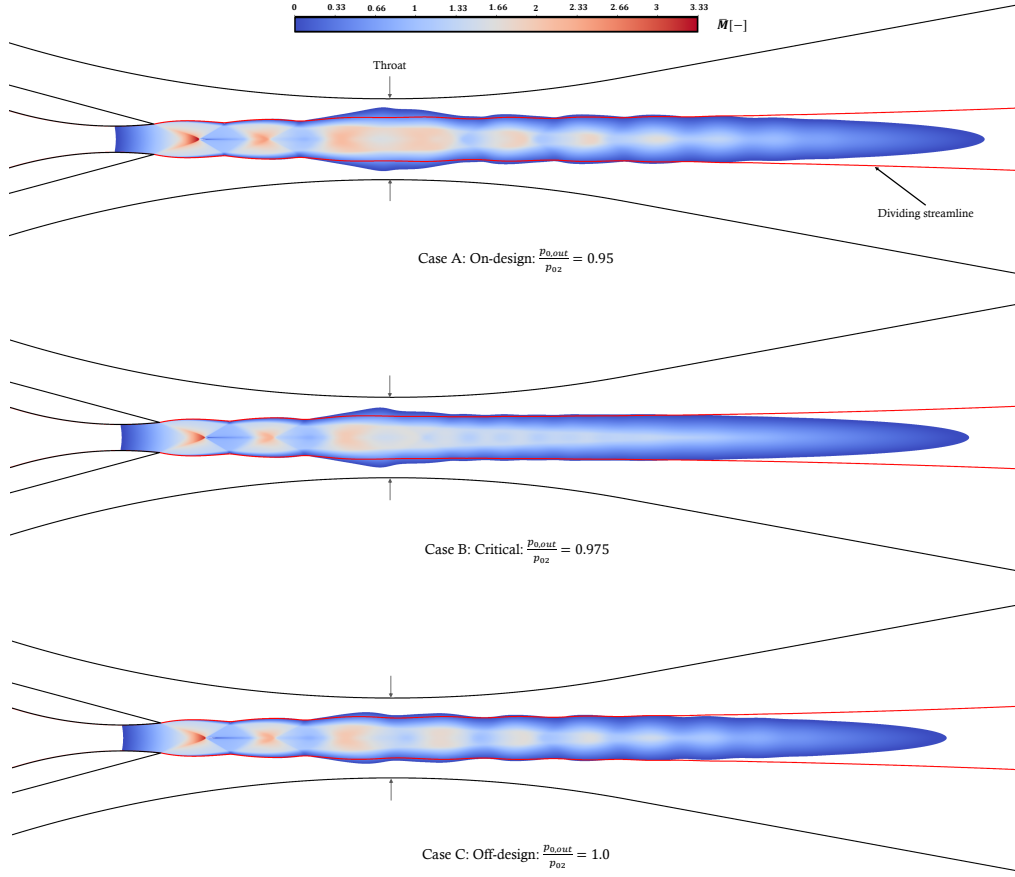


Figure D.1: Mean-flow Mach number field (supersonic region only) of the modified ejector, for three operating points. The location of the ejector shroud throat is indicated with arrows.

Appendix E

Ejector Performance

E.1 Fabri model fit

The following figure shows the fit performed using the model proposed by Chen et al. [19]. It exactly corresponds to the Figure of section 4.2 but here, the off-design curves are entirely represented.

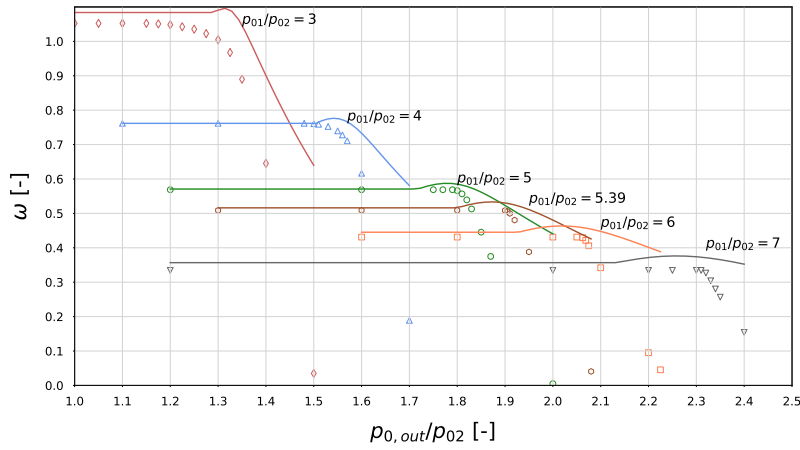


Figure E.1: Entrainment ratio comparison of W.Chen model and numerical results
 $\eta_p = 0.9764$, $\eta_s = 0.935$ and $\psi_m = 0.93$

E.2 Local flux and cumulative flux transfers between the primary and secondary streams

The purpose is to show a set of local fluxes and cumulative transfers for $p_{01}/p_{02} = 4$ (*compound-like*), the *Fabri-like* regime being shown in section 4.4.

