
Aerodynamic study of the high-pressure collector in a compressor test bench. BeCover

Auteur : Marc--Landouar, Tom

Promoteur(s) : Hillewaert, Koen

Faculté : Faculté des Sciences appliquées

Diplôme : Cours supplémentaires destinés aux étudiants d'échange (Erasmus, ...)

Année académique : 2025-2026

URI/URL : <http://hdl.handle.net/2268.2/25614>

Avertissement à l'attention des usagers :

Tous les documents placés en accès ouvert sur le site le site MatheO sont protégés par le droit d'auteur. Conformément aux principes énoncés par la "Budapest Open Access Initiative"(BOAI, 2002), l'utilisateur du site peut lire, télécharger, copier, transmettre, imprimer, chercher ou faire un lien vers le texte intégral de ces documents, les disséquer pour les indexer, s'en servir de données pour un logiciel, ou s'en servir à toute autre fin légale (ou prévue par la réglementation relative au droit d'auteur). Toute utilisation du document à des fins commerciales est strictement interdite.

Par ailleurs, l'utilisateur s'engage à respecter les droits moraux de l'auteur, principalement le droit à l'intégrité de l'oeuvre et le droit de paternité et ce dans toute utilisation que l'utilisateur entreprend. Ainsi, à titre d'exemple, lorsqu'il reproduira un document par extrait ou dans son intégralité, l'utilisateur citera de manière complète les sources telles que mentionnées ci-dessus. Toute utilisation non explicitement autorisée ci-avant (telle que par exemple, la modification du document ou son résumé) nécessite l'autorisation préalable et expresse des auteurs ou de leurs ayants droit.

FACULTY OF APPLIED SCIENCES



Aerodynamic Study of the High-Pressure Collector In a Compressor Test Bench

*Master thesis carried out as part of a student exchange program
at the University of Liège*

Author :
Tom MARC-LANDOUAR

Supervisor :
Pr. Koen HILLEWAERT

Academic Year 2025-2026

Acknowledgements

I would like to express my sincere gratitude to my supervisor at the University of Liège, Professor Koen Hillewaert, for his invaluable guidance, continuous availability, and insightful advice throughout the development of this master thesis.

I am deeply grateful to BeCOVER for providing me with the opportunity to conduct this research. I would particularly like to thank Mr. Olivier Servais, Director of BeCOVER, and Mr. Rémy Princivalle for their interest in my work and constructive follow-up. Although this internship did not take place physically on their premises, their support and the opportunity to work on this fascinating topic were essential to the successful completion of this thesis.

I also wish to extend my thanks to Mr. Hüseyin Güner and Mr. Nathan Deneffe for their warm welcome, valuable help, and technical guidance at the very beginning of this project.

Finally, I would like to thank the University of Liège for its hospitality during this academic year, as well as my home institution, École Centrale de Nantes, for making this rewarding Erasmus exchange possible.

Contents

Acknowledgements	i
List of Figures	v
1 Introduction	2
1.1 Context	2
1.2 Objectives of the Internship	3
2 Company Presentation	5
2.1 BeCOVER: A Strategic Joint Venture	5
2.2 Technical Capabilities	6
2.3 The ULiège Partnership	7
3 Theoretical Framework	8
3.1 Governing Equations of Fluid Dynamics	8
3.1.1 Conservation of Mass	8
3.1.2 Conservation of Momentum	8
3.1.3 Conservation of Energy	8
3.1.4 Equation of State	9
3.1.5 Boundary Layer Development	9
3.2 Turbulence Modeling	9
3.2.1 The RANS Approach and the Closure Problem	10
3.2.2 The Spalart–Allmaras (SA) Model	10
3.3 Physics of Pipe Bends	11
3.3.1 Centrifugal Forces and Radial Pressure Gradients	12
3.3.2 Secondary Flows: Dean Vortices	12
3.3.3 Impact on Velocity Profiles and Downstream Distortion	13
3.4 Orifice Plates and Multi-perforated Restrictors	13
3.4.1 Pressure Drop Mechanism and Vena Contracta	13
3.4.2 Multi-hole Jet Interaction and Coalescence	14
3.4.3 ISO 5167 Compliance and Geometric Limitations	15
3.4.4 Compressibility and Sonic Choking	15
3.4.5 Discharge Coefficient and Flow Capacity	16
3.5 Ejector Theory and Thermal Mixing	18

3.5.1	Mechanics of Entrainment and Mixing Layers	18
3.5.2	Thermal Mixing and Energy Conservation	18
3.5.3	Ejector Geometry and Flow Regions	19
3.6	Thermodynamics and Gas Properties	19
3.6.1	Ideal Gas Law and Compressibility	19
3.6.2	Sutherland’s Law and Viscosity Variations	20
3.6.3	Impact on Reynolds Number and Flow Regime	21
4	Numerical Methodology	22
4.1	Geometry Description	22
4.1.1	Investigated Configurations	22
4.1.2	Detailed Dimensional Specifications	24
4.2	Preprocessing with Pointwise	25
4.2.1	Near-Wall Resolution and T-Rex Algorithm	25
4.2.2	Diaphragm Initialization and Hybrid Topology	26
4.2.3	Volumetric Refinement and Local Sources	28
4.2.4	Final Mesh Statistics	29
4.3	Solver Configuration in SU2	30
4.3.1	Numerical Schemes: SLAU2 and Roe	30
4.3.2	Boundary Conditions and Color Mapping	30
4.3.3	High-Performance Computing (HPC) Resources	31
4.3.4	Convergence Monitoring and Residuals	34
5	Phase 1: Impact of Upstream Piping on Diaphragm Aerodynamics	36
5.1	Methodology of Analysis	36
5.1.1	Ideal Mass Flow and Normalization Strategy	36
5.1.2	Physical Scaling and Reynolds Sensitivity	36
5.2	Configuration 1: The Ideal Baseline (Reference)	37
5.2.1	Upstream Flow Uniformity	37
5.2.2	Global Performance and Discharge Coefficient	38
5.2.3	Jet Formation and Symmetry	40
5.2.4	Downstream Flow Development and Mixing	42
5.3	Configuration 2: Influence of the 90-degree Elbow	43
5.3.1	Flow Distortion and Dean Vortices	43
5.3.2	Penalization of the Discharge Capacity	44
5.3.3	Jet Asymmetry and Mass Imbalance	46
5.4	Sensitivity to Operating Temperature	47
5.4.1	Thermal Impact on Mass Flow Capacity	47
5.4.2	Reynolds Independence and C_d Stability	48
5.4.3	Synthesis of the Elbow Impact	49
6	Phase 2: Full System Analysis and Ejection Performance	50

6.1	Flow Topology in the Common Divergent	50
6.1.1	Primary Jet Expansion and Internal Recirculations	50
6.1.2	Secondary Air Suction and Mixing Uniformity	52
6.2	Quantifying the Suction Effect	55
6.2.1	Global Performance and Mass Flow Consistency	55
6.2.2	Entrainment Ratio Analysis	57
6.2.3	Impact of Temperature on Ejection Efficiency	57
6.2.4	Development of a Predictive Model for a Continuous Operating Envelope	59
6.3	Thermal Mixing Analysis and Pipe Integrity	62
6.3.1	Longitudinal Heat Dissipation and the "Cold Envelope"	62
6.3.2	Cross-Sectional Homogenization and Stratification	63
6.3.3	Theoretical Validation: The Adiabatic Mixing Limit	64
6.3.4	Operational Generalization: Pressure-Independent Cooling	65
7	Conclusions and Perspectives	66
7.1	Summary of Key Findings	66
7.1.1	Phase 1: Impact of Upstream Piping and Flow Distortion	66
7.1.2	Phase 2: Annular Ejector Dynamics and Thermal Mixing	67
7.1.3	Methodological Considerations and Mesh Sensitivity	67
7.2	Operational Envelope and Predictive Framework	67
7.3	Perspectives for Future Work	68
7.3.1	Alternative Layout: Single Orifice Plate Configuration	68
7.3.2	Foundation for a 1D Digital Twin	68
7.3.3	Experimental Commissioning and Model Validation	68
A	Solver and HPC Configuration	69
A.1	Technical Rationales for Solver Settings	69
A.2	Mesh Format and Data Efficiency	69
A.3	SU2 Configuration File Sample	70
B	High-Performance Computing Setup	71
B.1	Slurm Submission Script	71
B.2	Technical Rationale for the Workflow	72

List of Figures

1.1	The BeCOVER aerodynamic test center infrastructure in Herstal. [3]	3
1.2	CAD view of the downstream piping section highlighting the elbow and the diaphragm location.	4
2.1	General view of the BeCOVER aerodynamic test bench.	6
3.1	Development of the velocity profile from the pipe entrance to the fully developed region [4].	9
3.2	Distribution of modified turbulent viscosity and velocity profile in a wall-bounded flow using the Spalart–Allmaras model [5].	11
3.3	Radial pressure gradient and centrifugal force distribution in a 90-degree elbow [6].	12
3.4	Cross-sectional view of the secondary flow showing the pair of counter-rotating Dean vortices [7].	13
3.5	Static pressure distribution across an orifice plate, showing the vena contracta and permanent pressure loss [8].	14
3.6	Illustration of the sonic choking plateau using absolute mass flow [10]. In standard aerodynamic practice, such curves are normalized by the passage area and standard inlet conditions to ensure universal scalability.	16
3.7	Velocity profile evolution and momentum exchange in a jet ejector, illustrating the development of the mixing layer [12].	18
3.8	Evolution of air dynamic viscosity as a function of temperature according to Sutherland’s law, with markers at the three temperatures considered in the study.	20
4.1	Configuration 1: Reference geometry with a straight supply pipe.	23
4.2	Configuration 2: Geometry including the 90-degree elbow for distortion analysis.	23
4.3	Global CAD view of the complete Configuration 3 assembly.	24
4.4	Detailed view of the 4-hole restrictor geometry.	24
4.5	Zoom on the cooling funnel (suction chamber) where ambient air entrainment occurs.	25
4.6	Radial cross-section of the supply pipe showing the T-Rex prismatic layers along the circumference and the transition to the core flow.	26
4.7	Surface mesh of the 4-hole diaphragm initialized with the Advancing Front Ortho algorithm to ensure orthogonal cell distribution around the orifices.	27
4.8	Longitudinal cross-sections of the structured mesh around the restrictor plate. The flow direction is oriented from bottom (upstream) to top (downstream).	28
4.9	Detailed view of the 90-degree elbow showing the transition from structured hex-dominant pipes to the unstructured tetrahedral mesh within the bend.	28
4.10	Visualization of the volumetric sources used to maintain high mesh density along the primary jet paths within the divergent section.	29
4.11	Global longitudinal cross-section of the mesh	30

4.12	Spatial distribution of boundary conditions for Configuration 3: Yellow (Inlets), Orange (Walls), Green (Outlet), and Blue/Violet (Farfield).	31
4.13	View of the physical NIC5 supercomputer racks housed at the University of Liège.	32
4.14	Global network architecture of the NIC5 cluster illustrating the high-speed Infini-Band HDR100 topology [14].	33
4.15	Technical schematic of a NIC5 compute node highlighting the dual AMD EPYC socket configuration and memory interface [14].	34
4.16	Evolution of the RANS residuals showing a convergence level of 10^{-6} for the final simulation stages.	35
5.1	Velocity magnitude contours along the straight supply lines for Configuration 1.	37
5.2	Radial velocity profile at the diaphragm inlet demonstrating perfect flow symmetry.	38
5.3	Specific reduced mass flow ($\dot{m}_{spec}$) as a function of the pressure ratio Π for Configuration 1. The normalization by the passage area A demonstrates the universality of the restrictor's performance across the thermal series.	39
5.4	Discharge coefficient (C_d) evolution for the baseline configuration.	40
5.5	Velocity magnitude contours in a transverse plane immediately downstream of the diaphragm for Configuration 1 ($\Pi = 1.50$, $T = 288.15$ K).	41
5.6	Longitudinal velocity magnitude contours downstream of the diaphragm for Configuration 1.	42
5.7	Zoomed-in Surface LIC visualization of the recirculation zones and toroidal vortices immediately downstream of the diaphragm plate.	42
5.8	Longitudinal static pressure contours in the 90-degree elbow for Configuration 2, showing the centrifugal pressure gradient between the inner wall and the outer wall.	43
5.9	Cross-sectional visualization of Dean vortices at a 45° slice for Configuration 2, combining Surface LIC and velocity Glyphs.	44
5.10	Flow distortion at the diaphragm inlet for Configuration 2: Velocity magnitude contours (left) and radial velocity profile (right).	44
5.11	Comparison of specific reduced mass flow ($\dot{m}_{spec}$) between Configuration 1 (Straight) and Configuration 2 (Elbow) at 288.15 K.	45
5.12	Discharge coefficient (C_d) comparison between Configuration 1 and Configuration 2.	46
5.13	Longitudinal velocity magnitude contours downstream of the diaphragm for Configuration 2.	47
5.14	Specific reduced mass flow ($\dot{m}_{spec}$) as a function of the pressure ratio Π for three operating temperatures (Configuration 2).	48
5.15	Discharge coefficient (C_d) as a function of the pressure ratio for 288.15 K, 600 K, and 900 K.	49
6.1	Three-dimensional isometric view of the assembly, showing the longitudinal cutting plane location used for the extraction of the 2D flow fields.	51
6.2	Global longitudinal velocity magnitude (0–1000 m/s) showing the high-energy primary expansion.	51
6.3	Filtered longitudinal velocity (0–300 m/s) highlighting the wall-hugging jets and the large central recirculation zone.	52
6.4	Surface LIC zoom at the diaphragm exit, showing localized recirculation zones at the divergent entry corners.	52
6.5	Interaction between primary jets and secondary air at low scale (0–100 m/s).	53
6.6	Velocity magnitude contours of the primary core (0–190 m/s) on a transverse cross-section, showing the persistent "twin-spot" asymmetry.	53

6.7	Velocity magnitude contours of the secondary suction (0–10 m/s) on a transverse cross-section, demonstrating a uniform entrainment ring.	54
6.8	Surface LIC visualization of the in-plane velocity vectors on the transverse cross-section, highlighting secondary air convergence and the central vertical updraft topology.	54
6.9	Discharge coefficient C_d vs. inlet total pressure, showing the stability at 0.83 and the comparison with Phase 1 results.	56
6.10	Evolution of primary and secondary specific reduced mass flows vs. inlet total pressure, demonstrating the choked flow plateau and entrainment stability.	56
6.11	Comparison of the raw entrainment ratio (ω) between 288.15 K and 900 K. The ratio jumps from 0.62 to 1.00 primarily due to the reduction in driving primary mass flow.	58
6.12	Corrected entrainment ratio (ω_{corr}) showing the similarity breakdown. The 8% drop at high temperature reveals a degradation in momentum transfer efficiency.	59
6.13	Non-dimensional primary mass flow (Master Curve). The perfect collapse of the cold and hot datasets allows for a universal prediction of air consumption using the specific reduced mass flow scaling law.	60
6.14	Linear interpolation of the corrected entrainment ratio ω_{corr} . The regular spacing between curves represents the assumption of a linear degradation of mixing efficiency with temperature.	61
6.15	Final Reconstructed Performance Map (ω vs P_{in}). The non-linear spacing between curves demonstrates the respect of the $\sqrt{T}$ scaling law for momentum-driven suction.	62
6.16	Longitudinal total temperature distribution (T_t) from the ejector to the discharge pipe outlet.	63
6.17	Detailed view of the thermal shear layer at the ejector exit, highlighting the protective secondary air "shielding" the walls.	63
6.18	Evolution of total temperature radial distribution along the discharge pipe expressed in non-dimensional pipe diameters.	64

Chapter 1

Introduction

1.1 Context

The global aviation industry is currently facing its most significant transformation since the beginning of the jet age. For decades, the primary drivers of aeronautical engineering were speed, range, and capacity. However, the 21st century has shifted the paradigm toward environmental sustainability and carbon neutrality. This transition is not merely an evolution of existing technologies but a radical transformation of engine architectures and energy management.

This shift is largely dictated by international and regional regulatory frameworks. At the European level, the Flightpath 2050 goals, established by the Advisory Council for Aviation Research and Innovation in Europe (ACARE) [1][2], set extremely ambitious targets: a 75 percent reduction in CO₂ emissions per passenger kilometer, a 90 percent reduction in NO_x emissions, and a 65 percent reduction in perceived noise compared to the levels of the year 2000. These goals are further reinforced by the industry-wide commitment to achieve Net Zero carbon emissions by 2050, a pledge supported by major airlines and manufacturers under the guidance of the International Air Transport Association (IATA) and the International Civil Aviation Organization (ICAO).

To meet these requirements, engine manufacturers are forced to look beyond the incremental improvements of conventional turbofans. Current engines, such as the CFM LEAP, have reached a high level of maturity in terms of bypass ratio and thermal efficiency. To achieve the next leap in performance, a technological rupture is necessary. One of the most prominent initiatives in this regard is the RISE (Revolutionary Innovation for Sustainable Engines) program led by CFM International, which is the long-standing joint venture between Safran Aircraft Engines and GE Aerospace.

The RISE program serves as a technology demonstrator for the next generation of propulsion systems. The centerpiece of this program is the Open Fan architecture, which removes the traditional engine casing to allow for a significantly larger fan diameter. By doing so, the engine can achieve bypass ratios far exceeding those of current ducted engines, which is the primary lever for improving propulsive efficiency and reducing fuel consumption. Additionally, the RISE program explores hybrid-electric capabilities and the full integration of Sustainable Aviation Fuels (SAF) and hydrogen.

While the RISE program is a prominent example, it is part of a broader industry-wide technological race. Other major engine manufacturers are actively pursuing comparable validation paths, notably Rolls-Royce with the UltraFan architecture and RTX (Pratt & Whitney) with the GTF Advantage program.

All these revolutionary designs involve much higher pressure ratios and operating temperatures in the engine core to maintain thermodynamic efficiency. This evolution directly impacts

the requirements for ground testing. Validating the High-Pressure Compressor (HPC) for such architectures requires test facilities capable of handling extreme conditions that go beyond the original design limits of existing infrastructure. This is precisely where the HPCo-Upgrade project at BeCOVER becomes essential. By adapting the test bench to these new thermal and aerodynamic demands, BeCOVER ensures that the core components of the engines of tomorrow can be tested with the precision and safety required by such ambitious environmental goals.



Figure 1.1: The BeCOVER aerodynamic test center infrastructure in Herstal. [3]

1.2 Objectives of the Internship

The primary mission of this internship is to contribute to the technical validation of the HPCo-Upgrade project. This project represents a critical step in the evolution of the BeCOVER facility, as it aims to verify if the existing test bench infrastructure can safely withstand the significantly increased temperatures and pressures associated with High-Pressure Compressor testing. While the bench was initially sized for the aerodynamic loads of Low-Pressure Compressors, the transition to high-pressure ratios implies a major shift in the thermodynamic environment. Specifically, the work focuses on the aerodynamic and thermal behavior of the downstream piping system (see Figure 1.2), often referred to as the collector, located immediately after the test article.

The study is structured around several key technical objectives that follow the standard numerical simulation workflow, from geometry preparation to physical analysis.

The first major objective is the preprocessing and meshing phase. This step is fundamental, as the accuracy of any Computational Fluid Dynamics (CFD) simulation is directly linked to the quality of the underlying grid. Using Pointwise, the goal is to generate high-quality computational grids for a complex section of the downstream pipe. The geometry is particularly challenging because it includes a 90-degree elbow followed by a diaphragm, or orifice plate. This configuration requires a mesh that is fine enough to capture the boundary layer development along the pipe walls, the flow separation and secondary flows (such as Dean vortices) induced by the elbow, and the high-pressure gradients occurring across the diaphragm. Achieving a grid with proper cell orientation and stretching ratios is essential to ensure the convergence and stability of the subsequent calculations.

The second objective consists of a detailed aerodynamic analysis using the SU2 flow solver. The focus here is to investigate how the flow disturbances generated by the upstream elbow affect the velocity profile before it reaches the diaphragm. In ideal conditions, a diaphragm

requires a fully developed and uniform flow to provide accurate measurements. However, the proximity of the elbow introduces swirl and asymmetry. Since the characteristic curve of the diaphragm—which relates the pressure drop to the mass flow rate—is highly sensitive to flow uniformity, it is crucial to quantify the impact of these disturbances. By using SU2 to simulate various operating points, the study aims to determine the deviation from theoretical performance and to verify if the flow measurement remains within acceptable industrial tolerances under high-pressure conditions.

Finally, the project includes a comprehensive thermal assessment of the system. In the context of HP testing, the air exiting the compressor reaches temperatures that are significantly higher than those encountered in LP tests. It is vital to analyze the gas temperature distribution throughout the pipe and at the final exit of the exhaust system. This assessment is not only about the fluid behavior but also about ensuring the structural integrity of the facility. Predicting hot spots where the high-velocity gas impacts the pipe walls is necessary to prevent thermal fatigue or mechanical failure of the seals and support structures. This thermal mapping provides the necessary data to decide whether the current materials and cooling strategies of the BeCOVER bench are sufficient or if further modifications are required to complete the HP upgrade safely.

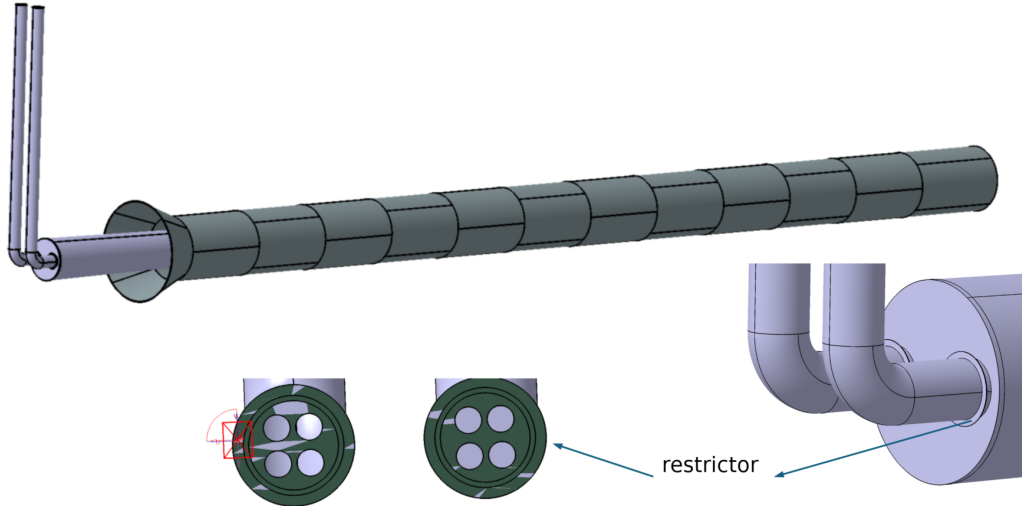


Figure 1.2: CAD view of the downstream piping section highlighting the elbow and the diaphragm location.

Chapter 2

Company Presentation

2.1 BeCOVER: A Strategic Joint Venture

BeCOVER is a cutting-edge aerodynamic test center specializing in aircraft engine compressors, located in Herstal, Belgium. Its creation marks a major milestone for the European aerospace industry, as it provides a sovereign and independent facility for high-level turbomachinery validation. The center was established through a strategic public-private partnership, taking the form of a joint venture between three major stakeholders: Safran Aero Boosters holding 25 percent, the Walloon Region via its investment arm Wallonie Entreprendre holding 50 percent, and the Belgian Federal State via the SFPIM holding 25 percent.

This partnership reflects a shared ambition to anchor high-tech industrial activity and high-value employment in the Liège region. By combining private industrial expertise with public investment, BeCOVER serves as a catalyst for innovation in the regional aerospace cluster. Representing an investment of approximately 50 million euros, the facility covers a total area of 3,000 square meters. Its construction and internal infrastructure were specifically designed to handle the massive mechanical and thermal loads generated by the next generation of aircraft engines. The choice of Herstal for its location is also strategic, placing it at the heart of an aerospace hub with close proximity to major industrial partners, specialized suppliers, and world-class research institutions.

What makes BeCOVER unique in Europe is its ability to perform Scale 1, or full-scale, testing within a closed air loop system. Most conventional test benches operate in an open circuit, which often limits their ability to control inlet conditions or requires massive amounts of energy to continuously treat new incoming air. BeCOVER's closed-loop architecture allows engineers to simulate a complete flight envelope with extreme precision, reproducing various atmospheric conditions from sea level up to high altitudes. This capability is essential for investigating the aerodynamic stability and efficiency of compressors under the low-pressure and low-temperature conditions encountered during cruise flight, as well as the high-pressure regimes of take-off and climb.

Furthermore, BeCOVER is designed as an open and independent test center. While Safran Aero Boosters is a key industrial partner and founder, the facility is strategically independent and accessible to all major Original Equipment Manufacturers and research institutions worldwide. By offering a multi-OEM platform, BeCOVER fosters a unique ecosystem where academic research and industrial application meet. This independence ensures that the most advanced aerodynamic concepts can be validated with extreme precision before being integrated into the engines of the future, all while maintaining strict confidentiality for each customer and their proprietary designs through isolated data management systems.

2.2 Technical Capabilities

The heart of the BeCOVER infrastructure is its massive 30 MW electric motor (see Figure 2.1). This power level makes it one of the most capable facilities in Europe, allowing for the testing of full-scale compressor components. By testing at a 1:1 scale, engineers can avoid the Reynolds number scaling effects that often complicate the transition from small-scale models to real engine parts. The motor is capable of driving compressors at rotational speeds reaching up to 25,000 RPM, which is essential to match the high-pressure ratios and peripheral speeds found in the latest generations of high-performance turbomachinery.

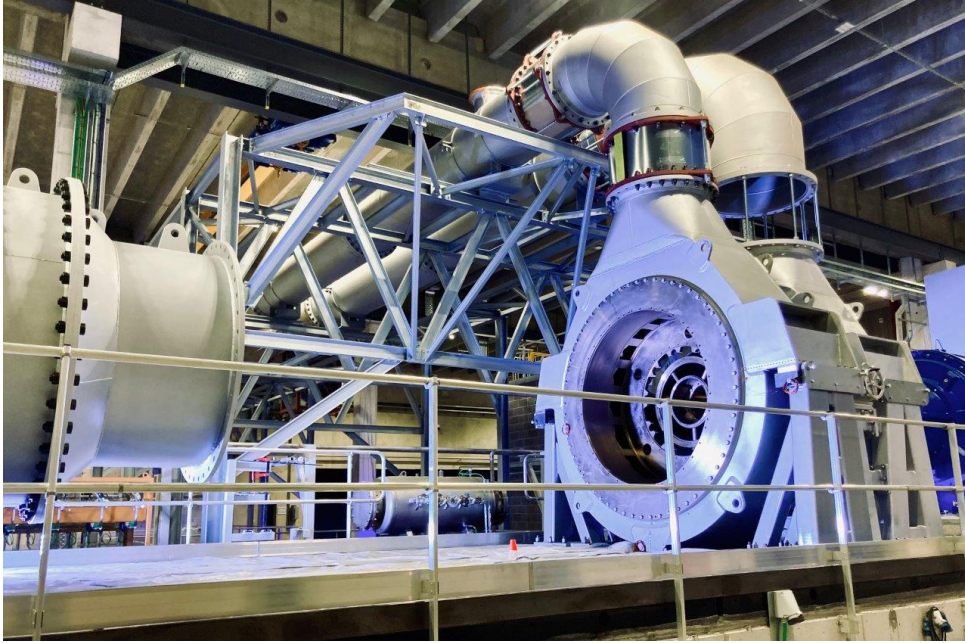


Figure 2.1: General view of the BeCOVER aerodynamic test bench.

