Cold Flow Simulation of Gas Flare Auxiliary Air

by

Faisal A. Al Qurooni

B.Sc, King Fahd University of Petroleum and Minerals (KFUPM), 2013

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF
THE REQUIREMENTS FOR THE DEGREE OF

MASTER OF APPLIED SCIENCES

in
THE FACULTY OF GRADUATE AND POSTDOCTORAL STUDIES
(Mechanical Engineering)

THE UNIVERSITY OF BRITISH COLUMBIA
(Vancouver)

August 2018

© Faisal A. Al Qurooni, 2018
The following individuals certify that they have read, and recommend to the Faculty of Graduate and Postdoctoral Studies for acceptance, a thesis/dissertation entitled:

**Cold Flow Simulation of Gas Flare Auxiliary Air**

submitted by Faisal A. Al Qurooni in partial fulfillment of the requirements for

the degree of Master of Applied Science

in Mechanical Engineering

**Examiner Committee:**

Dr. Ali Vakil

Supervisor

Dr. Sheldon Green

Supervisor

Dr. Boris Stoeber

Supervisory Committee Member

Dr. Carl Ollivier-Gooch

Supervisory Committee Member
Abstract

Flaring in oil and gas production is the controlled burning of unwanted exhaust gases to enhance safety. During flaring, there are two important concerns. The first is to prevent the flame from accumulating on the flare tip, which occurs during periods of high crosswinds. Such flame accumulation, which is known as flame capping, can cause premature failures and structural damage of the flare tip if sustained for long period of time. Excluding the combustion physics, a gas flare can be simplified as a jet in a crossflow (JICF). To avoid the considerable complexity associated with simulating a combusting flow, cold flow modeling can be used to investigate the critical flow conditions that would result in flame capping. The second concern is to avoid excessive smoke during combustion, which is important to meet the environmental regulations.

Saudi Aramco has therefore developed a flare system that ameliorates both concerns. The system uses supersonic air nozzles that are distributed evenly around the flare exit. Besides preventing flame capping, the high-speed jets of these nozzles introduce extra oxygen and improve mixing in the combustion zone, which reduces flare smoking. The focus of this thesis is the cold (non-combusting) flow in one of these flare systems.

Computational Fluid Dynamics (CFD) was used to study the flow within a gas flare. The capabilities of different turbulence models to simulate the flare flow field, particularly Large Eddy Simulation (LES) and Shear Stress Transport (SST) $k - \omega$ model, were tested by reproducing previous JICF experimental data. The flow within the auxiliary air nozzles was studied by computing several parameters such as the mass entrainment, axial velocity, turbulent kinetic energy and Mach number in different flow regimes. Additionally, the differences between the
results from different Reynolds-Averaged Navier–Stokes (RANS) models (including the SST $k - \omega$ and the Realizable $k - \varepsilon$) are investigated. For fixed mass flow rate, the detailed geometry of these nozzles is shown to have little impact on the mass entrainment rate, and therefore is expected to have little impact on the flare combustion characteristics. Finally, this work presents a preliminary study of a simplified full flare system.
Lay Summary

Gas flares are important safety devices that are used in the oil and gas industry to burn unwanted or excess gases safely. They are usually equipped with auxiliary systems that often use either air or stream to help obtaining a complete combustion at the top of the flare stack and consequently a smokeless combustion. This is important because it reduces the environmental impact of flaring. This research looks at methods to improve such an auxiliary system that uses high speed convergent-divergent nozzles to enhance the combustion of a gas flare. The research is done using Computational Fluid Dynamics (CFD), which uses numerical simulations to solve complex fluid mechanics problems.

The results provide a foundation that helps in improving this auxiliary system. Furthermore, the results present a preliminary study to avoid premature failures in gas flare tips, which are expensive to replace and result in ceasing the operations of plants.
Preface

This work was first proposed by my co-supervisor Dr. Ehab Elsaadawy from Research & Development Center of Saudi Aramco Company. The methodology and the work structure were proposed by my supervisor Dr. Ali Vakil and reviewed by my co-supervisors Dr. Sheldon Green and Dr. Ehab Elsaadawy.

The author of the introduction and literature review in Chapter 1 is Faisal Al Qurooni. Dr. Ali Vakil provided supervision and advices to conduct this work. Dr. Ali, Dr. Ehab and Dr. Green proposed the work in Chapter 2, which was done by Faisal Al Qurooni with the help and supervision of Dr. Ali Vakil.

The authors of Chapter 3 are Faisal Al Qurooni, Dr. Ali Vakil, Dr. Ehab Elsaadawy and Dr. Sheldon Green. Dr. Ali identified the need to study the flow within that specific nozzle and advised the methodology for this study, which was conducted by Faisal Al Qurooni and reviewed by Dr. Ali, Dr. Ehab and Dr. Green. A shorter version of this chapter has been accepted and appeared in the Canadian Society for Mechanical Engineering (CSME) 2018 Conference Proceeding.

The author of Chapter 4, Chapter 5 and Chapter 6 is Faisal Al Qurooni. The work presented in these chapters was supervised by Dr. Ali Vakil and reviewed by Dr. Sheldon Green and Dr. Ehab Elsaadawy. The thesis manuscript was written by Faisal Al Qurooni, supervised by Dr. Ali Vakil and reviewed by Dr. Green and Dr. Ehab.
# Table of Contents

Abstract........................................................................................................................................ iii

Lay Summary.................................................................................................................................. v

Preface........................................................................................................................................... vi

Table of Contents............................................................................................................................ vii

List of Tables .................................................................................................................................. xii

List of Figures .................................................................................................................................. xiii

List of Symbols ............................................................................................................................... xviii

List of Abbreviations ..................................................................................................................... xix

Acknowledgements ......................................................................................................................... xx

Dedication ....................................................................................................................................... xxi

Chapter 1: Introduction .................................................................................................................. 1

1.1 Background ............................................................................................................................... 1

1.2 Theory ....................................................................................................................................... 5

1.2.1 JICF Theory ......................................................................................................................... 5

1.2.2 Nozzle Flow Theory ............................................................................................................ 7

1.3 Literature Review ...................................................................................................................... 8

1.3.1 Jet in Crossflow ................................................................................................................... 8

1.3.2 Convergent Divergent Nozzle Flow ...................................................................................... 14

1.4 Thesis Objectives ...................................................................................................................... 16

Chapter 2: Validation of JICF against Experiments ....................................................................... 18

2.1 Simulation of RA Experiments ................................................................................................ 18
2.1.1 Numerical Method ................................................................. 18
  2.1.1.1 Computational Domain, Mesh and Boundary Conditions ........ 18
  2.1.1.2 Turbulence Modeling ...................................................... 20
2.1.2 Results and Discussions .......................................................... 20
2.1.3 Summary and Conclusion .......................................................... 21
2.2 Simulation of SP Experiments .......................................................... 22
  2.2.1 Numerical Method ................................................................. 22
    2.2.1.1 Computational Domain, Mesh and Boundary Conditions .... 22
    2.2.1.2 Turbulence Modeling ...................................................... 22
  2.2.2 Results and Discussions .......................................................... 22
    2.2.2.1 First Attempt ............................................................... 24
    2.2.2.2 Second Attempt ............................................................. 25
    2.2.2.3 Third Attempt ............................................................... 28
    2.2.2.4 Fourth Attempt ............................................................. 30
  2.2.3 Summary and Conclusion .......................................................... 31
2.3 Simulation of SM Experiments .......................................................... 33
  2.3.1 Numerical Method ................................................................. 33
    2.3.1.1 Computational Domain, Mesh and Boundary Conditions .... 33
    2.3.1.2 Turbulence Modeling ...................................................... 33
  2.3.2 Results and Discussions .......................................................... 34
  2.3.3 Summary and Conclusion .......................................................... 35
2.4 Conclusions .................................................................................. 37

Chapter 3: Subsonic and Supersonic Flow through a Specific Nozzle ............ 39
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Introduction</td>
<td>39</td>
</tr>
<tr>
<td>3.2 Numerical Method</td>
<td>40</td>
</tr>
<tr>
<td>3.2.1 Governing Equations</td>
<td>40</td>
</tr>
<tr>
<td>3.2.2 Mesh and Boundary Conditions</td>
<td>44</td>
</tr>
<tr>
<td>3.2.3 Mesh Independency Check</td>
<td>44</td>
</tr>
<tr>
<td>3.2.4 2D vs. 3D Simulations</td>
<td>46</td>
</tr>
<tr>
<td>3.2.5 Model Validation against Experimental Data</td>
<td>50</td>
</tr>
<tr>
<td>3.3 Results and Discussions</td>
<td>53</td>
</tr>
<tr>
<td>3.3.1 Mean Axial Velocity Profiles</td>
<td>53</td>
</tr>
<tr>
<td>3.3.2 Turbulent Kinetic Energy Profiles</td>
<td>56</td>
</tr>
<tr>
<td>3.3.3 Mach Number Profiles</td>
<td>58</td>
</tr>
<tr>
<td>3.3.4 Pressure Coefficient</td>
<td>62</td>
</tr>
<tr>
<td>3.3.5 Mass Entrainment Rate Profiles</td>
<td>63</td>
</tr>
<tr>
<td>3.3.6 Mass Flux Variation with NPR</td>
<td>65</td>
</tr>
<tr>
<td>3.4 Conclusions</td>
<td>66</td>
</tr>
<tr>
<td>Chapter 4: Comparison between Different Nozzles Performance</td>
<td>69</td>
</tr>
<tr>
<td>4.1 Introduction</td>
<td>69</td>
</tr>
<tr>
<td>4.2 Results and Discussions</td>
<td>69</td>
</tr>
<tr>
<td>4.2.1 Mean Axial Velocity Profiles</td>
<td>70</td>
</tr>
<tr>
<td>4.2.2 Turbulent Kinetic Energy Profiles</td>
<td>72</td>
</tr>
<tr>
<td>4.2.3 Mach Number Profiles</td>
<td>73</td>
</tr>
<tr>
<td>4.2.4 Mass Entrainment Rate Profiles</td>
<td>76</td>
</tr>
<tr>
<td>4.2.5 Mass Flux Variation with NPR</td>
<td>77</td>
</tr>
</tbody>
</table>
Chapter 5: Effect of Geometry Change on Nozzle Performance .................................. 80

5.1 Introduction ............................................................................................................ 80

5.2 Numerical Method .................................................................................................. 81
    5.2.1 Mesh and Boundary Conditions ....................................................................... 81
    5.2.2 Mesh Independence Check .............................................................................. 82

5.3 Results and Discussions ......................................................................................... 82

5.4 Conclusions ............................................................................................................ 84

Chapter 6: Simplified Full Flare System ..................................................................... 85

6.1 Introduction ............................................................................................................ 85

6.2 Numerical Method .................................................................................................. 85
    6.2.1 Computational Domain ..................................................................................... 85
    6.2.2 Mesh and Boundary Conditions ....................................................................... 86

6.3 Results and Discussions ......................................................................................... 88

6.4 Conclusions ............................................................................................................ 90

Chapter 7: Summary and Future Work ...................................................................... 91

7.1 Introduction ............................................................................................................ 91

7.2 Summary of Chapter 1 ........................................................................................ 92

7.3 Summary of Chapter 2 ........................................................................................ 92

7.4 Summary of Chapter 3 ........................................................................................ 94

7.5 Summary of Chapter 4 ........................................................................................ 95

7.6 Summary of Chapter 5 ........................................................................................ 96

7.7 Summary of Chapter 6 ........................................................................................ 96
Appendix A Mean longitudinal velocity profiles in RA experiments for $R = 0.5$ at location $y = 0$ and $x = 1D$ (top) and $x = 2D$ (bottom) as computed by the SST $k - \omega$ model and LES.

Appendix B Mean velocity profiles of SP experiments (Case 1) at $y = 0$ and two streamwise locations, $x = 3.670D$ (top) and $x = 5.505D$ (bottom).

Appendix C User Defined Function (UDF) to model a velocity profile that follows the $1/7$ power law.

Appendix D The normalized axial velocity profiles downstream of the CCTDN exit for both turbulence models at the other upstream pressures; Extraction line is at (top) $x = 2.256 \text{ rexit}$ (bottom) $x = 11.278 \text{ rexit}$ downstream of the nozzle exit.

Appendix E The normalized axial velocity profiles downstream of the CCTDN exit for the Realizable $k - \varepsilon$ model (top) and The SST $k - \omega$ model (bottom) at $P_{in} = 160 \text{ kPaa}$ at more extraction lines.

Appendix F Mach number profiles of the CCTDN along the jet symmetry axis as predicted by both turbulence models at $P_{in} = 120 \text{ kPaa}$ and $P_{in} = 180 \text{ kPaa}$.

Appendix G The entrainment rate as a function of distance along the jet for different turbulence models at all the considered upstream pressures.
List of Tables

Table 1.1: A list of previous JICF studies, depicting some information and dimensionless numbers
Table 2.1: Validation matrix using SP experiments [5]
Table 2.2: A summary of the simulated JICF experiments
Table 3.1: A list of the governing equations (Favre-averaging or density-weighed averaging)
Table 3.2: The variation of the average exit velocity of the CCTDN with the upstream pressure
Table 4.1: A comparison of the variation of the average exit velocity of the CCTDN and the CDN with the upstream pressure
Table 5.1: The exit diameter and the expansion ratio of the CCTDN at different divergence angles
Table 6.1: The velocity components of each nozzle
List of Figures

Figure 1.1: Flame bent over on the downwind side of a flare (left). Flame is operating normally (right). ............................................................................................................................................................................ 4

Figure 1.2: A flare with (right) and without (left) HPAAS, which shows the advantage of the tiny nozzles that are placed around the circumference of the flare exit [1] ............................................. 4

Figure 1.3: The vortices that form in JICF [3].......................................................................................................................................................... 6

Figure 2.1: A front view of the computational domain (top) and the structured mesh (bottom).. 19

Figure 2.2: The mean axial velocity profiles for $R = 0.5$ at location $x = 0$ and $y = 0$ as computed by the SST $k - \omega$ and LES models against experiment. .............................................. 21

Figure 2.3: A front view of the computational domain (top) and the structured mesh (bottom).. 23

Figure 2.4: Mean velocity profiles (Case 1) at $y = 0$ and two streamwise locations, $x = 0$ (left) and $x = 1.835D$ (right). ......................................................................................................................................................... 25

Figure 2.5: Mean velocity profiles at $x = 0$ (left) and $x = 1.835D$ (right). Experiment results are shown against the LES results in Case 1, Case 2A and Case 2B......................................................... 27

Figure 2.6: Mean velocity profiles at $x = 0$ (left) and $x = 1.835D$ (right). SP experiments results are shown against the LES results of Case 1, Case 2B and Case 3. ................................................................. 29

Figure 2.7: Measured mean velocity profiles from Kim’s PIV [11] and SP’s hot film [5] against the LES results of YSF [17] and Case 3. Left: at $x = 0$. Right: at $x = 1.835D$................................. 30

Figure 2.8: Measured mean velocity magnitude from PIV [11], and hot film [5], against the current LES results of Case 4................................................................................................................. 31

Figure 2.9: A front view of the 3D structured mesh of SM experiments. ........................................... 34
Figure 2.10: a) A simplified sketch that shows the lines where the computational results are measured (not to scale). b) Mean axial velocity profile along the streamwise direction at $z = 0.1 \, RD_j$. c) Mean axial velocity profile along the streamwise direction at $z = 0.5 \, RD_j$. d) Reynolds stresses along the streamwise direction at $z = 0.5 \, RD_j$.

Figure 3.1: The design of the nozzles used in gas flares (not to scale).

Figure 3.2: Setup of the computational domain (top) and the mesh (bottom). The computational domain is not to scale.

Figure 3.3: Velocity profile at $x = 2.256 \, rexit$ downstream of the nozzle exit for $Pin = 160 \, kPa$ at different mesh resolutions.

Figure 3.4: Normalized axial velocity profile (Top) and Mach number and pressure coefficient profiles along the nozzle symmetry axis for 2D axisymmetric and 3D simulation (bottom). ($Pin = 160 \, kPa$ and $Pout = 101 \, kPa$).