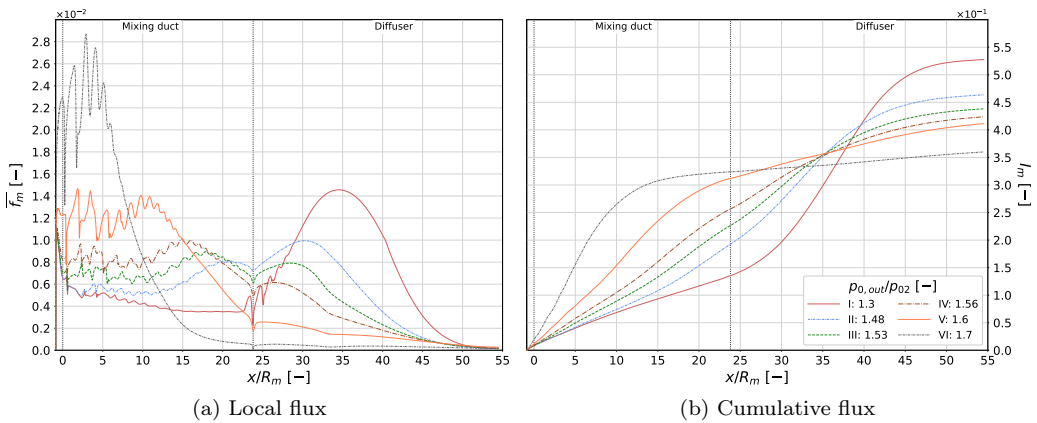
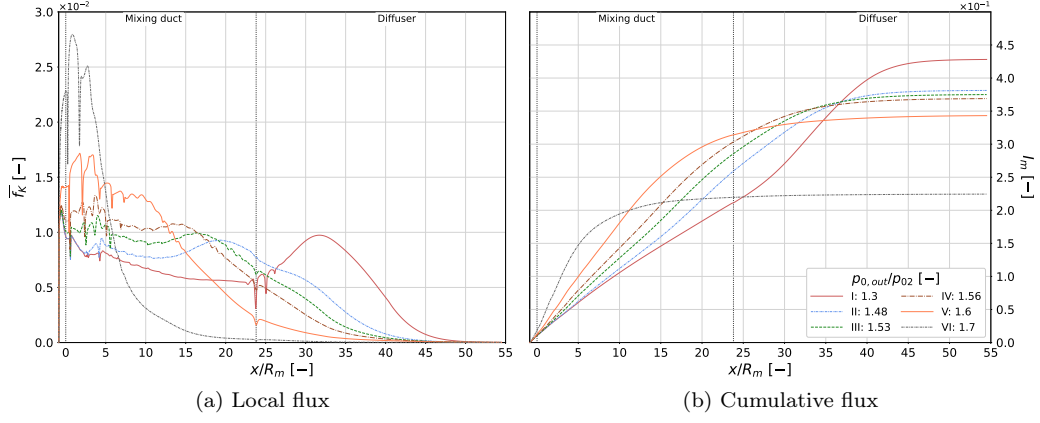
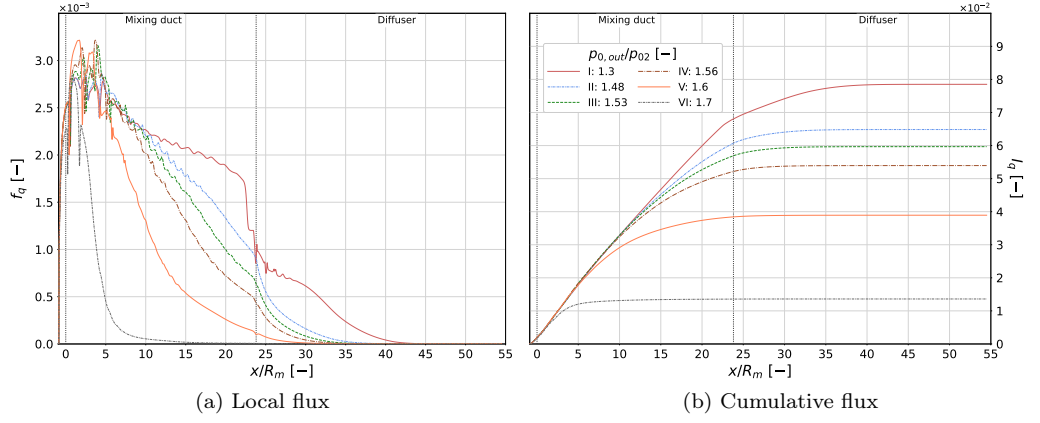
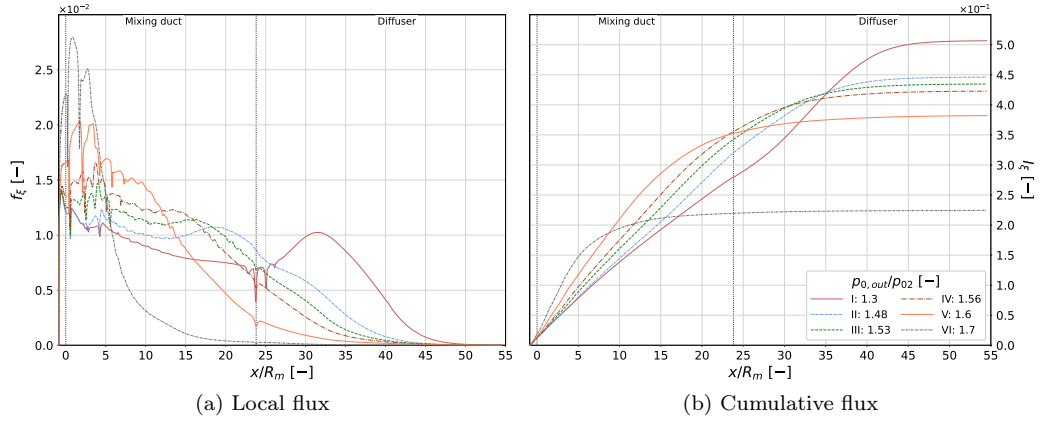


Figure E.2: Dimensionless Momentum flux - $p_{01}/p_{02} = 4$


 Figure E.3: Dimensionless mean-flow kinetic energy flux - $p_{01}/p_{02} = 4$

 Figure E.4: Dimensionless heat flux (exergy content) - $p_{01}/p_{02} = 4$

 Figure E.5: Dimensionless mean-flow total exergy flux - $p_{01}/p_{02} = 4$

Appendix F

Experimental setup: Additional drawings

This Appendix is a non-exhaustive list of the drawings used for the experimental project. The most important pieces are presented hereunder. These drawings were initially produced by the authors but some additional expertise from the CREDEM designer Mr. Lion was incorporated.

Adaptor preliminar design

The small sketch displayed in Figure F.1 shows a possible design of adaptor to fit between the T-connection piece and the first piece of the ejector assembly in order to modify the NXP value (i.e., the distance between the primary nozzle tip and the beginning of the mixing duct).

The width of the adaptor must be therefore adapted to fit the desired value. Depending on the chosen value, the length of the T-connection can also be reduced to accommodate these changes.