One of the most advanced technical features of the facility is the implementation of a closed air loop. Unlike open-circuit benches, the closed loop allows for the precise regulation of the thermodynamic state of the air. This is achieved through a complex system of heat exchangers and pressure control valves that can adjust the air density and temperature at the compressor inlet. By controlling these parameters, BeCOVER can simulate a wide range of flight envelopes, from sea-level take-off conditions to high-altitude cruise at different Mach numbers. This capability is crucial for validating the aerodynamic stability and efficiency of a compressor across its entire operating map without the need for actual flight testing, which is significantly more expensive and restrictive.

Furthermore, the facility was designed with a strong focus on environmental integration and operational safety. The semi-underground architecture of the building is a key engineering choice. It provides natural thermal stability for the test environment, which is beneficial for the precision of sensitive aerodynamic measurements. Combined with advanced acoustic insulation and specialized silencers, this design ensures a noise attenuation of approximately 77 dB. This minimizes the acoustic impact on the surrounding area in Herstal, even when the 30 MW motor is operating at full capacity. This focus on sustainability and community integration was a core requirement during the design phase of the infrastructure.

2.3 The ULiège Partnership

A key aspect of BeCOVER is its deep integration into the Belgian aerospace ecosystem, particularly with the University of Liège. This internship is physically hosted within the ULiège campus at Sart Tilman, specifically within the Aerospace and Mechanical Engineering department. This collaboration is part of the SOCRATE partnership, which stands for Scientific Organization for Compressor Research And Technology for a Better Environment. This initiative brings together BeCOVER, ULiège, Cenaero, and the von Karman Institute to share expertise and resources.

This synergy allows for a direct bridge between academic excellence in Computational Fluid Dynamics and industrial-scale experimental testing. The university provides the high-performance computing resources and the theoretical framework necessary for complex simulations using solvers like SU2, while BeCOVER provides the experimental validation data. This feedback loop between numerical prediction and experimental reality is essential for fostering innovation and developing the next generation of sustainable aviation technologies. Working within the university environment also allows for constant exchange with researchers and professors specialized in turbomachinery, providing a rich academic context to the industrial challenges of the HPCo-Upgrade project.

Chapter 3

Theoretical Framework

This chapter establishes the physical principles and mathematical frameworks necessary to analyze the aerodynamic and thermal behavior of the High-Pressure collector. The study transitions from fundamental governing equations to the complex interactions within the exhaust system, specifically focusing on internal flow singularities, turbulence modeling, and momentum transfer in ejector systems.

3.1 Governing Equations of Fluid Dynamics

The aerodynamic behaviour of the High-Pressure collector is governed by the compressible Navier–Stokes equations. These equations express the conservation of mass, momentum, and energy for a viscous, heat-conducting fluid. In the context of the HPCo-Upgrade, where strong pressure gradients, high temperatures, and potential choking phenomena occur, a fully compressible formulation is required to accurately capture the flow physics.

3.1.1 Conservation of Mass

The continuity equation ensures that mass is conserved within the control volume. For a compressible flow, it is written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \quad (3.1.1)$$

where ρ is the density and $\mathbf{u}$ the velocity vector. In the collector, density variations are significant due to the acceleration through the diaphragm and the thermal expansion of the hot primary jet.

3.1.2 Conservation of Momentum

The momentum equations express Newton’s second law applied to a fluid element:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau}, \quad (3.1.2)$$

where p is the static pressure and $\boldsymbol{\tau}$ the viscous stress tensor. These terms govern the prediction of pressure losses, wall shear stresses, and the secondary flows generated in the 90° elbow.

3.1.3 Conservation of Energy

The total energy equation accounts for compressibility, viscous dissipation, and heat transfer:

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot [(\rho E + p)\mathbf{u}] = \nabla \cdot (k\nabla T + \mathbf{u} \cdot \boldsymbol{\tau}), \quad (3.1.3)$$

where E is the total energy per unit mass, T the static temperature, and k the thermal conductivity. This equation is essential for predicting the thermal mixing between the hot primary jet and the entrained ambient air inside the ejector.

3.1.4 Equation of State

To close the system, air is modelled as a calorically perfect gas:

$$p = \rho RT, \quad (3.1.4)$$

with $R = 287.05 \text{ J kg}^{-1} \text{ K}^{-1}$. As temperature increases, density decreases, leading to higher velocities for a given mass flow rate. This effect is central to the sensitivity analysis performed at 288 K, 600 K, and 900 K.

3.1.5 Boundary Layer Development

As the flow progresses along the pipe walls, the no-slip condition induces the formation of a boundary layer in which viscous effects dominate. In turbulent regimes, this layer is composed of:

- a viscous sublayer close to the wall,
- a buffer region,
- a logarithmic region where the velocity profile follows the classical log-law.

The boundary layer thickness grows with downstream distance until a fully developed velocity profile is reached (see Figure 3.1). This behaviour strongly influences wall shear stress, pressure losses, and the accuracy of CFD simulations, which require appropriate near-wall mesh resolution.

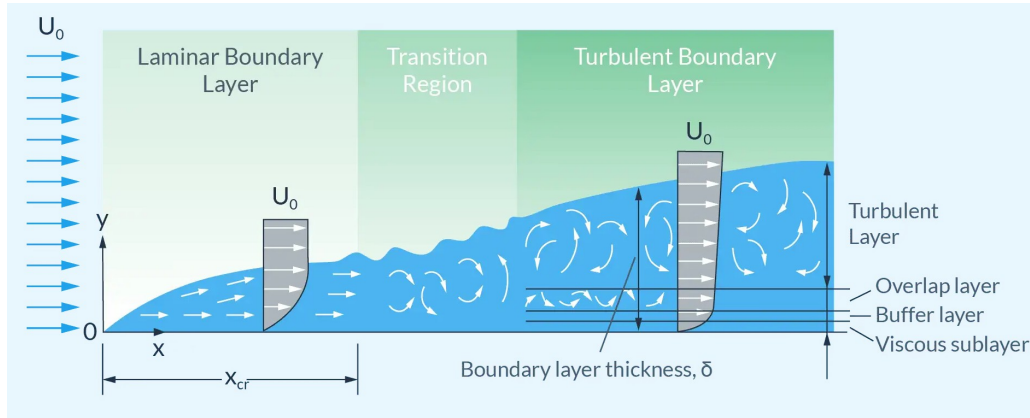


Figure 3.1: Development of the velocity profile from the pipe entrance to the fully developed region [4].

3.2 Turbulence Modeling

The flow inside the High-Pressure collector operates at Reynolds numbers of several millions, placing it firmly in the fully turbulent regime. In such conditions, the instantaneous motion of the fluid is dominated by chaotic, three-dimensional fluctuations spanning a wide range of spatial and temporal scales. Resolving all these scales directly through Direct Numerical Simulation (DNS) is computationally prohibitive for industrial geometries such as the exhaust line. A statistical approach is therefore required to model the effect of turbulence on the mean flow.

3.2.1 The RANS Approach and the Closure Problem

The Reynolds-Averaged Navier–Stokes (RANS) formulation decomposes each instantaneous quantity ϕ into a mean component $\bar{\phi}$ and a fluctuating component ϕ' :

$$\phi = \bar{\phi} + \phi'. \quad (3.2.1)$$

Applying this decomposition to the momentum equations introduces additional terms known as the Reynolds stresses, which represent the momentum transfer induced by turbulent fluctuations. These stresses take the form of a symmetric tensor:

$$-\rho \overline{u'_i u'_j}, \quad (3.2.2)$$

which adds six new unknowns to the system, creating the so-called *closure problem*.

To close the equations, the Boussinesq hypothesis is commonly used. It assumes that the turbulent stresses behave like an additional viscosity, the turbulent (eddy) viscosity μ_t , leading to:

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}, \quad (3.2.3)$$

where $k = \frac{1}{2} \overline{u'_k u'_k}$ is the turbulent kinetic energy and δ_{ij} is the Kronecker delta.

The role of a turbulence model is therefore to provide a transport equation or an algebraic relation for μ_t , and in some cases for additional variables such as k .

This approach drastically reduces the computational cost compared to scale-resolving methods such as LES or DNS, making RANS particularly suitable for large-scale industrial configurations like the present collector geometry.

However, the Boussinesq hypothesis relies on the assumption of isotropic turbulence, which may not hold in flows with strong anisotropy, curvature effects, or complex shear-layer interactions. As a result, RANS models may have limited accuracy in predicting highly three-dimensional and unsteady phenomena such as jet mixing, secondary flows, and recirculation regions encountered in the present study.

3.2.2 The Spalart–Allmaras (SA) Model

As discussed in the previous section, the closure of the RANS equations requires the introduction of a turbulence model to provide a formulation for the turbulent viscosity μ_t . In this framework, the Spalart–Allmaras (SA) model is selected. It is a one-equation eddy-viscosity model, based on the Boussinesq hypothesis, which directly provides a transport equation for a modified turbulent kinematic viscosity $\tilde{\nu}$.

Originally developed for external aerodynamics and wall-bounded flows, the SA model is particularly well suited for turbomachinery applications and internal ducts with strong adverse pressure gradients, as encountered in the present study.

The model solves a single transport equation for a modified turbulent kinematic viscosity $\tilde{\nu}$:

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = C_{b1} \tilde{S} \tilde{\nu} + \frac{1}{\sigma} [\nabla \cdot ((\nu + \tilde{\nu}) \nabla \tilde{\nu}) + C_{b2} (\nabla \tilde{\nu})^2] - C_{w1} f_w \left(\frac{\tilde{\nu}}{d} \right)^2. \quad (3.2.4)$$

The SA model is chosen for four main reasons:

- **Robustness in adverse pressure gradients:** it provides a stable prediction of boundary layer behavior and captures separation phenomena in the 90° elbow with sufficient accuracy for engineering purposes.

- **Numerical stability:** it converges reliably on large, complex meshes such as the full collector geometry, which contains nearly 28 million cells.
- **Computational efficiency:** as a one-equation model, it offers a favorable compromise between accuracy and computational cost compared to two-equation models such as $k-\omega$ SST.
- **Versatile wall treatment:** the model, as implemented in SU2, features an all- y^+ wall formulation. This allows for the use of semi-empirical wall functions when the near-wall mesh is relatively coarse ($y^+ > 30$), providing a reliable estimation of wall shear stress without the prohibitive cost of a fully wall-resolved approach.

Although more advanced models such as the $k-\omega$ SST model are known to better capture strong separations and complex shear-layer interactions, especially in highly distorted or mixing-dominated flows, the SA model was preferred in this study as a robust and efficient choice for large-scale industrial CFD simulations. The objective is to capture global aerodynamic trends and performance indicators rather than highly detailed turbulent structures.

However, it is important to highlight the main limitations of the SA model. As a Reynolds-Averaged Navier–Stokes (RANS) approach, it relies on turbulence modeling assumptions and provides time-averaged solutions. Consequently, it may have limited accuracy in flows involving:

- strong unsteady effects,
- highly anisotropic turbulence,
- complex jet interactions and mixing layers, such as those present downstream of the multi-hole restrictor and in the ejector.

Despite these limitations, the SA model remains a widely accepted and reliable choice for engineering applications where robustness, convergence, and computational cost are critical.

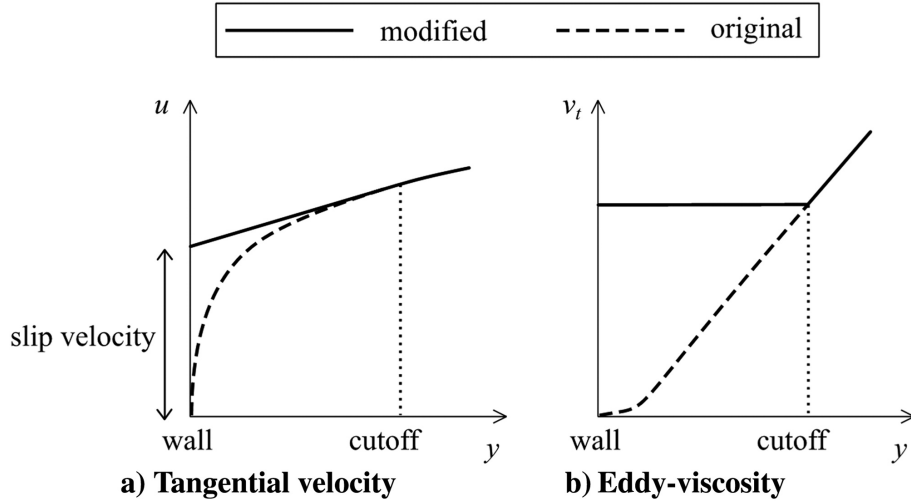


Figure 3.2: Distribution of modified turbulent viscosity and velocity profile in a wall-bounded flow using the Spalart–Allmaras model [5].

3.3 Physics of Pipe Bends

The 90-degree elbow located upstream of the diaphragm plays a central role in the aerodynamic behaviour of the High-Pressure collector. When a high-speed flow is forced to follow

a curved path, centrifugal effects, pressure imbalances, and secondary motions develop within the cross-section. These mechanisms generate persistent distortions that strongly influence the downstream velocity profile and, consequently, the performance of the restrictor plate.

3.3.1 Centrifugal Forces and Radial Pressure Gradients

As the flow enters the bend, fluid particles experience a centrifugal acceleration directed toward the outer radius of curvature. To balance this acceleration, a radial pressure gradient must be established across the pipe section:

$$\frac{\partial p}{\partial r} = \frac{\rho v^2}{R_c}, \quad (3.3.1)$$

where r is the radial coordinate and R_c the curvature radius of the elbow.

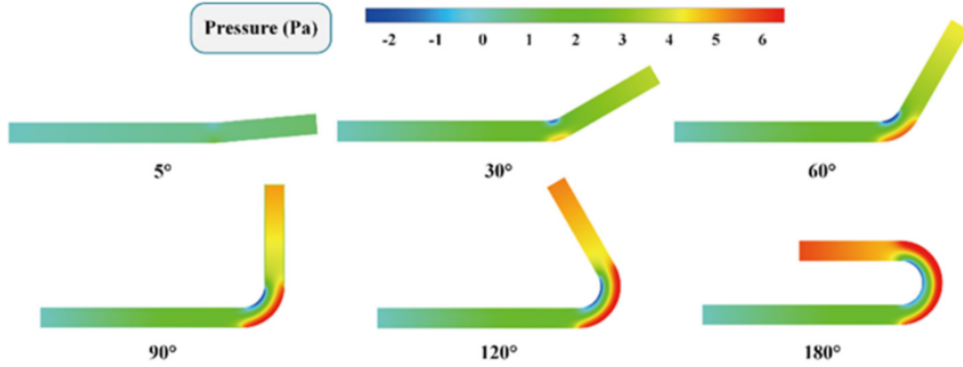


Figure 3.3: Radial pressure gradient and centrifugal force distribution in a 90-degree elbow [6].

This mechanism produces a high-pressure region at the extrados and a low-pressure region at the intrados. The resulting pressure imbalance is responsible for boundary-layer separation on the inner wall and for the onset of secondary flows.

3.3.2 Secondary Flows: Dean Vortices

The radial pressure gradient does not affect all fluid layers equally. The high-momentum core flow is able to resist the outward pressure force, whereas the slower boundary-layer fluid near the walls cannot. This imbalance induces a transverse motion from the extrados toward the intrados along the upper and lower walls.

These motions form a pair of counter-rotating vortical structures known as *Dean vortices*. Their intensity is characterized by the Dean number:

$$De = Re \sqrt{\frac{D}{2R_c}}, \quad (3.3.2)$$

where D is the pipe diameter. At the Reynolds numbers encountered in the HPCo-Upgrade, these vortices are strong, stable, and significantly modify the internal momentum distribution.

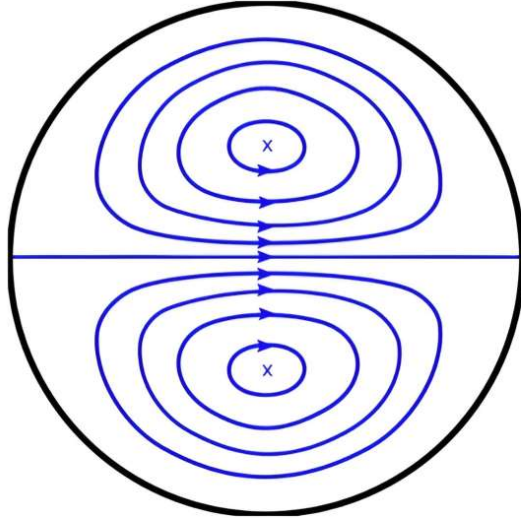


Figure 3.4: Cross-sectional view of the secondary flow showing the pair of counter-rotating Dean vortices [7].

3.3.3 Impact on Velocity Profiles and Downstream Distortion

The combined effect of centrifugal forces and Dean vortices leads to a highly asymmetric axial velocity profile at the elbow exit. The high-speed core is displaced toward the extrados, while the intrados region exhibits lower velocities and possible recirculation.

Moreover, the helical motion induced by the vortices generates a residual swirl that persists far downstream. In straight pipes, the flow typically requires several tens of diameters to return to a fully developed state. In the present configuration, the distance between the elbow and the diaphragm is insufficient to allow re-establishment of a uniform profile. As a result, the restrictor plate is subjected to a distorted, swirling inflow, which directly affects its discharge coefficient and the predictability of the pressure drop.

Understanding and quantifying this distortion through CFD is therefore essential for the accurate characterization of the collector's aerodynamic performance.

3.4 Orifice Plates and Multi-perforated Restrictors

In the BeCOVER High-Pressure collector, orifice plates (or diaphragms) are used as throttling elements to impose a controlled pressure drop. Unlike standard metering orifices, these components operate under high thermal and mechanical loads and must accommodate distorted inflow conditions generated by the upstream elbow. Their behaviour is therefore strongly influenced by compressibility effects, jet interactions, and installation geometry.

3.4.1 Pressure Drop Mechanism and Vena Contracta

As the flow approaches the orifice, the sudden reduction in cross-sectional area forces the fluid to accelerate. According to Bernoulli's principle, this acceleration is accompanied by a drop in static pressure. The minimum effective flow area is not located at the physical opening but slightly downstream, at the *vena contracta*, where the jet reaches its maximum velocity.

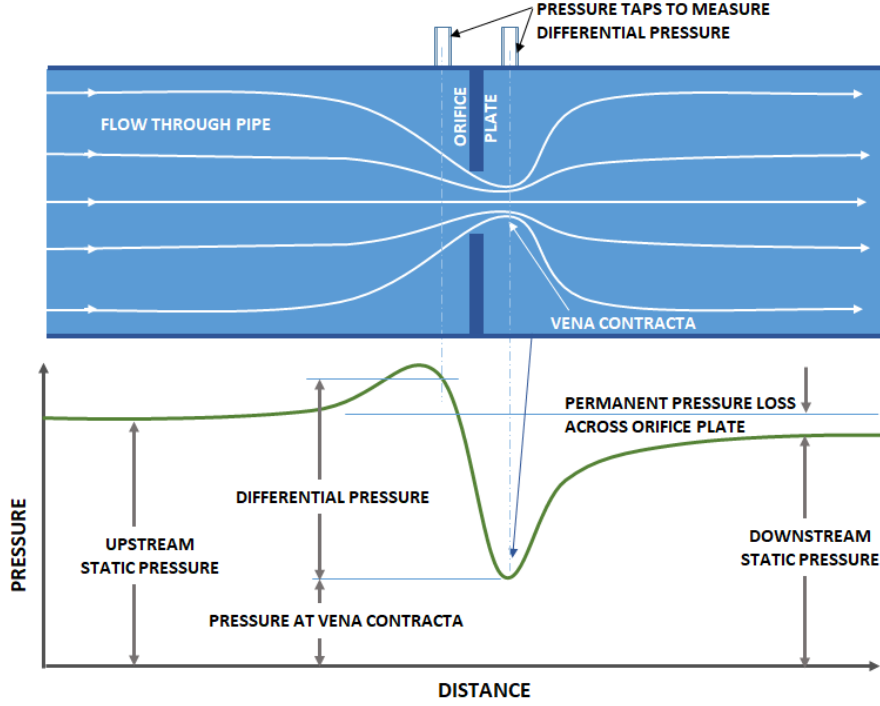


Figure 3.5: Static pressure distribution across an orifice plate, showing the vena contracta and permanent pressure loss [8].

The permanent pressure loss across the plate is expressed using the loss coefficient K :

$$\Delta P = K \cdot \frac{1}{2} \rho v^2, \quad (3.4.1)$$

where v is the mean upstream velocity. Accurately predicting this loss is essential for determining the back-pressure applied to the compressor under test.

3.4.2 Multi-hole Jet Interaction and Coalescence

The restrictors used in the HPCo-Upgrade feature four perforations instead of a single concentric opening. This configuration generates four individual high-speed jets that interact downstream of the plate.

In the near-field region, the shear layers of the jets merge, producing a highly turbulent mixing zone. This process, known as *jet coalescence*, leads to:

- redistribution of momentum across the pipe cross-section,
- enhanced mixing and turbulence levels,
- potential asymmetries if the upstream flow is distorted,
- increased sensitivity of the discharge coefficient to inflow conditions.

Such multi-jet behaviour cannot be captured by classical single-orifice correlations and therefore requires CFD to determine the effective mass flow characteristics.

3.4.3 ISO 5167 Compliance and Geometric Limitations

The ISO 5167-1:2022 standard provides empirical correlations for flow measurement using orifice plates. However, it imposes strict requirements on the upstream flow quality:

“The primary device shall be installed in the pipeline at a position such that the flow conditions immediately upstream of the primary device approximate to those of swirl-free, fully developed pipe flow.” [9]

For a 90-degree elbow, the standard recommends an upstream straight length of 30 to 44 pipe diameters to limit measurement uncertainty below 0.5%. In the present configuration, the available straight length is significantly shorter, and the use of a four-hole geometry deviates from the standard concentric orifice model.

As a result, ISO 5167 correlations cannot be applied directly, and CFD becomes necessary to:

- quantify the actual discharge coefficient,
- evaluate the impact of swirl and velocity distortion,
- recalibrate the restrictor behaviour under high-pressure conditions.

3.4.4 Compressibility and Sonic Choking

At high pressure ratios, compressibility effects dominate the flow through the restrictor. When the Mach number at the throat reaches unity, the flow becomes choked, meaning that further reductions in downstream pressure no longer increase the mass flow rate.

For a perfect gas with $\gamma = 1.4$, the critical pressure ratio is:

$$\frac{P_{\text{total}}}{P^*} = \left(\frac{\gamma + 1}{2} \right)^{\frac{\gamma}{\gamma - 1}} \approx 1.893. \quad (3.4.2)$$

Beyond this point, the mass flow rate depends solely on the upstream stagnation conditions. This phenomenon explains the plateau observed in the numerical flow curves around a pressure ratio of 1.9.

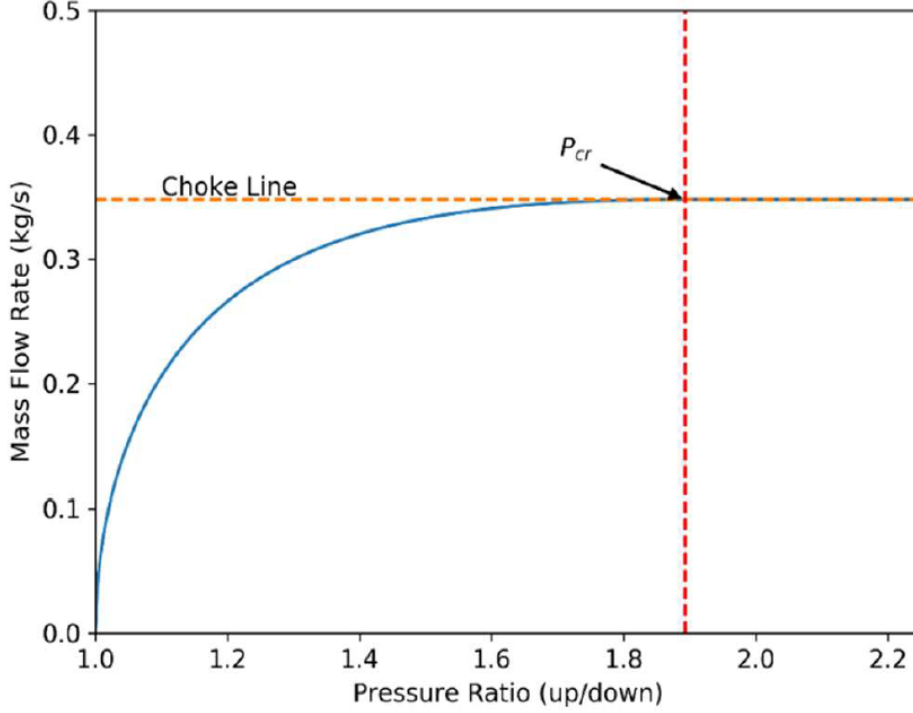


Figure 3.6: Illustration of the sonic choking plateau using absolute mass flow [10]. In standard aerodynamic practice, such curves are normalized by the passage area and standard inlet conditions to ensure universal scalability.

While Figure 3.6 illustrates the fundamental principle of sonic choking using absolute mass flow rates for a specific experimental setup, this dimensional representation is tied to a single geometric scale. To decouple the aerodynamic performance from the physical dimensions of the conduit and the absolute thermodynamic state, standard industrial practice requires expressing the flow as a specific reduced mass flow ($\dot{m}_{spec} = \frac{\dot{m}\sqrt{T_{t,in}}}{P_{t,in}A}$). This rigorous normalization is the approach adopted for all subsequent analyses in this study to ensure the universality of the BeCOVER performance models.

3.4.5 Discharge Coefficient and Flow Capacity

To provide a rigorous comparison between the different configurations and operating temperatures, it is essential to establish a theoretical reference for the mass flow rate. The performance of a restrictor is characterized by the discharge coefficient (C_d), which is the ratio of the actual mass flow rate ($\dot{m}_{real}$) to the ideal mass flow rate ($\dot{m}_{ideal}$) obtained under isentropic conditions:

$$C_d = \frac{\dot{m}_{real}}{\dot{m}_{ideal}} \quad (3.4.3)$$

According to Anderson [11], the mass flow rate through a section A is fundamentally linked to the local Mach number (M) and the stagnation properties ($P_{t,in}$, $T_{t,in}$). The general formulation is given by:

$$\dot{m} = \frac{P_{t,in}A}{\sqrt{T_{t,in}}} \sqrt{\frac{\gamma}{R}} M \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{\gamma+1}{2(\gamma-1)}} \quad (3.4.4)$$

To highlight the physical dependencies of the flow, Equation 3.4.4 can be rewritten by isolating the influence of the upstream stagnation conditions from the local kinematic state. This decomposition expresses the mass flow rate as the product of a reference mass flow and a non-dimensional function of the Mach number:

$$\dot{m} = \underbrace{\left(\frac{P_{t,in} A}{\sqrt{T_{t,in}}} \sqrt{\frac{\gamma}{R}} \right)}_{\text{Reference Flow (Total conditions)}} \times \underbrace{M \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{\gamma+1}{2(\gamma-1)}}}_{f(M)} \quad (3.4.5)$$

This representation reveals two distinct physical blocks:

1. The first term carries the dimensions of a mass flow rate $[\text{kg} \cdot \text{s}^{-1}]$. It represents the flow potential of the system, which scales linearly with the total pressure and the geometric area, and inversely with the square root of the total temperature.
2. The second term, $f(M)$, is a non-dimensional weighting function. It acts as a cursor that determines how much of the flow potential is effectively realized based on the expansion level (Mach number) of the fluid.