Figure 3.5: Comparison of sonic line (2D) and sonic surface (3D). The contours are colored by the density field ($kg/m^3$).

Figure 3.6: Comparison of axial velocity profile along the symmetry axis between current simulations, those in [43], and experiments [56].

Figure 3.7: Comparison of turbulent kinetic energy profile along the axis of symmetry between current simulations, those in [43], and experiments [56].

Figure 3.8: Comparison of the axial velocity profile downstream of the nozzle exit between current simulations and results in [43] and [56].

Figure 3.9: The normalized axial velocity profiles downstream of the CCTDN exit for both turbulence models; Extraction line is at (top) $x = 2.256 \, rexit$ (bottom) $x = 11.278 \, rexit$ downstream of the nozzle exit.
Figure 3.10: The normalized turbulent kinetic energy along the axis of symmetry predicted by the turbulence models at Pin = 140 kPaa and Pin = 160 kPaa .......................................................... 57

Figure 3.11: TKE development as computed by the Realizable $k - \varepsilon$ model at Pin = 160 kPaa in the entire flow domain (top) and inside the nozzle (bottom). ......................................................... 58

Figure 3.12: Mach number profiles of the CCTDN along the jet symmetry axis as predicted by both turbulence models at $Pin = 140\, kPaa$ and $Pin = 160\, kPaa$; The figure at the bottom shows the details of the variations in the immediate upstream and downstream of the nozzle exit ........................................................................................................................................ 60

Figure 3.13: Mach number contours as predicted by the Realizable $k - \varepsilon$ model at (a) $Pin = 140\, kPaa$ (c) $Pin = 160\, kPaa$, and as predicted by the SST $k - \omega$ model at (b) $Pin = 140\, kPaa$ (d) $Pin = 160\, kPaa$ .................................................................................................................................................. 61

Figure 3.14: Pressure coefficient evolution inside the nozzle at different upstream pressures as predicted by both turbulence models. .............................................................................................................. 63

Figure 3.15: The normalized mass entrainment variation as a function of distance along the jet for different turbulence models .............................................................................................................. 64

Figure 3.16: The entrainment rate coefficient $Ke$ as a function of distance along the jet for different turbulence models. .............................................................................................................. 65

Figure 3.17: The variation of the mass flux inside the CCTDN with the NPR as predicted by the RANS models .............................................................................................................................................. 66

Figure 4.1: CCTDN vs. CDN (not to scale) .......................................................................................................... 70

Figure 4.2: A comparison of the normalized axial velocity profiles downstream of the CDN and the CCTDN exits as computed by the Realizable $k - \varepsilon$ (left) and SST $k - \omega$ model (right) in the near field ($x = 2.256\, rexit$) ................................................................................................................................. 71
Figure 4.3: A comparison of the normalized axial velocity profiles downstream of the CDN and the CCTDN exits as computed by the Realizable k – ε (left) and the SST k – ω model (right) in the far field (x = 11.278 rexit). ................................................................. 71

Figure 4.4: Left: The TKE along the axis of symmetry of CDN as computed from the Realizable k – ε and the SST k – ω models. Right: A comparison of the TKE profiles in the CDN and the CCTDN as computed from the SST k – ω model. ................................................................. 73

Figure 4.5: Mach number profiles of the CDN along the jet symmetry axis as predicted by both turbulence models at Pin = 140 kPa and Pin = 160 kPa; The figure on the right shows the details of the variations in the immediate upstream and downstream of the nozzle exit............. 74

Figure 4.6: Mach number profiles of the CDN and the CCTDN along the jet symmetry axis when the flow is subsonic as predicted by the Realizable k – ε model (left) and the SST k – ω model (right). ................................................................. 75

Figure 4.7: Mach number profiles of the CDN and the CCTDN along the jet symmetry axis when the flow is supersonic as predicted by the Realizable k – ε and the SST k – ω models. The figure on the right shows the profiles inside the nozzle around the normal shock location........... 75

Figure 4.8: The computed entrainment rate coefficient, Ke, of both nozzles from both RANS models. ......................................................................................................................................................... 77

Figure 4.9: A comparison between the variations of the mass flux inside both nozzles with the NPR as predicted by both RANS models ......................................................................................................................................................... 78

Figure 5.1: The axial velocity profiles, normalized by the exit velocity, at x = 1 rinlet of the CCTDN with θ = 1° (left) and θ = 5°(right). The results were computed using the Realizable k – ε model at Pin = 140 kPa. ......................................................................................................................................................... 82
Figure 5.2: The variation of Mach number profiles along the symmetry axis with the divergence angle as computed from the Realizable k - ε model at Pin = 140 kPaa. ........................................... 83

Figure 5.3: The variation of the normalized mass entrainment with the divergence angle downstream of the three nozzles................................................................. 84

Figure 6.1: The computation domain (top) and the flare exit surrounded by the eight holes that represent the exit of the nozzles (bottom)............................................................................ 87

Figure 6.2: The mesh of the circular jet and its surrounding nozzles ........................................ 88

Figure 6.3: Velocity contours of the complete flow domain when the nozzles are off (top) and when the nozzles are on (bottom). ................................................................................. 89

Figure 6.4: The normalized mean velocity magnitude profiles at x = 0, 1D and 2D. The dashed lines represent the nozzles-off case, while the solid lines represent the nozzles-on case.......... 90
List of Symbols

D: Diameter
J: Momentum ratio
$K_e$: The entrainment rate coefficient
k: Turbulent kinetic energy
Ma: Mach number
R: Velocity ratio
r: Radius
Re: Reynolds number
S: Density ratio
T: Temperature
U: Mean velocity magnitude of jet in crossflow
$U_z$: Mean axial velocity of jet in crossflow
$U_j$: Jet velocity
$U_\infty$: Crossflow velocity
u: Axial velocity of nozzles
$\rho$: Density
$\mu$: Dynamic viscosity
$\delta$: Boundary layer thickness
List of Abbreviations

CCTDN: Convergent-Constant Throat-Divergent Nozzle
CDN: Convergent-Divergent Nozzle
CFD: Computational Fluid Dynamics
CVP: Counter Rotating Vortex Pairs
DNS: Direct Numerical Simulations
HPASS: High Pressure Air Assist System
JICF: Jet in Crossflow
LES: Large Eddy Simulation
NPR: Nozzle Pressure Ratio
PIV: Particle Image Velocimetry
PLIF: Planar laser-induced fluorescence
RANS: Reynolds-Averaged Navier–Stokes
SST: Shear Stress Transport
TKE: Turbulent Kinetic Energy
UDF: User Defined Function
Acknowledgements

First and foremost, I would like to express my sincere gratitude to my supervisor Dr. Ali Vakil for his exceptional patience, technical guidance, and continuous encouragement during the course of my research. I will always be in debt for his invaluable support. Not only is he a great research supervisor, but also a wonderful instructor in the classroom.

I would like also to thank my supervisor Dr. Sheldon Green for his kindness and endless support to my research, technically through his broad knowledge in Fluid Mechanics and also financially. My appreciation is also extended to my co-supervisor Dr. Ehab Elsaadawy from the R&DC of Saudi Aramco for proposing this research topic, which I found very interesting. Moreover, I would like to thank him for his continuous support for me as colleague in Saudi Aramco and as a supervisor of my research.

I would like also to thank the members of my examining committee, Dr. Carl Ollivier-Gooch and Dr. Boris Stoeber for their valuable inputs and comments on this work.

I would like to thank UBC for being the great school that it is. A special appreciation goes to Saudi Aramco, my sponsor and employer, for giving me such a great development assignment. In addition, I would like to acknowledge the support this research received from Coanda Research & Development and from NSERC.

I wish I could express in words alone my deepest thanks to all my friends in Vancouver: Saleh, Miguel, Eduardo, Claire, Sarah, Harsh, Juuso, Mike, Shahzaib, Aishwarya, Ratul, Somesh, Filipe, Nirmal, Omar, Abdullah, Majid, Adriana, Mustafa, Pouya, Sara, Allan, Dr. Farzad, Hatef, Justin and Alondra. Thank you friends! You made Vancouver more enjoyable.

Last but not least, I want to thank my parents, my sister and my brothers for everything they have done for me. I hope you are proud!
Dedication

To my mother Salma

To my family and my friends
Chapter 1: Introduction

1.1 Background

Gas flaring is a technique used in oil refineries, gas and oil separation plants (GOSPs), gas plants and other petrochemical plants in which unwanted or excess gases are burnt safely. As crude oil is extracted for production, raw associated natural gas is brought to the surface. Due to economic or operational reasons, such as lacking pipelines and other gas transportation infrastructure nearby the oil well, vast amounts of such associated gas are commonly flared. Gas flares are also used to protect the plants against the danger of over pressurizing in case of emergency or critical equipment trip. The gas flaring system is very important and plants operations should cease if it is not working properly. The flaring of associated gas usually takes place at the top of a vertical flare stack. Besides the flare stack, a gas flare system consists of a specially designed burner tip and auxiliary fuel. To meet increasingly strict environmental regulations, gas flares are also equipped with steam or air system to promote mixing for nearly complete combustion and destruction of the gas.

While gas flaring is required for safety reasons, it has many drawbacks. Besides the waste of potentially useful resources, gas flaring has negative impacts on the environment (primarily due to incomplete combustion of the gas). To mitigate these negative impacts, companies seek to eliminate or reduce gas flaring. In situations when flaring is unavoidable, it is desirable to have smokeless gas flaring. Because the gas flares in old facilities are not usually equipped with steam or air system that facilitates smokeless combustion, companies seek cost-effective solutions to retrofit these flares. Saudi Aramco, the largest oil company in the world, has made a thorough
analysis and study [1] to investigate a cost-effective and efficient solution to retrofit its hundreds of gas flares in Saudi Arabia. Several options were explored such as steam-assisted flare tips, sonic flare tips and low-pressure air-assist. However, none of these solutions was deemed to be cost-effective and, therefore, Saudi Aramco developed a new flare design named the High Pressure Air Assist System (HPAAS).

Briefly, the system consists of convergent-divergent nozzles that surround the circumference of the flare exit tip. These nozzles have a specific design in which the convergent and divergent sections that are connected via a throat section with a finite length and constant diameter. As these convergent-constant throat-divergent nozzles (CCTDNs) inject compressed air to the combustion zone, the resulting air entrainment from the atmosphere due to the high exit velocity provides the required air mass to yield smokeless flaring. Not only do these nozzles facilitate smokeless combustion, but they also prevent the very undesired accumulation of the flame over the flare tip, which is a common issue in gas flares operations (Figure 1.1). This occurs at a high crosswind velocity relative to the flared gas velocity at the flare tip and it can potentially cause premature failures in the flare tip and drastically reduces its life.

This thesis shows a study of an industrial gas flare system utilized in Saudi Aramco using Computational Fluid Dynamics (CFD) using ANSYS Fluent 18.0. Apart from the combustion, a gas flare can be simplified as a jet in crossflow (JICF), which are also known as transverse jets. The jet is the hydrocarbon gases exiting the flare and the crossflow is the air in the atmosphere. Although the problem should be handled from combustion performance point of view, it can also be simplified by utilizing a cold flow model to study only the hydrodynamics aspect of the
problem, which is directly linked to the combustion performance. To achieve that, the ratio of the exit velocity (or momentum) of the flared gas to that of the wind should be investigated to study the operating envelope of the flare system. Nevertheless, the investigated flare is not a simple JICF due to the presence of HPAAS nozzles, which play a critical role and make the problem more challenging.

The final objective of this study is to determine the critical flow conditions (gas flow rates and wind speeds) that might cause harm and premature failures to the flare tip. However, the current study that is presented here only sets forth the first part of this study and the rest is left for future work.

Chapter 1 shows an introduction that includes a literature review and provides the minimum required theoretical background. Chapter 2 presents the attempts to validate some simulations against JICF experiments. This is to test the adequacy of different turbulence models to correctly predict the flow field of JICF. Moreover, this will be instructive in scaling down the real flare system, which is too huge to be simulated as a whole. Chapter 3 is devoted to a detailed study of the flow within the special nozzle (CCTDN). The study sheds light on certain important parameters such as axial velocity, mass entrainment and Mach number at different flow regimes (subsonic and supersonic). In addition, Chapter 3 is concerned with the turbulence models used in simulating the flow in the nozzles and providing a comparison of the results obtained from each model. A comparison between the performance of different nozzles is presented in Chapter 4. This is to investigate which nozzle can better serve the HPAAS. The effect of geometry change on the
CCTDN performance is the topic of Chapter 5. Finally, Chapter 6 shows a preliminary study of a simplified full flare system.

Figure 1.1: Flame bent over on the downwind side of a flare (left). Flame is operating normally (right).

Figure 1.2: A flare with (right) and without (left) HPAAS, which shows the advantage of the tiny nozzles that are placed around the circumference of the flare exit [1]
1.2 Theory

This section provides the reader with the basic theories of JICF and nozzle flow and the minimum required knowledge to understand the thesis.

1.2.1 JICF Theory

When a jet is discharged into a crossflow, it bends towards the crossflow direction and a curved trajectory is created. This is known as a jet in crossflow (JICF) interaction or a transverse jet.

This interaction is important in various engineering applications. For instance, the JICF interaction is vital in V/STOL aircraft (vertical and/or short take off and landing). These aircraft can land and takeoff vertically by the very high speed jet in the ambient. Another example is in turbomachinery and specifically in gas turbine combustors where dilution holes issue cool air jets that mix with hot combustion to reduce the exit temperature. Another industrial application in which JICF interaction is gas flares. Two explanations for the reason of the deflection of JICF interaction exist in the literature [2]. The first one is due to the pressure gradient between the leading edge (windward) and the trailing edge (leeward) at the jet exit. The second one is due to the resulted entrainment of the jet into the crossflow.

The jet trajectory that is created in any JIFC is associated with the appearance of four main vortices [3]. The nature and development of these vortices were among the first aspects that researchers have studied in JICF problems. These vortices are Counter Rotating Vortex Pairs (CVP), horseshoe vortices, shear layer vortices and Wake vortices. Figure 1.3 shows these vortices and more details
about them will be briefly presented after introducing some dimensionless numbers that are usually very important in studying JICF.

![Figure 1.3: The vortices that form in JICF [3]](image)

The simplest, yet the most important, dimensionless number is the velocity ratio $R$, which is defined as follows:

$$R = \frac{U_j}{U_\infty}$$

Where $U_j$ and $U_\infty$ define the jet and crossflow mean velocities, respectively. This ratio plays an important role in determining the flow structure and the vortices as shown in many previous investigations like [4]. Another parameter that becomes important in case the jet density ($\rho_j$) and the crossflow density ($\rho_\infty$) are not equal is the momentum flux ratio ($J$), which is defined as:

$$J = SR^2 = \frac{\rho_j U_j^2}{\rho_\infty U_\infty^2}$$
Where $S$ represents the density ratio. In addition, the Mach numbers of the jet ($M_{a_j}$) and crossflow ($M_{a_\infty}$) are very important they have been considered in different ranges in the literature. They are defined as:

$$M_{a_j} = \frac{U_j}{c} \quad ; \quad M_{a_\infty} = \frac{U_\infty}{c}$$

Where $c$ is the local speed of sound. Reynolds numbers of the jet and the crossflow are also important parameters in JICF. They are usually defined as follows:

$$Re_j = \frac{U_j \rho_j L_j}{\mu_j} \quad ; \quad Re_\infty = \frac{U_\infty \rho_\infty L_j}{\mu_\infty}$$

Where $\mu$ is the dynamic viscosity of a fluid and $L_j$ is the characteristic length, which is usually the internal diameter of the jet, $D_j$, for circular jets [4][5] or the jet width for noncircular jets [6].

### 1.2.2 Nozzle Flow Theory

Nozzles are simple devices designed to increase the flow speed. They are widely used in several engineering applications such as rockets and piping systems. There are two main geometries of nozzles, Converging nozzles (CN) and Converging Diverging nozzles (CDN).