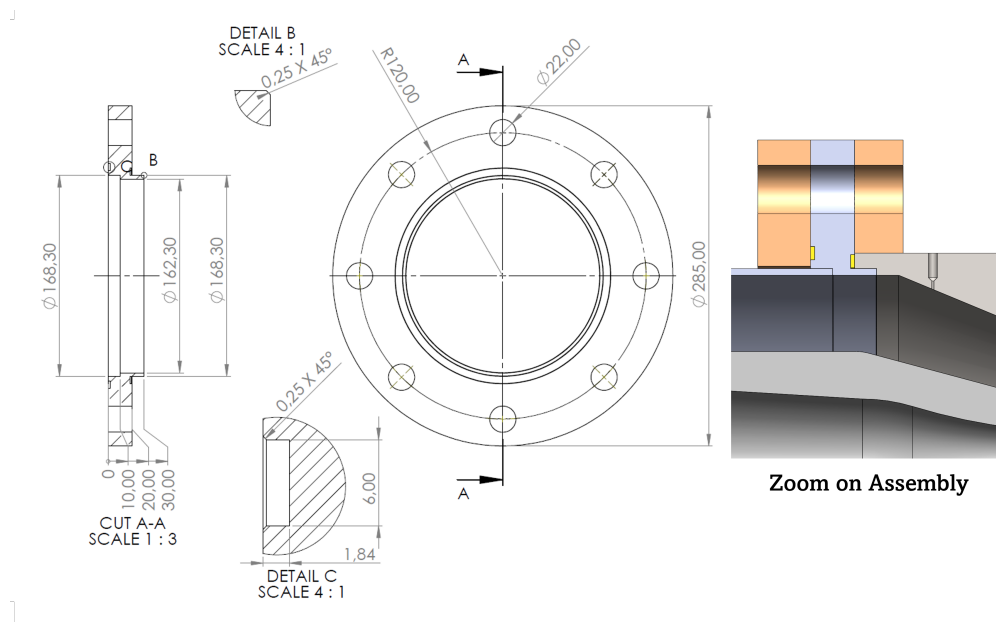


Figure F.1: Adaptor preliminary design - assembly representation scale doesn't correspond