In the BeCOvER collector, the Mach number at the orifice throats results from the pressure ratio $\Pi = P_{t,in}/P_{s,out}$. By utilizing the isentropic relation linking M and Π :

$$M^2 = \frac{2}{\gamma-1} \left[\left(\frac{P_{t,in}}{P_{s,out}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right] \quad (3.4.6)$$

The ideal mass flow rate can be expressed directly as a function of the pressure ratio, yielding the Saint-Venant relation:

$$\dot{m}_{ideal} = A \cdot \frac{P_{t,in}}{\sqrt{T_{t,in}}} \cdot \sqrt{\frac{2\gamma}{R(\gamma-1)}} \cdot \sqrt{\left(\frac{P_{s,out}}{P_{t,in}} \right)^{\frac{2}{\gamma}} - \left(\frac{P_{s,out}}{P_{t,in}} \right)^{\frac{\gamma+1}{\gamma}}} \quad (3.4.7)$$

When the pressure ratio reaches the critical threshold defined by the sonic condition ($M = 1$ at the throat), the flow becomes choked. At this stage, the weighting function $f(M)$ reaches its maximum value, and the mass flow rate becomes independent of the downstream pressure:

$$\dot{m}_{choked,ideal} = \frac{P_{t,in} A}{\sqrt{T_{t,in}}} \sqrt{\frac{\gamma}{R} \left(\frac{2}{\gamma+1} \right)^{\frac{\gamma+1}{\gamma-1}}} \quad (3.4.8)$$

The discharge coefficient C_d serves as a critical scaling parameter for the following physical reasons:

- **Thermodynamic Normalization:** By incorporating the $1/\sqrt{T_{t,in}}$ dependency, the C_d accounts for changes in gas density. For a purely geometric restriction, the C_d curves for all temperatures are expected to collapse onto a single profile.
- **Reynolds Independence:** At high mass flow rates, the flow is characterized by high Reynolds numbers where inertial forces dominate. In this regime, the C_d is primarily governed by the geometry of the orifices and the resulting vena contracta.
- **Viscous Effects:** Any shift in C_d values between temperatures can be attributed to secondary viscous effects. As temperature increases, the air viscosity μ increases (Sutherland's law) while density decreases, leading to a lower Reynolds number. This can induce a slight thickening of the boundary layer, marginally reducing the effective flow area.

3.5 Ejector Theory and Thermal Mixing

The final stage of the High-Pressure collector uses an annular ejector to cool the hot primary exhaust gases by entraining ambient air. Unlike mechanical pumps, ejectors rely solely on momentum transfer between two fluid streams, making them robust and passive devices capable of handling the extreme thermal loads associated with High-Pressure compressor testing.

3.5.1 Mechanics of Entrainment and Mixing Layers

When the high-speed primary jet exits the internal pipe into the larger mixing chamber, a shear layer forms at the interface between the primary flow and the stagnant secondary air. Within this shear layer, turbulent stresses transfer momentum from the primary jet to the surrounding fluid, accelerating the secondary air and generating a region of low static pressure at the ejector inlet. This pressure deficit drives the suction of ambient air through the cooling funnel.

The efficiency of this entrainment process depends strongly on:

- the velocity profile of the primary jet,
- the presence of swirl or distortion inherited from upstream elbows and diaphragms,
- the geometry of the suction chamber.

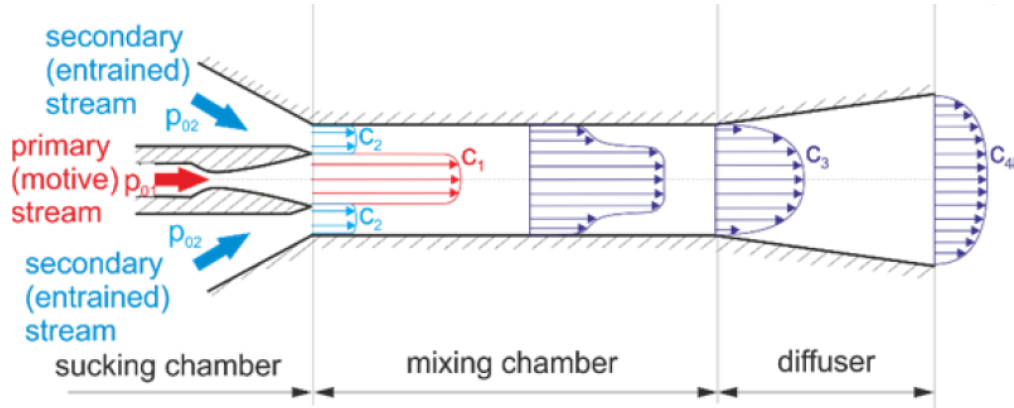


Figure 3.7: Velocity profile evolution and momentum exchange in a jet ejector, illustrating the development of the mixing layer [12].

3.5.2 Thermal Mixing and Energy Conservation

The ejector also acts as a thermal dilutor. The final mixed temperature T_m of the combined streams is obtained from the conservation of energy at constant pressure:

$$(\dot{m}_p + \dot{m}_s)c_p T_m = \dot{m}_p c_p T_p + \dot{m}_s c_p T_s, \quad (3.5.1)$$

where $\dot{m}_p$ and T_p are the mass flow rate and temperature of the primary jet, and $\dot{m}_s$ and T_s those of the entrained secondary air.

Introducing the entrainment ratio ω :

$$\omega = \frac{\dot{m}_s}{\dot{m}_p}, \quad (3.5.2)$$

the mixed temperature becomes:

$$T_m = \frac{T_p + \omega T_s}{1 + \omega}. \quad (3.5.3)$$

A higher entrainment ratio leads to a lower final temperature, reducing thermal loads on the downstream exhaust pipe. In CFD, the modelling of this process requires an appropriate turbulent Prandtl number Pr_t to correctly capture the coupling between momentum and thermal diffusion.

3.5.3 Ejector Geometry and Flow Regions

The ejector of the HPCo-Upgrade consists of two main functional regions:

1. **Suction Chamber:** Located at the cooling funnel inlet, where the primary jet emerges. The low-pressure region created by the jet induces the entrainment of ambient air.
2. **Mixing Pipe:** A long, constant-area duct where momentum and thermal mixing occur. The objective is to achieve a uniform temperature and velocity profile before the final discharge.

The length of the mixing pipe is a critical design parameter. If it is too short, the hot core of the primary jet may persist, creating thermal stratification and potential hot spots on the exhaust stack. The numerical simulations aim to verify that the current length is sufficient to ensure safe thermal homogenization under all operating conditions.

3.6 Thermodynamics and Gas Properties

The transition from conventional low-pressure testing to High-Pressure compressor validation involves a wide range of operating temperatures, from ambient conditions (288 K) up to 900 K at the collector inlet. Such variations significantly affect the thermodynamic and transport properties of air, particularly its density and dynamic viscosity. Understanding these effects is essential for predicting the aerodynamic behaviour of the collector and for evaluating the cooling requirements of the ejector system.

3.6.1 Ideal Gas Law and Compressibility

For the pressure and temperature ranges encountered in the HPCo-Upgrade, air is modelled as a calorically perfect gas. Its thermodynamic state is governed by the ideal gas law:

$$p = \rho RT, \quad (3.6.1)$$

where $R = 287.05 \text{ J kg}^{-1} \text{ K}^{-1}$ is the specific gas constant.

At constant pressure, an increase in temperature leads to a decrease in density. For a given mass flow rate, this implies:

- higher volumetric flow rates,
- higher velocities in the ducts,
- increased mechanical and thermal loads on the piping.

These effects are central to the sensitivity analysis performed at 288 K, 600 K, and 900 K.

3.6.2 Sutherland's Law and Viscosity Variations

The dynamic viscosity of air increases significantly with temperature, a behaviour that must be accounted for when analysing high-temperature compressor flows. This dependence is described by Sutherland's law:

$$\mu(T) = \mu_0 \left(\frac{T}{T_0} \right)^{3/2} \frac{T_0 + S}{T + S}, \quad (3.6.2)$$

where:

- μ_0 is the reference viscosity at temperature T_0 ,
- S is Sutherland's constant.

Figure 3.8 illustrates the evolution of air viscosity over the temperature range relevant to the HPCo-Upgrade, with emphasis on the three operating points investigated in this study (288 K, 600 K, and 900 K).

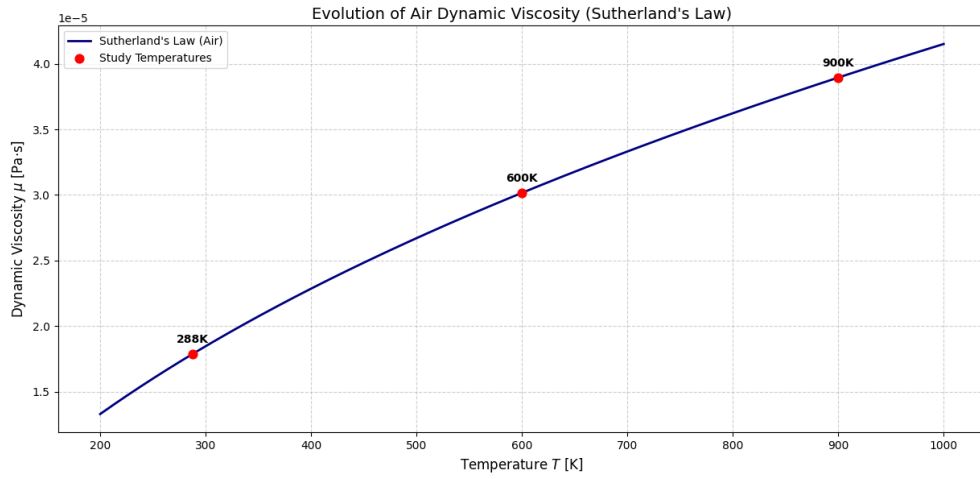


Figure 3.8: Evolution of air dynamic viscosity as a function of temperature according to Sutherland's law, with markers at the three temperatures considered in the study.

As temperature rises:

- viscosity increases,
- the Reynolds number decreases for a given velocity,
- boundary layers become thicker and more diffusive,
- turbulent mixing in the ejector becomes more effective due to enhanced momentum and thermal diffusion.

These trends directly influence the entrainment ratio and the thermal homogenization inside the long exhaust pipe, making viscosity modelling a key aspect of the CFD analysis.

3.6.3 Impact on Reynolds Number and Flow Regime

The Reynolds number is defined as:

$$Re = \frac{\rho v D}{\mu}. \quad (3.6.3)$$

Because density decreases and viscosity increases with temperature, the Reynolds number drops significantly between 288 K and 900 K. Although the flow remains fully turbulent, this reduction affects:

- the thickness and structure of the boundary layer,
- the development of secondary flows in the elbow,
- the mixing efficiency in the ejector,
- the pressure losses across the restrictors.

Accurately modelling these temperature-dependent effects is therefore essential for predicting the performance and thermal safety of the High-Pressure collector.

Chapter 4

Numerical Methodology

This chapter details the numerical tools, meshing strategies, and solver configurations employed to conduct the aerodynamic and thermal study of the High-Pressure collector. The methodology is designed to ensure high-fidelity results across the three investigated configurations, ranging from isolated component analysis to the simulation of the full exhaust system. For reasons of industrial confidentiality regarding the BeCOVER facility, the exact physical dimensions of the test bench have been intentionally omitted from this public document.

4.1 Geometry Description

The numerical domain is modeled based on the technical specifications of the BeCOVER High-Pressure collector. To isolate the physical phenomena and quantify the impact of each singularity, the study is structured around three distinct geometric configurations.

4.1.1 Investigated Configurations

The research strategy follows a progressive complexity to characterize the flow behavior:

- **Configuration 1 (Reference Case):** A straight supply pipe followed by the 4-hole diaphragm. This setup establishes the baseline performance of the restrictor under ideal inflow conditions.

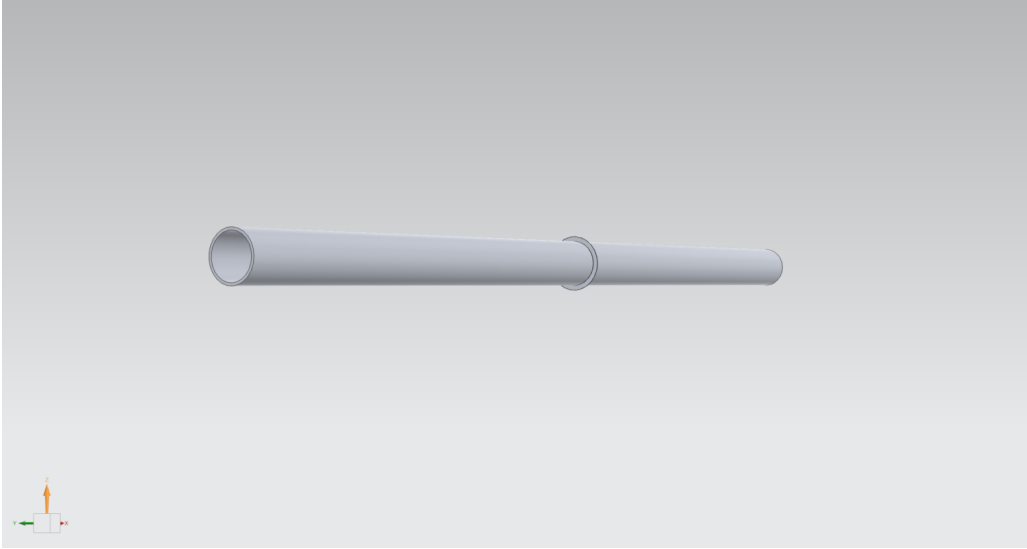


Figure 4.1: Configuration 1: Reference geometry with a straight supply pipe.

- **Configuration 2 (Distortion Analysis):** Integration of a 90-degree elbow upstream of the diaphragm. This configuration is specifically designed to isolate the impact of Dean vortices and flow asymmetry on the pressure drop.

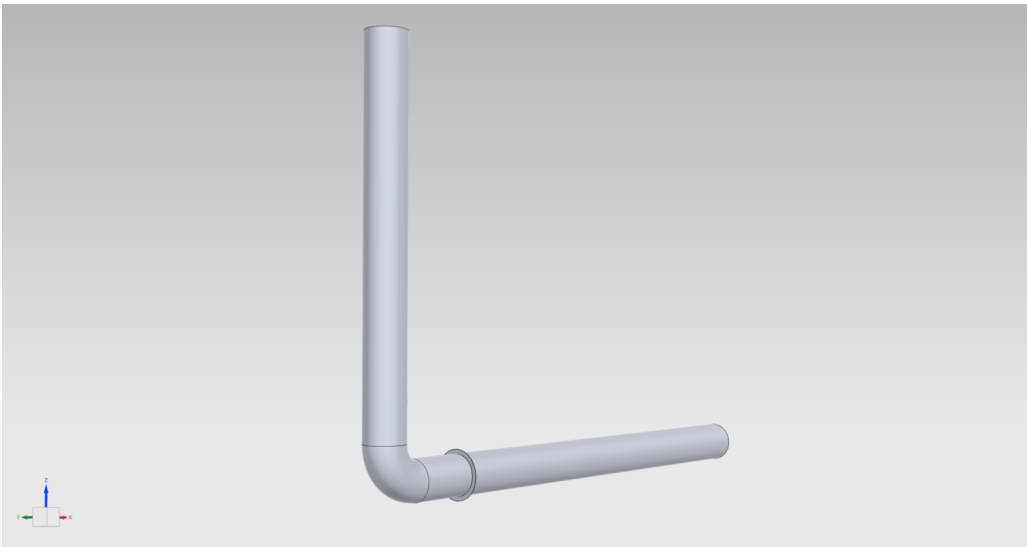


Figure 4.2: Configuration 2: Geometry including the 90-degree elbow for distortion analysis.

- **Configuration 3 (Full System):** The complete collector geometry, including the dual high-pressure supply lines, the large mixing section, the cooling funnel for ambient air entrainment, and the final discharge pipe.

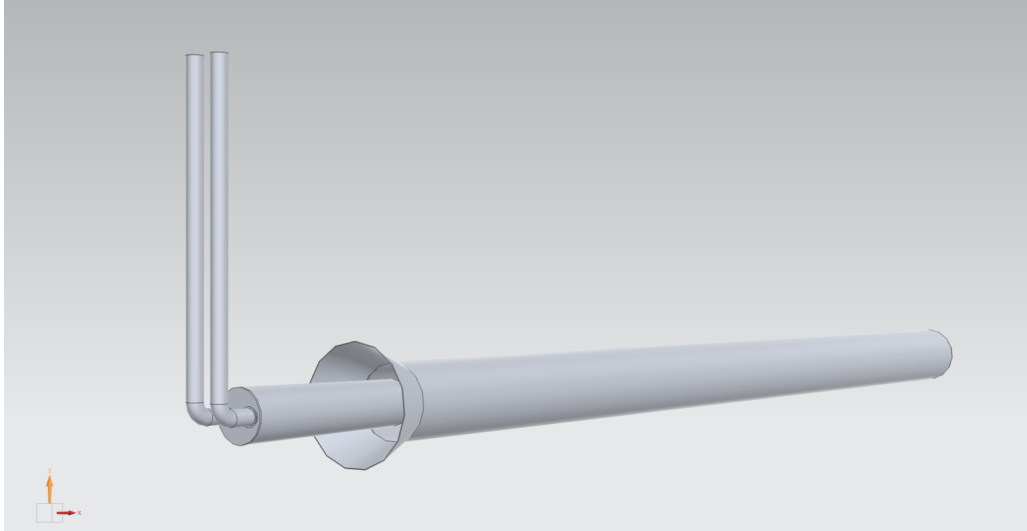


Figure 4.3: Global CAD view of the complete Configuration 3 assembly.

4.1.2 Detailed Dimensional Specifications

The internal geometry is defined by several critical dimensions that govern the aerodynamic behavior:

- **Primary Supply Lines:** The pipes leading to the diaphragms have a fixed internal radius. The 90-degree elbows feature a specific minimum radius of curvature.
- **Elbow-to-Restrictor Section:** For both supply lines, there is a straight section between the elbow exit and the restrictor plate.
- **Restrictor Plates (Figure 4.4):** Each throttling element consists of a multi-perforated plate with four circular orifices.

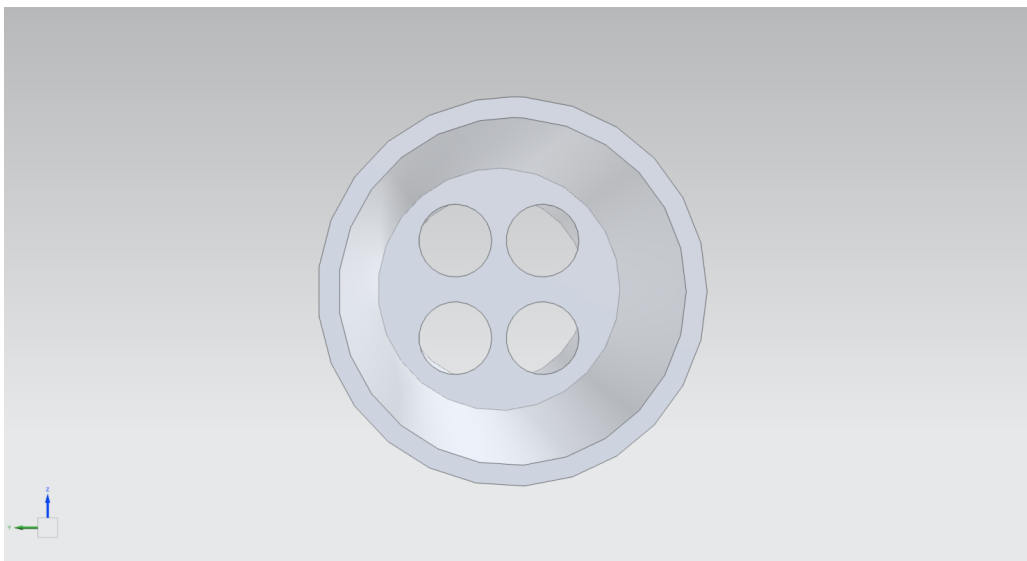


Figure 4.4: Detailed view of the 4-hole restrictor geometry.

- **Mixing Section:** Downstream of the plates, after a short transition, the flow enters a large mixing pipe. This section is slightly convergent, with its internal radius reducing from the inlet to the outlet.

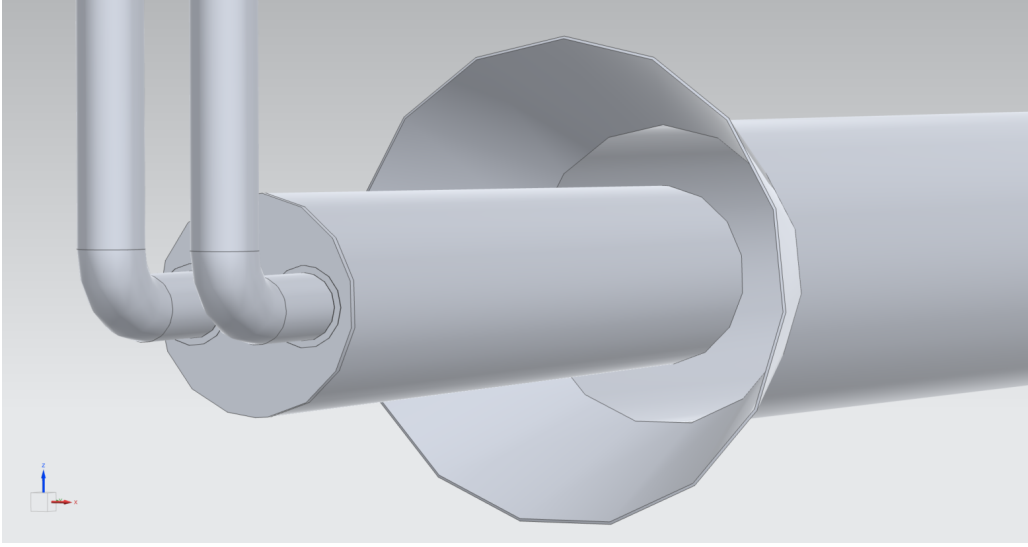


Figure 4.5: Zoom on the cooling funnel (suction chamber) where ambient air entrainment occurs.

- **Ejector and Exhaust:** The primary flow reaches the cooling funnel. This assembly leads to the final discharge pipe, which has a constant internal radius and a fixed total length.

4.2 Preprocessing with Pointwise

The generation of the computational grid was performed using **Pointwise** (Cadence Design Systems). Pointwise is a leading industrial mesh generation software specifically designed for high-fidelity Computational Fluid Dynamics (CFD). Widely utilized in the aerospace and power generation sectors, it is renowned for providing absolute control over grid topology and cell quality.

Unlike automated black-box meshers, Pointwise allows for a granular "bottom-up" approach, which was essential for the BeCOVER collector's requirements. This control was necessary to bridge the gap between the small-scale features of the restrictor orifices and the massive volumes of the discharge system while ensuring a high-quality transition. By leveraging its advanced suite of algorithms, a hybrid meshing strategy was developed to optimize the cell count while maintaining high accuracy in regions with extreme velocity gradients, specifically around the restrictor orifices and within the mixing zones.

4.2.1 Near-Wall Resolution and T-Rex Algorithm

To resolve the turbulent boundary layers, the *Anisotropic Tetrahedral Extrusion* (T-Rex) algorithm was employed on all internal surfaces. This technique extrudes high-aspect-ratio cells from the walls to ensure a high-quality transition toward the core flow.

The height of the first cell off the wall was fixed at 0.5 mm for all configurations. This setup results in a y^+ distribution ranging approximately from 1 to 50 across the investigated operating

conditions. As established in the theoretical framework (see 3.2.2), this range is perfectly handled by the all- y^+ formulation of the Spalart-Allmaras model, ensuring a continuous transition between the viscous sublayer and the log-law region. A geometric growth rate of 1.2 was applied to these layers to capture the steep velocity gradients within the viscous sublayer.

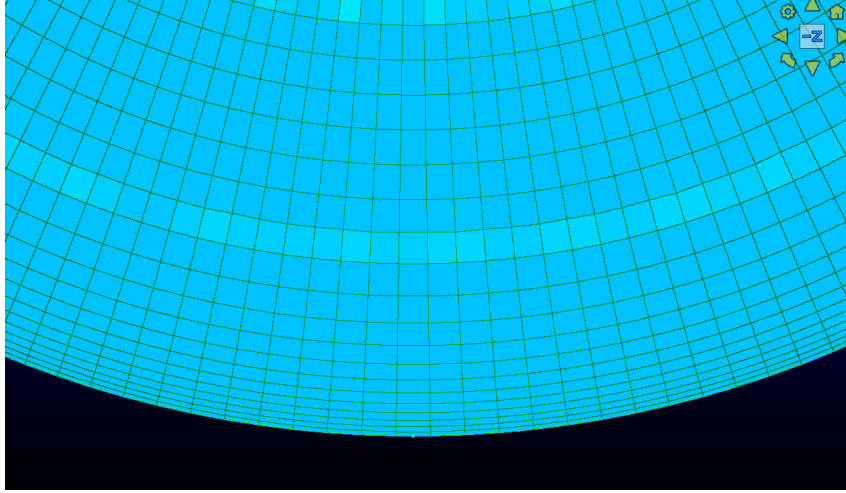


Figure 4.6: Radial cross-section of the supply pipe showing the T-Rex prismatic layers along the circumference and the transition to the core flow.

4.2.2 Diaphragm Initialization and Hybrid Topology

The mesh construction followed a directional logic centered on the restrictor plates, where the flow physics is most sensitive. This approach ensures that the high-gradient regions near the orifices are captured with structured cells before transitioning to unstructured zones in more complex geometric areas.

Diaphragm Surface Mesh

The domains representing the restrictor faces were initialized using the Advancing Front Ortho algorithm. This choice ensures that the surface cells near the orifice edges remain perfectly orthogonal and regular. This high-quality surface baseline is essential for the success of the subsequent volumetric extrusions, as it prevents cell skewness in the region where velocity is maximal.

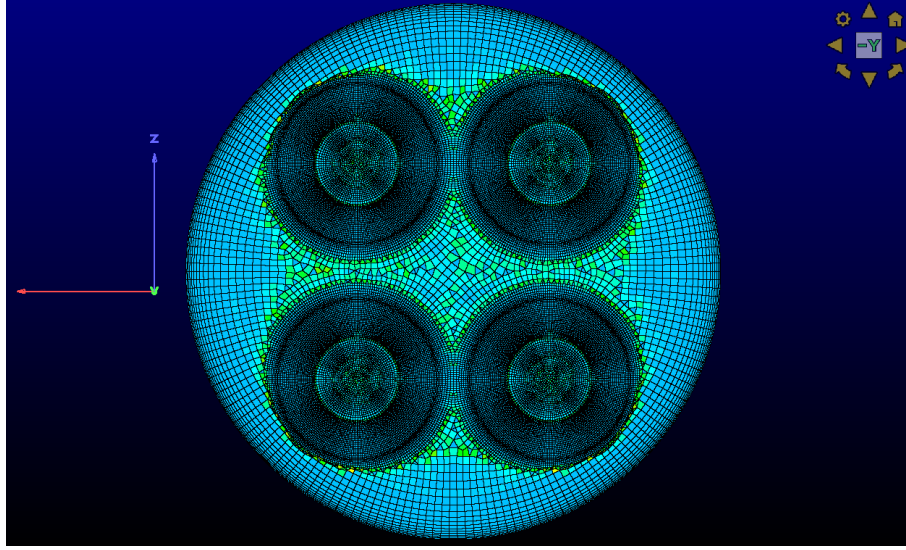


Figure 4.7: Surface mesh of the 4-hole diaphragm initialized with the Advancing Front Ortho algorithm to ensure orthogonal cell distribution around the orifices.

Structured Extrusion Strategy

To manage the flow acceleration and the development of the supersonic jets, a structured hexahedral extrusion strategy was adopted on both sides of the diaphragm. This creates a high-precision numerical corridor where cells are perfectly aligned with the flow direction.

