NUMERICAL FLUID DYNAMICS AND COMBUSTION STUDY OF  
EMULSIFIED CANOLA OIL DROPLETS IN A SWIRL PROMOTED  
COMBUSTION CHAMBER  

A Dissertation  

by  

QIBO LI  

Submitted to the Office of Graduate and Professional Studies of  
Texas A&M University  
in partial fulfillment of the requirements for the degree of  

DOCTOR OF PHILOSOPHY  

Chair of Committee, Jorge Alvarado  
Committee Members, Kalyan Annamalai  
Hamn-Ching Chen  
Othon Rediniotis  
Head of Department, Andreas Polycarpou  

December 2015  

Major Subject: Mechanical Engineering  

Copyright 2015 Qibo Li
ABSTRACT

Recently, straight vegetable oils (SVO) have been considered as combustion fuels due to their relatively high heating value and negligible content of sulfur and nitrogen, which is beneficial from the combustion point of view. However, the high viscosity of SVO has hindered their ability and applicability in combustion systems. In order to reduce the effect of high viscosity of SVO, methanol has been used as an additive to reduce its apparent viscosity. Furthermore, a twin-fluid atomizer has been used to form and dispense a fine spray of SVO to ensure complete combustion. To enhance the combustion characteristics of SVO, a multiple vane swirler has been adapted and used inside a conventional combustion chamber to enhance the mixing between the emulsified SVO fuel blend and the surrounding flowing air. However, the effects of vortex generation and droplet distribution on combustion emissions for SVO biofuels in a combustion chamber still are not well understood.

Recent studies have revealed that large eddy simulation (LES) as CFD technique is capable of capturing the unstable features of two-phase swirling flows. This technique provides an effective and economical approach to study the behavior of air and coupled with the other model to predict the fuel spray droplets and combustion characteristics in a conventional combustion chamber. In this study, blends of canola oil and methanol
have been selected as emulsified fuels for simulation purposes. To fully understand the
effects of swirl number and equivalence ratio on the distribution of emulsified canola oil
droplets, the discrete phase model (DPM), stochastic model and TAB model have been
adopted to simulate SVO blend droplets. Furthermore, the Eddy-Dissipation model has
been used to understand the effects of swirl number and equivalence ratio on combustion
characteristics of SVO. A robust and stable numerical algorithm has been selected and
validated. The interaction between the swirling air flow and fuel droplets have been
simulated using LES by considering two swirl numbers. The effects of swirl number and
equivalence ratio on droplet size distribution and combustion characteristics in the
combustion chamber have been considered and evaluated correspondingly. Results
reveal that turbulent flow, fuel blend droplet distribution, and combustion characteristics
are affected by the swirl number significantly. Moreover, the simulation results have the
same trend as the experimental work, but the error of simulation depends on the intensity
of turbulence.
DEDICATION

This dissertation is dedicated to my family, Dr. Jiding Li, Mrs. Jing Han, Mr. Fu Han, Mr. Yimeng Hu, Mrs. Qiao Shu, and Mrs. Xiaodan Hu, for their love, guidance, encouragement and continuous support throughout my vital educational years. This work would not have been possible without their blessings.
ACKNOWLEDGEMENTS

I would like to offer my sincere gratitude to several people who made this dissertation possible. First, I wish to thank my committee chair, Dr. Jorge L. Alvarado, for his guidance and mentorship. He has always been extremely helpful, and has generously shared his knowledge with me throughout the numerous discussions we had during the course of this research work. I would also like to thank the committee members for their kind assistance: Dr. Kalyan Annamalai, Dr. Hamn-Ching Chen, Dr. Waruna Kulatilaka and Dr. Othon Rediniotis.

In addition, I would like to thank my fellow members of Dr. Alvarado’s research team: Shreyas Bhimani, Gautam Sandesh, Hyunseok Nam, for all their contributions of combustion experiments; Chun-Wei Yao, Taolue Zhang, and Minsuk Kong, for their wholehearted support.