For a given nozzle shape, the flow controlling parameter is the nozzle pressure ratio (NPR). In CN, as the NPR increases, the maximum flow rate is achieved when the flow speed at the nozzle exit is sonic, corresponding to the choked flow condition. On the other hand, a CDN geometry allows the flow to achieve supersonic speeds. For a CDN that undergoes an isentropic expansion, the sonic condition is reached at the throat connecting an upstream subsonic condition to a downstream supersonic condition. In contrast, an isentropic compression within the nozzle connects an
upstream supersonic condition to a downstream subsonic condition via a sonic throat [7]. If the flow is choked at the throat but the NPR differs from that required to yield isentropic flow, a combination of normal and oblique shocks is formed downstream of the throat [8]. Numerical predictions of the compressible flow within CDN, shocks behavior, and the compressible turbulent shear flow of supersonic jets has been subject of extensive studies in the last century.

1.3 Literature Review

This thesis involved different literature reviews for different, yet interrelated, topics. Therefore, it is convenient to show each literature review separately in this section.

1.3.1 Jet in Crossflow

Because of its relevance in nature and many engineering applications, JICF interaction has been covered intensively in the literature both experimentally and computationally. The review shown here is influenced by the thorough and deep review of Mahesh [9]. Table 1.1 summarizes an extensive review of the range of the dimensionless numbers values that were considered in the literature. The table idea is influenced by a similar table in [10].

Kim and Yoon [11] have conducted Particle Image Velocimetry (PIV) measurements to examine the effect of changing $Re_\infty$ while holding a constant $R$. They found that despite holding $R$ constant, the variation of $Re_\infty$ led to a different flow structure in the near field. This indicated a big influence of $Re_\infty$ on JICFs. Additionally, their investigations revealed that the shear layer has a farther-reaching extent (i.e. thicker shear layer) in the case of higher $Re_\infty$ because turbulence facilitates higher mass entrainment from the surrounding fluid.
Table 1.1: A list of previous JICF studies, depicting some information and dimensionless numbers

<table>
<thead>
<tr>
<th>Ref</th>
<th>Method</th>
<th>Fluid</th>
<th>$D_j$(mm)</th>
<th>$U_j$ (m/s)</th>
<th>$U_\infty$ (m/s)</th>
<th>R</th>
<th>$Re_j$</th>
<th>$Re_\infty$</th>
<th>$Ma_j$</th>
<th>$Ma_\infty$</th>
<th>$S$</th>
<th>$J$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[4]</td>
<td>Exp.*</td>
<td>Air</td>
<td>50</td>
<td>6.95</td>
<td>13.9</td>
<td>0.5</td>
<td>20,500</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>13.9</td>
<td></td>
<td>1</td>
<td>41,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>27.8</td>
<td></td>
<td>2</td>
<td>82,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[5]</td>
<td>Exp.</td>
<td>Water</td>
<td>13.84</td>
<td>1.4</td>
<td>0.35</td>
<td>4</td>
<td>17,632</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[6]</td>
<td>DNS</td>
<td></td>
<td>A squared jet</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2.5 &amp; 3.5</td>
<td></td>
<td>225 &amp; 300</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[11]</td>
<td>Exp.</td>
<td>Air</td>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3.3</td>
<td></td>
<td>1,050 &amp; 2,100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[12]</td>
<td>DNS</td>
<td></td>
<td>Air &amp; $N_2$</td>
<td>4.53</td>
<td>16.9</td>
<td>2.95</td>
<td>5.7</td>
<td>5,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[13]</td>
<td>Exp. (PLIF)</td>
<td>Air</td>
<td>2, 2.5, 3.3, 5 &amp; 10</td>
<td>25, 50, 75, 100 &amp; 125</td>
<td>5</td>
<td>5, 10, 15, 20 &amp; 25</td>
<td>From 8,400 to 41,500</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[14]</td>
<td>Exp.</td>
<td>Water</td>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2 to 6</td>
<td></td>
<td>440 &amp; 6,200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[15]</td>
<td>Exp.</td>
<td>Air</td>
<td>4</td>
<td>2.4 x10^5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>1.6</td>
</tr>
<tr>
<td>[16]</td>
<td>Exp. (PIV)</td>
<td>Air</td>
<td>9.53</td>
<td>278, 280, 281 &amp; 290</td>
<td>286</td>
<td>0.972 to 1.014</td>
<td>1.9x10^5</td>
<td></td>
<td></td>
<td>0.8</td>
<td>47.1</td>
<td></td>
</tr>
<tr>
<td>[17]</td>
<td>LES</td>
<td>Water</td>
<td>13.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1,050 &amp; 2,100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[18]</td>
<td>LES</td>
<td>Water</td>
<td>13.84</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3.3</td>
<td></td>
<td>2,100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exp.</td>
<td>Water (PIV)</td>
<td>5</td>
<td>0.02</td>
<td>1.5 to 4.5</td>
<td>150 to 450</td>
<td>100</td>
<td>1, 2, 1, 2.8, and 3.5</td>
<td>2 to 3</td>
<td>0.1 to 0.93</td>
<td>1.7 to 8.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
<td>---</td>
<td>------</td>
<td>------------</td>
<td>-----------</td>
<td>-----</td>
<td>---------------------</td>
<td>-------</td>
<td>------------</td>
<td>-------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(hot wire &amp; LDV)</td>
<td>Air</td>
<td>25.4</td>
<td>12</td>
<td>2.3 &amp; 1.15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(PIV)</td>
<td>Air</td>
<td>39.7</td>
<td>5, 7.5 &amp; 10</td>
<td>10</td>
<td>0.5, 0.75 &amp; 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>k − ε</strong></td>
<td>Water</td>
<td>10</td>
<td>7.15 x 10^{-3}</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>7.15 x 10^{-3}</td>
<td></td>
<td>3x10^{-3}</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.532 x 10^{-3}</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>8.1 x 10^{-4}</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4 x 10^{-4}</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1.788</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exp.</td>
<td>Water elliptic jets</td>
<td>30 to 150</td>
<td>30</td>
<td>1 to 5</td>
<td>890 &lt; Re &lt; 4,400</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exp.</td>
<td>Water</td>
<td>32.47</td>
<td>4.6</td>
<td>1.600</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exp.</td>
<td>Air</td>
<td>5</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exp.</td>
<td>N₂ &amp; Air</td>
<td>7.5</td>
<td>3.1</td>
<td>1.2</td>
<td>2.58</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Experiment</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
After defining some important dimensionless parameters in JICF and illustrating, the next thing to consider is the flow structure of JICF. When a jet is discharged into a crossflow, the following vortex structures are formed:

1. Counter rotating vortex pair (CVP):
   Also known as bound vortices or Kidney-shape vortices, CVP is considered as the signature feature of transverse jets [9]. It is initiated in the near field and persists and it gets fully developed in the far field. It exists in JICF regardless of the value of $Re$, jet nozzle shape, and $R$ [2]. Unlike the other vortices, it appears in both the qualitative time-averaged behavior and the qualitative instantaneous behavior [9]. Its appearance in the mean velocity field makes its measurement easier. This advantage, besides its robustness and importance, explains why CVP is the most studied feature of JICF. Experimental investigations indicate that CVP forms faster at lower $R$ values by comparing JICF with $R = 10$ and $R = 20$ [9], [13].

There are different explanations for the CVP formation. One suggests that CVP formation is attributed to the impulse the jet creates on the crossflow, resulting in tip vortices just like flow over finite wings [9], [29]. Another explanation is that CVP formation is due to the tilting and folding the shear layer faces [9], [14].

2. Horseshoe vortices:
   With a vortex rotation direction opposite to that of CVP [4], the horseshoe vortices form because of the adverse pressure gradient in the windward side of the jet exit (which acts as an obstacle) encountered by the crossflow boundary layer [9]. In other words, a horseshoe vortex formation is
attributed to the transverse vortex stretching induced by the velocity gradient across the wall boundary layer [9], [28].

3. Shear layer vortices:
Also known as ring vortices because they shape closed rings, these vortices dominate the jet vicinity where they are typically observed. When the two streams meet, the generated shear creates a Kelvin-Helmholtz instability causing the flow to roll-up around the jet’s exit [30]. Shear layer vortices can be further classified as leading edge and trailing edge vortices. They are intrinsically unsteady and hence not observed in the mean velocity flow despite having a quasi-steady nature of growth in the near field [6]

4. Wake vortices
These vertically oriented vortices are also known as upright vortices, Zipper vortices, tornado-like vortices and Fric’s vortices. They are opposite to shear layer vortices in the sense that they are developed downstream the jet (and hence comes the name since wakes usually formed behind objects). However, just like the shear layer vortices, they are intrinsically unsteady [3].
Now that the flow structure of JICF has been introduced, a summary of previous experimental and numerical studies will be provided.

There are numerous experimental studies that have been conducted to understand transverse jets. For instance, Camussi [19] used PIV and flow visualization to study a jet in a crossflow at very low $Re$ range (from 5 to 200) and $R$ range (from 0.5 to 12). The objective of this experiment, which was conducted in a vertical wind tunnel of planer jets in crossflow, was to understand the
relationship between the shear layer vortices and CVP. It was also concluded that the vortex structure is insensitive to $Re$. Radhouane [31] have also used PIV in conjunction with Reynolds Stress Model (RMS) to study the interaction of twin tandem in-line jets of variable temperature with crossflow. Hot wire technique has also been used by many researchers such as Rivero [2] who investigated the different vortex structures. It was concluded that CVP appears in the mean field and that it forms due to the high pressure gradients close to the jet exit. Rodi [4] have also used hot wire technique to measure the mean axial and turbulent velocities at several streamwise locations and at several velocity ratios. It was found through the conducted measurements that for higher velocity ratios, the jet penetrates deeper and the bending process becomes slower.

With the leap advancements in computing resources in the past three decades, there was a notable increase in using CFD to address typical industrial and engineering problems and to gain a deeper understanding of underlying physics and mechanisms involved. For example, Mahesh [12] has used Direct Numerical Simulations (DNS) to investigate different methods to scale the jet trajectory. Lai [32] have used RANS to study multiple tandem jets interaction with crossflow. As means of validation, the simulation results were compared with an earlier experiment and that revealed a good agreement. Ruiz [33] used LES to study the interaction of JICF in their attempt to validate the previous experimental results by Su and Mungle [34]. The successful validation of parameters like the turbulent kinetic energy (TKE) and velocity magnitude have proved LES to be a good tool for JICF analysis, which is a great advantage due to the difficulties associated with measuring turbulent quantities experimentally.
In summary, the study of JICF has been widely covered in numerous experimental and numerical studies. Besides the study of a single jet, topics like jets in tandem and inclined jets have been also covered. However, to the best of the author’s knowledge, no study has focused on the flow of a subsonic jet surrounded by supersonic small jets in a subsonic crossflow.

1.3.2 Convergent Divergent Nozzle Flow

Eggers [35] conducted an analytical and experimental investigation of a supersonic jet ($Ma = 2.22$ nozzle) to understand turbulence quantities and jet entrainment. He concluded that an eddy viscosity formulation independent of the radial coordinate, employed by other researchers, for a compressible turbulent jet was not justified. Birch and Eggers [36] collated experimental data on developed free turbulent shear layers to facilitate the validation of the turbulence models and numerical predictions. They concluded that the available data were not sufficient to clearly establish the effect of density gradients on mixing rate. Lau et al. [37] conducted measurements of a 51 mm diameter turbulent jet at Mach 0.28, 0.90, and 1.37 using LDV and a hot wire anemometer. Their data indicate a decrease in the spreading rate of the mixing layer with increasing Mach number. Seiner et al. [38] investigated the effect of jet temperature on the mixing rate, and concluded that high temperatures resulted in enhanced density fluctuations to increase the mixing rate.

Reynolds-averaged Navier–Stokes (RANS) solvers have been used by many researchers to simulate supersonic exhausts. A comprehensive review of various turbulence modeling, advantages and limitations of each approach are presented in [39]. Koch conducted a study on subsonic axisymmetric jets with flow separation using a two-equation $k – \varepsilon$ turbulence model
While the mean flow values showed a good agreement with experiment, the mixing rate predicted by the numerical modeling was lower than the experimental results, and so were the turbulent kinetic energy levels. To incorporate compressibility effects in turbulence models, and consequently to better predict the decrease in the growth rate with increasing Mach number, an additional compressibility term should be included in the turbulence transport equations [41], [42]. Georgiadis et al. [43] assessed the accuracy of modified two-equation turbulence models in flow field predictions of a subsonic ($Ma = 0.5$) heated and unheated jet. They showed that all the modified equations provided improved predictions compared to the standard models. However, all the models underestimated initial jet mixing rate and the turbulence kinetic energy fields.

Xiao et al. [44] examined experimentally and numerically the effect of both the nozzle expansion ratio and the NPR on the entrainment from CDN. The computations showed that the nozzle expansion ratio has a greater effect on the entrainment than the NPR. Among the RANS models, the Shear Stress Transport (SST) $k - \omega$ model compared the best with experiment, which was attributed to its ability to predict the flow separation after the shock correctly. For a fixed nozzle expansion ratio of 1.5, their experimental study showed the best mixing was achieved when the NPR was in the range of 1.4 – 1.6. In a numerical study conducted on CDN with Mach number in the range $0.9 < Ma < 1.2$ and NPR of 4 and 6, DalBello et al. [45] found that the SST $k - \omega$ model could match well with experimental measurements. The RANS models prediction accuracy decreases as the flow compressibility effects become more important. To account for these effects, modified RANS models were considered and their predictions were compared against experiment [46][43].
Sarkar et al. [42], [47] studied compressible shear layers using a Reynolds Stress Model (RSM). When the compressibility effects were not considered explicitly, the model failed to predict the dependence of the shear layer growth rate on the Mach number. In comparison, when the compressibility effects were considered explicitly, the computed results compared better with experiment. Lijo et al. [48] used RSM to study transient flows in a rocket propulsion nozzle which involves free and restricted shock separation, and they found RSM model results were in good agreement with experiment. Balabel et al. [49] assessed RSM and different RANS models for gas flow in a two-dimensional CDN with steady, compressible and turbulent flow. In their investigation to predict the separation point and the shock wave location, they found that the SST \( k - \omega \) model gave the best results when compared to experiment.

In summary, the nozzle flow has long been studied in experimental and numerical investigations since it is a main and important topic in fluid mechanics. Nevertheless, to the best of the author’s knowledge, no study has detailed the flow characteristics in a nozzle similar to the investigated one in this thesis (Chapter 3) and the turbulence modeling involved in that.

1.4 Thesis Objectives

The objectives of this thesis are:

- Investigating the capabilities of different turbulence models to reproduce the results of previous JICF experiments. In addition, the validation of a model against experiments is important to scale down the huge flare system. This can be accomplished by scaling down the flare diameter (1.22 m) to the jet diameter in an experiment while matching the dimensionless numbers.
• Studying the flow within CCTDNs by computing several physical parameters such as the axial velocity profiles, Mach number, mass entrainment and TKE.

• To investigate whether the flow regime (subsonic and supersonic) of the CCTDNs has a significant impact on the mass entrainment or not.

• During the study of the CCTDN, it was observed that some RANS models predict an unexpected behavior close to the nozzle exit. This added one more objective, which is to compare the results of different turbulence models.

• To compare the flow within different nozzles. The comparison particularly looks at the mass entrainment downstream of each nozzle, which is critical to gas flare applications, and some other parameters such as the Mach number development, axial velocity profiles and the TKE.

• To investigate the effect of the divergence angle of the CCTDN on the mass entrainment.

This thesis divides the whole problem into parts (JICF and nozzles flow) that are investigated individually. The thesis later provides a preliminary study of the whole system to see how the presence of the nozzles affects the JICF. The next step, which is left for future work, is to study the combined flow field of the gas flare (as a whole or scaled) and the CCTDNs using cold flow modeling to investigate the hydrodynamics of the problem and to find out the critical velocity ratios that may result in flame capping.
Chapter 2: Validation of JICF against Experiments

Since a gas flare can be considered as a jet that is discharged into a crossflow (the atmosphere), it is instructive to validate JICF models against experimental results. Therefore, this chapter shows the simulation results of three JICF experiments. These experiments are:

- Rodi’s experiments [4] and it will be referred to as RA paper hereafter (Section 2.1).
- Sherif’s experiments [5] and it will be referred to as SP paper hereafter (Section 2.2).
- Su’s experiments [34] and it will be referred to as SM paper hereafter (Section 3.3).