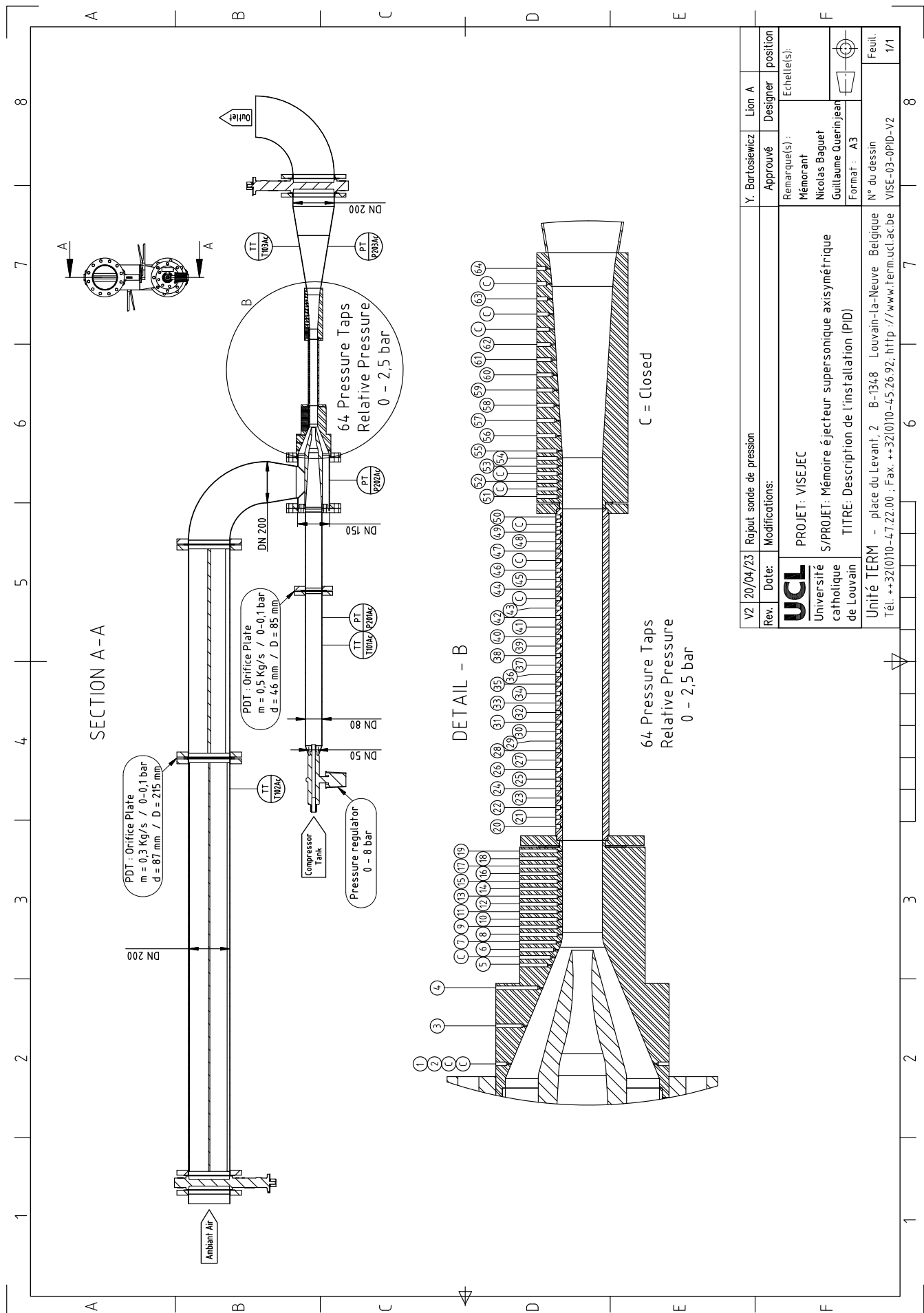


Figure F.2: Data acquisition - PID of the setup

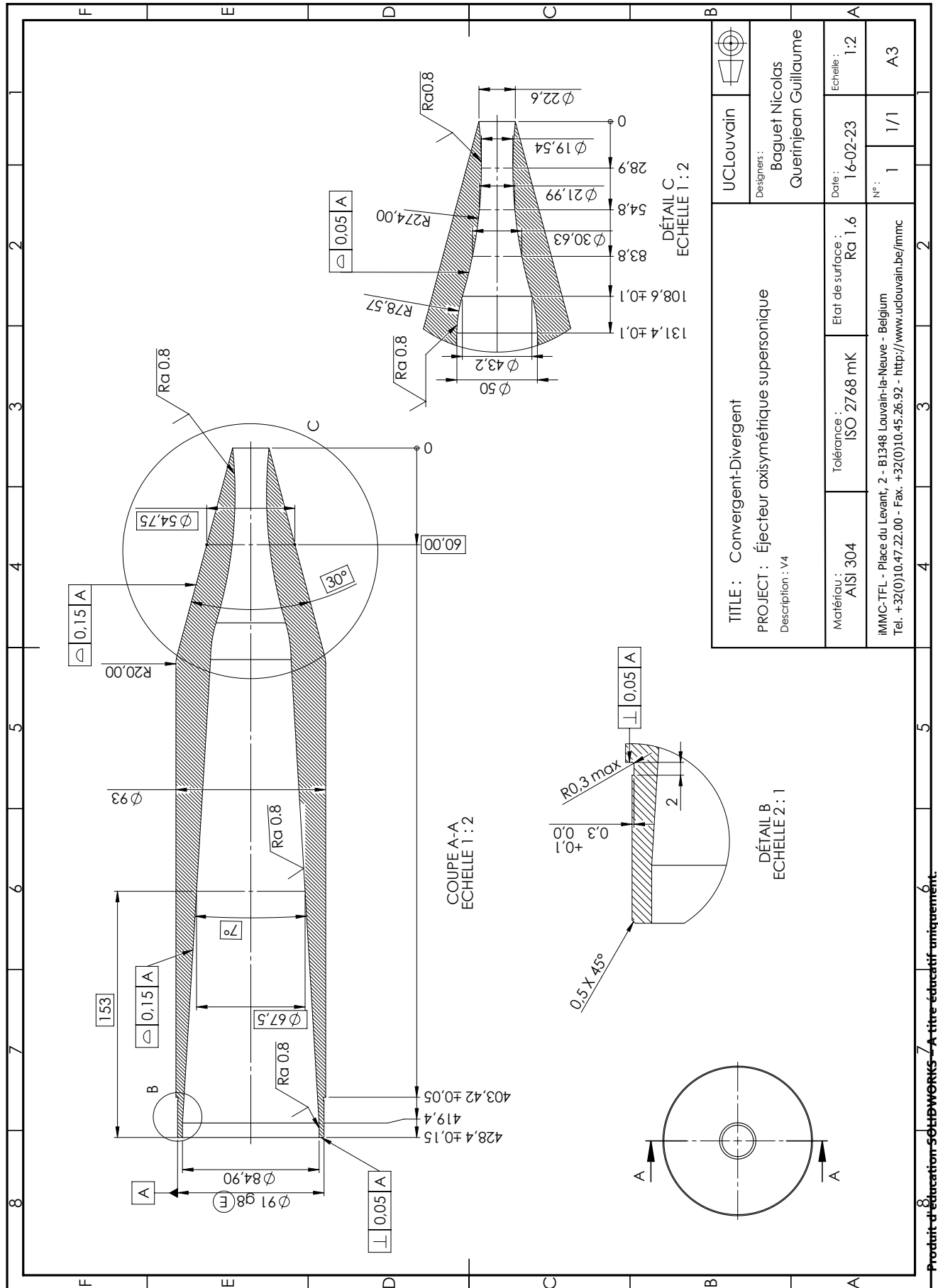


Figure F.3: Drawing of the primary nozzle piece - Dimensions are in millimeters

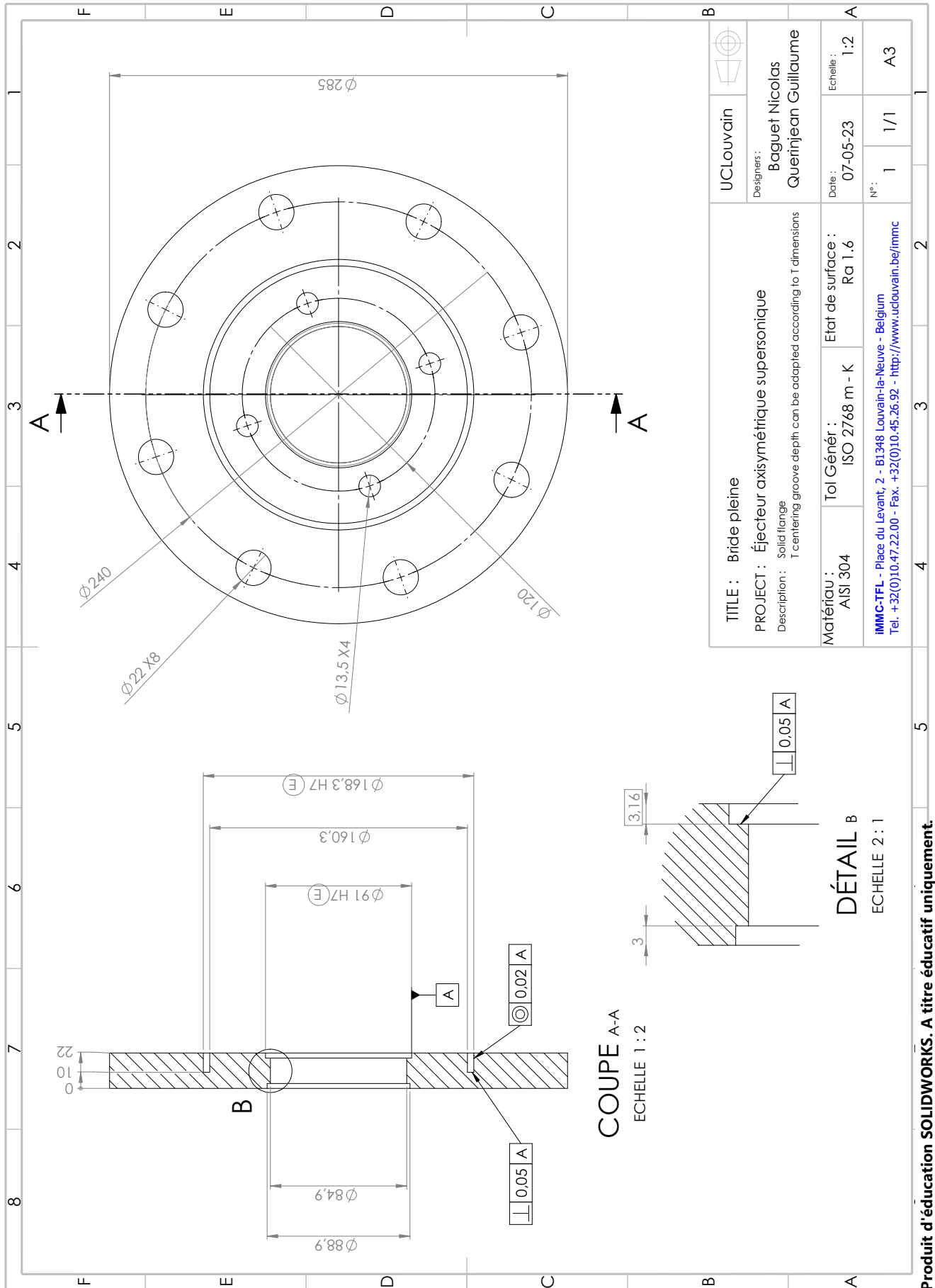


Figure F.4: Drawing of the centering nozzle flange - Dimensions are in millimeters