In closing, I would like to thank my parents for their encouragement and the unconditional love they have instilled in me right from the time when I was a little child. Furthermore, I wish to thank my lovely wife, Xiaodan Hu, for inspiring me to make the necessary effort to bring this dissertation to fruition.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>ii</td>
</tr>
<tr>
<td>DEDICATION</td>
<td>iv</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>v</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xiii</td>
</tr>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1. Motivation</td>
<td>2</td>
</tr>
<tr>
<td>1.2. Objective</td>
<td>3</td>
</tr>
<tr>
<td>1.3. Overview</td>
<td>4</td>
</tr>
<tr>
<td>2. LITERATURE REVIEW</td>
<td>5</td>
</tr>
<tr>
<td>2.1. Literature review of experimental studies on straight vegetable oil (SVO) combustion</td>
<td>5</td>
</tr>
<tr>
<td>2.2. Literature review of numerical studies on swirl-promoted turbulent flow in combustion chambers</td>
<td>8</td>
</tr>
<tr>
<td>2.3. Literature review of large eddy simulation on swirl-promoted turbulent combustion</td>
<td>10</td>
</tr>
<tr>
<td>3. METHODOLOGY OF THE STUDY</td>
<td>15</td>
</tr>
<tr>
<td>3.1. Description of combustion chamber geometry with boundary conditions</td>
<td>15</td>
</tr>
<tr>
<td>3.2. Description of fuel thermophysical properties</td>
<td>19</td>
</tr>
<tr>
<td>3.3. Description of grid generation and numerical algorithm</td>
<td>22</td>
</tr>
<tr>
<td>4. LARGE EDDY SIMULATION OF SWIRL PROMOTED TURBULENT COLD FLOW IN A COMBUSTION CHAMBER</td>
<td>43</td>
</tr>
<tr>
<td>4.1. Validation of numerical approach on turbulent flow field</td>
<td>43</td>
</tr>
<tr>
<td>4.2. Large eddy simulations of swirl-promoted turbulent cold flow in combustion chamber</td>
<td>52</td>
</tr>
</tbody>
</table>
4.2.1. Effects of swirl number on recirculation zones within the combustion chamber .................................................................53
4.2.2. Effects of swirl number on flow structure within the combustion chamber .................................................................58
4.2.3. Effects of swirl number on vorticity profile .................................................................64
4.3. Fuel droplet distribution in combustion chamber .................................................................68
4.3.1. Validation of fuel droplet distribution in combustion chamber ......68
4.3.2. Emulsified canola oil droplet distribution within swirl-promoted turbulent cold flow in combustion chamber.........................71

5. FLOW AND COMBUSTION OF SWIRL-PROMOTED FLOW OF EMULSIFIED CANOLA OIL USING LES-BASED NUMERICAL SCHEME.......85
5.1. Validation of numerically-based combustion scheme..........................85
5.2. Swirl-promoted turbulent combustion of emulsified canola oil........89
  5.2.1. Swirl-promoted turbulent flow during combustion.................90
  5.2.2. Effects of swirl number and equivalence ratio on combustion characteristics.................................................................104

6. CONCLUSIONS .................................................................................................................................................122
6.1. Large eddy simulations of swirl-promoted turbulent cold flow in combustion chamber – Conclusions........................................122
6.2. Fuel droplet distribution in combustion chamber – Conclusions........123
6.3. Swirl-promoted turbulent combustion of emulsified canola oil – Conclusions........................................................................124