The results in SM experiments were reproduced by Muppidi and Mahesh [50], which will be referred to as MM hereafter, using DNS. The current simulation results of this experiment will be compared to both SM experiments and MM DNS.

2.1 Simulation of RA Experiments

2.1.1 Numerical Method

2.1.1.1 Computational Domain, Mesh and Boundary Conditions

Figure 2.1 illustrates the computational domain, which was created according to the description of the wind tunnel given in [4]. The top wall of the tunnel, which is set as a pressure outlet, could also be set as a moving or stationary wall. This will have no impact on the results because the velocity ratio approaches one before reaching there as can be seen in Appendix A. Figure 2.1 also shows the structured mesh and it is clear that the grids are further refined in some critical locations.
such as the jet and crossflow interface and the areas close to the wall. The jet exit velocity $U_j = 6.95 \text{ m/s}$ with a crossflow velocity $U_\infty = 13.9 \text{ m/s}$, which results in a velocity ratio of $R = 0.5$. The jet diameter, $D_j$, is 50 mm.

Figure 2.1: A front view of the computational domain (top) and the structured mesh (bottom).
2.1.1.2 Turbulence Modeling

Both RANS (Reynolds-Averaged Navier–Stokes Equations) and LES were validated against the experimental data. The particular RANS model considered is the SST $k - \omega$ model.

2.1.2 Results and Discussions

Figure 2.2 shows the simulation results from both turbulence models against Rodi’s experimental results (hot wire measurements) of the mean axial velocity ($U_z$) profiles for $R = 0.5$ at location $x = 0$ and $y = 0$. The simulation results are shown for the finest considered mesh, in which the number of cells was in the order of 8 million. Results at other streamwise locations ($x = 1D$ and $2D$) are shown for reference in Appendix A.

The results from both turbulence models show a clear deviation close to the wall ($0 < z < 0.2D$). However, unlike LES, the SST $k - \omega$ shows a good agreement with the experiment between $0.2D < z < 0.4D$. Far away from the wall, at $z > 0.4D$, both models can predict the velocity profile with a reasonable accuracy. Although the RANS model shows a better agreement with experiments at this location, the LES results show a relatively better agreement with experiment in all the other downstream locations, such as $x = 1D$ and $x = 2D$.

The simulation attempt of this experiment was not successful. The reason of the deviation close to the wall might be because, as mention in [4], measurements could not be carried out there due to the presence of reversed flow and because of the high turbulence intensity in that region. The horizontal error bars, which show the uncertainty in velocity measurements, represent an error of
6%, but it is stated that the error could reach 12% or higher at areas close to the wall. Due to this possibility of large errors, it was decided to simulate another experiment.

![Graph showing mean axial velocity profiles](image)

Figure 2.2: The mean axial velocity profiles for $R = 0.5$ at location $x = 0$ and $y = 0$ as computed by the SST $k - \omega$ and LES models against experiment.

### 2.1.3 Summary and Conclusion

Validating the mathematical model proposed to simulate the flow field under study against this published experimental data showed a good agreement qualitatively as one can see from the general trends of the velocity profiles. However, there is a deviation that becomes clearer close to the jet exit. According to Rodi [4], there is a relatively high uncertainty in the measurements in that area. Therefore, it was decided to work on another experiment to see if better agreement could be obtained.
2.2 Simulation of SP Experiments

The work by Sherif [5] was selected for the CFD results validation to improve mesh resolution in the computational domain and reduce the run time of the simulations; the jet diameter is 13.84 mm (compared to 50 mm in RA experiments), which required a smaller computational domain size. Also, the same mesh count can yield a higher mesh resolution.

2.2.1 Numerical Method

2.2.1.1 Computational Domain, Mesh and Boundary Conditions

The computational domain, shown in Figure 2.3, was created according to the water tunnel description given in [5]. Figure 2.3 also shows the mesh, which is structured everywhere. The cells are refined near the boundary layer and at the jet exit. In comparison, a coarser structured mesh is used further away from the jet exit. This experiment was conducted in a water tunnel, in which the freestream and jet velocities are $U_\infty = 0.35$ m/s and $U_j = 1.4$ m/s, respectively. These values, which were measured using hot film, result in a velocity ratio $R = 4$. The boundary layer thickness, $\delta_{99}$, was observed to be 6.9 mm.

2.2.1.2 Turbulence Modeling

Similar to RA experiments, both LES and the SST $k-\omega$ turbulence models were used and validated against the experimental data of Sherif [5].

2.2.2 Results and Discussions

Several simulations run in an attempt to match SP experimental results. These simulations are also to test some reported reasons by other researchers for this deviation and to check whether
overcoming these reasons bring the results closer or not. These simulations and suggested deviation reasons will be explained in the coming sections.

Figure 2.3: A front view of the computational domain (top) and the structured mesh (bottom).
2.2.2.1 First Attempt

Initially, a computational domain that follows the same setup described in [5] was created. The simulations that were conducted for this setup will be referred to as Case 1. Figure 2.4 shows the mean velocity profiles at two streamwise locations, \(x = 0\) and \(x = 1.835D\). Qualitatively, the figure shows a good agreement between the results from the simulations and the experimental data. However, the graphs do not match quantitatively. The graphs of the mean velocity profiles do not collapse in the far field either (i.e. at \(x = 3.760D\) and \(x = 5.505D\), which can be found for reference in Appendix B).

At \(x = 0\), discrepancies are observed near the jet exit (\(z \leq 3D\)). They diminish at higher elevations where the boundary layer effects disappear and the normalized velocity profile approaches 1.

At \(x = 1.835D\), both turbulence models overestimate the location of the maximum velocity, or velocity overshoot. While the overshoot is experimentally observed to be at \(z \approx 3D\), the SST \(k - \omega\) model and LES predict it to be at \(z \approx 4.5D\) and at \(z \approx 4.75D\), respectively. On the other hand, LES correctly predicts the extent of the velocity overshoot, which is \(U \approx 2.2U_\infty\). This velocity overshoot is underestimated by the SST \(k - \omega\) model (\(U \approx 1.9U_\infty\)).

The same observations regarding the velocity overshoots hold true in the other downstream locations. Therefore, it is concluded that LES can predict the velocity overshoot more accurately.
2.2.2.2 Second Attempt

A thorough literature survey revealed that other research groups have also faced difficulties to validate their simulations with SP experiments. For instance, Yuan [17], hereafter referred to as YSF, found a discrepancy between their LES and SP experiments. YSF have reported two main reasons for the deviation. First, they suspected that the jet pipe was not long enough for the flow to be fully developed. This suspicion was confirmed upon personal communications with Sherif. Thus, the assumption of a fully developed flow in [5] is a possible source of error [17].

The other reason behind the deviation is the assumption of a uniform velocity profile inside the jet, although the Reynolds number ($Re_j = 19,000$) is high enough to warrant the assumption of a turbulent velocity profile that follows the $1/7$ power law [17]. To test the validity of these reasons...
and to check if they could actually improve the results, two more simulations were conducted. These simulations represent the following cases:

- **Case 2A**

  In this case, the same computational domain and the same jet length in SP experiments are considered. The only difference was the use of a User Defined Function (UDF) to model a turbulent velocity profile that follows the $1/7$ power law instead of a uniform velocity profile inside the jet. The utilized UDF can be found in Appendix C.

- **Case 2B**

  Here, a UDF is also used to model a turbulent velocity profile that follows the $1/7$ power law. Moreover, a shorter pipe is used in the setup to check if developing flow profile would lead to a better agreement with experiments.

  In the LES of YSF, the velocity ratio was reduced from $R = 4$ to $R = 3.3$ due to the deviation that was observed at $R = 4$. This reduction was explained as an attempt to match the jet centerline velocity that was predicted by the simulation to the one reported by SP [17]. This leads to changing the velocities as follows:

  \[
  Re_\infty = \frac{\rho U_\infty D_j}{\mu} = \frac{998.2 \times U_\infty \times 13.84 \times 10^{-3}}{0.001096} = 2100 \rightarrow U_\infty = 0.166 \text{ m/s}
  \]

  Since $R = 3.3$, the jet velocity should be $U_j = 3.3 \times 0.166 = 0.55 \text{ m/s}$. Simulations of Case 2A and Case 2B were conducted using LES and the mesh density is similar to the one in Case 1.
Figure 2.5 shows the results of Case 1, Case 2A, Case 2B and experiments. At \( x = 0 \), Case 2B (which is horizontally scaled by 0.166 \( m/s \), not 0.35 \( m/s \) like the previous cases) shows some improvement near the jet exit. At \( x = 1.835D \), the extent of the maximum velocity is correctly predicted only in Case 1 (\( U \approx 2.2U_\infty \)). On the other hand, the maximum velocity location shows improvement (\( z = 4D \)) in Case 2B.

In general, the deviation still exits and none of the attempted simulations results in a quantitative agreement. This conclusion is also reached by YSF [17], in which they observed that Case 2B, which is called Case 3II in YSF [17], relatively shows the best agreement with experiments.

Figure 2.5: Mean velocity profiles at \( x = 0 \) (left) and \( x = 1.835D \) (right). Experiment results are shown against the LES results in Case 1, Case 2A and Case 2B.
2.2.2.3 Third Attempt

In the previous simulations, the created computational domain follows the setup in SP experiments and the only difference was in Case 2B in which the pipe length was reduced. The entrance length, which is the distance from the leading edge of the water tunnel to the jet center, was set to $33D$ in all of the presented cases. It was observed that the entrance length was reduced to $2.7D$ in YSF setup without a clear reason. To explore its effect on the results, another simulation (Case 3) was run with an entrance length of $2.7D$. The only difference between Case 3 and Case 2B is the entrance length.

As shown in Figure 2.6, Case 3 shows a better agreement with the experiment than Case 2B at $x = 1.835D$ because it shows a better prediction of both the location and the magnitude of the velocity overshoot. The difference in the results between Case 2B and Case 3 indicates that the entrance length is important and hence the setup in YSF simulations should have been kept as described in SP experiments.

Unfortunately, in spite of all these attempts, the simulations of SP experiments do not show a good quantitative agreement, especially close to the jet exit. However, most of the simulations show a reasonable prediction of the general flow behavior as the predicted trends of the velocity profiles are very similar to the experiments.
Figure 2.6: Mean velocity profiles at $x = 0$ (left) and $x = 1.835D$ (right). SP experiments results are shown against the LES results of Case 1, Case 2B and Case 3.

A second research group, Kim [11], conducted experiments using PIV measurements on a similar geometry to SP and compared the results to the LES results of YSF. Digitized from [11] with the addition of Case 3, Figure 2.7 shows the results of Kim’s PIV, YSF’s LES, SP’s hot film measurements and Case 3.

The PIV results of the velocity profiles deviate from SP results near the jet exit ($\frac{z}{D} \leq 1$) at position $x = 0$. At $x = 1.835D$, they also show a lower velocity overshoot relative to SP experiments. Therefore, it is concluded that there are indeed some experimental errors in SP results, specifically in the region $z/D \leq 1$. The location of the velocity overshoot from PIV and LES almost coincides ($z/D \approx 3.8$). However, the PIV results show a lower velocity overshoot relative to the present LES. This difference might be attributed to the fact that the PIV measurements considered only
two velocity components in calculating the velocity magnitude [11], whereas the present LES and YSF LES considered all the three components.

![Figure 2.7](image)

Figure 2.7: Measured mean velocity profiles from Kim’s PIV [11] and SP’s hot film [5] against the LES results of YSF [17] and Case 3. Left: at $x = 0$. Right: at $x = 1.835D$.

### 2.2.2.4 Fourth Attempt

Figure 2.8 shows the PIV results, SP results and a final simulation attempt (Case 4), which is similar to Case 3 except that the UDF was expanded so that the velocity profiles of both the jet and the crossflow follow the $1/7$ power law. The difference in the computed results between Case 4 and Case 3 is negligible and it is therefore concluded that this expansion of the UDF to model turbulent boundary layers in both the jet and the crossflow has no effect on the velocity profile results.
Figure 2.8: Measured mean velocity magnitude from PIV [11], and hot film [5], against the current LES results of Case 4.

2.2.3 Summary and Conclusion

Several simulations were run to reproduce the results of SP experiments. Although a good qualitative agreement was obtained by using the SST $k - \omega$ model, a quantitative agreement was not obtained even with LES. It was observed upon investigations that the deviation between the simulations and the experiments might be attributed to some inaccurate assumptions as reported in [17] and [11]. For example, it was suspected that the pipe length is not enough to guarantee a fully developed flow. Another issue is that although $Re_j \approx 19,000$ is a turbulent velocity profile that follows the well-known 1/7 power law, a uniform velocity profile was assumed by SP. Therefore, several simulations were conducted to check if overcoming these issues would actually improve the agreement between simulations and experiments. A summary of the conducted simulations is presented in Table 2.1. In general, the later simulations show a better agreement with the experiments. However, the deviation still persists. The fact that the results of PIV
measurements [11] do not match with the hot film results of SP makes the accuracy of the reported results of SP questionable.

Table 2.1: Validation matrix using SP experiments [5].

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>This case represents the LES results of SP experiments with a uniform velocity profile. ( R = 4 ) is considered with ( U_\infty = 0.35 ) m/s and ( U_j = 1.4 ) m/s.</td>
</tr>
<tr>
<td>Case 2A</td>
<td>This case represents the LES results for same setup and jet length in SP experiments, but with the use of a UDF to model a turbulent velocity profile that follows the 1/7 power law instead of a uniform velocity profile. ( R = 4 ) with ( U_\infty = 0.35 ) m/s to follow YSF [17].</td>
</tr>
<tr>
<td>Case 2B</td>
<td>In addition to the UDF that models a velocity profile that follows the 1/7 power law, a shorter pipe is used in the setup to check if a developing flow profile would lead to results matching. ( R = 3.3 ) is considered with ( U_\infty = 0.166 ) m/s to follow YSF [17].</td>
</tr>
<tr>
<td>Case 3</td>
<td>While the entrance length is ( 33D ) in SP experiments, the setup in YSF [17] has a reduced entrance length of ( 2.7D ) for unclear reasons. Therefore, Case 3 is similar to Case 2B but with a smaller entrance length (( 2.7D ) versus ( 33D ))</td>
</tr>
<tr>
<td>Case 4</td>
<td>This case is similar to Case 3, but the UDF was expanded to have a turbulent boundary layer for both the jet and the crossflow.</td>
</tr>
</tbody>
</table>
2.3 Simulation of SM Experiments

So far, the presented simulation results did not coincide with experiments. Due to some reported inaccuracies in these experiments, another experiment was chosen to validate the simulations. The chosen experiment, SM [34], was later successfully simulated by another research group [50] using DNS (which is referred to as MM DNS).

2.3.1 Numerical Method

2.3.1.1 Computational Domain, Mesh and Boundary Conditions

SM experiments were conducted in a wind tunnel (fluid is air) with $U_\infty = 2.95 \, m/s$, $U_j = 16.9 \, m/s$, $R = 5.7$ and $Re_j = 5000$. The jet fluid is $N_2$ and consequently the density ratio is $s = 1.10$. The internal diameter of the pipe is $D_j = 4.53 \, mm$ and its length is $70.64D$. Measurements were obtained using simultaneous planar laser-induced fluorescence (PLIF) and particle image velocimetry (PLV) techniques.

A structured mesh (Figure 2.9) with about 7.5 million cells was used and a similar meshing strategy has been followed as finer cells are used in critical areas such as the area close to the walls and the interface between the jet and the crossflow.

2.3.1.2 Turbulence Modeling

Both LES and SST $k – \omega$ were used to predict the turbulence features in SM experiments [34]. The simulation results were compared to both references [50] and [34].
2.3.2 Results and Discussions

The velocity profiles are extracted along two different horizontal lines placed on the vertical axis above the jet exit. These lines are $z = 0.1 \text{RD}_j$ and $z = 0.5 \text{RD}_j$ as shown in Figure 2.10a. Figure 2.10b and Figure 2.10c show the normalized axial velocity profiles along the streamwise direction at $z = 0.1 \text{RD}_j$ and $z = 0.5 \text{RD}_j$, respectively. The figures depict the results of SM experiments, DNS, and the current simulation as computed from LES and the SST $k - \omega$ model. A very good agreement can be observed. In fact, the current results, from both turbulence models, match with the experiment even better than the DNS results.