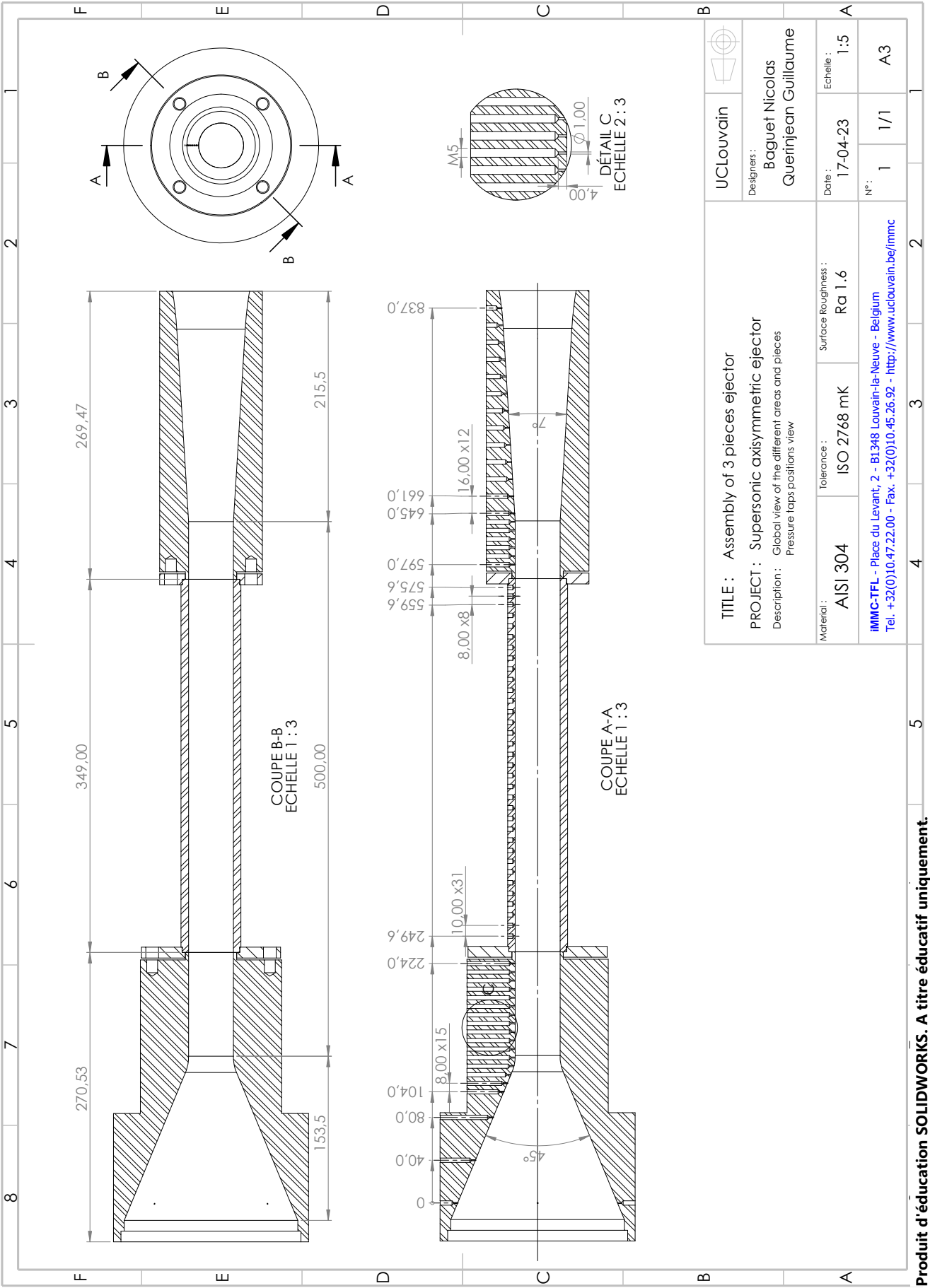


Figure F.5: Drawing of the ejector assembly. Dimensions are in millimeters



Figure F.6: Drawing of the first piece of the ejector. Dimensions are in millimeters