- **Diaphragm-to-Elbow Section (Upstream):** A structured extrusion starts from the diaphragm and extends upstream through the straight pipe section. This ensures that the flow distortion approaching the orifices is captured without numerical diffusion.
- **Downstream Buffer Zone:** Immediately downstream of the orifices, a short structured extrusion was performed. This zone stabilizes the initial expansion of the jets and the formation of the vena contracta before the flow enters the fully unstructured domain of the divergent section.

As requested by the refinement analysis, Figure 4.8 illustrates this strategy. The longitudinal cross-section shows the perfect alignment of the hex-dominant blocks, while the zoomed view confirms the high density of points and the presence of T-Rex layers within the orifice throats to resolve the thin internal boundary layers.

Bend Transition and Unstructured Zones

Once the flow is sufficiently far from the restrictor singularities, the mesh transitions to unstructured tetrahedral domains to accommodate complex geometries:

- **90-Degree Elbow:** Within the bend, the mesh transitions to a T-Rex unstructured tetrahedral domain. This provides the flexibility to resolve the secondary Dean vortices without the topological constraints of a pure extrusion.
- **Inlet Section:** Further upstream, before the elbow, the mesh returns to a structured hexahedral extrusion up to the inlet boundary.

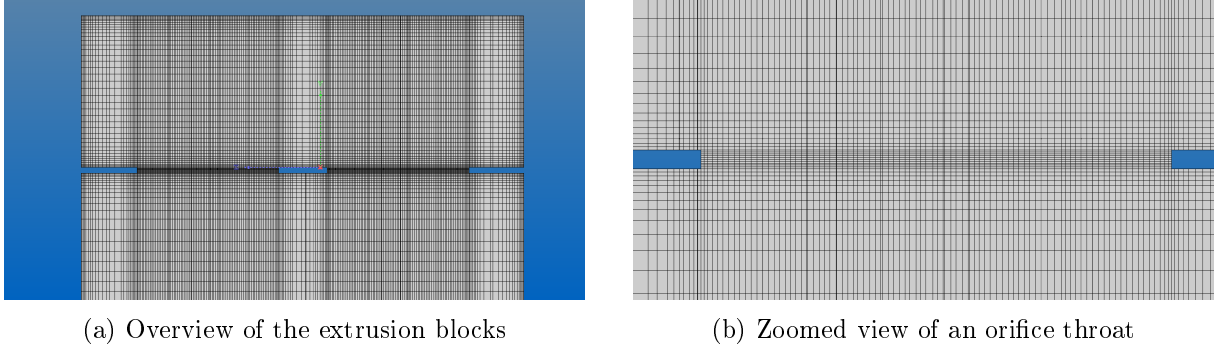


Figure 4.8: Longitudinal cross-sections of the structured mesh around the restrictor plate. The flow direction is oriented from bottom (upstream) to top (downstream).

The transition between the structured extruded blocks and the unstructured tetrahedral zones is characterized by a maximum included angle of 160 degrees, ensuring a smooth numerical interface and preventing artificial pressure reflections at the block boundaries.

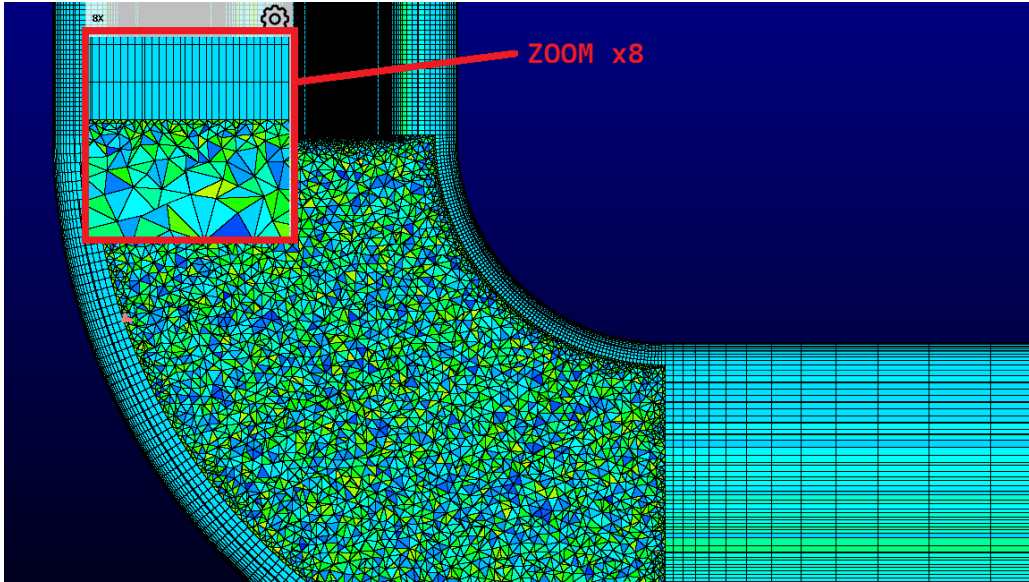


Figure 4.9: Detailed view of the 90-degree elbow showing the transition from structured hex-dominant pipes to the unstructured tetrahedral mesh within the bend.

4.2.3 Volumetric Refinement and Local Sources

For the large volumes, the core was filled with isotropic tetrahedral cells. To track the high-velocity jets, manual volumetric sources were placed downstream of the orifices. These sources prevent the mesh from coarsening too rapidly, capturing the momentum transfer between the primary and secondary air.

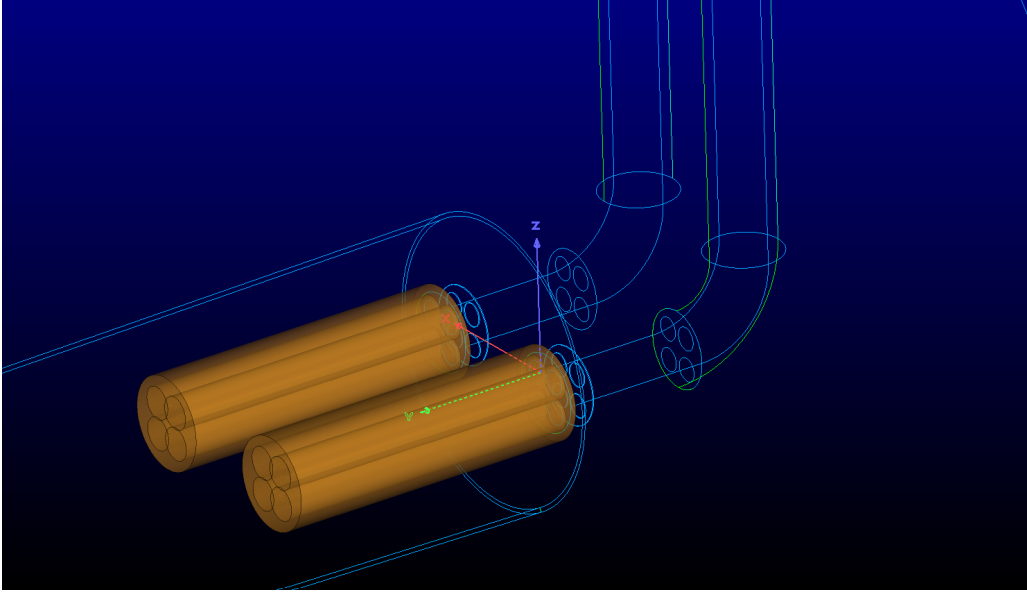


Figure 4.10: Visualization of the volumetric sources used to maintain high mesh density along the primary jet paths within the divergent section.

4.2.4 Final Mesh Statistics

The complete system results in a highly refined grid of approximately 28 million cells. The breakdown highlights the high proportion of hexahedral cells resulting from the extrusion strategy:

- **Total Volumetric Cells:** 27 994 750
- **Total Grid Points:** 18 146 108
- **Cell breakdown:**
 - **Hexahedra (Hexes):** 15 645 172 — Dominating the structured supply pipes and wall layers.
 - **Tetrahedra (Tets):** 11 269 382 — Constituting the unstructured elbows, divergent, and farfield core.
 - **Prisms & Pyramids:** 1 080 196 — Facilitating the topological transitions.

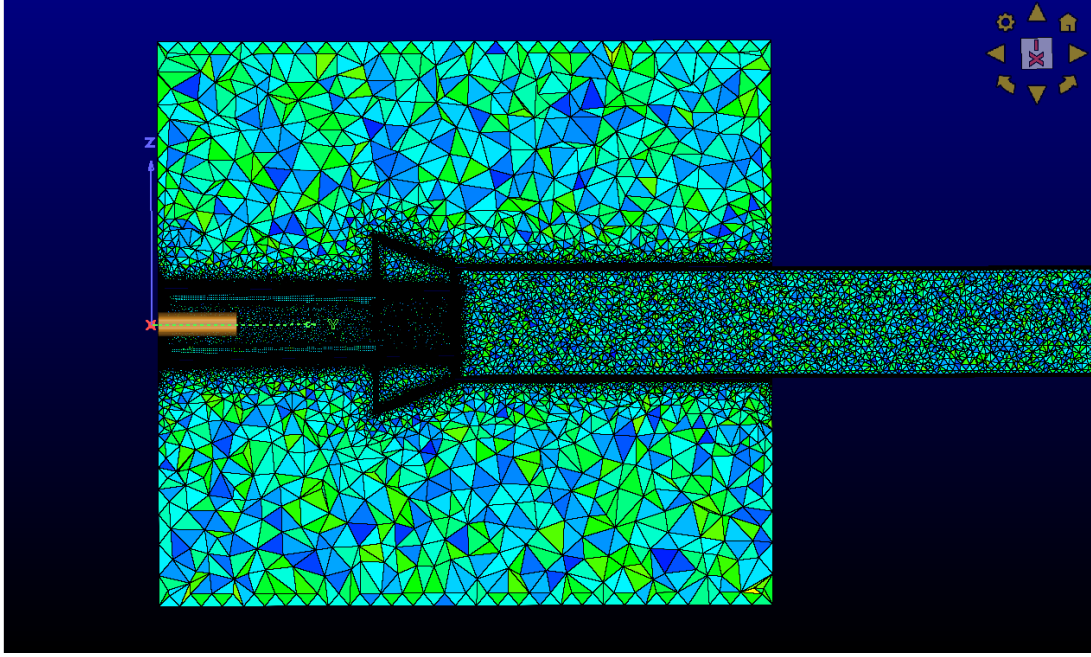


Figure 4.11: Global longitudinal cross-section of the mesh

4.3 Solver Configuration in SU2

The aerodynamic and thermal simulations were conducted using SU2 (Stanford University Unstructured), an open-source collection of software tools written in C++ for the solution of partial differential equations (PDEs) on unstructured meshes. SU2 is widely recognized in the aerospace community for its efficiency in solving compressible Reynolds-Averaged Navier-Stokes (RANS) equations and its robust implementation of turbulence models.

4.3.1 Numerical Schemes: SLAU2 and Roe

To ensure both numerical stability and high-fidelity results, a two-step approach was adopted for the spatial discretization of the convective fluxes.

The SLAU2 (Simple Low-dissipation Advection Upstream Splitting Method) scheme was primarily used during the initial stages of the calculation. Its low-dissipation properties at low Mach numbers and its superior stability make it ideal for starting simulations from scratch, especially in the presence of the complex recirculation zones found in the divergent section.

Once a stable flow field was established, the solver was switched to the Roe Upwind scheme with second-order spatial accuracy using the MUSCL (Monotonic Upstream-centered Scheme for Conservation Laws) approach. This ensures a more precise capture of the pressure gradients across the restrictor orifices and the shear layers within the ejector. Detailed solver parameters and the complete SU2 configuration file used for these simulations are provided in Appendix A for reproducibility purposes.

4.3.2 Boundary Conditions and Color Mapping

The definition of boundary conditions (see Figure 4.12) is critical for matching the experimental setup of the BeCOVER bench. These conditions are defined in the SU2 configuration file using specific markers that link the geometric surfaces to the physical properties of the flow.

- **Inlets (Yellow):** Defined by the `MARKER_INLET` tag, a total pressure and a total temperature are imposed at the start of the dual supply lines. These parameters are adjusted according to the investigated operating points to characterize the system's performance across its entire envelope. This boundary condition is applied to both symmetric supply lines; consequently, the total mass flow rate recorded by the solver represents the cumulative flux of the dual-pipe system.
- **Walls (Orange):** Defined by the `MARKER_HEATFLUX` tag, all internal surfaces of the collector are treated as adiabatic walls with a no-slip condition.
- **Outlet (Green):** Defined by the `MARKER_OUTLET` tag, a static pressure of 101,325 Pa is imposed at the exit of the discharge pipe to simulate atmospheric exhaust.
- **Farfield (Blue/Violet):** Defined by the `MARKER_FAR` tag, this condition is exclusively used for Configuration 3 (full system, see Figure 4.3). It establishes a non-reflecting boundary based on Riemann invariants to simulate the ambient atmosphere surrounding the cooling funnel. This condition fixes the static pressure at 101 325 Pa and the ambient temperature at 288.15 K. In the absence of a prescribed freestream velocity vector, the flow direction is determined by the local pressure gradient, allowing for the natural entrainment of secondary air into the suction chamber. These values represent the standard ambient conditions of the facility, which acts as an infinite reservoir maintaining a constant pressure and temperature.

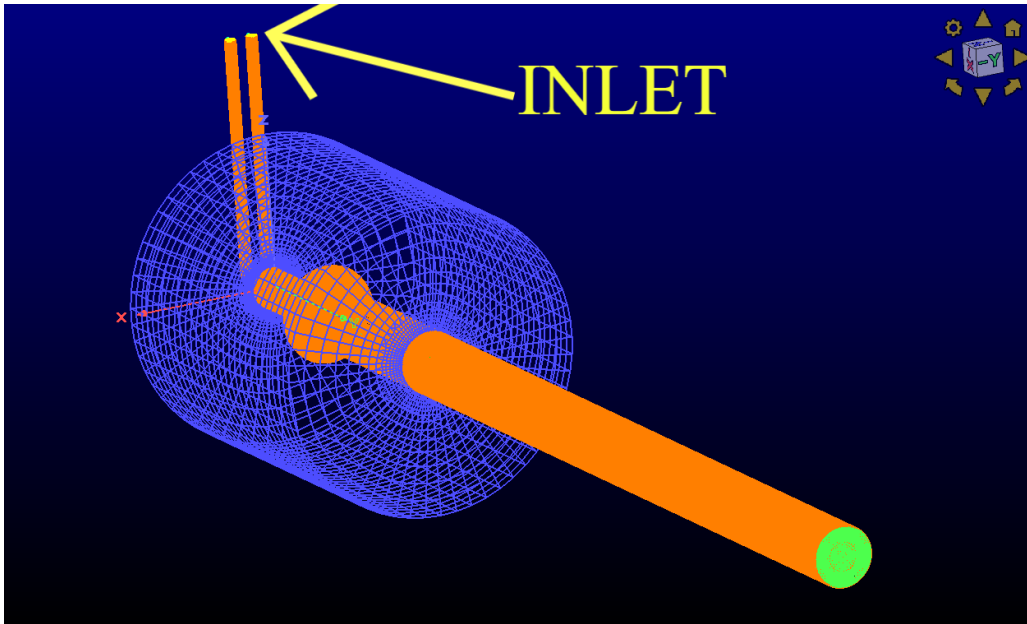


Figure 4.12: Spatial distribution of boundary conditions for Configuration 3: Yellow (Inlets), Orange (Walls), Green (Outlet), and Blue/Violet (Farfield).

4.3.3 High-Performance Computing (HPC) Resources

The computational intensity required for this study, particularly for the 28-million-cell Configuration 3, necessitated the use of high-performance computing resources. All simulations were performed on the NIC5 supercomputer, hosted at the University of Liège and managed within the

framework of the Consortium des Équipements de Calcul Intensif (CÉCI). The CÉCI is a cross-university infrastructure supported by the Fonds de la Recherche Scientifique (F.R.S.-FNRS) that pools high-performance computing power across five Belgian universities [13].

As shown in the physical installation view (Figure 4.13), NIC5 is a high-density cluster providing massive parallel scientific computational power.



Figure 4.13: View of the physical NIC5 supercomputer racks housed at the University of Liège.

The cluster is architected to handle large-scale RANS simulations with high parallel efficiency. As illustrated in the global network topology (Figure 4.14), the infrastructure relies on a non-blocking InfiniBand HDR100 interconnect. This high-speed communication fabric ensures the necessary low-latency data exchange between the partitioned domains of the mesh, the login nodes, and the multi-tier storage systems (NFS and BeeGFS).

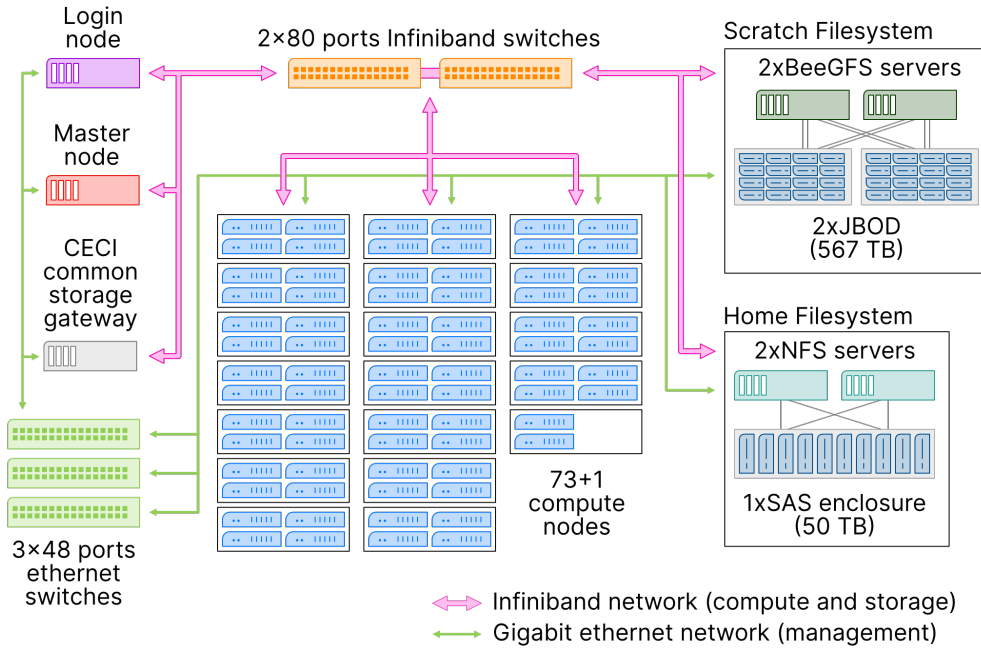


Figure 4.14: Global network architecture of the NIC5 cluster illustrating the high-speed InfiniBand HDR100 topology [14].

Each individual compute node is optimized for throughput. As detailed in the technical schematic (Figure 4.15), nodes are based on a dual-socket configuration featuring AMD EPYC 7763 "Milan" processors. These CPUs, utilizing the Zen 3 architecture, provide 64 cores per socket, totaling 128 cores per node, supported by an advanced Global Memory Interconnect and a dedicated InfiniBand HCA.

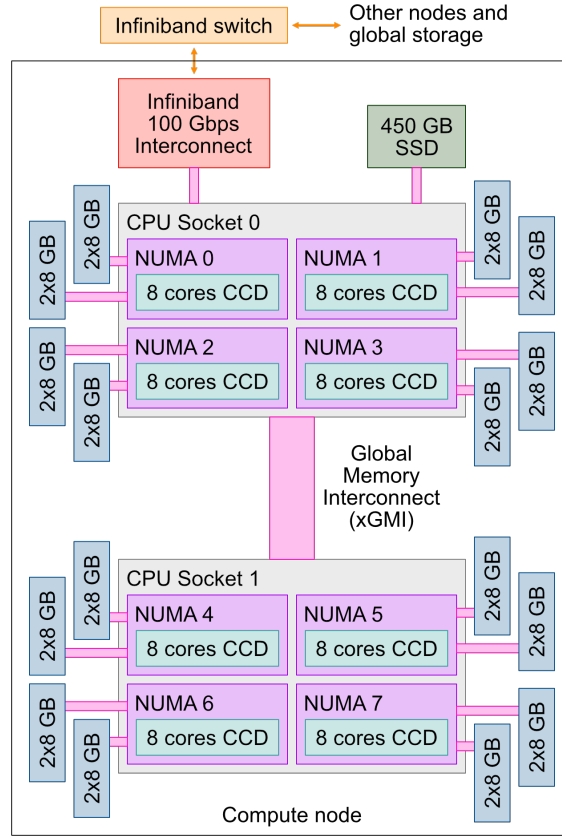


Figure 4.15: Technical schematic of a NIC5 compute node highlighting the dual AMD EPYC socket configuration and memory interface [14].

To manage these resources, a Slurm workload manager was used to interface with the cluster scheduler. For the final Configuration 3 simulations, the allocation was configured as follows:

- **Parallelization:** 256 CPU cores were utilized via OpenMPI, distributing the computational load across two complete compute nodes.
- **Memory Allocation:** 4 GB of RAM was allocated per CPU core, resulting in a total memory footprint of 1024 GB (1 TB).
- **Computational Efficiency:** This distribution ensured that approximately 109,350 cells were managed per core, optimizing scalability during the matrix inversion phases.
- **Wall Time:** A typical run for 10,000 iterations required approximately 14 hours of continuous calculation on this allocation.

The complete submission script, including the environment modules and the MPI execution commands, is provided in Appendix B for reproducibility purposes.

4.3.4 Convergence Monitoring and Residuals

The convergence of the steady-state simulations was monitored through two primary indicators to ensure the reliability of the numerical results.

First, the mathematical convergence was assessed via the evolution of the RANS residuals. For all investigated cases, the density and energy residuals were required to drop by at least six to seven orders of magnitude (10^{-6} to 10^{-7}) to ensure that the solution had sufficiently stabilized.

Second, a global mass flow balance was performed as a physical consistency check. Using custom output markers in the SU2 configuration, the mass flow was monitored at the primary inlets, the secondary suction opening, and the final exhaust exit. Convergence was only considered reached when the conservation of mass was satisfied: $\dot{m}_{primary} + \dot{m}_{secondary} = \dot{m}_{total}$. For Configuration 3, the numerical error in this balance remained below 0.1 % for all simulation points.

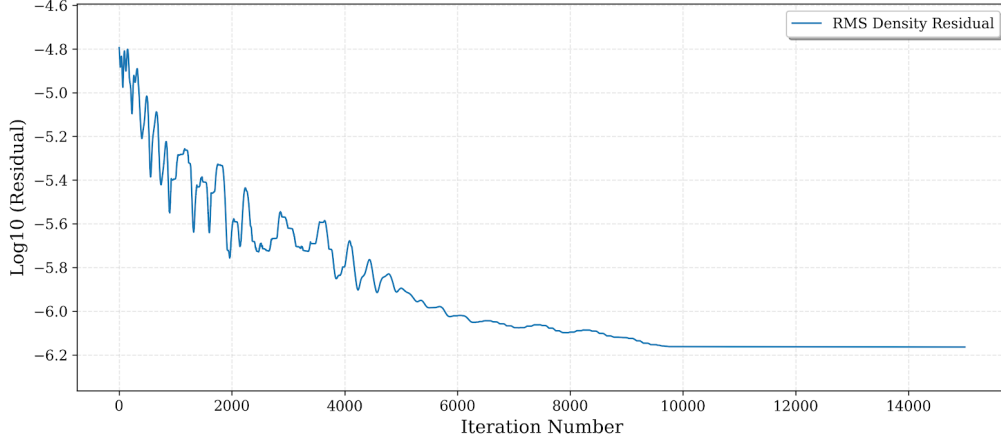


Figure 4.16: Evolution of the RANS residuals showing a convergence level of 10^{-6} for the final simulation stages.

Chapter 5

Phase 1: Impact of Upstream Piping on Diaphragm Aerodynamics

The first phase of this study focuses on the aerodynamic interaction between the supply piping and the restrictor plate. In the industrial configuration of the BeCOVER collector, the presence of 90-degree elbows immediately upstream of the diaphragm is a major geometric constraint. This chapter aims to quantify the flow distortions induced by these elbows and their subsequent impact on the discharge capacity and jet symmetry.

5.1 Methodology of Analysis

To provide a rigorous comparison between the different configurations and operating temperatures (288.15 K, 600 K, and 900 K), it is essential to normalize the raw mass flow data. This normalization allows for the isolation of purely aerodynamic effects, such as flow distortion induced by elbows, from thermodynamic variations related to air density and viscosity.

5.1.1 Ideal Mass Flow and Normalization Strategy

The performance of the restrictor plate is evaluated by comparing the actual mass flow rate ($\dot{m}_{real}$) obtained from the SU2 simulations with the theoretical maximum throughput under isentropic conditions, referred to as the ideal mass flow rate ($\dot{m}_{ideal}$).

As established in the theoretical framework (see Section 3.4.5), the ideal mass flow rate is calculated using the Saint-Venant relation, which accounts for the upstream stagnation properties ($P_{t,in}$, $T_{t,in}$) and the pressure ratio across the orifices. By utilizing this reference, the discharge coefficient (C_d) is derived:

$$C_d = \frac{\dot{m}_{real}}{\dot{m}_{ideal}} \quad (5.1.1)$$

In this study, $\dot{m}_{ideal}$ is calculated for each simulation point using the geometric area A of the four orifices. This normalization is critical because it decouples the mass flow capacity from the $1/\sqrt{T_{t,in}}$ thermodynamic dependency, allowing the results from all three thermal cases to be compared on a single non-dimensional scale.

5.1.2 Physical Scaling and Reynolds Sensitivity

The use of the discharge coefficient as the primary metric for Phase 1 serves three strategic purposes in the analysis of the BeCOVER collector:

- **Global Performance Benchmarking:** It allows for the direct quantification of the mass flow penalty induced by the 90-degree elbow in Configuration 2 relative to the straight-pipe baseline of Configuration 1.
- **Flow Regime Characterization:** At high pressure ratios, the flow reaches a plateau where inertial forces dominate. If the C_d remains constant across temperatures, it confirms that the restrictor's performance is governed by geometry and compressibility rather than viscous effects.
- **Viscous Impact Assessment:** Any observed shift in the C_d profiles at 600 K or 900 K can be precisely attributed to the increase in air viscosity (Sutherland's law). A lower Reynolds number at high temperatures may lead to thicker boundary layers within the orifices, slightly reducing the effective flow area (vena contracta), which would be reflected by a marginal drop in the discharge coefficient.

By adopting this methodology, the subsequent analysis focuses on the aerodynamic signature of the installation, ensuring that the conclusions drawn regarding flow distortion and jet asymmetry are independent of the specific test bench conditions.

5.2 Configuration 1: The Ideal Baseline (Reference)

The analysis begins with Configuration 1, which serves as the aerodynamic benchmark for the entire study. By utilizing straight supply lines, this setup eliminates any upstream geometric perturbations, allowing for the characterization of the restrictor plate's intrinsic performance.

5.2.1 Upstream Flow Uniformity

In this baseline configuration, the flow travels through two straight cylindrical conduits before entering the diaphragm chamber. The primary objective is to ensure that the flow reaching the orifices is fully developed and axisymmetrical.

As the fluid progresses through the pipes, a standard turbulent boundary layer develops along the walls. Given the length of the straight sections, the momentum is distributed symmetrically around the pipe axis. This is clearly visualized in the longitudinal velocity contours (Figure 5.1), where the acceleration remains centered.



Figure 5.1: Velocity magnitude contours along the straight supply lines for Configuration 1.

To quantify this uniformity, radial velocity profiles were extracted at a short distance upstream of the diaphragm plate. As shown in Figure 5.2, the profile perfectly matches a theoretical power-law distribution for turbulent pipe flow. There is no evidence of flow separation or

velocity bias toward any side of the pipe. This uniform "feeding" of the diaphragm ensures that the subsequent expansion through the four orifices is balanced, providing a clean reference for the jet formation analysis.