Reproducing some turbulence quantities such as Reynolds stresses may be daunting due to the high sensitivity of these quantities to the mesh resolution and because these quantities need longer
time to converge. However, the LES results of the normalized Reynolds stresses, $\overline{u_x u_z}$, at $z = 0.5 \text{RD}_j$ agree reasonably well with the experimental data as shown in Figure 2.10d.

2.3.3 Summary and Conclusion

The validation of the mathematical model proposed to simulate the flow field under study against this published experimental data showed a good quantitative agreement from both turbulence models. While both models can predict the velocity profiles with a good accuracy, LES has the advantage of predicting the turbulence quantities.
Figure 2.10: a) A simplified sketch that shows the lines where the computational results are measured (not to scale). b) Mean axial velocity profile along the streamwise direction at $z = 0.1 \, RD_j$. c) Mean axial velocity profile along the streamwise direction at $z = 0.5 \, RD_j$. d) Reynolds stresses along the streamwise direction at $z = 0.5 \, RD_j$. 
2.4 Conclusions

This chapter showed the simulation results of three different JICF experiments. Table 2.2 summarizes some information about the simulated experiments. The objective was to validate some results, such as the mean velocity profiles at different downstream positions, from different turbulence models. This is to test the capabilities of these different turbulence models to capture the complex flow fields that result when a jet is discharged into a crossflow. This will help in choosing the turbulence model to simulate the flare system. A summary of the simulations of each experiment was provided at the end of each section.

Table 2.2: A summary of the simulated JICF experiments

<table>
<thead>
<tr>
<th>Experiments</th>
<th>Date</th>
<th>Apparatus</th>
<th>Method</th>
<th>(D_j) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rodi (RA) [4]</td>
<td>1984</td>
<td>Wind tunnel</td>
<td>Hot wire</td>
<td>50</td>
</tr>
<tr>
<td>Sherif (SP) [5]</td>
<td>1989</td>
<td>Water tunnel</td>
<td>Hot film</td>
<td>13.84</td>
</tr>
<tr>
<td>Su (SM) [34]</td>
<td>2004</td>
<td>Wind tunnel</td>
<td>PIV</td>
<td>4.53</td>
</tr>
</tbody>
</table>

The simulations of the RA and SP experiments, as computed from both LES and the SST \(k - \omega\), revealed a qualitative agreement only. However, it was hard to obtain a quantitative agreement and some discrepancies were persistent in spite of all the attempts to overcome the suspected deviation reasons (as shown in SP experiments). Since other research groups failed to simulate SP experiments or validate it experimentally, this suggests some non-negligible errors in the experiments.
On the other hand, there is a very good agreement between the simulation results (from both LES and the SST $k - \omega$ model) and the data from SM experiments. This indicates that the SST $k - \omega$ model, which has the advantage of less computational time relative to LES, can be used in cold modeling of a gas flare with an acceptable accuracy.
Chapter 3: Subsonic and Supersonic Flow through a Specific Nozzle

3.1 Introduction

This chapter shows a study of the nozzles used in the gas flare system to entrain air mass, which are shown in Figure 3.1. These nozzles are of a specific design in which the converging section is connected to the divergent section via a finite-length throat section that has a constant-diameter, referred to as convergent-constant throat-divergent nozzle (CCTDN). The exit diameter, throat diameter and inlet diameters of the nozzle are $D_{exit} = 3.378 \, mm$, $D_{throat} = 2.921 \, mm$ and $D_{inlet} = 7.620 \, mm$, respectively. The expansion area ratio, exit area to throat area, is 1.34.

![Diagram of nozzle design](image)

Figure 3.1: The design of the nozzles used in gas flares (not to scale)

Because of the specific design of these nozzles, it was important to study some important parameters within it. Therefore, this chapter shows the variation of Mach Number, pressure
coefficient, turbulent kinetic energy, mass entrainment and axial velocity profiles at different boundary conditions.

3.2 Numerical Method

In this section, the numerical formulations employed to obtain the CFD results are presented.

3.2.1 Governing Equations

The governing-equations require special treatment, i.e. closure, when the flow is turbulent. In the time-averaging of the governing equations, the physical quantity $\phi$ is decomposed into its mean component $\bar{\phi}$ and fluctuating component $\phi'$ via the Reynolds decomposition as $\phi = \bar{\phi} + \phi'$. To account for the effects of the density fluctuations due to turbulence, Favre-averaging or density-weighed averaging is adopted in turbulent compressible flows. The density-weighted average of $\phi$ is obtained as $\phi = \bar{\phi} + \phi''$, where $\bar{\phi} = \bar{\rho}\bar{\phi}/\bar{\rho}$. The details of the Favre averaging can be found in [51].

The mass-averaged conservation of mass in terms of mean velocity vector $\bar{u}_i$ is given by Equation (1). The governing equations for the transport of momentum are given by Equation (2). $p$ is the mean pressure and related to mean temperature and density as $p = \bar{\rho}R\bar{T}$. The viscous stress tensor $\tau_{ij}$ for an isentropic Newtonian fluid is defined as given by Equation (3) where $\delta_{ij}$ is the Kronecker delta function, and $\mu$ is the dynamic viscosity. The Reynolds stresses $\bar{R}_{ij} = -\bar{\rho}u'_iu'_j$, arising from the Favre-averaging of the momentum equations, require modeling. One of the most common approaches to close the set of equations is to use the Boussinesq hypothesis to relate the Reynolds stresses to the mean deformation rates and the turbulent kinetic energy $k$ as given in
Equation (4). The turbulent viscosity $\mu_t$ is given by $\mu_t = \rho C_\mu k^2 / \varepsilon$, where $\varepsilon$ is the dissipation of turbulent kinetic energy and $C_\mu$ is a constant.

The Realizable $k - \varepsilon$ employs Equation (5) and Equation (6) as transport equations for $k$ and $\varepsilon$, respectively. $\bar{R}_{ij} \partial \bar{u}_i / \partial x_j$ is the generation of turbulence kinetic energy due to the mean velocity gradients. $Y_M$ is the contribution of fluctuating dilation to the dissipation rate. $S = \sqrt{2S_{ij}S_{ij}}$ is the modulus of the mean deformation rate tensor. $\sigma_k = 1.0, \sigma_\varepsilon = 1.2$ are, respectively, the turbulent Prandtl numbers for $k$ and $\varepsilon$. In contrast to the standard $k-\varepsilon$ model, $C_\mu$ is no longer a constant but rather depends on the mean deformation and rotation rates, the angular velocity of the rotation frame, and the turbulence quantities [52].

The SST $k - \omega$ employs Equations (8) and (9), respectively, as transport equations for $k$ and the specific dissipation rate $\omega$. $\sigma_k$ and $\sigma_\omega$ are the turbulent Prandtl numbers for $k$ and $\omega$, respectively. The turbulent viscosity is computed as $\mu_t = f \rho k / \omega$, where $f$ is a function of the strain rate magnitude, the specific dissipation rate, a limiter function, and a Reynolds number defined as $Re_t = \rho k / \mu \omega$. The model constants, which can be found in [53], are $\sigma_k = 0.85$ and $\sigma_\omega = 0.5$. This formulation accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosity.

This formulation accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosity. The Favre-averaged energy equation is given by Equation (10). The turbulent flux $-\bar{\rho u_j''h}$ is given based on the averaged enthalpy, turbulent Prandtl number, and the turbulent
eddy viscosity as given in Equation (11). The Favre-averaged material derivative is defined in Equation (12).
Table 3.1: A list of the governing equations (Favre-averaging or density-weighed averaging)

<table>
<thead>
<tr>
<th>Conservation of Mass</th>
<th>[ \frac{\partial \tilde{\rho}}{\partial t} + \frac{\partial}{\partial x_i}(\rho \tilde{u}_i) = 0 ]</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Conversation of Momentum</th>
<th>[ \frac{\partial}{\partial t}(\rho \tilde{u}_i) + \frac{\partial}{\partial x_j}(\rho \tilde{u}<em>j \tilde{u}<em>i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tilde{r}</em>{ij} - \tilde{R}</em>{ij}) ]</th>
</tr>
</thead>
</table>

| Realizable k - ε | \[ \frac{\partial}{\partial t}(\tilde{\rho} k) + \frac{\partial}{\partial x_j}(\tilde{\rho} \tilde{u}_i k) = \frac{\partial}{\partial x_j}\left[ \frac{\mu}{\alpha_{k}} \frac{\partial k}{\partial x_j} \right] + \frac{\partial \tilde{R}_{ij}}{\partial x_j} - \rho \tilde{\rho} - \rho_{\gamma} \tilde{V}_M \] |

| SST k - ω | \[ \frac{\partial}{\partial t}(\tilde{\rho} \omega) + \frac{\partial}{\partial x_j}(\tilde{\rho} \tilde{u}_i \omega) = \frac{\partial}{\partial x_j}\left[ \frac{\mu_{t}}{\alpha_{\omega}} \frac{\partial \omega}{\partial x_j} \right] + \frac{\alpha_{\omega}}{u_t} \frac{\partial \tilde{u}_i}{\partial x_j} - \rho_{\omega} \omega^2 + \frac{\gamma_{p}}{\omega_{\sigma_{\omega}}^{2}} \frac{1}{\partial x_j} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \] |

| Conservation of Energy | \[ \frac{\partial}{\partial t}(\tilde{\rho} \tilde{h}) + u_j \frac{\partial}{\partial x_j}(\tilde{\rho} \tilde{u}_i \tilde{h}) = \frac{\partial p}{\partial t} + u_j \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\rho \tilde{u}_i \tilde{h}) + \tilde{r}_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} + \tilde{r}_{ij} \frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}(\frac{\mu_{t}}{\rho_{\gamma}} \frac{\partial \tilde{h}}{\partial x_j}) \] |

\[ \bar{D} = \frac{\partial}{\partial t} + \tilde{u}_j \frac{\partial}{\partial x_j} \]
3.2.2 Mesh and Boundary Conditions

Figure 3.2 also shows the computational domain used for the simulations with the associated boundary conditions. The domain extends for 50 $r_{exit}$ downstream of the nozzle exit, where $r_{exit}$ is the nozzle exit radius. Detailed nozzle dimensions were given in Figure 3.1. The lateral walls, considered as the pressure outlet boundary condition, are extended to 15 $r_{exit}$ to assure their impact on the jet flow is insignificant. The typical computational mesh used in the simulations, shown in Figure 3.2, highlights the small mesh size used in regions of high velocity gradients and around the shock location.

3.2.3 Mesh Independence Check

To ensure that the simulations are mesh-independent, several flows were simulated with different mesh densities (number of cells was $1 \times 10^5$, $1.5 \times 10^5$ and $3 \times 10^5$ for the coarse, medium and fine mesh, respectively). In the refinement procedure, we refined the boundary layer mesh close to the nozzle wall, the flow region around the jet shear layer, and the shock areas. Additionally, we slightly refined the mesh in the outer region of the flow. The velocity profiles at $x = 2.256 r_{exit}$ downstream of the nozzle exit for $P_{in} = 160$ kPaa for different mesh sizes are shown in Figure 3.3. To estimate the discretization error, we followed the procedure recommended in [54] for the velocity profiles. The numerical uncertainty in the fine-grid solution was found to be 1.6% for the velocity profiles.
Figure 3.2: Setup of the computational domain (top) and the mesh (bottom). The computational domain is not to scale.
Figure 3.3: Velocity profile at $x = 2.256\ r_{\text{exit}}$ downstream of the nozzle exit for $P_{\text{in}} = 160$ kPa at different mesh resolutions

### 3.2.4 2D vs. 3D Simulations

For a highly over-expanded flow within a converging-diverging nozzle, the flow is inherently not axisymmetric [55]. The complex pattern of the shock surfaces (oblique and normal shocks) is three-dimensional, which means that the azimuthal derivatives of quantities should be accounted for in the governing equations. However, for the range of operating conditions of interest in this study, the flow remains attached to the nozzle walls and no flow separation was observed. These characteristics should permit one to use with reasonable accuracy a 2D axisymmetric model instead of full three-dimensional modeling. Using the 2D model avoids long computational times. To verify that the 2D axisymmetric modeling is capable of capturing the shock surfaces and flow
details, some 3D simulations were conducted to compare the simulations results with those of the corresponding 2D axisymmetric simulations. The SST $k – \omega$ turbulence model was chosen for both sets of simulations because, as will be shown in Section 3.2.5, simulations of a nozzle flow using this model have been validated experimentally. The Mach number and pressure coefficient profiles along the nozzle symmetry axis for the same upstream pressure are shown in Figure 3.4. The 2D profiles are almost identical to their 3D counterparts. Figure 3.5 shows the sonic lines inside the nozzle.
Figure 3.4: Normalized axial velocity profile (Top) and Mach number and pressure coefficient profiles along the nozzle symmetry axis for 2D axisymmetric and 3D simulation (bottom). ($P_{\text{in}} = 160 \text{ kPaa}$ and $P_{\text{out}} = 101 \text{ kPaa}$).
Figure 3.5: Comparison of sonic line (2D) and sonic surface (3D). The contours are colored by the density field ($kg/m^3$).
3.2.5 Model Validation against Experimental Data

This section contains a comparison between the simulation results on the Acoustic Reference Nozzle (ARN) [56] from the present ANSYS Fluent solver and those reported in [43] and [56]. This is to validate the simulations of a nozzle flow using RANS model. The geometry details are reported in [56]. The jet is issued from a nozzle with an exit diameter of 50.8 mm. Setpoint 3 [56], where the jet is unheated ($T_j/T_\infty = 0.950$) and the compressibility effects ($Ma = 0.513$) on the flow evolution could be insignificant, was chosen for the comparison reported here. The NPR of this point is 1.197.

Figure 3.6 and Figure 3.7, respectively, show a comparison of the axial velocity and turbulent kinetic energy along the symmetry axis between the current simulations and those reported in [43] and [56]. In Figure 3.6, the present simulation shows a good agreement with both the experimental data and previous simulation [43] results.

Both the present simulation and the previous simulation [43] predict a longer potential core than that observed experimentally. The current simulation, however, predicts a potential core shorter than the previous simulation [43]. Furthermore, the current simulation predicts a decay rate faster than both the experimental results and the previous simulation [43]. On the other hand, the turbulent kinetic energy (TKE) predicted by the current simulation using the SST $k - \omega$ turbulence model is in better agreement with experimental data than the previous simulations [43].

A comparison of the velocity profile at different positions downstream of the nozzle exit is shown in Figure 3.8. Very good agreement exists in the dimensions of the potential core and shear layer between the experimental data, the earlier simulation [43], and the current simulation. The above
results show that the utilized modeling approach in this paper is adequate to capture the flow features of the flow field studies in this investigation, especially at Mach number of 0.5.

There are some differences between the results from the current simulation and the results from the simulation reported in [43] although the same turbulence model was used (the SST $k - \omega$). This is because an additional diffusivity term was used in the $\omega$ equation in the previous simulations. In addition, the diffusion coefficient in the same equation was altered in [43].

![Figure 3.6: Comparison of axial velocity profile along the symmetry axis between current simulations, those in [43], and experiments [56].](image)
Figure 3.7: Comparison of turbulent kinetic energy profile along the axis of symmetry between current simulations, those in [43], and experiments [56].

Figure 3.8: Comparison of the axial velocity profile downstream of the nozzle exit between current simulations and results in [43] and [56].
3.3 Results and Discussions

In this section, the simulation results of the converging-constant throat-diverging nozzle, as shown in Figure 3.1, are presented. The nozzle has an exit to throat area ratio of 1.34. The flow fields are studied at different nozzle pressure ratios (NPR) in the range \(1.18 \leq \text{NPR} \leq 1.78\), corresponding to pressure inlet in the range \(120 \text{kPa} \leq P_{\text{in}} \leq 180 \text{kPa}\). The downstream pressure is set to standard atmospheric pressure at sea level (101 kPa) and the working fluid is air. Velocity, TKE, pressure coefficient, Mach number and mass entrainment coefficient variations within and downstream of the nozzle are presented.