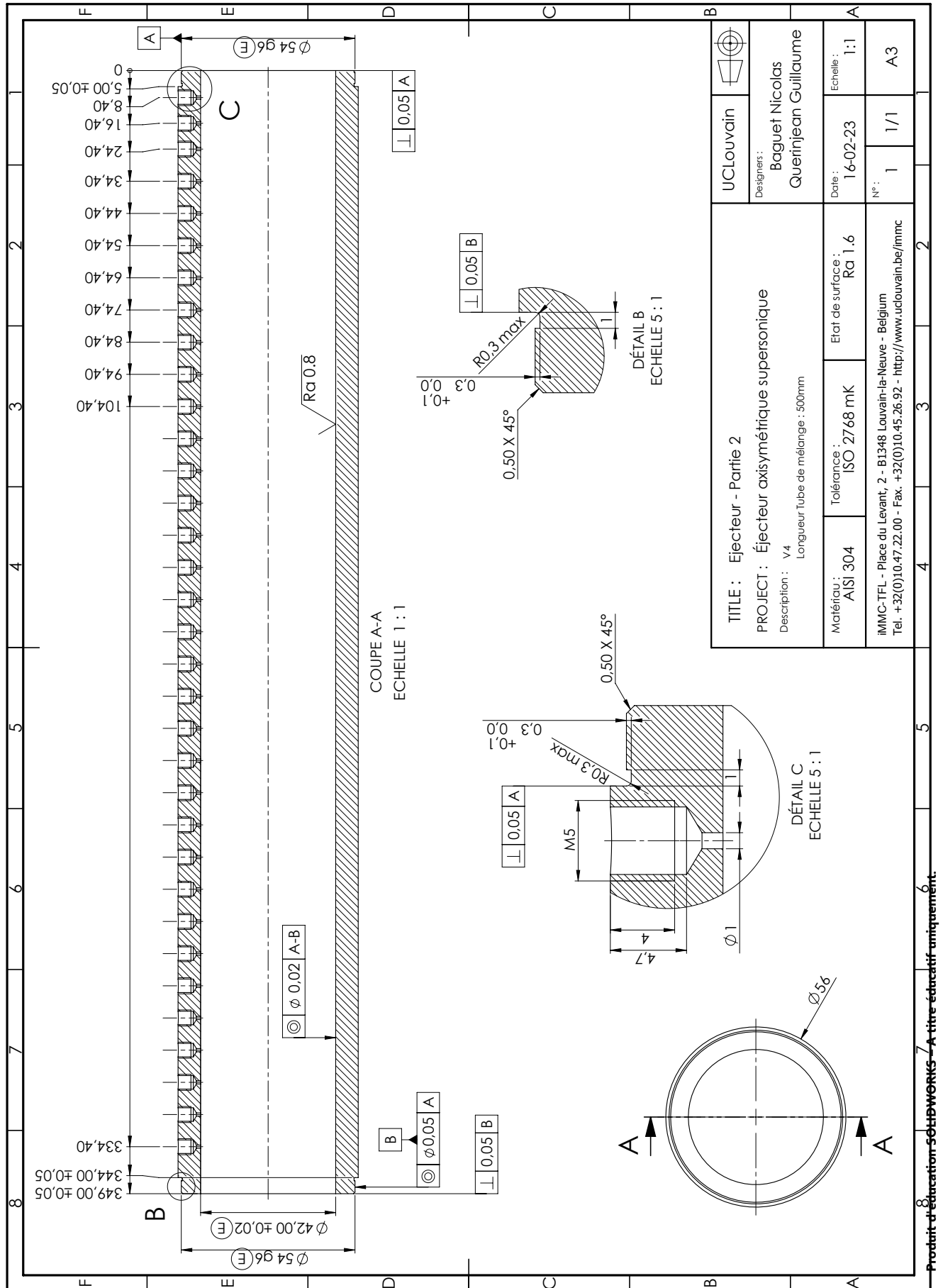


Figure F.7: Drawing of the second piece of the ejector. Dimensions are in millimeters

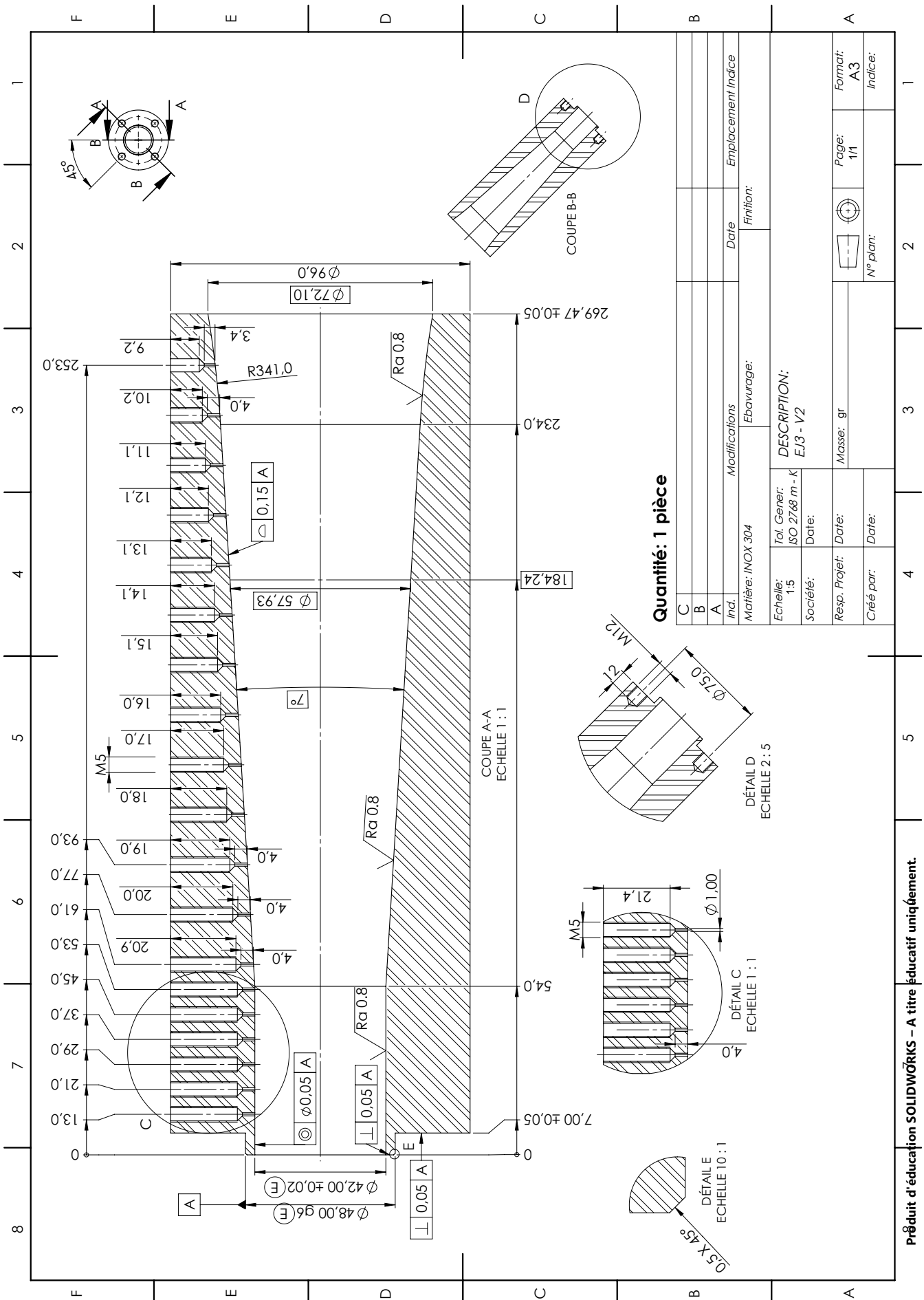


Figure F.8: Drawing of the third piece of the ejector. Dimensions are in millimeters

Bibliography

- [1] A.Petrovic, M.Z.Jovanovic, S.Genic, U.Bugaric, and B.Delibasic. Evaluating performances of 1-d models to predict variable area supersonic gas ejector performances. 163:270–289, November 2018.
- [2] Napier JR. The ejector condenser for steam engines. 88.357-IN311., 1869.
- [3] B. KK. Study of ejectors, 1935 (In Russian).
- [4] B. KK. Calculation of ejectors. p. 14-7, 1938.
- [5] F. Duncan Berkeley. Ejectors have a wide range of uses. 37:95–100, December 1958.
- [6] M. Alperin and J. J. Wu. Thrust augmenting ejectors, part i. 21(10):1428–1436, 1983.
- [7] M. Alperin and J. J. Wu. Thrust augmenting ejectors, part ii. 21(12):1698–1706, 1983.
- [8] T. G. Tillman and Jr W. M. Precz. Thrust characteristics of a supersonic mixer ejector. 11(5):931–937, 1995.
- [9] S. K. Karthick, Srisha M. V. Rao, G. Jagadeesh, and K. P. J. Reddy. Experimental parametric studies on the performance and mixing characteristics of a low area ratio rectangular supersonic gaseous ejector by varying the secondary flow rate. 161:832–845, 2018.
- [10] Ll Macia, R. Castilla, P. J. Gamez-Montero, S. Camacho, and E. Codina. Numerical simulation of a supersonic ejector for vacuum generation with explicit and implicit solver in openfoam. 12(18):3553, 2019.
- [11] Chong DT, Yan JJ, Wu GS, and Liu JP. Structural optimization and experimental investigation of supersonic ejectors for boosting low pressure natural gas. 29:2799–807, 2009.
- [12] A.A.Askalany and E.S. Ali. A new approach integration of ejector within adsorption desalination cycle reaching cop higher than one. Volume 41, 2020.
- [13] Rao SMV and G.Jagadeesh. Studies on the effects of varying secondary gas properties in a low entrainment ratio supersonic ejector. 78:289–302, 2015.
- [14] Zhao Li X and Hu X C. Thermodynamic analysis of organic rankine cycle with ejector. 42:342–9, 2012.
- [15] K. Chunnanond and S. Aphornratana. Ejectors: applications in refrigeration technology. 129-155, 2004.
- [16] 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. 29(89):1523 – 1531, 2009.
- [17] G. Besagni, R. Mereu, and F. Inzoli. Ejector refrigeration: A comprehensive review. 53:373 – 407, 2016.