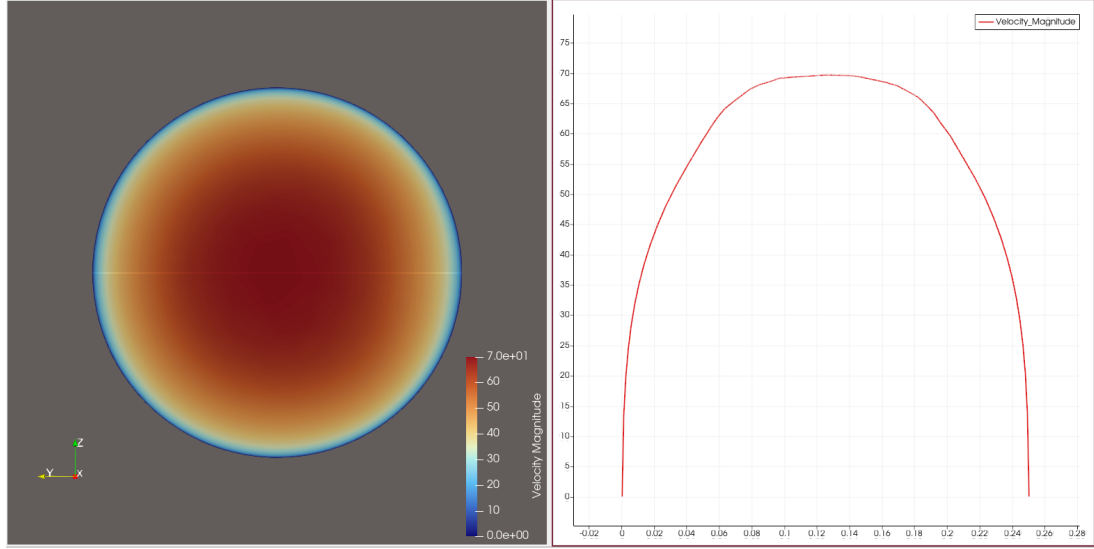


Figure 5.2: Radial velocity profile at the diaphragm inlet demonstrating perfect flow symmetry.

5.2.2 Global Performance and Discharge Coefficient

The global performance of Configuration 1 is assessed by analyzing the relationship between the mass flow capacity, the pressure ratio, and the stagnation temperature. This section establishes the baseline discharge behavior of the restrictor plate, which serves as the reference for quantifying the aerodynamic penalties in subsequent configurations.

Similarity Verification and Specific Reduced Mass Flow

To rigorously decouple the influence of the test bench's absolute dimensions and upstream stagnation conditions from the intrinsic aerodynamic behavior of the restrictor, the raw mass flow data is normalized. According to gas dynamics similarity laws, comparing a dimensional quantity ($\dot{m}$) to an adimensional one (Π) is insufficient. Instead, the capacity of the flow restriction is characterized by the specific reduced mass flow rate, defined as:

$$\dot{m}_{spec} = \frac{\dot{m}_{real} \sqrt{T_{t,in}}}{P_{t,in} \cdot A} \quad (5.2.1)$$

Where A represents the total geometric passage area of the orifices. By expressing the flow in this normalized form, not only are the thermodynamic effects of pressure and temperature neutralized, but the performance is also completely decoupled from the physical scale of the installation. This universal representation ensures that the derived aerodynamic models can be applied independently of the specific sizing of the BeCOVER facility.

As illustrated in Figure 5.3, the data points for all three temperature levels (288.15 K, 600 K, and 900 K) exhibit a characteristic compressible flow evolution.

This specific normalization provides critical physical validation for the study:

- It proves that the numerical model correctly captures the compressible flow physics across a wide thermal range.

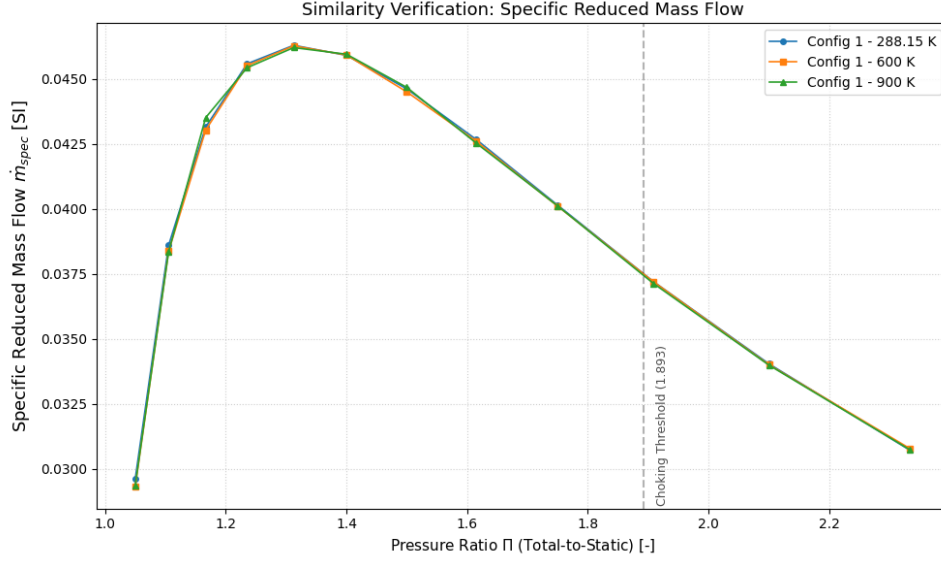


Figure 5.3: Specific reduced mass flow ($\dot{m}_{spec}$) as a function of the pressure ratio Π for Configuration 1. The normalization by the passage area A demonstrates the universality of the restrictor’s performance across the thermal series.

- It confirms that the massive variations in raw absolute mass flow at higher temperatures are strictly driven by changes in fluid density and speed of sound, rather than an inherent change in the discharge mechanism.
- It allows for a direct visual assessment of secondary Reynolds number effects: any minute vertical deviation between the temperature asymptotes directly reflects the impact of temperature-dependent dynamic viscosity on the boundary layer thickness and the effective *vena contracta*.
- It justifies the use of the discharge coefficient (C_d) as a universal performance indicator for the geometry.

Comparison with Ideal Throughput

Table 5.1 provides the numerical comparison between the simulated performance and the theoretical ideal capacity, both expressed as specific reduced mass flows ($\dot{m}_{spec}$). By normalizing the mass flow against the total geometric passage area (A) and standardizing to upstream stagnation conditions, the physical scale of the facility is completely abstracted while preserving the exact aerodynamic behavior.

For pressure ratios above the sonic limit ($\Pi > 1.893$), the ideal specific mass flow reaches its choked maximum.

Discharge Coefficient and Scaling

By taking the ratio of the real and ideal specific mass flows, the discharge coefficient (C_d) is directly obtained. As a consequence of the similarity observed in the specific reduced mass flow analysis, the three temperature series in Figure 5.4 collapse onto a single master curve.

Table 5.1: Baseline performance and discharge coefficient for Configuration 1, normalized as specific reduced mass flows.

Temp. [K]	Ratio Π [-]	$\dot{m}_{spec,real}$ [SI]	$\dot{m}_{spec,ideal}$ [SI]	C_d [-]
288.15	1.05	0.0148	0.0175	0.845
	1.50	0.0319	0.0382	0.835
	2.10	0.0341	0.0399	0.854
600.00	1.05	0.0147	0.0175	0.837
	1.50	0.0318	0.0382	0.833
	2.10	0.0340	0.0399	0.852
900.00	1.05	0.0147	0.0175	0.838
	1.50	0.0319	0.0381	0.836
	2.10	0.0340	0.0399	0.852

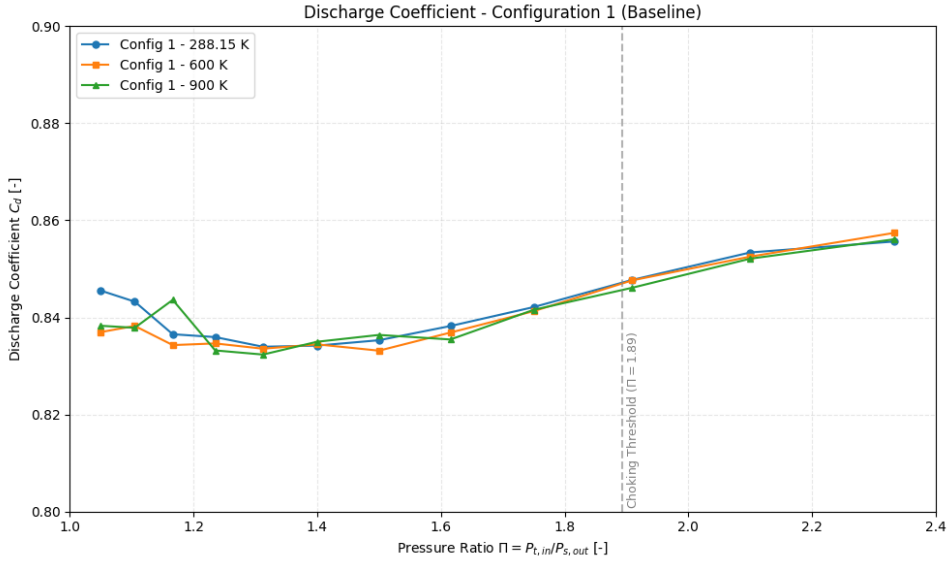


Figure 5.4: Discharge coefficient (C_d) evolution for the baseline configuration.

The discharge coefficient exhibits a slight variation at low pressure ratios due to Reynolds number effects and flow development, before reaching a stable value of approximately 0.84–0.85 in the compressible regime. This stability confirms that, in the absence of upstream disturbances, the restrictor’s performance is primarily governed by its geometry and the resulting *vena contracta*.

5.2.3 Jet Formation and Symmetry

The flow expansion through the restrictor plate represents the most critical aerodynamic event in the collector’s primary line. As the fluid reaches the four orifices, the potential energy (pressure) is rapidly converted into kinetic energy, resulting in the formation of four distinct high-velocity jets. In this baseline configuration, the absence of upstream disturbances is expected to produce a perfectly balanced flow distribution among the orifices.

To visualize the quality of this expansion, a transverse slice of the velocity magnitude was extracted immediately downstream of the diaphragm plate. As illustrated in Figure 5.5, the four jets exhibit identical topologies, both in terms of core velocity and spatial development. The

velocity magnitude within the jet cores reaches approximately the same peak values across all four orifices, confirming that the "feeding" of the diaphragm is uniform.

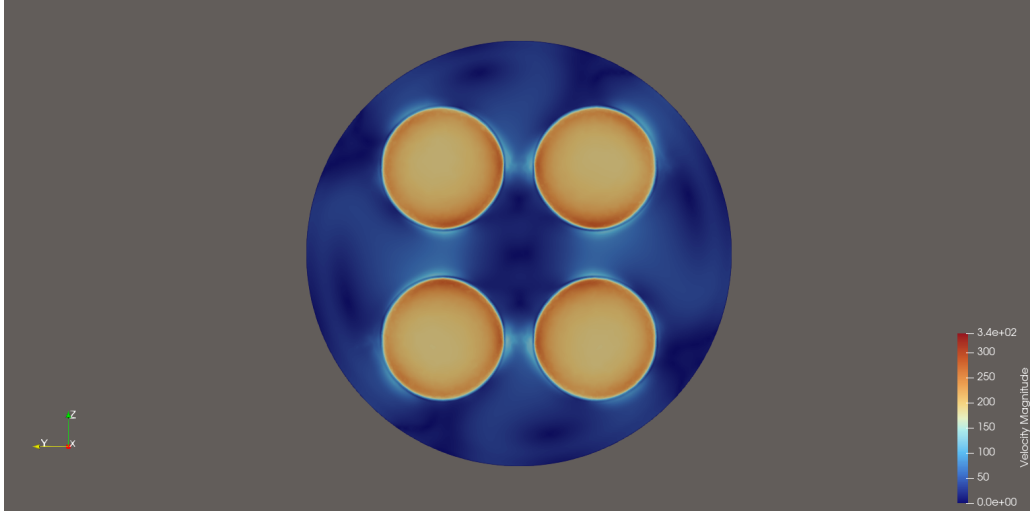


Figure 5.5: Velocity magnitude contours in a transverse plane immediately downstream of the diaphragm for Configuration 1 ($\Pi = 1.50$, $T = 288.15$ K).

Beyond the qualitative visual symmetry, a quantitative mass flow balance was performed by integrating the mass flow rate through each individual orifice (H_1 to H_4). To maintain consistency with the universal scaling approach, these local flows are expressed as specific reduced mass flows ($\dot{m}_{spec}$), normalized by the respective area of each orifice ($A/4$). This data, summarized in Table 5.2, demonstrates the intrinsic balance of the system. Each hole accounts for approximately 25.0% of the total mass flow, with the specific flow reaching a uniform value of approximately 0.0444 across all quadrants, accompanied by a negligible numerical deviation (less than 0.03 %).

Table 5.2: Mass flow distribution across the four orifices for the baseline configuration, expressed as specific reduced mass flow ($\dot{m}_{spec}$).

Orifice	Specific Mass Flow $\dot{m}_{spec}$ [SI]	Distribution [%]
Hole 1 (Top-Left)	0.04444	25.00
Hole 2 (Top-Right)	0.04446	25.01
Hole 3 (Bottom-Left)	0.04439	24.97
Hole 4 (Bottom-Right)	0.04446	25.01
Total (Global)	0.04444	100.00

This perfect partitioning is a direct consequence of the symmetrical boundary layer development in the straight supply pipes analyzed in Section 5.2.1. In the absence of any centrifugal forces or flow detachment (vortex of Dean), the pressure field upstream of the plate is uniform, ensuring that each hole is subjected to the same local total pressure. These results establish a robust reference state. Any deviation from this 25 % per hole distribution in the subsequent phases of the study will therefore be solely attributable to the geometric perturbations introduced by the 90-degree elbow.

5.2.4 Downstream Flow Development and Mixing

Immediately after the expansion through the four orifices, the flow enters a complex mixing phase. This region is characterized by the interaction between the high-velocity jets and the large stagnant zones created by the solid parts of the diaphragm.

As illustrated in the longitudinal velocity contours (Figure 5.6), the four jets emerge into the downstream chamber with a perfectly symmetrical velocity distribution. In this baseline configuration, the flow development remains balanced, and the four individual jets gradually expand as they exchange momentum with the surrounding fluid through turbulent shear layers.



Figure 5.6: Longitudinal velocity magnitude contours downstream of the diaphragm for Configuration 1.

To better understand the internal flow structure, a detailed view using Surface LIC (Line Integral Convolution) was captured immediately behind the plate (Figure 5.7). This visualization highlights the recirculation zones forming directly behind the solid sections of the diaphragm. These toroidal vortices are the primary regions of entropy production and are responsible for the permanent pressure loss quantified by the discharge coefficient.

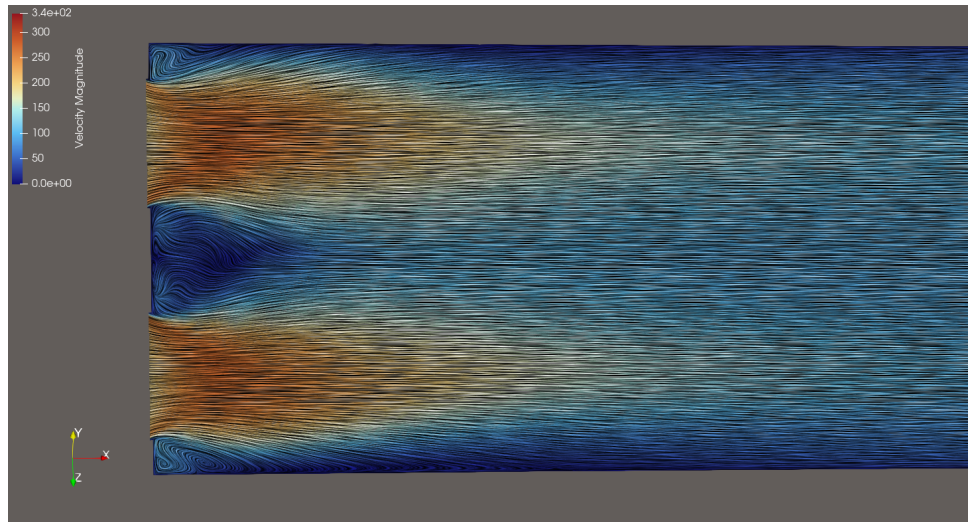


Figure 5.7: Zoomed-in Surface LIC visualization of the recirculation zones and toroidal vortices immediately downstream of the diaphragm plate.

A short distance downstream of the plate, the jets begin to merge toward the pipe centerline.

Because the mass flow is equally distributed between the orifices (as shown in Section 5.2.3), the pressure recovery is uniform across the pipe section. This results in a balanced mixing process where no single jet dominates the flow. This symmetrical behavior serves as the necessary reference point for Phase 2, where the flow distortions induced by an upstream elbow will disrupt this balance.

5.3 Configuration 2: Influence of the 90-degree Elbow

In this second phase, the straight supply lines are replaced by 90-degree elbows, reflecting the actual geometric constraints of the BeCOVER test bench. The objective is to characterize how the centrifugal forces induced by the curvature modify the flow structure before it reaches the restrictor plate.

5.3.1 Flow Distortion and Dean Vortices

The transition from a straight pipe to a curved section introduces a radial pressure gradient required to balance the centrifugal force ($F_c = \rho v^2/r$). As the fluid enters the elbow, it is subjected to a significant pressure imbalance: a high-pressure zone develops at the outer radius while a low-pressure zone forms at the inner radius.

This phenomenon is visualized in Figure 5.8, where the longitudinal pressure contours highlight the "compression" of the fluid against the outer wall. This pressure gradient is the fundamental driver of the subsequent flow distortions.

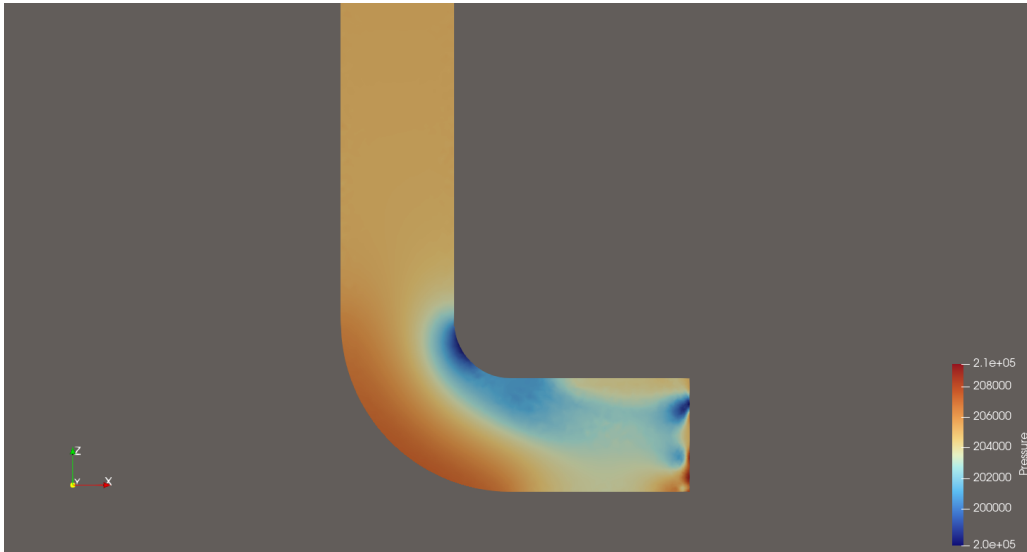


Figure 5.8: Longitudinal static pressure contours in the 90-degree elbow for Configuration 2, showing the centrifugal pressure gradient between the inner wall and the outer wall.

This imbalance triggers a secondary flow phenomenon known as **Dean vortices**. Due to the radial pressure gradient, the slower fluid within the boundary layers is pushed back toward the inner wall along the pipe walls, creating a pair of counter-rotating vortices. This secondary motion is captured in Figure 5.9 using a combination of Surface LIC for the flow structure and velocity Glyphs for directionality. These vortices transport low-momentum fluid toward the center of the pipe, further disrupting the flow uniformity.

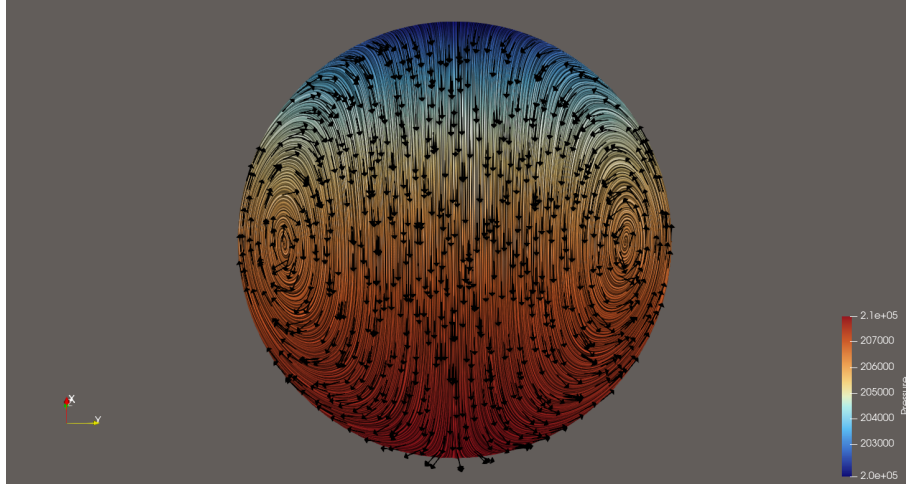


Figure 5.9: Cross-sectional visualization of Dean vortices at a 45° slice for Configuration 2, combining Surface LIC and velocity Glyphs.

To quantify the resulting distortion at the diaphragm inlet, Figure 5.10 presents a dual-view analysis of the flow field just before the restriction. Unlike the perfectly symmetrical profile observed in the straight pipe of Configuration 1, the velocity contours here (left) show a clear migration of the high-velocity core toward the outer wall. This is confirmed by the radial line plot (right), where the velocity distribution is highly skewed, with the peak velocity significantly shifted away from the pipe centerline. This "distorted feeding" of the restrictor plate is expected to break the symmetry of the downstream jets and penalize the overall discharge capacity.

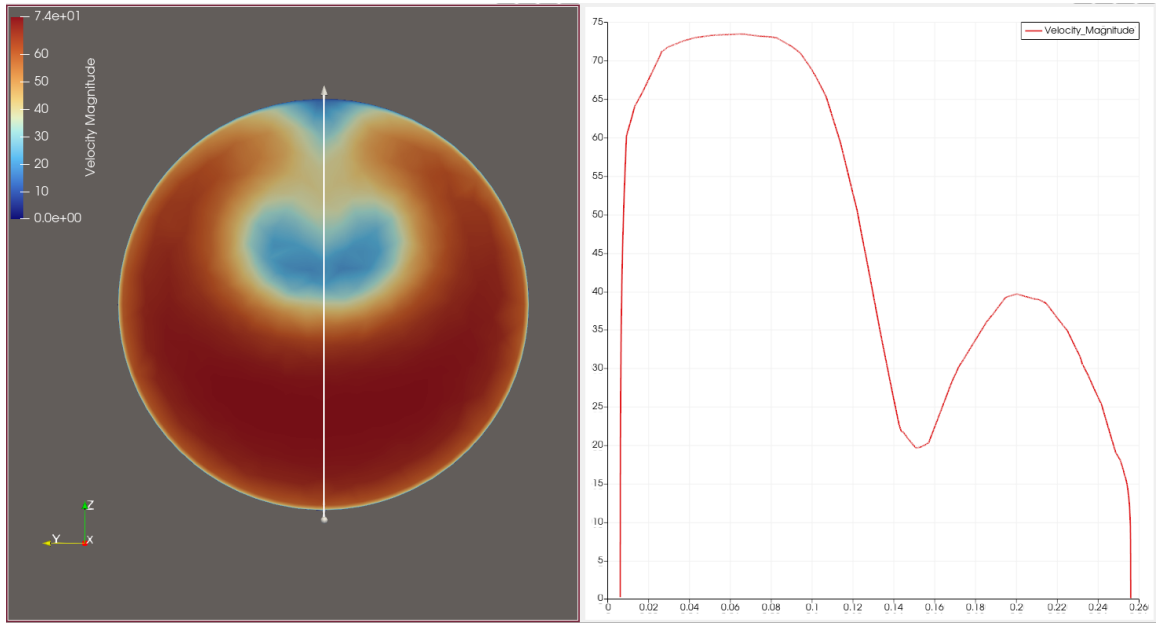


Figure 5.10: Flow distortion at the diaphragm inlet for Configuration 2: Velocity magnitude contours (left) and radial velocity profile (right).

5.3.2 Penalization of the Discharge Capacity

The flow distortions and secondary flows (Dean vortices) characterized in the previous section have a direct impact on the global performance of the system. The energy dissipated in the

elbow, combined with the non-uniform "feeding" of the restrictor plate, leads to a measurable reduction in the mass flow rate compared to the ideal straight-pipe baseline.

Mass Flow Rate and Global Efficiency

Figure 5.11 illustrates the evolution of the specific reduced mass flow ($\dot{m}_{spec}$) as a function of the pressure ratio Π for both configurations. Normalizing the mass flow rate by the geometric passage area (A) and standardizing to upstream stagnation conditions allows for a direct, non-dimensional comparison of the system's capacity. While the general trend remains consistent with compressible flow physics, the curve for Configuration 2 is systematically shifted downward. At a constant pressure ratio of $\Pi = 1.50$, the presence of the 90-degree elbow induces a specific mass flow penalty of approximately 4%.

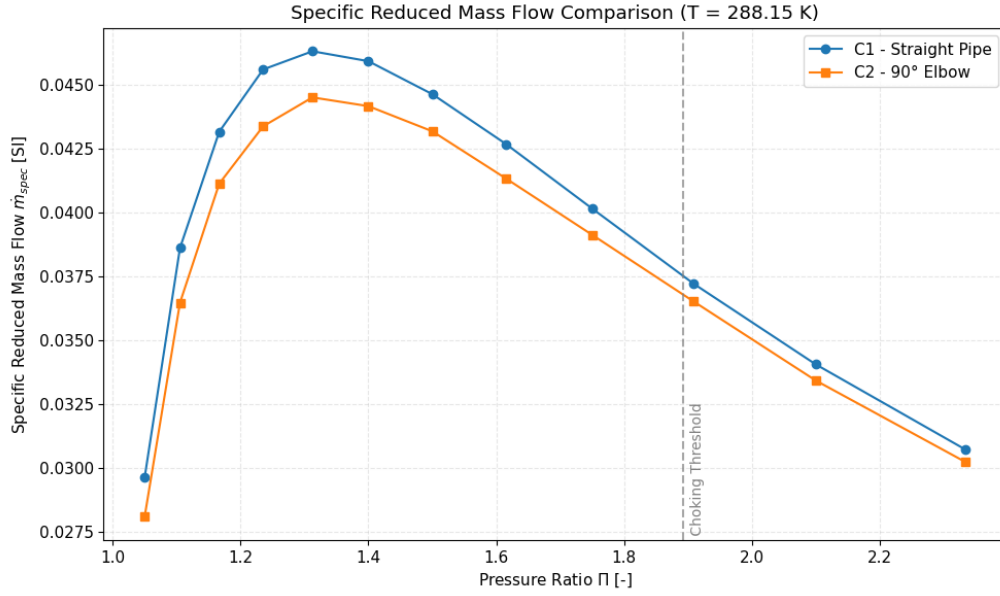


Figure 5.11: Comparison of specific reduced mass flow ($\dot{m}_{spec}$) between Configuration 1 (Straight) and Configuration 2 (Elbow) at 288.15 K.