3.3.1 Mean Axial Velocity Profiles

Figure 3.9 shows a comparison of the normalized axial velocity profiles as computed by the different turbulence models. For simplicity, the graphs are shown for only two upstream pressures: \(P_{\text{in}} = 140 \text{kPa}\) (at which the flow is subsonic along the centerline) and \(P_{\text{in}} = 160 \text{kPa}\) (at which the flow is supersonic along the centerline). This is to compare the profiles at different flow regimes. The graphs at the other upstream pressure values follow the presented graph closely and can be found in Appendix D. The axial velocity profiles are extracted at two locations downstream of the nozzle exit, \(x_1 = r_{\text{inlet}} = 2.256 \, r_{\text{exit}}\) and \(x_2 = 5r_{\text{inlet}} = 11.278 \, r_{\text{exit}}\) (profiles at other locations could be found for reference in Appendix E).

The normalized velocity profiles predicted by different turbulence models follow each other closely at \(P_{\text{in}} = 140 \text{kPa}\); they start with a potential core close to the jet centerline and transition to the slow moving outer flow through a shear layer. The potential core at \(x = 2.256 \, r_{\text{exit}}\) exists in \(0 < y/r_{\text{exit}} < 0.6\). The extent of the potential core in the cross-stream direction at
\( x = 11.278 \ r_{\text{exit}} \) shrinks to \( 0 < y/r_{\text{exit}} < 0.1 \) due to the mixing. The SST \( k - \omega \) model predicts a slightly faster decay in the potential core compared to the Realizable \( k - \varepsilon \) model. The shear layer is contained in \( 0.6 < y/r_{\text{exit}} < 1.5 \) at \( x = 2.256r_{\text{exit}} \), while it has a far-reaching extent \( 0.1 < y/r_{\text{exit}} < 3.5 \) at \( x = 11.278 \ r_{\text{exit}} \) (for comparison purposes, the profiles at \( x = 11.278 \ r_{\text{exit}} \) are clipped to \( y/r_{\text{exit}} = 2.0 \)). Along the centerline, the flow at this inlet pressure approaches sonic conditions at the throat.

At \( P_{\text{in}} = 160 \ \text{kPaa} \) the velocity profiles predicted by the Realizable \( k - \varepsilon \) model are similar to those at the lower pressure except that the potential core reaches a higher normalized peak velocity. The SST \( k - \omega \), however, predicts a velocity deficit in the far field in the potential core, which seems unphysical. This can be seen at the centerline \( (y = 0) \) as velocity profile is dented backward between \( 0 < y/r_{\text{exit}} < 0.4 \). Note that the flow within the nozzle at this pressure is supersonic (both turbulence models predict a peak Mach number of \( \approx 1.45 \) in the diverging section), where the compressibility effects play a critical role in the flow evolution. This velocity deficit, however, disappears in the far field (after \( x \geq 11.278 \ r_{\text{exit}} \)). In fact, both turbulence models predict almost the same normalized axial velocity profile in the far field at \( x = 11.278 \ r_{\text{exit}} \) regardless of the flow condition.

Since the velocity profiles were normalized by the average exit velocity, Table 3.2 illustrates the values of the average exit velocity at different \( P_{\text{in}} \) values as predicted by both turbulence models.
Figure 3.9: The normalized axial velocity profiles downstream of the CCTDN exit for both turbulence models; Extraction line is at (top) $x = 2.256 r_{exit}$ (bottom) $x = 11.278 r_{exit}$ downstream of the nozzle exit.
Table 3.2: The variation of the average exit velocity of the CCTDN with the upstream pressure

<table>
<thead>
<tr>
<th>( P_{\text{in}} ) (kPa)</th>
<th>( U_{\text{exit}} ) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Realizable ( k - \varepsilon )</td>
</tr>
<tr>
<td>120</td>
<td>131.8</td>
</tr>
<tr>
<td>140</td>
<td>175.6</td>
</tr>
<tr>
<td>160</td>
<td>199.7</td>
</tr>
<tr>
<td>180</td>
<td>217.8</td>
</tr>
</tbody>
</table>

3.3.2 Turbulent Kinetic Energy Profiles

The normalized turbulent kinetic energy (TKE) along the axis of symmetry at \( P_{\text{in}} = 140 \) kPa and \( P_{\text{in}} = 160 \) kPa is compared between different turbulence models in Figure 3.10. The transport of TKE to the jet centerline and the evolution of TKE predicted by the two turbulence models is almost the same. The Realizable \( k - \varepsilon \) model, however, predicts some turbulence upstream and downstream of the nozzle exit for \( x/r_{\text{exit}} \lesssim 12 \). This level of turbulence is absent in the predictions of the SST \( k - \omega \) model, which predicts almost zero turbulence along the symmetry axis up to \( x/r_{\text{exit}} \approx 12 \), where the potential core exists.

After the end of the potential core, in the shear layer region, the diffusion of turbulence reaches its maximum value at a downstream distance of \( x/r_{\text{exit}} \approx 18 \), where the normalized TKE graph attains a maximum value. This maximum value is slightly greater than 0.04 for the Realizable \( k - \)
ε model and slightly less than 0.04 for the SST k − ω model. The dissipation of the TKE after the peak point is slightly higher for the SST k − ω model for \( x/r_{\text{exit}} < 22 \), and thereafter the Realizable k − ε model predicts a slightly faster decay, and consequently more mixing, farther downstream.

The contours in Figure 3.11 show the TKE development in the entire flow domain as computed by the Realizable k − ε model at \( P_{\text{in}} = 160 \) kPaa, which is shown to be maximized in the shear layer. While the contours do not show any TKE inside the nozzle along the centerline (as shown in Figure 3.10), they show a clear TKE development close to the walls in the divergent section.

Figure 3.10: The normalized turbulent kinetic energy along the axis of symmetry predicted by the turbulence models at \( P_{\text{in}} = 140 \) kPaa and \( P_{\text{in}} = 160 \) kPaa
Figure 3.11: TKE development as computed by the Realizable $k-\varepsilon$ model at $P_{in} = 160 \text{kPa}$ in the entire flow domain (top) and inside the nozzle (bottom).

### 3.3.3 Mach Number Profiles

Figure 3.12 shows the Mach number profiles along the symmetry axis predicted by both turbulence models at $P_{in} = 140 \text{kPa}$ and $P_{in} = 160 \text{kPa}$ (profiles at the other considered upstream pressure values can be found for reference in Appendix F). Generally, both turbulence models predict almost the same Mach number evolution inside and outside the nozzle. The location of the shocks in the nozzle (when $P_{in} \geq 160 \text{kPa}$) predicted by the two models is the same. In addition, both models show that the sonic flow condition along the centerline is always reached at the constant throat around $x/r_{exit} = -3.2$, just upstream of the divergent section (which starts at $x/r_{exit} = -2.556$).
However, when the flow is subsonic ($P_{in} \leq 140 \ kPa$), the SST $k - \omega$ model predicts slightly higher flow velocities than the Realizable $k - \varepsilon$ model in the converging and constant-throat sections (between $-9.3 < x/r_{exit} < -2.6$). The difference disappears after that and the profiles perfectly match.

When the flow is supersonic ($P_{in} \geq 160 \ kPa$), both turbulence models predict the same Mach number profiles inside the nozzle. Outside the nozzle, there are some slightly higher values predicted by the Realizable $k - \varepsilon$ model within the potential core, i.e., $0 \leq x/r_{exit} \leq 10$, followed by a slightly earlier decay in velocity. The difference disappears after that and the profiles match. Another difference between the SST $k - \omega$ and Realizable $k - \varepsilon$ models is that the former model predicts a sharp velocity decay; whereas the latter model predicts a much smoother velocity decay at the end of the potential core. Figure 3.13 shows the Mach number contours in the entire flow domain at $P_{in} = 140 \ kPa$ and at $P_{in} = 160 \ kPa$ as computed by both turbulence models.
Figure 3.12: Mach number profiles of the CCTDN along the jet symmetry axis as predicted by both turbulence models at $P_{in} = 140 \, kPa$ and $P_{in} = 160 \, kPa$; The figure at the bottom shows the details of the variations in the immediate upstream and downstream of the nozzle exit
Figure 3.13: Mach number contours as predicted by the Realizable $k - \varepsilon$ model at (a) $P_{in} = 140 \, kPa$, (c) $P_{in} = 160 \, kPa$, and as predicted by the SST $k - \omega$ model at (b) $P_{in} = 140 \, kPa$ (d) $P_{in} = 160 \, kPa$. 
3.3.4 Pressure Coefficient

Figure 3.14 shows the evolution of pressure coefficient along the axis of symmetry of the nozzle as predicted by the different turbulence models at various NPR. The pressure coefficient is defined as follows:

\[
\text{Pressure coefficient} = \frac{P_s - P_\infty}{\frac{1}{2} \rho_{exit} U_{exit}^2}
\]

Where \( P_s \) is the pressure along the centerline, \( P_\infty \) is the pressure outlet (\( P_{\text{outlet}} \)) and \( \rho_{exit} \) is the density at the nozzle exit. The computed results from the different turbulence models follow each other closely, with slightly higher results computed by the Realizable \( k - \varepsilon \) model. This is expected since this model predicts a slightly lower velocity in the converging part compared to the SST \( k - \omega \) model. For any \( P_{in} \), the pressure will decrease in the converging part \((-9.3 < x/r_{exit} < -7.3)\), followed by a pressure drop along the constant-diameter throat section. \((-7.3 < x/r_{exit} < -2.56)\). For \( P_{in} \leq 140 \text{ kPa} \), the pressure in the diverging section gradually expands to the downstream pressure, while for the pressure \( 160 \text{ kPa} \leq P_{in} \) there is a sharp change in the pressure profiles as the flow passes through the shock region before adjusting to the downstream pressure. Both turbulence models predict the expected movement of the normal shock closer to the nozzle exit as the NPR increases.
3.3.5 Mass Entrainment Rate Profiles

This specific nozzle is used to provide gas flares with air to facilitate smokeless combustion. Therefore, it is instructive to study the variation of the mass entrainment along the jet centerline with the upstream pressure. The total mass flow rate through a given cross-section downstream of the nozzle is denoted by $M$ and the mass flow rate at the nozzle exit is denoted by $M_{exit}$. The mass entrainment is defined as follows:

$$\text{Mass Entrainment} = M - M_{exit}$$

Figure 3.15 shows that the mass entrainment, normalized by $M_{exit}$, varies parabolically along the centerline of the jet. Consequently, the entrainment rate coefficient, $K_e$ [57], which is defined as:
\[ K_e = \frac{dM_{r_{exit}}}{dx} \left( \frac{\rho_e}{\rho_x} \right)^{0.5} \]

where \( \rho_e \) and \( \rho_x \) are the initial and entrained fluid densities, respectively, increases linearly as shown in Figure 3.16. For sake of illustrations, \( K_e \) graphs at lower pressure (where nozzle flow is subsonic with \( 0.5 < Ma < 0.9 \)) and higher pressure (\( 1.5 < Ma < 2.0 \)) are not shown on the graph. However, the normalized entrainment graphs at operating pressures of \( 120 \) kPaa \( \leq P_{in} \leq 180 \) kPaa almost collapse on the presented graphs with an insignificant difference and they can be found in Appendix G.

![Figure 3.15: The normalized mass entrainment variation as a function of distance along the jet for different turbulence models.](image)
3.3.6 Mass Flux Variation with NPR

The mass flux is defined as the mass flow rate inside the nozzle divided by the throat area, which is calculated as follows:

$$A_t = \pi r_t^2 = \pi \times 1.461^2 = 6.70 \times 10^{-6} \, m^2$$

$$\text{Mass flux} = \frac{M_{\text{inlet}}}{A_t} = \frac{M_{\text{exit}}}{A_t}$$

The graph of the mass flux versus the NPR is shown in Figure 3.17 to give a quantitative picture of the total mass flow rate downstream of the nozzle exit. From Figure 3.17, $M_{\text{exit}}$ at any NPR can be calculated and then used in Figure 3.15 to quantify the mass entrainment downstream of the nozzle. Figure 3.17 shows that the mass flow rate through the CCTDN is directly proportional to
the NPR. There is a small difference between the results from the two turbulence models and this difference (5% or less, depending on the NPR) is a modeling difference.

![Figure 3.17](image)

**Figure 3.17:** The variation of the mass flux inside the CCTDN with the NPR as predicted by the RANS models

### 3.4 Conclusions

This chapter showed the results of the conducted numerical studies of compressible flow in a specific converging-diverging nozzle. The nozzle consists of converging and diverging sections connected via a throat section that has a finite length and constant diameter. The jet generated by the nozzle is used to control the performance of typical industrial flare systems by entraining the right amount of air into the flame to assure a smokeless flame. The numerical studies characterized the nozzle performance at various pressure ratios (NPR) in the range $1.18 \leq \text{NPR} \leq 1.78$ for two different turbulence models. The mean axial velocity profiles at different streamwise locations, mass entrainment rates, the turbulent kinetic energy, Mach number and pressure coefficient along
the jet centerline were presented. The purpose was to study this nozzle and test if the flow regime affects the amount of entrained air. In summary:

- In the case of subsonic nozzle flow, the mean axial velocity profiles were found to be insensitive to the turbulence models used. For supersonic flow, however, this insensitivity was seen only downstream of the nozzle exit and outside the potential core region, while within this region the SST $k - \omega$ model showed an unexpected deficit in the mean axial velocity profiles close to the centerline.

- Therefore, it is found that the SST $k - \omega$ model, which is a superior model when the flow contains a separation region, predicts an unrealistic velocity deficit in the potential core when the flow became supersonic inside the nozzle. This velocity deficit was not associated with any flow separation at the wall as the diverging section angle in this nozzle was relatively small and no flow separation was observed for the flow.

- It is hypothesized that this unrealistic behavior is due to the lack of diffusivity in the SST $k - \omega$ model (i.e. source and sink) to dampen out this behavior.

- The turbulent kinetic energy profiles for different turbulence models follow each other closely, with the Realizable $k - \varepsilon$ model predicting a slightly higher peak value.

- For this nozzle design (CCTDN), the flow along the centerline becomes supersonic at the end of the finite-length throat right before entering the diverging part. The turbulence models capture the same normal shock location along the nozzle symmetry axis. The SST $k - \omega$ predicts a slightly higher Mach number in the converging and constant-throat parts of the nozzle when the flow is subsonic. The Realizable $k - \varepsilon$ model, on the other hand, predicts a slightly higher velocity in the potential core when the flow is supersonic. The
classical pressure coefficient graph along the nozzle behaves, as expected, opposite to the Mach number profile.

- The entrainment coefficient increases linearly with the distance along the jet centerline with an insignificant difference between the turbulence models. The same holds true for the mass flux inside the nozzle at different NPR.

- The NPR, or the flow regime, has an inconsiderable effect on the mass entrainment downstream of the CCTDN.
Chapter 4: Comparison between Different Nozzles Performance

4.1 Introduction

In the previous chapter, the objective was to study the flow within the CCTDN at different flow regimes and boundary conditions using different RANS models. In this chapter, on the other hand, the objective is to compare the flow within the CCTDN to another convergent-divergent nozzle that has the same dimensions but without the constant-throat section. In other words, the converging and diverging parts in the other nozzle are connected directly and the finite length throat section is eliminated. This other nozzle, which is referred to as CDN, is shown in Figure 4.1.

This comparison is mainly to investigate which nozzle helps in entraining more mass to facilitate a smokeless combustion. In addition, the comparison is to investigate the effect of the constant throat section on some other parameters such as the axial velocity, TKE and Mach number. The numerical method, mesh independency and model validation that were shown in the previous chapter are also applicable to this chapter. Therefore, this chapter will start with the results.