- [18] B.J. Huang, J.M. Chang, C.P. Wang, and V.A. Petrenko. A 1-d analysis of ejector performance. 22(5):354 – 364, 1999.
- [19] 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.
- [20] 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.
- [21] A.Metsue, R.Debroeyer, S.Poncet, and Y.Bartosiewicz. An improved thermodynamic model for supersonic real-gas ejectors using the compound-choking theory. 121856, 2021.
- [22] J. Fabri and R. Siestrunck. Supersonic air ejectors. 5:1–34, 1958.
- [23] A. Bernstein, W. H. Heiser, and C. Hevenor. Compound compressible nozzle flow. 34(3):548– 554, 09 1967.
- [24] O.Lamberts, P.Chatelain, N.Bourgeois, and Y.Bartosiewicz. The compound-choking theory as an explanation of the entrainment limitation in supersonic ejectors. 158:524–536, September 2018.
- [25] Keenan JH and Newman EP. A simple air ejector. 64:A75–81, 1942.
- [26] Keenan JH, Newman EP, and Lustwerk F. An investigation of ejector design by analysis and experiment. *J Appl Mech, Trans ASME*, 1950.
- [27] M.John T and Bagster DF. A new ejector theory applied to steam jet refrigeration. 16:442–9, 1977.
- [28] B.J. Huang, C.P. Wang J.M. Chang, and V.A. Petrenko. A 1-d analysis of ejector performance. 22(5):354 – 364, 1999.
- [29] Li F, Tian Q, Wu C, Wang X, and Lee J-M. Ejector performance prediction at critical and subcritical operational modes. *Appl Therm Eng*, 2017.
- [30] Y. Zhu, C. Cai, C. Wen, and Y. Li. Shock circle model for ejector performance evaluation. 48(9):2533 – 2541, 2007.
- [31] Z. Ma, H. Bao, , and A. P. Roskilly. Thermodynamic modelling and parameter determination of ejector for ejection refrigeration systems. 75:117 – 128, 2017.
- [32] R. Yapc and H.K. Ersoy. Performance characteristics of the ejector refrigeration system based on the constant area ejector flow model. 46(18):3117 – 3135, 2005.
- [33] N. Galanis and M. Sorin. Ejector design and performance prediction. 104:315 – 329, 2016.
- [34] W. Chen, S. Zhang C. Shi, H. Chen, D. Chong, and J. Yan. Theoretical analysis of ejector refrigeration system performance under overall modes. 185:2074 – 2084, 2017.
- [35] 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. 55:72 – 84, 2015.
- [36] Chaoyin Shi, Huiqiang Chen, Weixiong Chen, Shuangping Zhang, Daotong Chong, and Junjie Yan. 1d model to predict ejector performance at critical and sub-critical operation in the refrigeration system. 75:1477–1483, August 2015.
- [37] 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. 35(4):772– 784, 2012.
- [38] S. He, Y. Li, and R.Z.Wang. Progress of mathematical modeling on ejectors. 13(8):1760 –1780, 2009.

-
- [39] S. Croquer, Y. Fang, A. Metsue, Y. Bartosiewicz, and S. Poncet. Compound-choking theory for supersonic ejectors working with real gas. 120396, 2021.
 - [40] G. Besagni, R. Mereu, and F. Inzoli. Ejector refrigeration: A comprehensive review. 53:373 – 407, 2016.
 - [41] S.Croquer, O.Lamberts, S.Poncet, S.Moreau, and Y.Bartosiewicz. Large eddy simulation of a supersonic air ejector. 118177, 2022.
 - [42] J. Gagan, K. Smierciew, D. Butrymowicz, and J. Karwacki. Comparative study of turbulence models in application to gas ejectors. 78:9 – 15, 2014.
 - [43] G.Besagni, N.Cristiani, L.Croci, G.R.Guédon, and F.Inzoli. Computational fluid-dynamics modelling of supersonic ejectors: Screening of modelling approaches, comprehensive validation and assessment of ejector component efficiencies. *Applied Thermal Engineering*, 186:116431, 2021.
 - [44] S. Croquer, S. Poncet, and Z. Aidoun. Turbulence modeling of a single-phase r134a supersonic ejector. part 1: Numerical benchmark. 61:140 – 152, 2016.
 - [45] S. Croquer, S. Poncet, and Z. Aidoun. Turbulence modeling of a single-phase r134a supersonic ejector. part 2: Local flow structure and exergy analysis. 61:153 – 165, 2016.
 - [46] K. Pianthong, W. Seehanam, M. Behnia, T. Sriveerakul, and S Aphornratana. Investigation and improvement of ejector refrigeration system using computational fluid dynamics technique. 48, 2556–2564, 2007.
 - [47] 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. 29(89):1523 – 1531, 2009.
 - [48] 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. 29(1415):2990 – 2998, 2009.
 - [49] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier. Cfd experiments integration in the evaluation of six turbulence models for supersonic ejectors modeling, 2003.
 - [50] 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. 63:1–21, 2010.
 - [51] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier. Numerical and experimental investigations on supersonic ejectors. 26(1):56 – 70, 2005.
 - [52] S. Garimella F. Mazzelli, A. B. Little and Y. Bartosiewicz. Computational and experimental analysis of supersonic air ejector: Turbulence modeling and assessment of 3d effects. 56:305 – 316, 2015.
 - [53] 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. 63:133 – 145, 2013.
 - [54] G. Besagni and F. Inzoli. Computational fluid-dynamics modeling of supersonic ejectors: Screening of turbulence modeling approaches. 117:122 – 144, 2017.
 - [55] 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. 89:795 – 811, 2015.
 - [56] Y. Zhu and P. Jiang. Experimental and numerical investigation of the effect of shock wave characteristics on the ejector performance. 40:31 – 42, 2014.