7. FUTURE WORK .........................................................................................................................................................126
REFERENCES ..........................................................................................................................................................128
APPENDIX A ........................................................................................................................................................135
### LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig. 1. 1973 and 2013 fuel shares of OECD [1]</td>
<td>1</td>
</tr>
<tr>
<td>Fig. 2. Schematic of combustion chamber</td>
<td>16</td>
</tr>
<tr>
<td>Fig. 3. (a) Front view of swirler, (b) Assembly drawing of swirler in guiding pipe</td>
<td>17</td>
</tr>
<tr>
<td>Fig. 4. Computational mesh of (a) Cone and Swirler, (b) Entire domain</td>
<td>22</td>
</tr>
<tr>
<td>Fig. 5. Schematic diagram for fuel sprays generation</td>
<td>29</td>
</tr>
<tr>
<td>Fig. 6. Particle distortion for the TAB model</td>
<td>32</td>
</tr>
<tr>
<td>Fig. 7. Overview of the pressure-based solver</td>
<td>38</td>
</tr>
<tr>
<td>Fig. 8. Schematic diagram of the PRESTO! Scheme</td>
<td>41</td>
</tr>
<tr>
<td>Fig. 9. (a) Schematic of a chamber with an inlet duct. Dimensions are in mm, (b) a coarse grid mesh of sudden expansion chamber used for validation, (c) a fine grid mesh of sudden expansion chamber used for validation</td>
<td>44</td>
</tr>
<tr>
<td>Fig. 10. Comparison of simulation and experimental results for turbulent flow without swirler at axial location of $z/H = 10.5$</td>
<td>47</td>
</tr>
<tr>
<td>Fig. 11. Schematic of a sudden expansion chamber with swirler. Dimensions area in mm</td>
<td>48</td>
</tr>
<tr>
<td>Fig. 12. (a) Coarse mesh with tetrahedron grid elements; (b) Fine mesh with tetrahedron grid elements; (c) Fine mesh with a combination grid</td>
<td>49</td>
</tr>
<tr>
<td>Fig. 13. Comparison of three simulation cases and experimental data for the chamber with swirler at axial location ($z/H$) of 0.38</td>
<td>50</td>
</tr>
<tr>
<td>Fig. 14. Radial distribution of mean velocity components at two axial locations</td>
<td>51</td>
</tr>
<tr>
<td>Fig. 15. Schematic of domain for swirl promoted turbulent simulation. Dimensions is in mm</td>
<td>53</td>
</tr>
<tr>
<td>Fig. 16. Combination mesh for swirl promoted turbulent simulation</td>
<td>53</td>
</tr>
<tr>
<td>Fig. 17. Radial distribution of time-averaged normalized velocity components at (a) axial direction, (b) radial direction, (c) tangential direction, respectively</td>
<td>55</td>
</tr>
</tbody>
</table>
Fig. 18. Radial distribution of normalized turbulent kinetic energy ...

Fig. 19. Iso-Surface of time-averaged axial velocity on $<u_t>$ = 0 for (a) SN of 1.0 and (b) SN of 1.4 ..................................................

Fig. 20. Snapshot of vorticity profile at the $z$-$r$ plane on SN of 1.0 and SN of 1.4 at flow time of 500 ms..................................................

Fig. 21. Streamlines at the $r$-$\theta$ planes of $z/H= 0.38$, $z/H=1$, and $z/H=4$ on (a) SN of 1.0 (b) SN of 1.4..................................................

Fig. 22. Instantaneous streamlines at the $r$-$\theta$ planes ($z/H$) of 1 when SN is 1.4 at time of (a) 450 ms, (b) 500 ms, (c) 550 ms, (d) 600 ms, (e) 650 ms, (f) 700 ms ..................................................

Fig. 23. Instantaneous streamlines at the $r$-$\theta$ planes ($z/H$) of 4 when SN is 1.4 at time of (a) 450 ms, (b) 500 ms, (c) 550 ms, (d) 600 ms, (e) 650 ms, (f) 700 ms ..................................................

Fig. 24. Snapshot of the iso-surface based on Q-criterion ($Q= 10000$ s$^{-2}$) at flow time of 500 ms..................................................

Fig. 25. Snapshot of the iso-surface based on Q-criterion ($Q= 10000$ s$^{-2}$) when SN is 1.4 at flow time of (a) 300 ms, (b) 400 ms, (c) 500 ms, and (d) 600 ms ....

Fig. 26. Schematic of the combustor chamber by Jones et al. [66]..........................

Fig. 27. Computational mesh for droplet spray validation..........................

Fig. 28. Droplet axial velocity for different droplet sizes at three radial positions ($r$) at an axial position ($x$) of 50 mm [66]..........................

Fig. 29. Fuel droplets spatial distribution for SN of 1.4 and $\phi$ of 0.81 at time of (a) 40 ms, (b) 50 ms, (c) 60 ms ..................................................

Fig. 30. Fuel droplets spatial distribution on SN of 1.0 in 60 ms (a) $\phi$ of 0.81, (b) $\phi$ of 0.97..................................................

Fig. 31. Fuel droplets spatial distribution on SN of 1.4 in 60 ms (a) $\phi$ of 0.81, (b) $\phi$ of 0.97..................................................