4.2 Results and Discussions

Similar to the CCTDN, the CDN has an exit to throat area ratio of 1.34. The flow fields are studied at $1.18 \leq \text{NPR} \leq 1.78$, which corresponds to $120 \text{ kPa} \leq P_{\text{in}} \leq 180 \text{ kPa}$. The downstream pressure is set to standard atmospheric pressure at sea level ($101 \text{ kPa}$) and the working fluid is air. Velocity, TKE, pressure coefficient, Mach number and mass entrainment coefficient variations within and downstream of the nozzle are presented.
4.2.1 Mean Axial Velocity Profiles

Figure 4.2 illustrates the computed results of axial velocity profiles at $x = 2.256 \, r_{exit}$ (i.e. the near field), whereas Figure 4.3 illustrates the velocity profiles at $x = 11.278 \, r_{exit}$ (i.e. the far field). Similar trends are observed for both nozzles and the graphs follow each other closely.

The velocity deficit that is predicted by the SST $k - \omega$ model in the potential core when the flow is supersonic in the CCTDN also exists in the CND case. The potential core and shear layer in both nozzle designs appear at almost the same radial locations ($y = 0.6 \, r_{exit}$ at $x = 2.256 \, r_{exit}$ and $y = 0.2 \, r_{exit}$ at $x = 11.278 \, r_{exit}$) from both turbulence models. Figure 4.2 and Figure 4.3 show that the maximum normalized velocity is slightly higher in the CCTDN profiles. For instance, at $P_i = 160 \, kPa$, the Realizable $k - \varepsilon$ predicts a maximum velocity of $\frac{u}{u_{exit}} \approx 1.35$ in the
CCTDN. In contrast, the same model at the same upstream pressure predicts a maximum $\frac{u}{u_{exit}} \approx 1.25$. Table 4.1 shows an expansion of Table 3.2 to include the average exit velocity of the CDN.

Figure 4.2: A comparison of the normalized axial velocity profiles downstream of the CDN and the CCTDN exits as computed by the Realizable $k - \varepsilon$ (left) and SST $k - \omega$ model (right) in the near field ($x = 2.256 \ r_{exit}$).

Figure 4.3: A comparison of the normalized axial velocity profiles downstream of the CDN and the CCTDN exits as computed by the Realizable $k - \varepsilon$ (left) and the SST $k - \omega$ model (right) in the far field ($x = 11.278 \ r_{exit}$).
Table 4.1: A comparison of the variation of the average exit velocity of the CCTDN and the CDN with the upstream pressure

<table>
<thead>
<tr>
<th>$P_{in}$ (kPa)</th>
<th>$U_{exit}$ of the CCTDN (m/s)</th>
<th>$U_{exit}$ of the CDN (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Realizable $k - \varepsilon$</td>
<td>SST $k - \omega$</td>
</tr>
<tr>
<td>140</td>
<td>175.6</td>
<td>180.3</td>
</tr>
<tr>
<td>160</td>
<td>199.7</td>
<td>201.5</td>
</tr>
</tbody>
</table>

4.2.2 Turbulent Kinetic Energy Profiles

It was shown in the previous chapter that along the centerline, both RANS models predict negligible TKE inside the CCTDN and that the TKE peak is reached in the shear layer (Figure 3.10). This holds true in the CDN only if using the SST $k - \omega$ model. The Realizable $k - \varepsilon$ model, in comparison, predicts a seemingly unphysical spike in the TKE inside the CDN at higher pressure values ($P_{in} \geq 160 \text{ kPa}$) as shown in Figure 4.4 (left).

In fact, the same spike was observed in the TKE profiles of the CCTDN when the flow is supersonic, but this spike disappears upon changing the wall function from Standard to Non-Equilibrium. This does not help in the case the CDN as the spike appears regardless of wall function. Because the TKE is very sensitive to the mesh resolution, several mesh refinements were conducted but this spike was found to be mesh independent.

Since the SST $k - \omega$ model does not predict any turbulence inside any of the nozzles (CCTDN or CDN), Figure 4.4 (right) shows the computed results from the SST $k - \omega$ model for both nozzles.
at \( P_{in} = 140 \, kPa \) and \( P_{in} = 160 \, kPa \). Although the TKE peaks are slightly higher in the CCTDN, the TKE graphs of both nozzles follow each other closely.

![Graphs showing TKE along the axis of symmetry and comparison of TKE profiles in CDN and CCTDN](image)

Figure 4.4: Left: The TKE along the axis of symmetry of CDN as computed from the Realizable \( k-\varepsilon \) and the SST \( k-\omega \) models. Right: A comparison of the TKE profiles in the CDN and the CCTDN as computed from the SST \( k-\omega \) model.

### 4.2.3 Mach Number Profiles

Similar to the CCTDN case, the SST \( k-\omega \) model always predicts Mach number values inside the CDN that are slightly higher than the predictions of the Realizable \( k-\varepsilon \) along the centerline as shown in Figure 4.5. In the CCTDN, it was shown that both RANS models predict a subsonic flow for \( P_{in} = 140 \, kPa \). Similarly, the Realizable \( k-\varepsilon \) model also predicts a subsonic flow inside the CDN at \( P_{in} = 140 \, kPa \). However, the SST \( k-\omega \) model predicts a supersonic flow (\( Ma = 1.2 \)) inside the CDN at \( P_{in} = 140 \, kPa \) as shown in Figure 4.5. Therefore, it is observed that the
SST $k-\omega$ model predicts that the CDN design requires a smaller NPR to reach to the supersonic condition.

Both RANS models predict the same Mach number development downstream of the nozzle exit (i.e. $x/r_{exit} \geq 0$) in both the CDN and CCTDN. The is true regardless of the flow condition as depicted in Figure 4.6 (which shows subsonic profiles) and Figure 4.7 (which shows supersonic profiles). Inside the nozzles (i.e. $x/r_{exit} \leq 0$), the two turbulence models also predict the same maximum Mach number values when the flow is subsonic (Figure 4.6). However, when the flow becomes supersonic, both RANS models predict that the maximum Mach number inside the CDN will always be higher than the CCTDN as illustrated in Figure 4.7.

Figure 4.5: Mach number profiles of the CDN along the jet symmetry axis as predicted by both turbulence models at $P_{in} = 140 \text{kPa}$ and $P_{in} = 160 \text{kPa}$; The figure on the right shows the details of the variations in the immediate upstream and downstream of the nozzle exit.
Figure 4.6: Mach number profiles of the CDN and the CCTDN along the jet symmetry axis when the flow is subsonic as predicted by the Realizable $k - \varepsilon$ model (left) and the SST $k - \omega$ model (right).

Figure 4.7: Mach number profiles of the CDN and the CCTDN along the jet symmetry axis when the flow is supersonic as predicted by the Realizable $k - \varepsilon$ and the SST $k - \omega$ models. The figure on the right shows the profiles inside the nozzle around the normal shock location.
4.2.4 Mass Entrainment Rate Profiles

The main objective of the comparison between the CCTDN and the CDN is to investigate which nozzle design would help in entraining more air mass from the surrounding to serve the flare system better.

The simulations indicate that both nozzles result in almost the same normalized mass entrainment. The entrainment rate in the CDN, normalized by $M_{exit}$, also varies parabolically just like the case in the CCTDN. Therefore, the entrainment rate coefficient $K_e$ varies linearly as shown in Figure 4.8 and the mass entrainment graphs of the CDN and the CCTDN any $P_{tn}$ collapse perfectly on each other.

It is concluded that the normalized mass entrainment and its rate of change are the same. Hence, at any NPR within the considered range, one graph can predict the mass entrainment with a very good accuracy.
Figure 4.8: The computed entrainment rate coefficient, $K_e$, of both nozzles from both RANS models.

### 4.2.5 Mass Flux Variation with NPR

Figure 4.9 depicts the mass flux variation with NPR in both nozzles. It is shown that both RANS models predict approximately the same mass flux inside both nozzles at different NPR and with less than 5% difference. As expected, the mass flux is directly proportional to the NPR.
Figure 4.9: A comparison between the variations of the mass flux inside both nozzles with the NPR as predicted by both RANS models

4.3 Conclusions

In this chapter, a thorough comparison between the CCTDN and the CDN was conducted. The comparison included some parameters such as axial velocity, Mach number and TKE profiles. The objective was to investigate how the addition of a throat section that has a constant diameter and a finite length to a CDN would affect and change the physics and the flow behavior. In addition, it was desired to find out which nozzle design would facilitate more mass entrainment downstream of the nozzle exit.

For a given NPR, both RANS models predict similar axial velocity profiles inside both nozzles. The extent of the potential core is seen to be slightly higher in the CCTDN with less than 7%
difference. The axial velocity deficit that was predicted by the SST $k - \omega$ model when the flow is supersonic close to the CCTDN exit is also seen close to the CDN exit.

The SST $k - \omega$ model indicates that the TKE profiles follow each other closely, with almost the same dissipation rate. However, slightly higher TKE peaks are observed inside the CCTDN, whether the flow is supersonic or subsonic. It is also observed that this model predicts some turbulence inside the CDN and in its potential core, which is absent in the TKE profile of the CCTDN. The Realizable $k - \varepsilon$ model also predicts similar trends. However, when the flow becomes supersonic, this model predicts a high TKE spike inside the nozzle, which does not seem physical.

When analyzing Mach number development in the CCTDN last chapter, it was concluded that the SST $k - \omega$ model predicts a slightly higher Mach number inside the nozzle. This is also true in the case of the CDN. In this chapter, it is also observed that, at the same boundary conditions, the Mach number will be higher inside the CDN than the CCTDN if the flow is supersonic. In fact, the CDN needs a smaller NPR to reach to the sonic state. On the other hand, both of the RANS models predict approximately the same Mach number inside both nozzles if the flow is subsonic. The main conclusion is that the resulted mass entrainment from both nozzles, at the same boundary conditions, differs insignificantly regardless of the flow regime (i.e. subsonic, sonic or supersonic) and regardless of the turbulence model. Therefore, both nozzle designs are capable of entraining the same amount of air mass from the surrounding when used in gas flare systems.
Chapter 5: Effect of Geometry Change on Nozzle Performance

5.1 Introduction

This chapter examines the effect of changing the divergence angle of the CCTDN on the mass entrainment. The other parameters that were studied previously, such as Mach number and TKE, will not be considered in this chapter because this has been covered in other studies [44].

For a nozzle with a given shape, the controlling parameter of the flow is the NPR. However, if the nozzle shape is to be changed, the main controlling parameter is the expansion ratio, or the divergence angle, which has a greater effect on the nozzle flow than the NPR [44].

In this chapter, the mass entrainment obtained downstream of CCTDN jet will be studied at three different divergence angles. Besides \( \theta = 3^\circ \) (which represents the original design), the mass entrainment will be computed for \( \theta = 1^\circ \) and \( \theta = 5^\circ \). By changing the divergence angle, all the dimensions are kept unchanged except for the exit diameter. Therefore, the exit to throat area ratio will change accordingly.

For the original design, which was shown in Figure 3.1, \( \theta \) was calculated as follows:

\[
\theta = \tan^{-1} \left( \frac{r_{exit} - r_{throat}}{\text{Length of the diverging section}} \right) = \tan^{-1} \left( \frac{3.378 - 2.921}{4.318} \right) = 3^\circ
\]

For \( \theta = 5^\circ \):

\[
r_{exit} - r_{throat} = \text{Length of diverging section} \times \tan(5^\circ) = 4.318 \tan(5^\circ) = 0.378 \text{ mm}
\]

Consequently,
\[ D_{exit} = \left( 0.378 + \frac{2.921}{2} \right)^2 = 3.677 \text{ mm} \]

Similarly, for \( \theta = 1^\circ \):

\[ r_{exit} - r_{throat} = \text{Length of the diverging section} \times \tan(\theta) = 4.318 \tan(1^\circ) = 0.075 \text{ mm} \]

Hence,

\[ D_{exit} = \left( 0.075 + \frac{2.921}{2} \right)^2 = 3.071 \text{ mm} \]

The new exit diameters that result from changing the divergence angle along with the associated change in the expansion ratio are summarized in Table 5.1 below.

### Table 5.1: The exit diameter and the expansion ratio of the CCTDN at different divergence angles

<table>
<thead>
<tr>
<th>Divergence Angle</th>
<th>Exit Diameter (mm)</th>
<th>Expansion ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \theta = 1^\circ )</td>
<td>3.071</td>
<td>1.11</td>
</tr>
<tr>
<td>( \theta = 3^\circ )</td>
<td>3.378</td>
<td>1.34</td>
</tr>
<tr>
<td>( \theta = 5^\circ )</td>
<td>3.677</td>
<td>1.58</td>
</tr>
</tbody>
</table>

#### 5.2 Numerical Method

##### 5.2.1 Mesh and Boundary Conditions

The numerical method and model validation that were shown in Chapter 3 are also applicable to this chapter. The mesh is structured everywhere and the results are shown for \( P_{in} = 140 \text{ kPaa} \) only using the Realizable \( k - \varepsilon \) model.
5.2.2 Mesh Independency Check

To ensure that the results in this study are independent on the mesh resolution, the simulations were conducted at three different mesh resolutions: Coarse ($1.76 \times 10^3$ cells), intermediate ($2.6 \times 10^3$ cells) and fine ($3.5 \times 10^3$ cells). Figure 5.1 shows the axial velocity profiles of the CCTDN with $\theta = 1^\circ$ (left) and $\theta = 5^\circ$ (right) from three different mesh resolutions. The figure shows the mesh independency of the results.

![Figure 5.1: The axial velocity profiles, normalized by the exit velocity, at $x = 1 r_{inlet}$ of the CCTDN with $\theta = 1^\circ$ (left) and $\theta = 5^\circ$ (right). The results were computed using the Realizable k – $\varepsilon$ model at $P_{in} = 140$ kPaa.](image)

5.3 Results and Discussions

First, it is desired to see how changing the divergence angle and consequently changing the nozzle expansion ratio would affect the flow condition. Figure 5.2 shows how the Mach number profiles along the symmetry axis change with the divergence angle at the same boundary conditions. Since the nozzles have different exit radii, the x-axis is normalized with the inlet radius of these nozzles,
which is equal. As expected, a higher Mach number is obtained inside the nozzle as the divergence angle increases. However, the simulations suggest that flow exits the nozzle with the same Mach number ($Ma = 0.7$). It is also observed that the potential core extent is inversely proportional to the divergence angle.

It was shown in the previous chapters that the normalized mass entrainment downstream of this nozzle is not highly sensitive to the flow condition. Similarly, the normalized mass entrainment is only slightly affected by changing the divergence angle as shown in Figure 5.3, which shows the normalized entrainment rate downstream of the nozzle at $P_{in} = 140$ kPaa as predicted by the Realizable $k – \varepsilon$ model.

![Figure 5.2: The variation of Mach number profiles along the symmetry axis with the divergence angle as computed from the Realizable $k – \varepsilon$ model at $P_{in} = 140$ kPaa.](image)
Figure 5.3: The variation of the normalized mass entrainment with the divergence angle downstream of the three nozzles.

5.4 Conclusions

This chapter focused on the effect of changing the divergence angle of the CCTDN on the mass entrainment by comparing the variation of the mass entrainment at divergence angles $\theta = 1^\circ$ and $\theta = 5^\circ$ to the divergence angle of the original design of the CCTDN, which is $\theta = 3^\circ$. Since one of the conclusions in Chapter 3 was that the NPR and the turbulence model have a negligible effect on the computed mass entrainment, this study was conducted at $P_{\text{in}} = 140$ kPaa using the Realizable $k - \varepsilon$ model only.

The computed results show that the divergence angle of this specific CCTDN has less than a 5% effect on the normalized mass entrainment graphs.
Chapter 6: Simplified Full Flare System

6.1 Introduction
The whole study so far investigates and analyzes individual components (mainly JICF and nozzles) of the full system. This chapter shows a preliminary study of the whole system, with the JICF (gas flare) and the CCTDNs integrated together. Unfortunately, the actual system that is used in the field is huge as the gas flare has a diameter of 1.22 m and sufficiently meshing the flow domain requires very expensive computations that cannot be handled by the available resources. Scaling down the system is not easy either due to the huge difference between the nozzles and the flare dimensions. Therefore, a mock setup was created with a jet diameter of 30 mm and nozzles diameter of 1 mm. This is just to show how the presence of the nozzles will affect the flow in the gas flare vicinity.