-
- [57] W. Chen, D. Chong, J. Yan, and J. Liu. Numerical optimization on the geometrical factors of natural gas ejectors. 50(8):1554 – 1561, 2011.
 - [58] T. Sriveerakul, S. Aphornratana, and K. Chunnanond. Performance prediction of steam ejector using computational fluid dynamics: Part 1. validation of the cfd results. 46(8):812 – 822, 2007.
 - [59] S. Varga, A. C. Oliveira, and B. Diaconu. Numerical assessment of steam ejector efficiencies using cfd. 32(6):1203 – 1211, 2009.
 - [60] J. Yan, W. Cai, and Y. Li. Geometry parameters effect for air-cooled ejector cooling systems with r134a refrigerant. 46(Supplement C):155 – 163, 2012.
 - [61] S.Varga, A.C.Oliveira, and B.Diaconu. Influence of geometrical factors on steam ejector performance – a numerical assessment. 32(7):1694–1701, 2009.
 - [62] 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. 40:328 – 337, 2014.
 - [63] D. Chong, M. Hu, W. Chen, J. Wang, J. Liu, and J. Yan. Experimental and numerical analysis of supersonic air ejector. 130(C):679–684, 2014.
 - [64] S. Croquer, S. Poncet, and N. Galanis. Comparison of ejector predicted performance by thermodynamic and cfd models. 68:28 – 36, 2016.
 - [65] M. Ji, T. Utomo, J. Woo, Y. Lee, H. Jeong, and H. Chung. Cfd investigation on the flow structure inside thermo vapor compressor. 35(6):2694 – 2702, 2010.
 - [66] F. Kong and H.D. Kim. Analytical and computational studies on the performance of a two-stage ejector-diffuser system. 85:71 – 87, 2015.
 - [67] F. Mazzelli and A. Milazzo. Performance analysis of a supersonic ejector cycle working with r245fa. 49:79 – 92, 2015.
 - [68] O.Lamberts. Choking and mixing phenomena in supersonic ejectors. *Doctoral dissertation, Université catholique de Louvain*, 2018.
 - [69] O. Lamberts, P. Chatelain, and Y. Bartosiewicz. New methods for analyzing transport phenomena in supersonic ejectors. 64:23-40, 2017.
 - [70] 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.
 - [71] 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*, 2013.
 - [72] A.Soenen. "numerical simulation of a supersonic ejector : influence of the geometry and study of a transient scenario, 2021.
 - [73] Optimal nozzle exit position for a single-phase ejector (experimental, numerical and thermodynamic modelling). *International Journal of Refrigeration*, 144:108–117, 2022.
 - [74] G.Winckelmans. Pertes de charge singulières. *LMECA2322 - Fluid Mechanics 2*, pages 7–8.
 - [75] MI Ilmy. Grabcad - pressure regulator 3d model, 2015.
 - [76] Cheng ho CHEN. Grabcad - butterfly valve 3d model, 2021.
 - [77] E.L. Upp and Paul J. LaNasa. *Fluid Flow Measurement*. 2002.

- [78] G.Pradeep, R.Srisha, and K.Pramod. Effect of mixing duct geometry of supersonic ejector in the critical flow regime. 07 2019.
- [79] A. Ricordeau and P. Compain-Mefray. *Méthode active de dessin technique: lycées professionnels, lycées techniques, formation continue*. Casteilla, 2000.
- [80] ERIKS. Frenzelit plaque d’étanchéité fibres novapress 815, 2017.
- [81] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. 32(8), 1598-1605, 1994.
- [82] W. Sutherland. The viscosity of gases and molecular force. (5):507–531, 1893.
- [83] ANSYS. Inc. Ansys fluent theory guide, 2021-R2.
- [84] D.R. Gist, Trevor Foster, and Dianne Deturris. Examination of fabri choking in a simulated air augmented rocket. 5392, 2007.
- [85] R.Debroeyer. Internal communication.
- [86] J. Jiménez. Free shear flows.
- [87] O.Cleynen. The pressure distribution along a pipe with converging-diverging section, for various cases of decreasing back pressure. https://commons.wikimedia.org/wiki/File:Converging-diverging_nozzle,_pressure_distribution_during_acceleration_of_flow.svg, 2015.
- [88] K. R. Hedges and P. G. Hill. Compressible flow ejectors: Part ii— flow field measurements and analysis. 96(3):282–288, 1974.
- [89] Jr. E.F Kurtz. Theoretical model for predicting steam-ejector performance. 98(2):645–651, 1976.
- [90] A. Bejan. Fundamentals of exergy analysis, entropy generation minimization, and the generation of flow architecture. 26(7):0–43, 2002.
- [91] K.Virendra, PMV.Subbarao, and S.Gaurav. Effect of nozzle exit position (nxp) on variable area mixing ejector. *SN Applied Sciences*, 1:1473, 11 2019.
- [92] Ullah KR, Saidur R, Ping HW, Akikur RK, and Shuvo NH. A review of solar thermal refrigeration and cooling methods. 24:499–513, 2013.
- [93] Otanicar T, Taylor RA, and Phelan PE. Prospects for solar cooling—an economic and environmental assessment. 86:1287–99, 2012.
- [94] Anand S, Gupta A, and Tyagi SK. Solar cooling systems for climate change mitigation: a review. 41:143–61, 2015.
- [95] Calm JM. The next generation of refrigerants – historical review, considerations, and outlook. 31:1123–33, 2008.
- [96] E. Rusly, L. Aye, W.W.S. Charters, and A. Ooi. Cfd analysis of ejector in a combined ejector cooling system. 28:1092–1101, 2005.
- [97] R. L. Yadav and A. W. Patwardhan. Design aspects of ejectors: Effects of suction chamber geometry. 63(15):3886 – 3897, 2008.
- [98] X. Yang, X. Long, and X. Yao. Numerical investigation on the mixing process in a steam ejector with different nozzle structures. 56:95 – 106, 2012.
- [99] Y. Li S. He and R.Z. Wang. Progress of mathematical modeling on ejectors. 13(8):1760 – 1780, 2009.
- [100] A.Petrovic, MZ.Jovanovic, S.Genic, U.Bugaric, and B.Delibasic. Evaluating performances of 1-d models to predict variable area supersonic gas ejector performances. *Energy*, 163:270–289, 2018.

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

Rue Archimède, 1 bte L6.11.01, 1348 Louvain-la-Neuve, Belgique | www.uclouvain.be/epl