To isolate the aerodynamic penalty, the discharge coefficient (C_d) is analyzed in Figure 5.12. The results show a clear degradation of the plate's efficiency in Configuration 2, with C_d dropping from approximately 0.84 to 0.80. This decrease indicates that the distorted velocity profile provided by the elbow modifies the global *vena contracta* effect. In this configuration, the orifices are no longer fed by a uniform axial flow, which increases local turbulence and reduces the overall effective flow area.

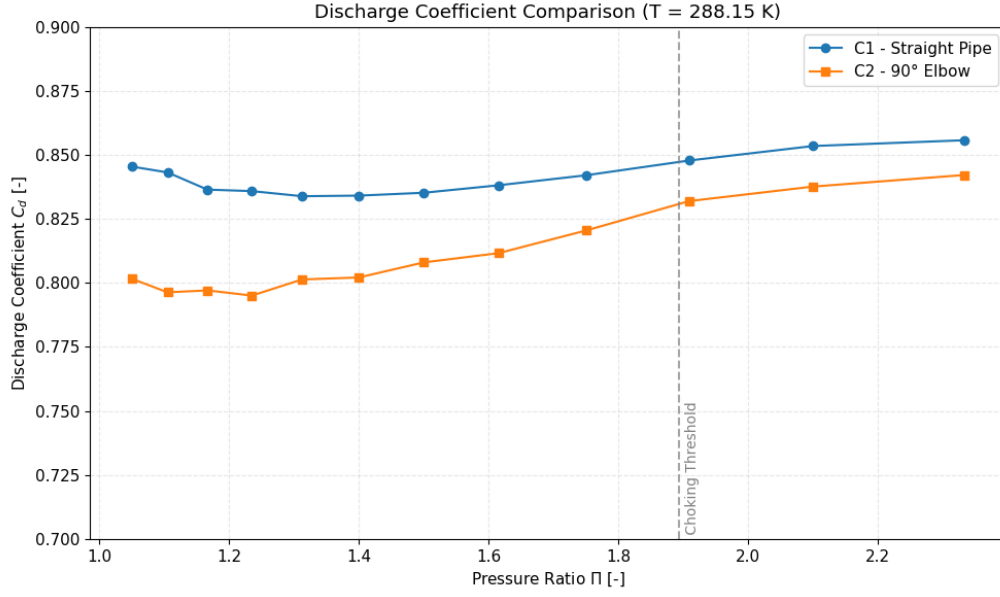


Figure 5.12: Discharge coefficient (C_d) comparison between Configuration 1 and Configuration 2.

5.3.3 Jet Asymmetry and Mass Imbalance

Beyond the global loss of efficiency, the most critical impact of the 90-degree elbow is the breakdown of flow distribution between the orifices. In the ideal Configuration 1, the symmetry of the velocity profile ensures that each orifice processes exactly 25% of the total mass flow. In Configuration 2, the interaction between the centrifugal pressure gradient and the plate geometry leads to a significant redistribution.

Quantification of Mass Flow Distribution

To quantify this imbalance, the mass flow rate through each individual orifice was integrated for a pressure ratio of $\Pi = 1.50$, and expressed as specific reduced mass flow ($\dot{m}_{spec}$) normalized by the individual orifice area ($A/4$). The results, presented in Table 5.3, reveal a counter-intuitive phenomenon: the orifices located near the bottom side process a higher specific mass flow than those at the top side.

Table 5.3: Detailed mass flow distribution between the four orifices ($\Pi = 1.50$) for Configuration 2, expressed as specific reduced mass flow ($\dot{m}_{spec}$).

Position	Orifice	Specific Mass Flow $\dot{m}_{spec}$ [SI]	Distribution [%]
Top	Top Right	0.04079	24.66
	Top Left	0.04064	24.57
Bottom	Bottom Left	0.04226	25.55
	Bottom Right	0.04172	25.22
Total		0.04135	100.0

The data confirms a vertical imbalance, with the two lower orifices processing 50.77% of the total flow. This suggests that the local mass flow is primarily driven by the static pressure

gradient. Despite the higher approach velocity at the outer wall, the significant static pressure deficit at the inner wall (required to balance centrifugal forces in the elbow) creates a larger local pressure drop across the lower orifices, effectively "drawing" more fluid through the bottom section of the diaphragm.

Visual Evidence of Symmetry Breaking

This imbalance is visually evident in the longitudinal development of the jets. Figure 5.13 presents a vertical cross-section of the velocity magnitude as the flow exits the orifices.



Figure 5.13: Longitudinal velocity magnitude contours downstream of the diaphragm for Configuration 2.

The inner wall jets exhibit a more robust structure compared to the outer wall jets. This confirms that the pressure-driven feeding at the bottom of the pipe results in higher momentum flux for the lower pair of orifices. This disparity creates a non-uniform velocity field in the mixing chamber, which could induce asymmetrical thermal or mechanical stresses on any equipment located downstream.

5.4 Sensitivity to Operating Temperature

After characterizing the impact of the 90-degree elbow at ambient temperature, it is essential to evaluate how these aerodynamic penalties evolve under high-temperature conditions (600 K and 900 K). This analysis determines whether the flow distortions induced by the curvature are exacerbated or attenuated by thermodynamic changes in the fluid. To ensure a universal comparison across temperatures, the mass flow data is presented as specific reduced mass flow ($\dot{m}_{spec} = \frac{\dot{m}\sqrt{T}}{PA}$), effectively decoupling the aerodynamic performance from the facility scale.

5.4.1 Thermal Impact on Mass Flow Capacity

The evolution of the specific reduced mass flow ($\dot{m}_{spec}$) for the three investigated temperatures in Configuration 2 is presented in Figure 5.14. As dictated by the Saint-Venant relations, the mass flow capacity of the system is strongly dependent on the stagnation temperature; the normalization by the passage area A allows for a direct comparison of the restrictor performance regardless of the thermal regime.

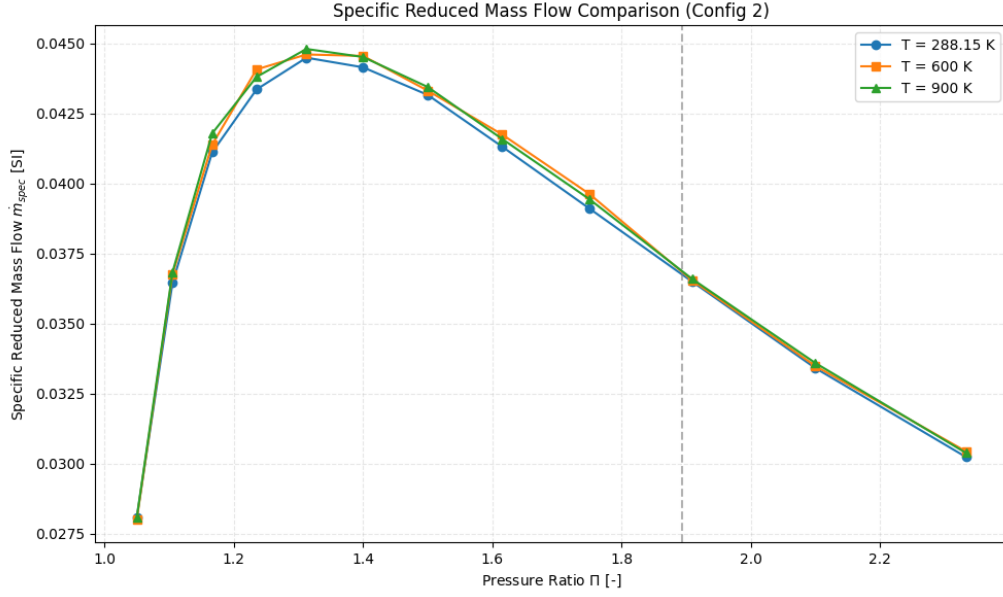


Figure 5.14: Specific reduced mass flow ($\dot{m}_{spec}$) as a function of the pressure ratio Π for three operating temperatures (Configuration 2).

For a constant pressure ratio, the collapse of the thermal curves indicates that the flow capacity is dominated by compressibility and choking effects rather than scaling variance. The variations observed at lower pressure ratios are physically driven by the reduction in gas density ($\rho \propto 1/T$), which is accounted for in this specific normalization.

5.4.2 Reynolds Independence and C_d Stability

To isolate the purely aerodynamic effect of the elbow from these thermodynamic variations, the discharge coefficient (C_d) is analyzed. Figure 5.15 shows the comparison of C_d curves for the three temperature levels.

The results demonstrate a remarkable stability of the discharge coefficient across the entire thermal range. Despite the significant changes in fluid properties—notably the increase in dynamic viscosity μ with temperature according to Sutherland’s law—the C_d curves remain almost perfectly superimposed, fluctuating within a narrow band between 0.79 and 0.81.

This collapse of the curves onto a single master profile leads to two major physical conclusions:

- **Inertial Dominance:** In the investigated operating range, the flow is characterized by high Reynolds numbers. At these scales, the pressure losses and flow redistributions induced by the elbow are dominated by inertial and centrifugal forces rather than viscous effects.
- **Universal Penalty:** The performance degradation caused by the 90-degree elbow is a purely geometric and aerodynamic phenomenon. The 4% mass flow penalty and the 0.04 drop in C_d (relative to the straight-pipe baseline) can be considered constant for this installation, regardless of the thermal conditions of the primary air line.

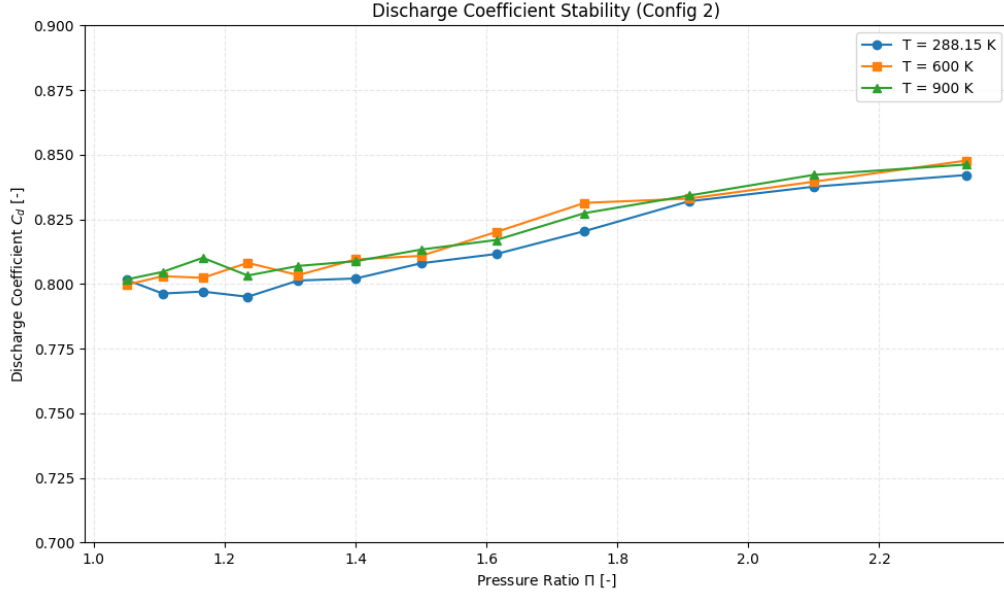


Figure 5.15: Discharge coefficient (C_d) as a function of the pressure ratio for 288.15 K, 600 K, and 900 K.

5.4.3 Synthesis of the Elbow Impact

The study of Configuration 2 confirms that the performance of a multi-orifice restrictor plate is highly sensitive to its installation environment but relatively insensitive to temperature variations once normalized by the C_d . The centrifugal forces in the elbow trigger a persistent distortion and a pair of Dean vortices that break the symmetry of the downstream jets and reduce the overall efficiency. These findings provide a robust basis for the final phase of this study, where the complete system including the annular ejector will be characterized.

Chapter 6

Phase 2: Full System Analysis and Ejection Performance

Following the characterization of the upstream piping effects, this final phase investigates the complete BeCOVER collector system. This chapter analyzes the interaction between the primary high-speed jets, the divergent section geometry, and the secondary air entrainment within the annular ejector.

6.1 Flow Topology in the Common Divergent

The common divergent section acts as the interface where the high-pressure streams from the two supply lines interact. Due to the extreme velocity gradients—ranging from near-stagnant zones to supersonic jets—a multi-scale visual analysis is required to capture the full complexity of the flow propagation.

6.1.1 Primary Jet Expansion and Internal Recirculations

The expansion of the eight primary jets represents the "engine" of the ejector system. To establish a clear spatial reference for the subsequent two-dimensional cross-sectional analyses, a global three-dimensional isometric view illustrating the exact location and orientation of the longitudinal cutting plane within the full collector assembly is presented in Figure 6.1. This plane passes directly through the centerlines of the multi-hole restrictor diaphragms, capturing the complete development axis of the high-speed streams.

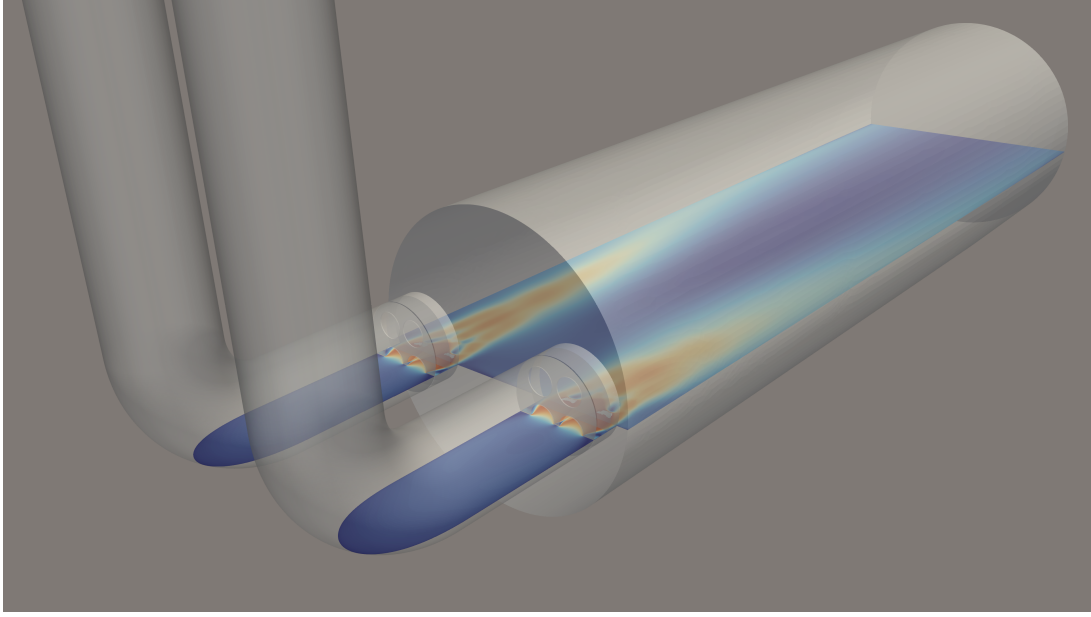


Figure 6.1: Three-dimensional isometric view of the assembly, showing the longitudinal cutting plane location used for the extraction of the 2D flow fields.

The global flow structure along this plane is first analyzed using a full velocity scale (Figure 6.2), where the supersonic cores originating from the diaphragms are clearly identified.

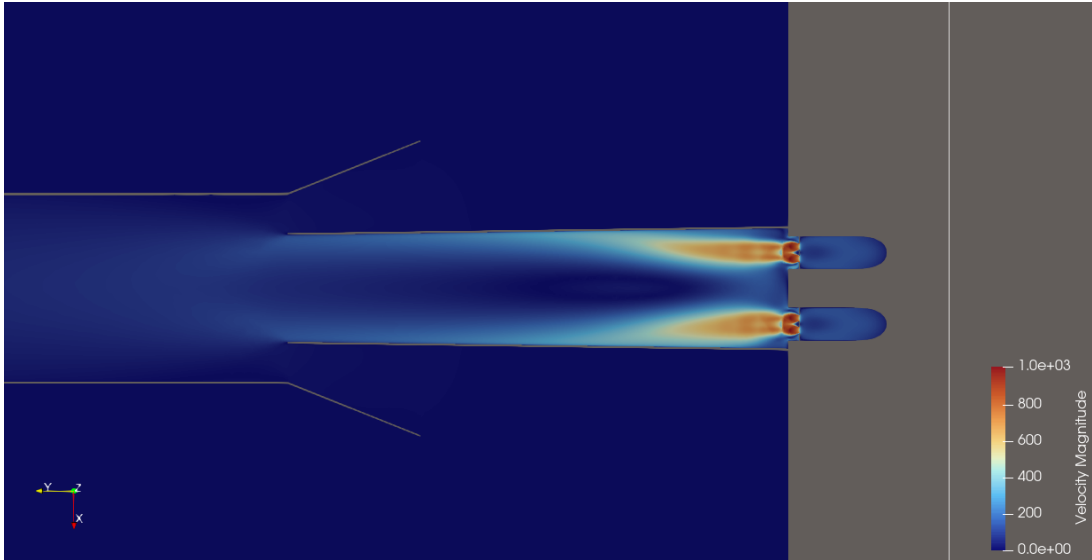


Figure 6.2: Global longitudinal velocity magnitude (0–1000 m/s) showing the high-energy primary expansion.

To understand the interaction between these jets and the divergent geometry, a filtered scale up to 300 m/s is employed (Figure 6.3). This visualization reveals a critical topological feature: the most energetic jets tend to "follow" the divergent walls. This behavior, induced by the dual-line entry geometry, leaves a massive, low-velocity recirculation region in the center of the chamber. This central "dead zone" indicates that the two primary streams do not merge immediately upon entry.

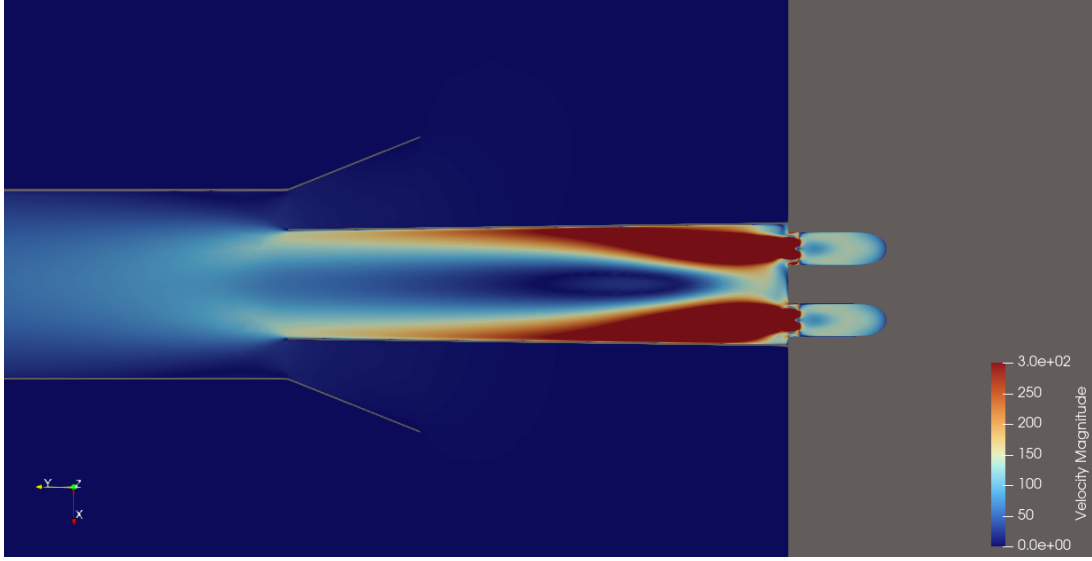


Figure 6.3: Filtered longitudinal velocity (0–300 m/s) highlighting the wall-hugging jets and the large central recirculation zone.

At the very start of the expansion, localized flow structures are visible at the diaphragm-divergent interface. A detailed view using Surface LIC (Figure 6.4) shows small toroidal vortices forming at the entry corners. These separations are the initial sites of energy dissipation and play a role in the initial spreading rate of the primary jets.

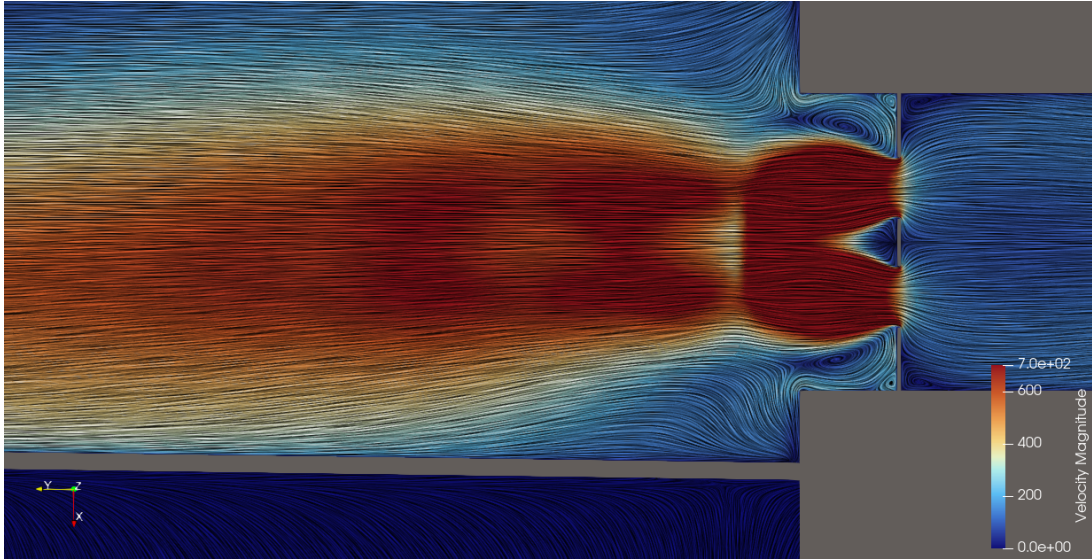


Figure 6.4: Surface LIC zoom at the diaphragm exit, showing localized recirculation zones at the divergent entry corners.

6.1.2 Secondary Air Suction and Mixing Uniformity

The actual ejection effect occurs at much lower velocity magnitudes. By filtering the longitudinal view to 100 m/s (Figure 6.5), the shear layers between the high-speed primary air and the entrained ambient air become apparent. This is where the momentum transfer drives the secondary air toward the mixing pipe.

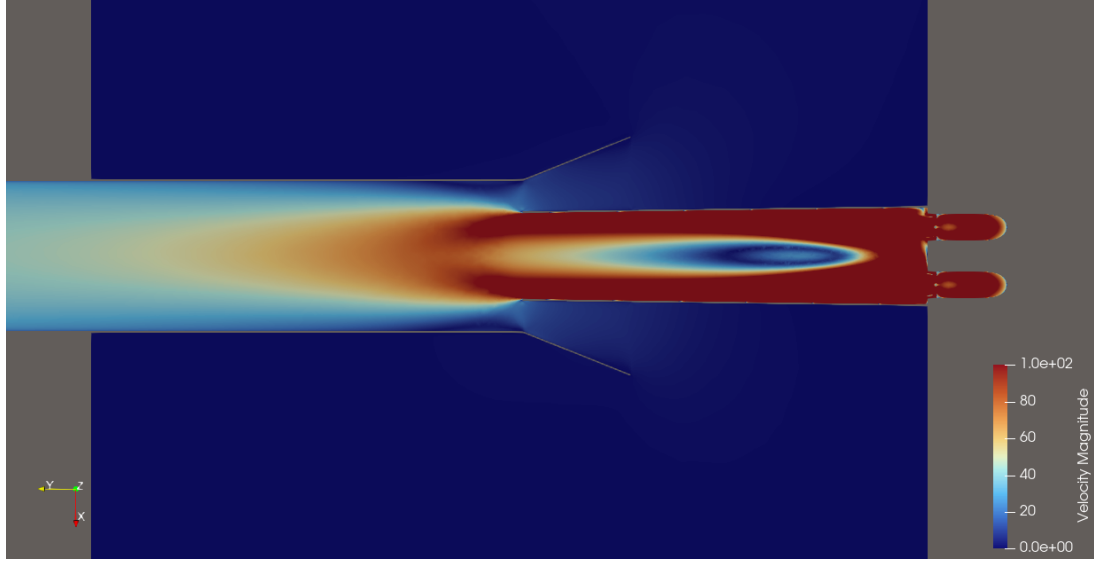


Figure 6.5: Interaction between primary jets and secondary air at low scale (0–100 m/s).

This interaction is further quantified via transverse cross-sections at the ejector throat. Due to the high velocity contrast, separate scales are utilized to confirm the aerodynamic decoupling between the primary jet cores and the surrounding secondary entrainment layer. Note that these contours represent the overall velocity magnitude captured on the transverse plane.

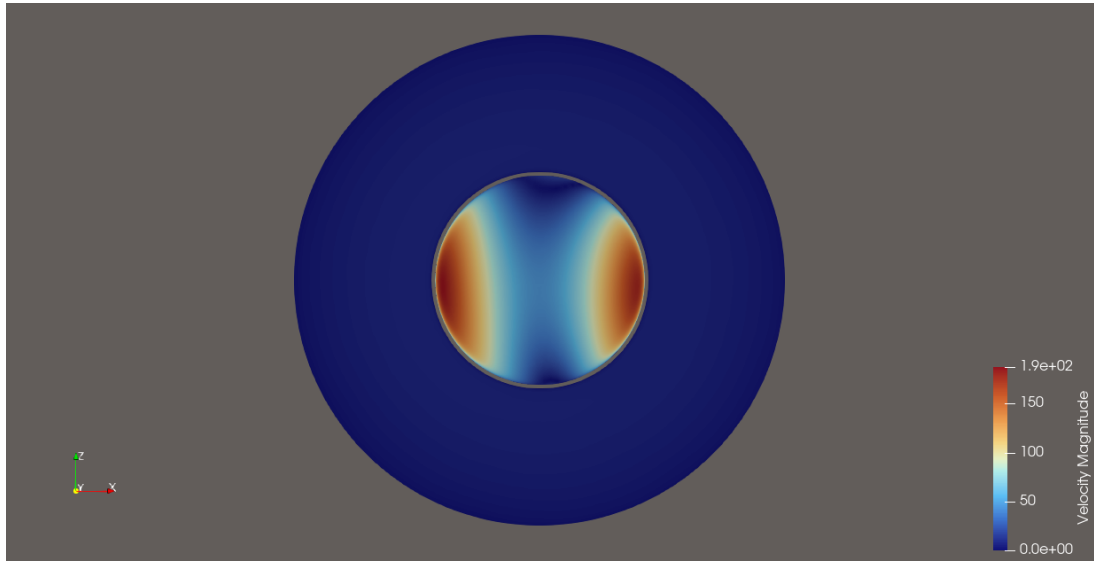


Figure 6.6: Velocity magnitude contours of the primary core (0–190 m/s) on a transverse cross-section, showing the persistent "twin-spot" asymmetry.