Fig. 32. Histogram of droplet size distribution on $\phi$ of 0.81 and 0.97 when (a) swirl number is 1.0, (b) swirl number is 1.4 ..................................
Fig. 33. Time-averaged droplet axial velocity profiles of $SN$ of 1.0 at three axial locations for (a) $\phi$ of 0.81, (b) $\phi$ of 0.97 .................................................................80

Fig. 34. Time-averaged droplet radial velocity profiles of $SN$ of 1.0 at three axial locations on (a) $\phi$ of 0.81, (b) $\phi$ of 0.97 .................................................................81

Fig. 35. Time-averaged droplet axial velocity profiles of $\phi$ of 0.81 at three axial locations on (a) $SN$ of 1.0, (b) $SN$ of 1.4 .........................................................83

Fig. 36. Time-averaged droplet radial velocity profiles of $\phi$ of 0.81 at three axial locations on (a) $SN$ of 1.0, (b) $SN$ of 1.4 .........................................................84

Fig. 37. Jones et al. experimental setup [43] ...........................................................................86

Fig. 38. Computational mesh used for combustion validation .................................................87

Fig. 39. Comparison of simulation results and experimental data [43] of the time-averaged temperature at an axial location of $z/R=2.0$ ........................................................................88

Fig. 40. Comparison of simulation results and experimental data [43] of time-averaged $CO_2$ concentration at an axial location of $z/R=2.0$ .................................89

Fig. 41. Simulation pressure probes within the simulation domain ..........................................91

Fig. 42. Frequency spectrum of pressure oscillation at locations of $P1$, $P2$, $P3$, and $P4$ for $SN$ of 1.0 at (a)–(d) $\phi$ of 0.81, (e)–(h) $\phi$ of 0.97 .................................93

Fig. 43. Frequency spectrum of pressure oscillation at locations of $P1$, $P2$, $P3$, and $P4$ for $\phi$ of 0.81 at (a)-(d) $SN$ of 1.0, (e)-(h) $SN$ of 1.4 ...............................95

Fig. 44. Schematic diagram of three axial locations within the combustion chamber .........................96

Fig. 45. The mean temperature profile at 2000 K for (a) $SN$ of 1.0, (b) $SN$ of 1.4 ....97

Fig. 46. Mean temperature profile for $SN$ of 1.0 and $\phi$ of 0.81 at the crossing plane of $z/H=4$ .........................................................................................98

Fig. 47. Time-averaged normalized turbulent kinetic energy (TKE) for $SN$ of 1.0 at three axial locations at (a) $\phi$ of 0.81, (b) $\phi$ of 0.97 .................................................100

Fig. 48. Time-averaged normalized turbulent kinetic energy (TKE) for $\phi$ of 0.81 at three axial locations when (a) $SN$ is 1.0, (b) $SN$ is 1.4 .................................101
Fig. 49. Center recirculation zones for SN of 1.0 (a) when $\phi$ is 0.81, (b) when $\phi$ is 0.97

Fig. 50. Center recirculation zones for $\phi$ of 0.81 (a) when SN is 1.0, (b) when SN is 1.4

Fig. 51. Mean temperature profile for SN of 1.0 when (a) $\phi$ is 0.81, (b) $\phi$ is 0.97

Fig. 52. Mean temperature profile for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 53. Iso-surface of mean temperature of 2000 K at 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 54. Snapshot of iso-surface of mean temperature of 2000 K with Q-criteria value of 40000 s$^{-2}$ at 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 55. Distribution of emulsified canola oil on a mass basis after 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 56. Distribution of CO after 60 ms for SN of 1.0 when (a) $\phi$ is 0.81, (b) $\phi$ is 0.97

Fig. 57. Distribution of CO after 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 58. Distribution of $O_2$ and center recirculation zone after 60 ms for $\phi$ of 0.81 and SN of 1.4

Fig. 59. Distribution of NO$_x$ after 60 ms for SN of 1.0 when (a) $\phi$ is 0.81, (b) $\phi$ is 0.97

Fig. 60. Distribution of NO$_x$ after 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 61. Distribution of thermal NO$_x$ after 60 ms coupled with iso-surface of temperature at 2400 K for (a) SN of 1.0 and $\phi$ of 0.81, (b) SN of 1.0 and $\phi$ of 0.97, (c) SN of 1.4 and $\phi$ of 0.81, (d) SN of 1.4 and $\phi$ of 0.97

Fig. 62. Comparison between simulation work and experimental work [4] in terms of CO emissions