6.2 Numerical Method
6.2.1 Computational Domain
The diameter of the jet was chosen to be 30 mm, whereas the diameter of each of the eight supersonic nozzles that surround the flare is 1 mm. These nozzles are distributed evenly around the flare exit on a 45° intervals. As shown in Figure 6.1, the distance from the crossflow inlet to the jet inlet is $2.18D$ and the domain extends for $4.85D$ downstream of the jet edge. The domain also extends vertically up for $6.67D$. 
6.2.2 Mesh and Boundary Conditions

A total number of 9.5 million cells was used in the mesh shown in Figure 6.2. However, this mesh is coarse and its accuracy needs to be verified by a finer mesh. It is hoped that this mesh, which was the best one could get with the available resources, is sufficient to provide at least a good qualitative picture of the flow field.

The jet inlet velocity \((U_j)\) and the crossflow velocity \((U_\infty)\) are both equal to 1 \(m/s\). The magnitude of the exit velocity \((W)\) of each nozzle is 10 \(m/s\), pointing towards the flare jet. Since the nozzles make an angle of 30° with respect to the vertical dimension, the calculated velocity components of each nozzle are shown in Table 6.1. \(\theta\) and \(\phi\) are defined as the angles with respect to the horizontal \((x-axis)\) and vertical \((z-axis)\) dimensions, respectively.

<table>
<thead>
<tr>
<th>Nozzle number</th>
<th>(\theta)</th>
<th>(\phi)</th>
<th>(W_x)</th>
<th>(W_y)</th>
<th>(W_z)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Degrees</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>-30</td>
<td>-5.00</td>
<td>0</td>
<td>8.66</td>
</tr>
<tr>
<td>2</td>
<td>45</td>
<td></td>
<td>-3.54</td>
<td>-3.54</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>90</td>
<td></td>
<td>0</td>
<td>-5.00</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>135</td>
<td></td>
<td>3.54</td>
<td>-3.54</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>180</td>
<td></td>
<td>5.00</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>225</td>
<td></td>
<td>3.54</td>
<td>3.54</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>270</td>
<td></td>
<td>0</td>
<td>5.00</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>315</td>
<td></td>
<td>-3.54</td>
<td>3.54</td>
<td></td>
</tr>
</tbody>
</table>

The following is a sample of calculations for the velocity components of nozzle 1:

\[
W_x = W \cos \theta \sin \phi = 10 \cos(0°) \sin(-30°) = -5.00 \text{ m/s}
\]

\[
W_y = W \sin \theta \sin \phi = 10 \sin(0°) \sin(-30°) = 0 \text{ m/s}
\]

\[
W_z = W \cos \phi = 10 \cos(-30°) = 8.66 \text{ m/s}
\]
In another simulation, the holes that represent the exits of the nozzles are set as walls in order to represent a simple JICF setup without the nozzles to compare flow field with and without them. The time step size is 0.001 s and the turbulence model used is the SST $k - \omega$ model since it was shown in chapter 2 that it can simulate such a flow with a reasonable accuracy.

Figure 6.1: The computation domain (top) and the flare exit surrounded by the eight holes that represent the exit of the nozzles (bottom).
6.3 Results and Discussions

Figure 6.3 shows an iso-surface of velocity magnitude equal to 2 m/s. The figure shows the field for two cases; with and without the nozzles. With the nozzles on, the zone in which the combustion should take place has a higher velocity magnitude. Also, the contours indicate that the jet will have a higher maximum velocity when the nozzles are on as can be obviously predicted. Figure 6.4 shows the normalized axial velocity profiles at three positions for both cases (with and without the nozzles). The velocity overshoots are higher when the nozzles are on as illustrated by the solid lines. In addition, it can be seen that these overshoots are shifted up when the nozzles are on by about $z \approx 0.3D$. 

Figure 6.2: The mesh of the circular jet and its surrounding nozzles.
Figure 6.3: Velocity contours of the complete flow domain when the nozzles are off (top) and when the nozzles are on (bottom).
Figure 6.4: The normalized mean velocity magnitude profiles at $x = 0, 1D$ and $2D$. The dashed lines represent the nozzles-off case, while the solid lines represent the nozzles-on case.

### 6.4 Conclusions

This chapter showed a preliminary study of the flare system. Due to the limited available computational resources, it was not feasible to simulate the whole real system. To get at least a qualitative picture of how the presence of these nozzles can affect the flow field, a simplified system was created and studied. Although the created setup was relatively small, it was not possible with the available resources to create a fine mesh. Therefore, the results obtained in this chapter need to be verified by a finer mesh. The results show that the nozzles, as expected, help increasing the jet velocity and that the overshoots of the normalized mean velocity profiles are shifted up.
Chapter 7: Summary and Future Work

7.1 Introduction

The study of a jet into a crossflow (JICF) is very important and it has numerous engineering applications. Therefore, several experimental and numerical studies have focused on it. One industrial example of JICF is gas flares in which the flare (jet) issues hydrocarbons into the atmosphere (crossflow).

This thesis looks at a typical gas flare system used in Saudi Aramco fields. To protect the flare exit tip from failures that result from flare capping, which is the undesired flame accumulation on the flare tip, and to ensure having smokeless combustion to meet the environmental regulations, gas flares are retrofitted with a system called the High Pressure Air Assist System (HPAAS) [1]. This system consists of supersonic air nozzles that are distributed evenly around the circumference of the flare exit. The high velocity jets exiting these nozzles entrain air from the surrounding and thus provide the required air mass to obtain smokeless combustion. In addition, these high velocity jets protect the flare tip by preventing the flame accumulation on it. These nozzles are convergent-divergent nozzles with a specific design in which the converging section and the diverging section are connected via a throat section that has a finite length and a constant diameter.

The objective of this work is to study of such a flare system to investigate the critical flow conditions that would result in the undesired flame accumulation. Therefore, the thesis starts with a validation of different turbulence models against data from JICF experiments, which will be helpful in choosing a model to simulate the flare system. In addition, the thesis studies the flow
within the specific nozzle, which is referred to as CCTDN. Due to some observed differences in the computed results from different RANS turbulence models, the thesis also compares the results from different RANS models. In addition, one objective is to compare performance of different nozzles and also investigating the effect of geometry change on the nozzles performance. A summary of each chapter is given next sections.

### 7.2 Summary of Chapter 1

Chapter 1 started with an introduction about gas flares and the HPAAS used in Saudi Aramco. In addition, it provided the reader with the minimum required theoretical knowledge to understand the rest of the work by briefly explaining the compressible fluid dynamics of nozzles and the JICF theories and main dimensionless numbers. Moreover, it shed light on some previous studies (both experimental and numerical) on nozzles flow and JICF. In addition, Chapter 1 includes a table that summarizes the range of the dimensionless numbers that have been considered in previous JICF studies, which could serve as a good reference for related future work.

### 7.3 Summary of Chapter 2

Because the results of any CFD work must be validated against experimental data to ensure its accuracy, Chapter 2 illustrated the simulation results of three different JICF experiments. In summary, the simulations of each experiment are summarized as follows:

- Rodi’s experiments (RA [4])

The simulations of this experiment were performed using both LES and RANS turbulence models. In spite of the good agreement that has been observed away from the wall, the near-wall results match only qualitatively. Rodi has mentioned some faced difficulties in measuring the velocity.
near the wall and stated that the percentage of error may double in that area. Therefore, it was concluded that this deviation is most likely due to experimental errors. Hence, it was decided to simulate another experiment.

- Sherif’s experiments (SP [5]):
The simulations of this experiment were also conducted using both LES and RANS turbulence models and the results compare qualitatively well with the experimental data. However, there is some quantitative deviation and a literature review has revealed that some other researcher groups have also failed to obtain a quantitative agreement with SP experiments. In fact, these research groups have some suspicions on the accuracy of SP results. They reported that the discrepancies were attributed to some experimental errors and wrong assumptions. These include the measurement method (hot film), the inaccurate assumption of a fully developed flow inside the jet although the pipe was not long enough and the seemingly invalid assumption of a uniform velocity profile inside the jet despite the high Reynolds number [17] [11]. Therefore, several simulations (summarized in Table 2.1) were run to see if overcoming the reported deviation reasons by, for example, reducing the jet length would help in getting a better agreement. Although some modifications helped in bringing the graphs closer to each other, some discrepancies persisted. In fact, even a more recent experiment in which PIV measurements were used showed results that do not agree with SP experiments. Therefore, it was decided to simulate one more experiment.

- Su’s experiments (SM [34]):
The simulation results of this experiment, computed from LES and the SST $k – \omega$, models, have revealed a good quantitative agreement with both the experimental data [34] and the previous DNS [50]. Since both models match well with the experiment, it was concluded that the RANS model,
which is computationally less expensive, is a reasonable option to conduct a cold modeling study of a gas flare.

7.4 Summary of Chapter 3

Since the CCTDNs play an important role in the flare system, Chapter 3 looked at the flow of air within these specific nozzles. The study was conducted at different nozzle pressure ratios (NPR) in the range $1.18 \leq \text{NPR} \leq 1.78$, which corresponds to a pressure inlet in the range $120 \text{ kPa} \leq P_{\text{in}} \leq 180 \text{ kPa}$ and a pressure outlet of $101 \text{ kPa}$ (standard atmospheric pressure at sea level). The 2-D axisymmetric simulations have been conducted using two RANS models (the SST $k - \omega$ and the Realizable $k - \epsilon$ models) to compute several important parameters.

The mass entrainment and its variation downstream of the nozzle at different NPRs have been studied to see how the flow regime affects the mass entrainment. The Mach number, pressure coefficient and TKE profiles along the axis of symmetry have been also computed. In addition, the axial velocity profiles at several downstream positions have been extracted. As far as the mass entrainment is concerned, it was concluded that the flow regime and the range of the considered NPRs have an insignificant effect. Unlike the mass entrainment, the Mach number, pressure coefficient and TKE profiles showed some sensitivities to the used RANS models. For example, it was found that the Realizable $k - \epsilon$ model predicts some turbulence inside the nozzle. This turbulence is absent from the predictions of the SST $k - \omega$. The Mach number profiles have some differences inside the nozzle as well. However, there is generally a very good agreement between the results from both models. The axial velocity profiles, as computed from both models, show a very good matching. The only exception is close to the nozzle exit only when the flow is supersonic, as the SST $k - \omega$ predicts a seemingly unphysical deficit in the axial velocity close to
the centerline. It is hypothesized that the lack of diffusivity in the SST $k - \omega$ model (i.e. source and sink) to dampen out this behavior is the reason for this behavior.

### 7.5 Summary of Chapter 4

Chapter 4 showed a comparison of the flow within the CCTDN and another nozzle (CDN). The comparison was held at the same boundary conditions to investigate the effect of the constant-throat section on the nozzle flow, which was interesting as the author is not aware of any similar study. More importantly, the objective was to see which nozzle could facilitate more mass entrainment. The main conclusions are:

- The normalized mass entrainment downstream of both nozzles, at the same boundary conditions, is almost the same.
- The normalized axial velocity profiles downstream of both nozzles follow each other closely. The CCTDN has a slightly higher normalized velocity. The velocity deficit that is predicted by the SST $k - \omega$ model close to the CCTDN exit is also observed in close to the CDN.
- The TKE peak, which is observed in the shear layer, is higher in the CCTDN. The negligible turbulence that is predicted by the Realizable $k - \varepsilon$ model inside the CCTDN becomes high inside the CDN when the flow is supersonic.
- At the same boundary conditions, both nozzles predict the same Mach number at the nozzle exit at any flow regime. This holds true inside the nozzle only when the flow is subsonic. When the flow is supersonic, however, the maximum Mach number inside the CCTDN is about 25% less than the maximum Mach number inside the CDN. This difference disappears after the normal shock.
7.6 Summary of Chapter 5

This chapter depicted the effect of the divergence angle on the mass entrainment downstream of the CCTDN and consequently it showed the effect of changing the exit diameter of the nozzle. It was concluded that changing the divergence angle over a range from $1^\circ \leq \theta \leq 5^\circ$ results in an insignificant difference in the normalized mass entrainment graphs.

7.7 Summary of Chapter 6

Due to the limited available resources, the study of the whole system (with the gas flare combined with the CCTDNs) is left for future work and this thesis focused on some aspects that were believed to be instructive prior to the simulation of the whole system. However, a preliminary study of a simplified geometry of the real flare system was shown in Chapter 6. The mesh is relatively coarse and hence the results should be verified by a finer mesh. The chapter compared the flow field of a simple JICF with and without the nozzles around the jet exit. It was shown that with the nozzles on, the overshoots of the normalized mean velocity magnitude profiles are higher than the case in which the nozzles are off. This is attributed to the fact that the nozzles add momentum to the jet exiting the pipe.

7.8 Future Work

In summary, the following items are suggested for future work:

- Investigating the reason behind the deficit in the axial velocity profiles which predicted by the SST $k-\omega$ model close to the nozzle exit in case of supersonic flow.
- Conducting cold flow modeling of the flare and the nozzles to find the critical flow conditions that would result in flame capping.
- Conducting hot flow modeling of the real system. This way, the combustion and its physics will be included in the investigations and hence it will make the study more comprehensive.
Bibliography


[22] A. Ben-Yakar, M. G. Mungal, and R. K. Hanson, “Time evolution and mixing


Appendices

Appendix A  Mean longitudinal velocity profiles in RA experiments for $R = 0.5$ at location $y = 0$ and $x = 1D$ (top) and $x = 2D$ (bottom) as computed by the SST $k-\omega$ model and LES.
Appendix B  Mean velocity profiles of SP experiments (Case 1) at \( y = 0 \) and two streamwise locations, \( x = 3.670D \) (top) and \( x = 5.505D \) (bottom).
Appendix C  User Defined Function (UDF) to model a velocity profile that follows the $1/7$ power law

```c
#include "udf.h"

DEFINE_PROFILE(Inlet_Velocity_BC, thread, position)
{
    real x[ND_ND];
    real radial_coor;
    face_t f;

    real U_mean = 0.5064;  /* area weighted average velocity in m/s */
    real U_max = 0.0;      /* Max velocity for power law profile in m/s */
    real n = 7.0;          /* power law which could be 1/8 or 1/9 */
    real R = 0.5*13.84e-3; /* Pipe radius in meters */

    U_max = U_mean*(n+1)*(2*n+1)/(2*n*n); /* converts U_mean to U_max */

    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        radial_coor = pow(x[0]*x[0]+x[1]*x[1],1.0/2.);
        /* note the inlet plane is assumed to be perpendicular to x-axis */
        /* If the inlet plane in perpendicular to z-axis uncomment */
        /* the following and comment the above line */
        /* radial_coor = pow(x[0]*x[0]+x[1]*x[1],1.0/2.); */

        F_PROFILE(f, thread, position) = U_max*pow(1-radial_coor/R,1.0/n);
    }
    end_f_loop(f, thread)
}
```
Appendix D  The normalized axial velocity profiles downstream of the CCTDN exit for both turbulence models at the other upstream pressures; Extraction line is at (top) $x = 2.256 \, r_{exit}$ (bottom) $x = 11.278 \, r_{exit}$ downstream of the nozzle exit.
Appendix E The normalized axial velocity profiles downstream of the CCTDN exit for the Realizable $k - \varepsilon$ model (top) and The SST $k - \omega$ model (bottom) at $P_{in} = 160 \text{kPa}$ at more extraction lines

![Graph of normalized axial velocity profiles downstream of the CCTDN exit for the Realizable $k - \varepsilon$ model (top) and The SST $k - \omega$ model (bottom) at $P_{in} = 160 \text{kPa}$ at more extraction lines.](image-url)
Appendix F  Mach number profiles of the CCTDN along the jet symmetry axis as predicted by both turbulence models at $P_{in} = 120 \text{kPaa}$ and $P_{in} = 180 \text{kPaa}$
Appendix G  The entrainment rate as a function of distance along the jet for different turbulence models at all the considered upstream pressures.