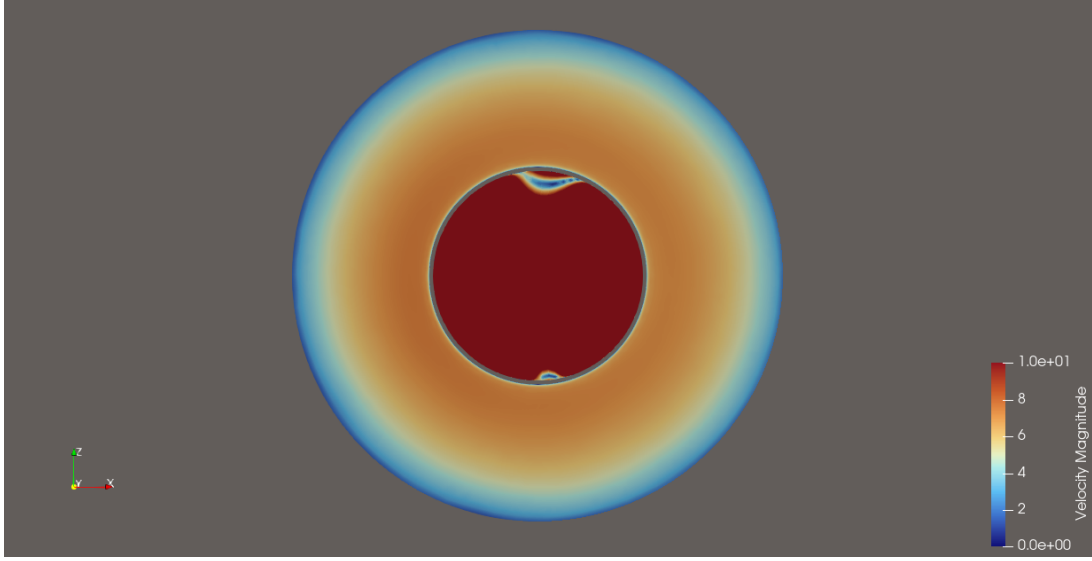


Figure 6.7: Velocity magnitude contours of the secondary suction (0–10 m/s) on a transverse cross-section, demonstrating a uniform entrainment ring.

The comparison between Figure 6.6 and Figure 6.7 is a key finding for the BeCOVER system. It proves that despite the residual asymmetry of the hot primary jets, the suction of cool ambient air remains perfectly balanced along the outer periphery. This outer uniformity ensures the formation of a homogeneous "cold envelope" that protects the mixing pipe walls from direct contact with the asymmetric primary cores at the start of the mixing process.

To resolve the exact kinematic paths driving this entrainment, a Surface Line Integral Convolution (LIC) visualization of the in-plane velocity vectors is presented in Figure 6.8.

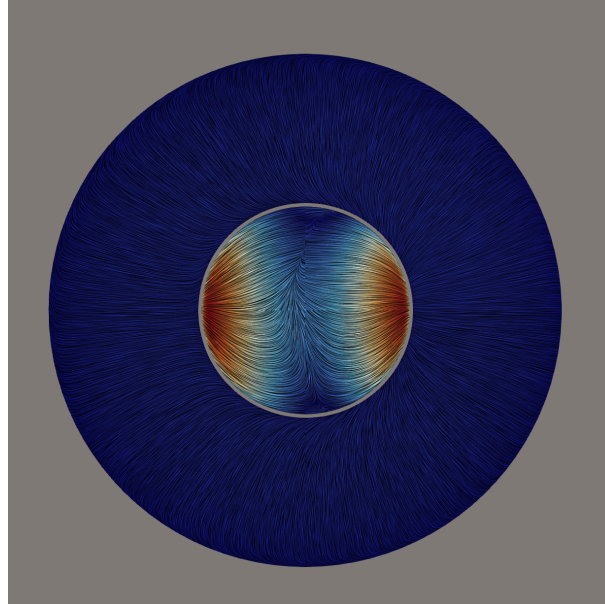


Figure 6.8: Surface LIC visualization of the in-plane velocity vectors on the transverse cross-section, highlighting secondary air convergence and the central vertical updraft topology.

The LIC streamlines visually map how the ambient secondary air is drawn radially inward from the outer boundaries, tracking the viscous momentum transfer. Near the central axis, the streamlines interact with the asymmetric twin primary cores, generating a highly distinct topological feature: the fluid paths originate from the high-velocity zones, converge toward the vertical symmetry axis, and merge into a strong upward motion. This behavior establishes a pair of symmetric, counter-rotating secondary flow structures—resembling a standard and a mirrored "J-shaped" streamline pattern on either side of the vertical axis. This central updraft demonstrates that the momentum mismatch between the two main primary supply lines triggers localized cross-flow transport and fluid folding at the throat before complete axisymmetrical homogenization is achieved further downstream.

6.2 Quantifying the Suction Effect

The primary objective of the annular ejector is to entrain sufficient ambient air to cool the exhaust gases. This section quantifies the performance of the system by analyzing the relationship between the primary supply pressure and the resulting specific reduced mass flow rates, while validating the numerical consistency across the investigated thermal range using universal scaling.

6.2.1 Global Performance and Mass Flow Consistency

For each simulation point, the mass flow balance was verified at the outlets using the conservation of mass principle. The numerical error remained below 0.1% for all cases, confirming the high level of convergence of the SU2 solver on the 28-million-cell mesh.

The discharge capacity of the complete system is characterized by the discharge coefficient (C_d) shown in Figure 6.9. For both temperatures, the C_d remains remarkably stable around 0.83 for all pressure ratios above 3.0. In comparison, the results from Phase 1 (Configuration 2 with elbow) showed a C_d starting at 0.79 for low pressure ratios before stabilizing between 0.84 and 0.85 for pressure ratios above 2.0. The consistency between these values confirms that the integration of the annular ejector and the discharge pipe has a negligible impact on the primary discharge capacity, validating the modularity of the design.

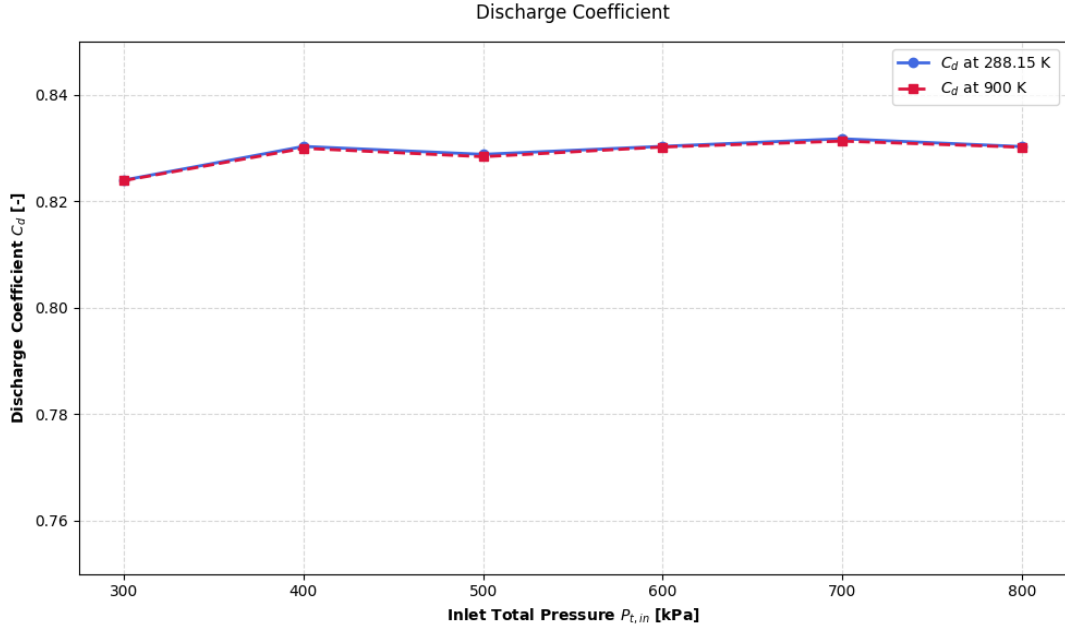


Figure 6.9: Discharge coefficient C_d vs. inlet total pressure, showing the stability at 0.83 and the comparison with Phase 1 results.

As illustrated in Figure 6.10, both the primary and secondary flows, expressed as specific reduced mass flows ($\dot{m}_{spec}$), exhibit a stable characteristic behavior across the inlet pressure range. For the primary flow, the reaching of an asymptotic plateau confirms that the diaphragm orifices are operating in a fully choked (sonic) regime. For the secondary flow, this stability indicates that the entrainment mechanism effectively scales with the primary momentum, maintaining a consistent suction efficiency across the investigated pressure range.

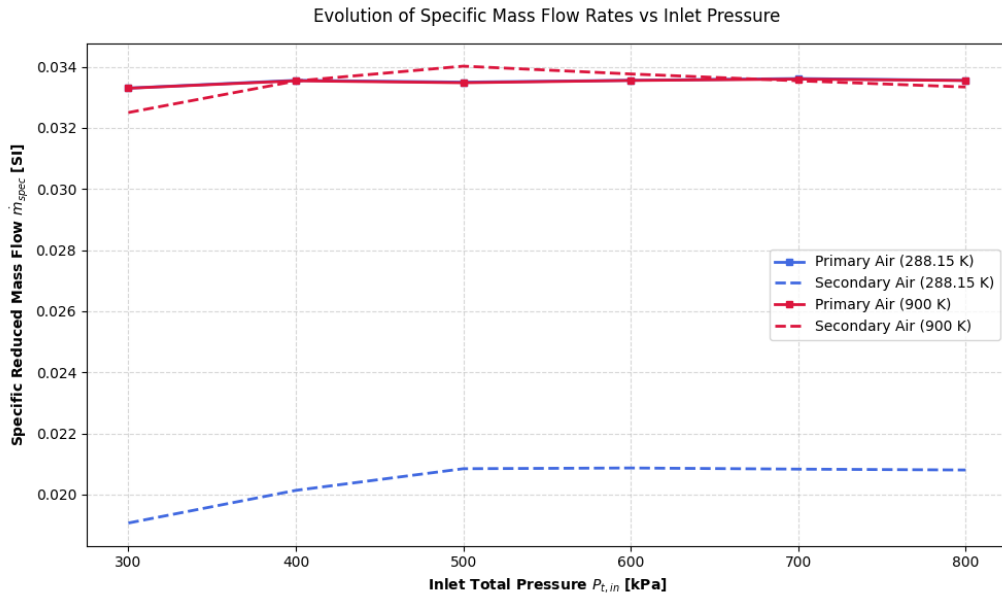


Figure 6.10: Evolution of primary and secondary specific reduced mass flows vs. inlet total pressure, demonstrating the choked flow plateau and entrainment stability.

Furthermore, comparing the non-dimensionalized data at 288.15 K and 900 K validates the isentropic scaling law. The specific reduced mass flow ($\dot{m}_{spec}$) is defined as:

$$\dot{m}_{spec} = \frac{\dot{m}\sqrt{T}}{P_{in} \cdot A} \quad (6.2.1)$$

The CFD results show a deviation of less than 0.01 % between the specific reduced mass flow calculated at 288.15 K and 900 K. This agreement confirms that the thermal variations are fully captured by the similarity laws, establishing a robust link between the two investigated temperatures without relying on absolute mass flow measurements.

6.2.2 Entrainment Ratio Analysis

The efficiency of the ejector is quantified by the entrainment ratio (ω), which represents the mass of ambient air suctioned per unit of primary driving fluid. The geometric configuration of the annular ejector is characterized by a primary-to-secondary area ratio of $AR \approx 0.49$. To ensure the analysis is independent of the facility scale, all mass flow rates are expressed as specific reduced mass flows ($\dot{m}_{spec} = \frac{\dot{m}\sqrt{T}}{P \cdot A}$).

Table 6.1 summarizes the performance data for the complete system. For the 900 K case, the simulation at 200 kPa did not achieve a stable converged state, as the primary momentum was insufficient to overcome the internal resistances of the discharge pipe.

Table 6.1: Summary of specific reduced mass flow rates and entrainment ratios for the complete system (Area Ratio $AR \approx 0.49$).

P_{in} [kPa]	288.15 K			900 K		
	$\dot{m}_{pri,spec}$	$\dot{m}_{sec,spec}$	ω [-]	$\dot{m}_{pri,spec}$	$\dot{m}_{sec,spec}$	ω [-]
200	0.0345	0.0186	0.54	—	—	—
300	0.0360	0.0206	0.57	0.0345	0.0337	0.98
400	0.0364	0.0218	0.60	0.0345	0.0345	1.00
500	0.0363	0.0226	0.62	0.0345	0.0351	1.02
600	0.0363	0.0226	0.62	0.0345	0.0347	1.01
700	0.0363	0.0225	0.62	0.0345	0.0345	1.00
800	0.0363	0.0225	0.62	0.0345	0.0343	0.99

6.2.3 Impact of Temperature on Ejection Efficiency

A significant finding of this study is the impact of gas temperature on the entrainment ratio. While the absolute value of the entrained secondary flow remains relatively stable between the two temperatures at a given pressure, the entrainment ratio ω undergoes a drastic shift, as shown in Figure 6.11.

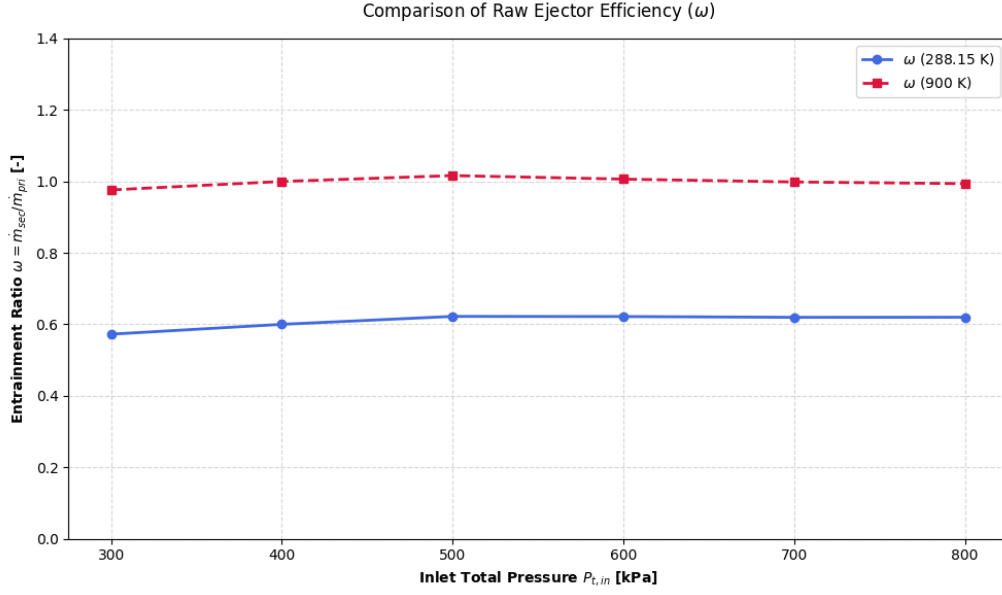


Figure 6.11: Comparison of the raw entrainment ratio (ω) between 288.15 K and 900 K. The ratio jumps from 0.62 to 1.00 primarily due to the reduction in driving primary mass flow.

This shift from 0.62 to 1.00 is explained by the change in the momentum-to-mass ratio. At higher temperatures and constant pressure, the primary mass flow rate decreases ($\dot{m} \propto 1/\sqrt{T}$), while the jet exit velocity increases ($V \propto \sqrt{T}$). Since the suction effect is driven by momentum transfer ($\dot{m}V$), the "pumping force" remains nearly constant while the mass of "driving" fluid is reduced, resulting in a higher raw ratio.

However, to assess the true aerodynamic efficiency of the mixing process, the entrainment ratio must be corrected by the square root of the temperature ratio:

$$\omega_{corr} = \omega \cdot \sqrt{\frac{T_{amb}}{T_{pri}}} \quad (6.2.2)$$

As illustrated in Figure 6.12, the corrected curves do not perfectly collapse. The corrected efficiency at 900 K ($\omega_{corr} \approx 0.57$) is approximately 8% lower than at 288.15 K ($\omega_{corr} \approx 0.62$).

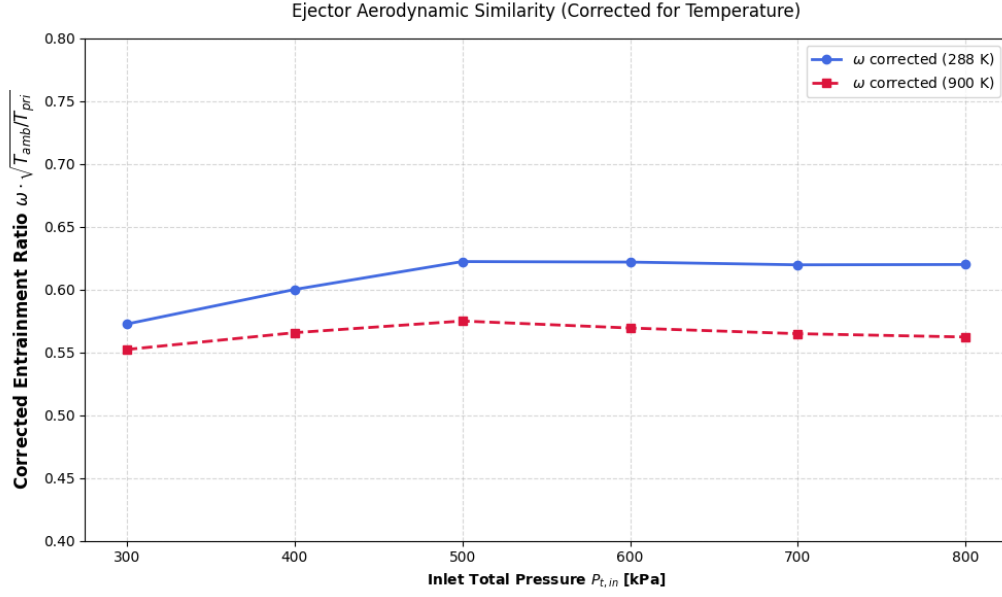


Figure 6.12: Corrected entrainment ratio (ω_{corr}) showing the similarity breakdown. The 8 % drop at high temperature reveals a degradation in momentum transfer efficiency.

This 8 % discrepancy reveals a fundamental difference in the mixing layer physics. At 900 K, the primary jet is significantly less dense than the ambient air ($\rho_{pri}/\rho_{amb} \approx 0.32$). This high density gradient, combined with the increased kinematic viscosity (Sutherland's law), alters the shear layer development. The hot, lighter jet occupies more volume in the mixing chamber and is slightly less effective at entraining the heavy ambient air compared to a cold jet of equal momentum.

This finding is critical: it proves that a simple linear interpolation of raw mass flow rates would be physically inaccurate, as it would fail to account for this thermal degradation of ejection efficiency.

6.2.4 Development of a Predictive Model for a Continuous Operating Envelope

The numerical results presented in the previous sections provide a high-fidelity characterization of the system at specific operating points. However, for the operational needs of the BeCOVER test bench, it is essential to define a continuous model capable of predicting mass flows and cooling efficiency across any combination of inlet pressure (P_{in}) and temperature (T).

Thermodynamic Scaling and Master Curve

The primary mass flow prediction is based on the "Master Curve" established through non-dimensional analysis. As demonstrated in Figure 6.13, the specific reduced mass flow ($\dot{m}_{spec}$) remains remarkably constant across the investigated pressure range, validating the use of a single scaling law for the entire facility, independent of scale.

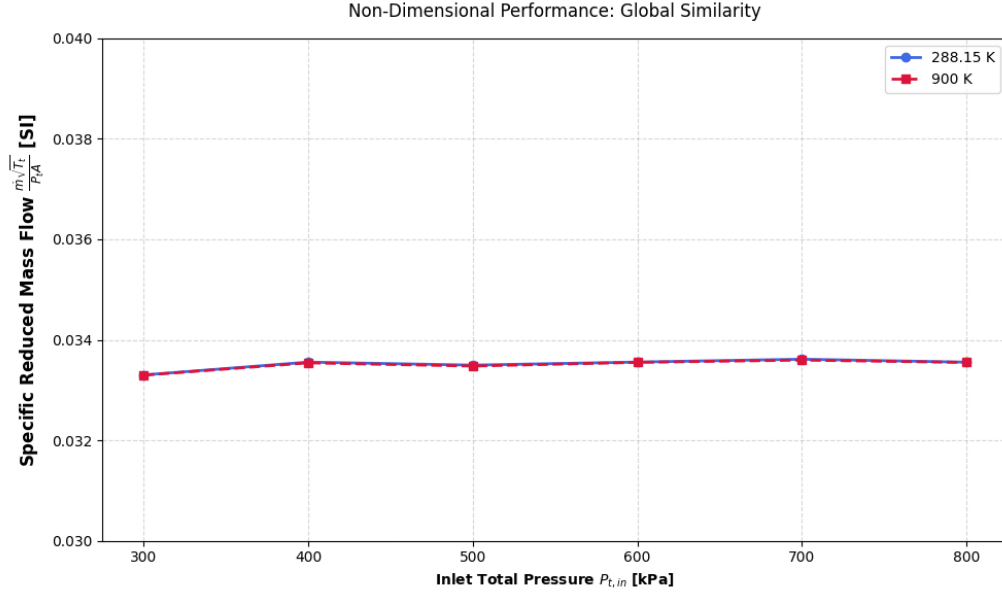


Figure 6.13: Non-dimensional primary mass flow (Master Curve). The perfect collapse of the cold and hot datasets allows for a universal prediction of air consumption using the specific reduced mass flow scaling law.

By utilizing the stabilized specific reduced mass flow value ($\dot{m}_{spec} \approx 0.0335$), the primary air consumption for any arbitrary condition (P_x, T_x) is calculated following the Saint-Venant similarity:

$$\dot{m}_{pri}(P_x, T_x) = \dot{m}_{spec} \cdot \frac{P_x \cdot A}{\sqrt{T_x}} \quad (6.2.3)$$

This approach physically accounts for the square-root dependence of the choked flow on temperature, while the inclusion of the passage area (A) ensures universal scaling. Given the stability of the discharge coefficient, this formula is sufficient for operational predictions across any facility scale.

Physics-Based Interpolation Strategy

To create a continuous model for the entrainment ratio ω , the tool must account for the non-linear relationship between temperature and momentum transfer. First, the **corrected efficiency** (ω_{corr}) is interpolated linearly between 288.15 K and 900 K. As shown in Figure 6.14, this isolates the density-driven degradation of the mixing process, resulting in an equidistant distribution of the temperature curves.

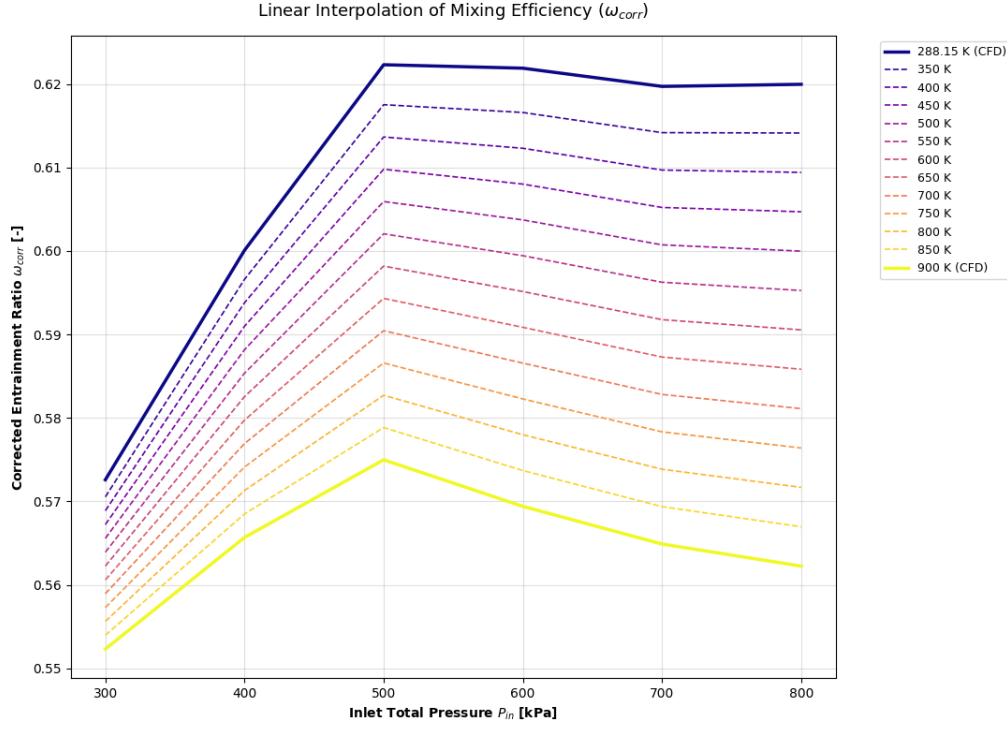


Figure 6.14: Linear interpolation of the corrected entrainment ratio ω_{corr} . The regular spacing between curves represents the assumption of a linear degradation of mixing efficiency with temperature.

Second, the operational entrainment ratio is reconstructed by reintroducing the thermodynamic scaling:

$$\omega(T_x) = \omega_{corr}(T_x) \cdot \sqrt{\frac{T_x}{288.15}} \quad (6.2.4)$$

Operational Implementation for the BeCOVER Bench

The final result is the **Ejector Efficiency Operating Map** (Figure 6.15). Unlike the corrected map, the curves of the real entrainment ratio ω exhibit a non-linear distribution, where the performance gain per Kelvin decreases at higher temperatures, as evidenced by the tightening of the upper curves.

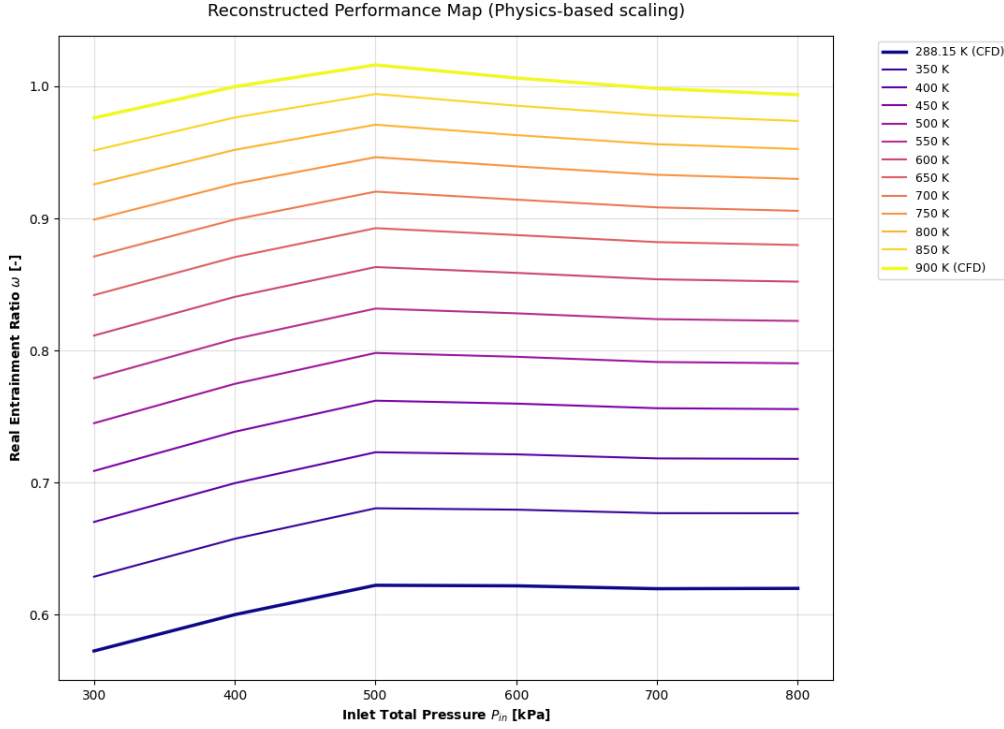


Figure 6.15: Final Reconstructed Performance Map (ω vs P_{in}). The non-linear spacing between curves demonstrates the respect of the $\sqrt{T}$ scaling law for momentum-driven suction.

Both maps are restricted to $P_{in} \geq 300$ kPa to exclude the unstable regime below the priming threshold. By consulting these tools, operators can estimate the total specific exhaust flow ($\dot{m}_{out,spec} = \dot{m}_{spec} \cdot (1 + \omega)$) and predict the mixing temperature via the enthalpy balance. For higher numerical precision, the analytical equations are directly implemented in the control system.