Fig. 63. Comparison between simulation work and experimental work [4] in terms of the concentration of NO$_x$ in the combustion chamber
Fig. A-1. Simulation results of molar fraction of $CO_2$ at the axial location ($z/H$) of 0.38, 1, and 4 at 60 ms (a) SN of 1.0 and $\phi$ of 0.81, (b) SN of 1.0 and $\phi$ of 0.97, (c) SN of 1.4 and $\phi$ of 0.81, and (d) SN of 1.4 and $\phi$ of 0.97 ....... 137
## LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 1. Flow Conditions for SN of 1.0 and 1.4</td>
<td>18</td>
</tr>
<tr>
<td>Table 2. Thermophysical properties of pure canola oil at 25 °C (% mass)</td>
<td>20</td>
</tr>
<tr>
<td>Table 3. Thermophysical properties of emulsified canola oil at 25 °C (% mass)</td>
<td>21</td>
</tr>
<tr>
<td>Table 4. Two steps chemical reaction mechanism of emulsified canola oil</td>
<td>36</td>
</tr>
<tr>
<td>Table 5. Description of Simulation Models</td>
<td>37</td>
</tr>
<tr>
<td>Table 6. Comparison of simulation results with analytical solution for the chamber without swirler</td>
<td>46</td>
</tr>
<tr>
<td>Table 7. Experimental data of Adiabatic Flame and Furnace Temperature [4]</td>
<td>121</td>
</tr>
<tr>
<td>Table A-1. Stoichiometric coefficients for complete combustion of emulsified canola oil</td>
<td>135</td>
</tr>
</tbody>
</table>
1. INTRODUCTION

In the past several decades, the tremendous demand for energy has led to high costs of traditional or conventional fossil fuels, such as oil, coal, and natural gas. This has resulted in greater pollutant emissions generated during combustion process of fuels. In recent years, researchers have shown renewed interest in development and use of alternative fuels such as biodiesel and other fuels derived from biomass. According to the latest data of the International Energy Agency (IEA) as shown in Fig. 1 [1], the usage of biofuels and waste within the Organization for Economic Cooperation and Development countries (OECD) has increased from 86.02 Mtoe (Million Tonners of Oil Equivalent) in 1973 to 279.469 Mtoe in 2013, and the supply of nuclear energy has increased from 48.62 Mtoe in 1973 to 511.481 Mtoe in 2013.

![Fig. 1. 1973 and 2013 fuel shares of OECD [1]](image-url)

However, the growing concern about the safety of nuclear energy especially after the 2011 Tohoku Earthquake has impeded the development of nuclear energy around the
world. Consequently, biofuels and organic-based waste are now being considered to be among the best alternative fuels in the near future.

1.1. Motivation

Over the last 20 years, a significant emphasis has been given to the production and use of biofuels including straight vegetable oils as alternative options to fossil fuels. One of the advantages of biofuels is that they can be considered carbon-neutral and less polluting than conventional fuels since many of them possess very small amount of sulfur or nitrogen, which are essential ingredients in the formation of $SO_x$ and $NO_x$. Recently, straight vegetable oils (SVO) have also been considered as liquid biofuels given their negligible content of sulfur and nitrogen, which is one of the primary reasons why they have great potential to replace conventional fuels [2]. However, the high viscosity of SVO has hindered their ability and applicability in combustion systems. Furthermore, most SVO are characterized by having complex molecular structures, which routinely lead to incomplete combustion under normal conditions. In order to reduce the effect of high viscosity of SVO, methanol has been used as additive to reduce its apparent viscosity. Blends of SVO and methanol have been burnt using a twin-fluid atomizer in a swirl-promoted combustion environment with moderate success [3, 4]. Although biofuel combustion has been studied and discussed by many researchers, most studies have only focused on the combustion of biofuels in engines, and few researchers have considered the combustion of SVO in boilers directly. Furthermore, little is known about the effects of SVO viscosity, SVO blend composition and swirl number on
combustion. Therefore, it is very important to understand the effects of SVO viscosity and swirl number on combustion emissions.