6.3 Thermal Mixing Analysis and Pipe Integrity

While the previous sections validated the mass flow entrainment, this section examines the thermal consequences of mixing a 900 K primary stream with ambient air. The primary concern for the BeCOVER facility is ensuring that the discharge pipe walls are not subjected to excessive temperatures that could compromise their structural integrity.

6.3.1 Longitudinal Heat Dissipation and the "Cold Envelope"

The thermal mixing process begins with the formation of a protective "buffer zone" of secondary air. As shown in Figure 6.16, the 900 K primary jets do not immediately contact the divergent walls. Instead, they expand into the low-velocity secondary stream, creating a high-gradient shear layer.

A detailed view of the ejector throat (Figure 6.17) reveals that the core of the jets remains at 900 K for a short distance downstream of the diaphragms (the "potential core" of the mixing zone). Beyond this point, the turbulent kinetic energy generated by the velocity mismatch ($\Delta V \approx 500$ m/s) triggers intense macroscopic mixing, causing the centerline temperature to decay. This dissipation is further enhanced by the temperature-dependent transport properties

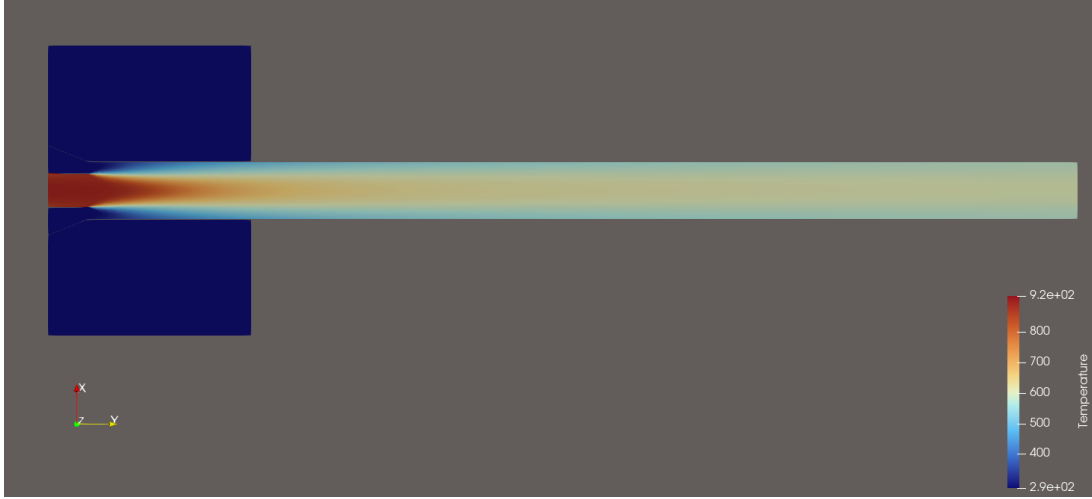


Figure 6.16: Longitudinal total temperature distribution (T_t) from the ejector to the discharge pipe outlet.

of the air. Following Sutherland's law, the dynamic viscosity of the 900 K primary core is significantly higher than that of the ambient stream. This viscous effect accelerates the expansion of the turbulent shear layer, promoting more rapid thermal diffusion from the core into the cold buffer zone.

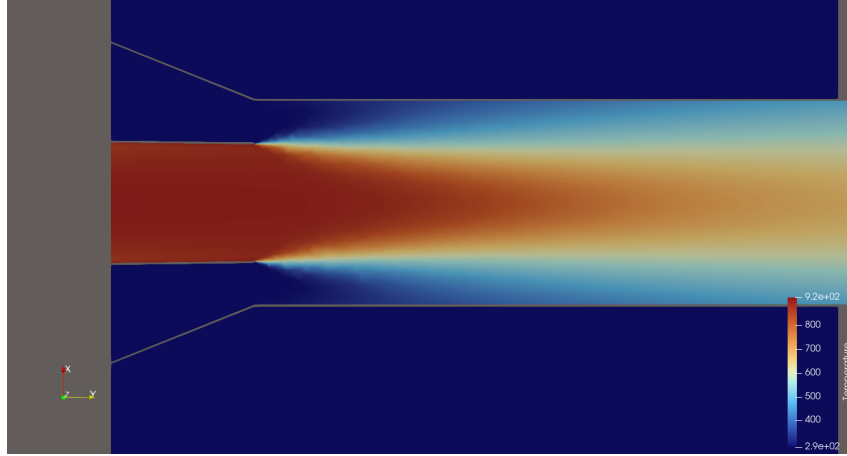


Figure 6.17: Detailed view of the thermal shear layer at the ejector exit, highlighting the protective secondary air "shielding" the walls.

6.3.2 Cross-Sectional Homogenization and Stratification

The evolution of the temperature profile is best captured through radial cross-sections (Figure 6.18). This visualization tracks the transition from discrete high-temperature spots to a quasi-homogeneous mixture.

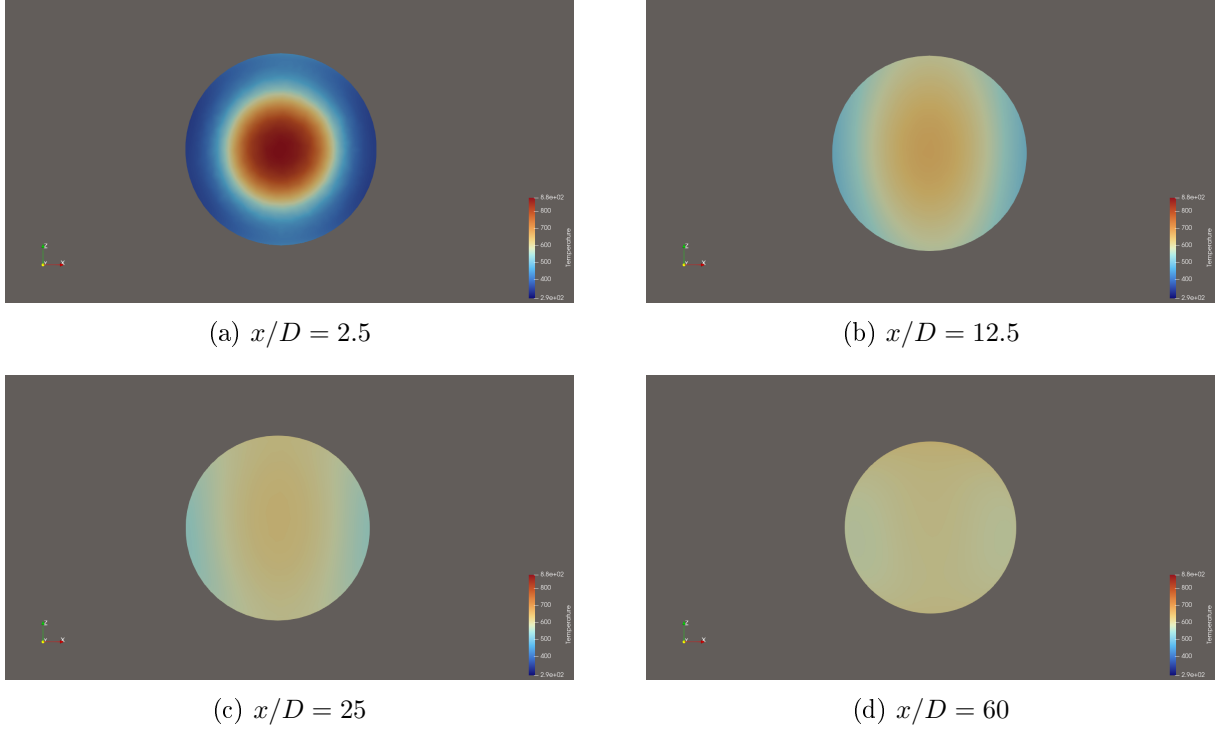


Figure 6.18: Evolution of total temperature radial distribution along the discharge pipe expressed in non-dimensional pipe diameters.

At the upstream measurement positions, the primary hot jet is clearly visible as a dominant central core, surrounded by a developing shear layer of entrained air. Further downstream, the thermal field evolves, and the initial core begins to broaden as it merges with the secondary stream, a direct consequence of the mixing dynamics within the divergent section. At the final outlet, while the mixing process is largely complete, a residual thermal stratification is observed. The thermal profile confirms that the core has been sufficiently diffused by the secondary air.

6.3.3 Theoretical Validation: The Adiabatic Mixing Limit

To validate the CFD results, the outlet temperature is compared to the theoretical adiabatic mixing temperature (T_{mix}). By expressing the enthalpy balance entirely in terms of the dimensionless entrainment ratio (ω), the final mixture temperature is given by:

$$T_{mix} = \frac{T_{pri} + \omega \cdot T_{sec}}{1 + \omega} \quad (6.3.1)$$

For the 600 kPa case ($T_{pri} = 900$ K, $\omega \approx 1.006$), the theoretical limit is approximately 593.2 K. The CFD simulation yields a mass-averaged temperature of 590.6 K at the outlet, with a local maximum of 610 K within the residual hot lobes.

The slight deviation (less than 0.5%) between the theoretical mean and the CFD result confirms the energy conservation of the numerical model. The difference between the peak temperature (610 K) and the mean (594 K) quantifies the "mixing efficiency": at the exhaust outlet, the system has achieved over 95% of its potential thermal homogenization. From an industrial standpoint, reducing the exhaust temperature from 900 K (627 °C) to a peak of 610 K (337 °C) ensures that the discharge piping operates well within the safe temperature limits for standard structural steels.

6.3.4 Operational Generalization: Pressure-Independent Cooling

A fundamental conclusion arises from the coupling between the entrainment ratio plateau and the thermal mixing physics. As established in Section 6.2, the ratio ω remains nearly constant for inlet pressures above 400 kPa.

Because the enthalpy balance is driven strictly by this ratio, it becomes evident that on the operational plateau, the final equilibrium temperature is independent of the absolute mass flow magnitude (and thus the inlet pressure). Whether the system operates at 400 kPa or 800 kPa, the cooling efficiency is solely dictated by the primary gas temperature T_{pri} and the ambient T_{sec} .

This self-regulating characteristic is a major asset for the BeCOVER facility. It ensures that increasing the primary supply pressure to achieve higher test Reynolds numbers does not lead to a linear increase in discharge temperature. As long as the ejector remains in its "saturated" regime, the thermal safety margin of the discharge piping remains constant, significantly simplifying the operational envelope of the test bench.

Chapter 7

Conclusions and Perspectives

This master thesis investigated the complex aerodynamic and thermal behavior of the BeCOVER high-pressure collector system through high-fidelity three-dimensional Computational Fluid Dynamics (CFD) simulations using the open-source SU2 solver. The study provided a comprehensive characterization of the highly distorted flow structures, pressure drop penalties, and turbulent thermal mixing phenomena across a wide and demanding operational envelope, converting raw numerical data into practical engineering solutions for the facility.

7.1 Summary of Key Findings

The research framework was executed in two distinct, sequential phases to isolate specific fluid dynamics challenges before evaluating the overall performance of the full collector facility under real-world operating conditions.

7.1.1 Phase 1: Impact of Upstream Piping and Flow Distortion

The first phase of the project quantified the severe aerodynamic penalty imposed by the 90° supply elbow situated immediately upstream of the multi-hole restrictor plates:

- **Dean Vortices and Velocity Asymmetry:** The strong curvature of the piping induces a secondary flow characterized by a counter-rotating vortex pair (Dean vortices). This secondary motion forces the high-velocity core toward the extrados of the elbow, resulting in a highly asymmetric, distorted velocity profile at the inlet of the restrictor plate diaphragms.
- **Discharge Capacity Penalization:** The severe flow distortion heavily degrades the vena contracta and restricts the effective flow area within the diaphragms. Consequently, the discharge coefficient (C_d) is reduced by approximately 4% in the elbow configuration compared to the idealized straight pipe reference case, directly penalizing the total mass flow capacity for identical upstream total pressures.
- **Reynolds and Thermal Sensitivity:** While the sonic choking threshold remains unaffected by the upstream layout ($\Pi = 1.89$), temperature sensitivity analyses demonstrated that higher gas temperatures slightly reduce the spatial footprint of the secondary separations. This variation is physically driven by the increase in fluid dynamic viscosity with temperature, which alters the local Reynolds number and dampens the recirculation bubbles.

7.1.2 Phase 2: Annular Ejector Dynamics and Thermal Mixing

The second phase evaluated the complete collector and exhaust assembly, focusing on the complex flow interactions inside the common divergent section and the annular mixing pipe:

- **Supersonic Jet Expansion:** The expansion of the eight supersonic primary jets into the collector creates a strong central recirculation zone ("dead zone"). The jets tend to closely follow the divergent walls due to a Coandă-like effect, delaying the momentum transfer and core shear interaction with the entrained secondary stream.
- **Symmetrical Suction Buffer:** Despite the residual flow asymmetry inherited from the dual upstream supply lines, the suction of cool secondary air from the ambient atmosphere self-stabilizes into a perfectly axisymmetric annular shield. This profile generates a continuous "cold envelope" that protects the structural integrity of the mixing pipe walls.
- **Entrainment Ratio Saturated Plateau:** The ejector enters a saturated aerodynamic regime for primary pressures above 400 kPa. The entrainment ratio (ω) stabilizes at 0.62 under cold conditions (288.15 K) and increases to a plateau of 1.006 under high-temperature conditions (900 K). This behavior confirms the thermodynamic balance where primary mass flow drops with $1/\sqrt{T}$ while the momentum-driven suction capacity remains invariant.
- **Thermal Homogenization:** At the outlet of the discharge pipe, the mixing efficiency exceeds 95 %. The mass-averaged temperature from the CFD settles at 590.6 K, deviating by less than 0.5 % from the ideal theoretical adiabatic mixing limit (593.2 K). The local peak temperature within the residual hot lobes reaches a maximum of 610 K, proving that the exhaust air is cooled well within the safe structural limit of standard industrial steels.

7.1.3 Methodological Considerations and Mesh Sensitivity

Given the rigorous constraints of a 16-week master thesis timeframe, a formal, multi-grid mesh convergence study (such as the Grid Convergence Index method) could not be fully completed. However, an intrinsic validation of the numerical consistency was achieved throughout the project.

As the author's proficiency with advanced meshing tools (Pointwise) matured during the internship, the mesh generation strategy for the Phase 2 full assembly became significantly more refined, high-density, and structured than the initial grids used in Phase 1. Despite this notable evolution in mesh refinement and spatial resolution, the core aerodynamic trends—most notably the restrictor discharge coefficients (C_d) and the choking pressure ratios—remained highly consistent between the phases. This cross-configuration stability strongly implies that the baseline spatial discretizations were already well-positioned within the asymptotic convergence region of the solver, ensuring that further grid refinement would yield negligible variations in engineering predictions.

7.2 Operational Envelope and Predictive Framework

To translate discrete, computationally expensive CFD data points into a ready-to-use engineering tool for the BeCOVER control room, a continuous analytical model was developed. Rather than relying on a naive linear interpolation of raw parameters, the framework implements a physics-based approach. It performs a linear interpolation on the corrected mixing efficiency (ω_{corr}) to isolate viscous and density variations, and subsequently reconstructs the operational entrainment ratio (ω) by reintroducing the non-linear $\sqrt{T}$ scaling law.

To guarantee maximum clarity for the bench operators, this predictive tool is decoupled into two complementary operational performance maps:

1. **The Primary Air Consumption Map:** Allows immediate evaluation of the specific reduced mass flow ($\dot{m}_{spec}$) against the inlet total pressure (P_{in}) across the entire thermal range. This ensures universal predictions independent of the facility scale, utilizing the choked flow linearity and the isentropic scaling law.
2. **The Ejector Efficiency Operating Map:** Displays the real entrainment ratio (ω) against P_{in} for temperatures ranging from 288.15 K to 900 K, clearly illustrating the non-linear "tightening" of the performance curves at high thermal regimes.

These tools allow operators to instantly predict specific exhaust mass flows and final mixing temperatures without the need for additional numerical simulations.

7.3 Perspectives for Future Work

Based on the numerical models and findings established in this thesis, several distinct paths are proposed for future industrial and academic work.

7.3.1 Alternative Layout: Single Orifice Plate Configuration

An important industrial evolution is currently being considered by the BeCOVER engineering team: the baseline 4-hole multi-hole restrictor plate may potentially be replaced by a single, large-diameter orifice plate design. While this modification might simplify manufacturing, it will fundamentally alter the development of the supersonic jets, the pressure drop characteristics, and the downstream flow distortion profile entering the collector. A crucial perspective for future work is therefore to adapt the developed SU2 CFD framework to this alternative single-orifice geometry, establishing a direct performance comparison against the multi-hole layout analyzed in this study.

7.3.2 Foundation for a 1D Digital Twin

The continuous analytical laws, scaling correlations, and performance curves derived in this work are structured to serve a broader purpose. They provide the foundational boundary conditions and aerodynamic characterization laws required to build a system-level 1D Digital Twin of the complete BeCOVER facility. This thesis directly prepares the groundwork for a subsequent intern, whose objective will be to implement these physics-based equations into a 1D transient simulation environment, enabling real-time performance monitoring and fast predictive scheduling of test campaigns.

7.3.3 Experimental Commissioning and Model Validation

Finally, the upcoming physical commissioning phase of the BeCOVER test bench offers an ideal opportunity for direct experimental verification. Comparing the pressure drop penalties, mass flow master curves, and cross-sectional temperature profiles generated in this work with physical sensor data will serve as the ultimate validation of the numerical methodology. This feedback loop will allow for further calibration of the turbulence closure models and solidify the predictive framework as a certified operational standard for the facility.

Appendix A

Solver and HPC Configuration

This appendix details the numerical setup used for the RANS simulations within the SU2 environment. Beyond the raw configuration file, specific technical choices regarding numerical stability and data management are justified hereafter.

A.1 Technical Rationales for Solver Settings

While the configuration file (Section A.3) follows standard RANS practices, three specific settings were crucial for the success of the BeCOVER study:

- **Low-Mach Preconditioning:** The `LOW_MACH_PREC = YES` option was enabled to ensure numerical accuracy in the large mixing sections and the atmospheric farfield, where velocities are significantly below Mach 0.1. Without preconditioning, the pressure-velocity coupling in compressible solvers becomes poorly conditioned at low speeds, leading to slow convergence and potential inaccuracies in the pressure field.
- **Slope Limiter:** The `VENKATAKRISHNAN` limiter was selected for the second-order spatial discretization. It provides an excellent compromise between numerical stability—preventing oscillations in high-gradient regions such as the restrictor orifices—and the preservation of second-order accuracy in the shear layers of the ejector.
- **Hybrid Marker Management:** The definition of markers using both `tri` and `quad` elements (e.g., `MARKER_INLET`) reflects the hybrid nature of the T-Rex unstructured mesh. This allows SU2 to correctly identify the prismatic layers generated near the walls for boundary layer resolution, ensuring that the anisotropic refinement is fully exploited by the solver.

A.2 Mesh Format and Data Efficiency

The CGNS (CFD General Notation System) format was preferred over the native SU2 mesh format for all simulations. This choice was primarily driven by storage efficiency and the management of large-scale datasets.

As the complexity of the configurations increased, the binary nature of CGNS provided a significant reduction in file size without any loss of geometric precision. For instance, in Configuration 1, the file size comparison was as follows:

- **SU2 Format:** 198 574 kB ($\approx$ 198.6 MB)
- **CGNS Format:** 129 200 kB ($\approx$ 129.2 MB)

This represents a reduction of approximately **35%** in disk space usage, which was critical for managing the 28-million-cell grids of the full system analysis (Configuration 3) on the NIC5 cluster.

A.3 SU2 Configuration File Sample

The following listing presents the configuration parameters for the steady-state simulation of the full collector at 900 K.

```
% --- SOLVER SETUP ---
SOLVER= RANS
KIND_TURB_MODEL= SA
MATH_PROBLEM= DIRECT
RESTART_SOL= NO

% --- FLUID PHYSICS AND INITIALIZATION ---
REF_DIMENSIONALIZATION= DIMENSIONAL
INIT_OPTION = TD_CONDITIONS
FREESTREAM_TEMPERATURE= 288.15
FREESTREAM_PRESSURE= 101325.0
MACH_NUMBER= 0.001
REF_LENGTH= 1.0
LOW_MACH_PREC= YES

% --- BOUNDARY CONDITIONS ---
MARKER_INLET= ( tri_Inlet, 900, 300000.0, 0.0, 0.0, -1.0, quad_Inlet,
  900, 300000.0, 0.0, 0.0, -1.0 )
MARKER_OUTLET= ( tri_Outlet, 101325.0, quad_Outlet, 101325.0 )
MARKER_FAR= ( tri_Farfield, quad_Farfield )
MARKER_HEATFLUX= ( tri_Walls, 0.0, quad_Walls, 0.0 )
MARKER_MONITORING= ( tri_Inlet, quad_Inlet, tri_Outlet, quad_Outlet,
  tri_Farfield, quad_Farfield, tri_Walls, quad_Walls )

% --- NUMERICAL CONTROLS ---
ITER= 1000
CFL_NUMBER= 50
CFL_ADAPT= YES
CFL_ADAPT_PARAM= ( 0.5, 1.2, 0.1, 200.0, 0.001 )
CONV_FIELD= RMS_DENSITY
CONV_RESIDUAL_MINVAL= -10

% --- CONVECTIVE SCHEMES ---
CONV_NUM_METHOD_FLOW= SLAU2
MUSCL_FLOW= YES
SLOPE_LIMITER_FLOW= VENKATAKRISHNAN
TIME_DISCRE_FLOW= EULER_IMPLICIT

% --- I/O AND PERFORMANCE ---
MESH_FILENAME= ConfigFinale.cgns
MESH_FORMAT= CGNS
OUTPUT_WRT_FREQ= 100
OUTPUT_FILES= ( RESTART, PARAVIEW, SURFACE_PARAVIEW, SURFACE_CSV )
CUSTOM_OUTPUTS= 'Debit_Inlet : MassFlowInt{1}[tri_Inlet, quad_Inlet];
  Debit_Outlet : MassFlowInt{1}[tri_Outlet, quad_Outlet];
  Debit_Farfield : MassFlowInt{1}[tri_Farfield, quad_Farfield]'
```

Appendix B

High-Performance Computing Setup

The computational demands of this study, particularly for the 28-million-cell mesh of Configuration 3, required the use of the NIC5 supercomputer. This appendix provides the exact Slurm submission script used to manage the resources and the data workflow on the cluster.

B.1 Slurm Submission Script

The following script requests 256 cores on the `batch` partition. A key feature of this setup is the use of a local `SCRATCH` directory to minimize I/O (Input/Output) overhead on the global file system during the calculation.

```
#!/bin/bash
#SBATCH --job-name=900K3
#SBATCH --ntasks=256
#SBATCH --cpus-per-task=1
#SBATCH --mem-per-cpu=4G
#SBATCH --time=0-24:00:00
#SBATCH --partition=batch

% --- Environment Setup ---
module --force purge
module load releases/2023b
module load GCC/13.2.0
module load OpenMPI/4.1.6-GCC-13.2.0

export PATH=$HOME/.local/bin:$PATH

% --- Scratch Directory Management ---
SCRATCH=/scratch/ulg/desturb/tmarc/job_${SLURM_JOB_ID}
mkdir -p $SCRATCH

% Copying input files to scratch
cp ConfigFinale.cfg ConfigFinale.cgns solution.dat $SCRATCH/
cd $SCRATCH

% --- Execution ---
mpirun -n $SLURM_NTASKS SU2_CFD ConfigFinale.cfg

% Transferring results back to the submission directory
cp history.csv flow.vtu restart.dat $SLURM_SUBMIT_DIR/
```

B.2 Technical Rationale for the Workflow

The submission strategy includes several advanced HPC practices:

- **Module Stack:** The use of `releases/2023b` and `GCC/13.2.0` ensures compatibility with the latest optimized libraries for high-speed interconnects on NIC5.
- **I/O Optimization:** By creating a temporary directory on the `/scratch` partition, the simulation avoids "bottlenecking" the main storage during the frequent writing of `history.csv` and large `restart.dat` files.
- **Scalability:** With 256 tasks (`ntasks=256`), the computational load is balanced to maintain high parallel efficiency for the SU2 solver's domain decomposition.

Bibliography

- [1] *About Us*. Acare. URL: <https://www.acare4europe.org/about-us/> (visited on 05/04/2026).
- [2] Directorate-General for Mobility and Transport (European Commission) and Directorate-General for Research and Innovation (European Commission). *Flightpath 2050 :Europe's Vision for Aviation : Maintaining Global Leadership and Serving Society's Needs*. Publications Office of the European Union, 2011. ISBN: 978-92-79-19724-6. URL: <https://data.europa.eu/doi/10.2777/50266>.
- [3] *BeCover - Compressor Test Facility Promoting Environment & Research*. BeCover. URL: <https://becover.eu/> (visited on 05/04/2026).
- [4] Charlie Young P.E. *Boundary Layer in Pipe Flow: A Comprehensive Analysis*. EngineerExcel. Apr. 4, 2023. URL: <https://engineerexcel.com/boundary-layer-in-pipe-flow/> (visited on 04/15/2026).
- [5] Yoshiharu Tamaki, Motoshi Harada, and Taro Imamura. "Near-Wall Modification of Spalart–Allmaras Turbulence Model for Immersed Boundary Method". In: *AIAA Journal* 55.9 (Sept. 2017), pp. 3027–3039. ISSN: 0001-1452. DOI: 10.2514/1.J055824. URL: <https://arc.aiaa.org/doi/10.2514/1.j055824> (visited on 04/20/2026).
- [6] Jianyi Zhang et al. "Numerical Investigation and Optimization of the Flow Characteristics of Bend Pipe with Different Bending Angles". In: *Processes* 10 (Aug. 1, 2022), p. 1510. DOI: 10.3390/pr10081510.
- [7] *Researchers Quantify the Onset of Turbulence in a Pipe Bent Back on Itself*. URL: <https://phys.org/news/2023-12-quantify-onset-turbulence-pipe-bent.html> (visited on 04/15/2026).
- [8] Admin. *Orifice Plates & Orifice Flanges: Complete Guide to Flow Measurement Systems*. Oct. 8, 2025. URL: <https://www.piping-world.com/orifice-plates-and-orifice-flanges-for-flow-measurement> (visited on 04/15/2026).
- [9] *ISO 5167-1:2022(En), Measurement of Fluid Flow by Means of Pressure Differential Devices Inserted in Circular Cross-Section Conduits Running Full — Part 1: General Principles and Requirements*. URL: <https://www.iso.org/obp/ui/#iso:std:iso:5167:-1:ed-3:v1:en> (visited on 05/22/2026).
- [10] Clinton Bedick, Don Ferguson, and Peter Strakey. *Validation and Application of a Reduced-Order Rotating Detonation Engine Inlet and Fill Zone Model*. Jan. 11, 2021. DOI: 10.2514/6.2021-0193.
- [11] John David Anderson. *Fundamentals of Aerodynamics*. Sixth edition. McGraw-Hill Series in Aeronautical and Aerospace Engineering. New York, NY: McGraw-Hill Education, 2017. 1130 pp. ISBN: 978-1-259-12991-9.
- [12] Lukas Vojta, Jan Kracik, and Vaclav Dvorak. *Introductory Numerical Study on Supersonic Ejector Working with R32*. Vol. 213. Jan. 1, 2019. 02096 pp. DOI: 10.1051/epjconf/201921302096.

- [13] *NIC5*. CECI. URL: <https://www.cec-hpc.be/clusters/nic5/> (visited on 05/12/2026).
- [14] *NIC5 Cluster - INFO 0939 - High Performance Scientific Computing - Fall 2025*. URL: <https://people.montefiore.uliege.be/geuzaine/INF00939/compenv/nic5/> (visited on 05/12/2026).