1.2. Objective

The purpose of this study is to understand the effects of induced vorticity (swirl number) and SVO fuel atomization on combustion by following a comprehensive numerical approach, which takes into account fluid dynamics, fluid atomization and combustion kinetics, flame stability simultaneously. To achieve this objective, a robust and stable numerical algorithm should be selected and validated. Firstly, the effects of swirl number on vortex generation should be investigated through numerical simulations when using swirler with flat vanes. The induced vorticity should be studied carefully in order to illustrate the relationship between fluid velocity and vortex structure in a combustion chamber. Furthermore, the effects of vortex generation on fuel droplet distribution should be elucidated. Once induced vorticity has been simulated and analyzed, combustion simulation of well-defined biofuels should be undertaken. This includes a comprehensive numerical effort where the chemical reactions of biofuel species coupled with pressure, velocity, and temperature fields are considered simultaneously. The simulation results should include vorticity strength, the fluid temperature profile inside the combustion chamber as well as the concentrations of combustion products for different swirl numbers and equivalence ratios.

In summary, the study should lead to a better understanding of the effects of vortex generation and droplet distribution on combustion emissions for SVO biofuels. It
is also hoped that this research would be used to identify better combustion strategies for biofuels with high viscosity.

1.3. Overview

This dissertation is divided into six chapters. Chapter II presents a review of literature in the area of numerical simulation of turbulence flow and fuel combustion. Chapter III describes the methodology used in the study, including the computational domain, boundary conditions, properties of fuel, and fundamental review of numerical schemes used in the study. Chapter IV includes the validation of the numerical approach used for turbulent cold flow and droplet spray, as well as numerical results for a swirl-promoted turbulent cold flow and the fuel droplets spray. Chapter V includes the validation of the numerical approach used for fuel droplets combustion, as well as the numerical results for turbulent flow with combustion. The chapter also includes numerical results such as temperature profile, and CO and NOx emissions. In Chapter VI and Chapter VII, the conclusions and recommendations for future studies is presented, respectively.
2. LITERATURE REVIEW

2.1. Literature review of experimental studies on straight vegetable oil (SVO) combustion

The experimental combustion studies of vegetable oil have been investigated since the 1970s due to the oil crisis and other factors [5]. One of the first research studies that considered the use of vegetable oils as alternative fuels was for diesel engines. It was done in the early 1980’s by Strayer et al. [6]. This work suggested that up to 13% (vol.) anhydrous alcohol can be blended with canola oil in order to reduce and obtain lower viscosity and better flow properties from the oil, respectively. The tests conducted on a small diesel engine showed that 90% canola oil – 10% ethanol blend led to reduced power and higher fuel consumption. Krumdieck et al. [7] sprayed and combusted a blend of 90% (vol.) pyrolysis oil from poplar wood and 10% (vol.) ethanol in a laboratory scale boiler. They observed fewer visible burning droplets when 10% ethanol was added to the oil as compared to the case of pure oil combustion. This indicated that ethanol addition improved the performance of the combustor and flame stability. Nettles-Anderson et al. [8] studied the emissions of straight vegetable oil in a direct injection IC engine, using sunflower, canola, camellia, and soybean oils, and concluded that higher levels of polyunsaturated fats (e.g. C18:1, C18:2, and C18:3) produce higher levels of NOx.

Emulsified biofuels have not only been thought as alternate fuels for boilers and diesel engines but also for turbines. Asea Brown Boveri (ABB) was the first company to
test emulsified fuels in turbines in 1985. The paper published by Chellini [9] illustrates the experimental work done on a GEC Alstom, MS9001B 100 MW gas turbine with water-in-diesel emulsion as the fuel. The authors monitored the performance of the turbine for different output levels including 20, 40, 60, 80 and 92.4 MW. The results of the experiments showed that \( \text{NO}_x \) emissions reduced constantly with increasing amount of water in the emulsion at all the power output levels. The combustion efficiency was same as in case of diesel fuel combustion. The authors suggest that fuel emulsification has proved itself as a new method for reducing emissions as compared to water/steam injection. Chiariello et al. [10] studied the gaseous and particulate emissions from a micro gas turbine that used straight vegetable oil - kerosene blends as energy fuels. The fuel droplets were injected into chamber using a plain-jet air-blast atomizer. Air was introduced into the chamber via a helical swirler. The authors reported that \( \text{NO}_x \) and \( \text{CO} \) emissions were practically insensitive to the composition of the fuel within the experimental uncertainty of the instrumentation. Al-Shudeifat et al. [11] reported on the combustion of waste trap grease oil in gas turbine generator. In their research, the fuel was injected into chamber using an atomizer, and the results showed that the trap grease led to less \( \text{NO}_x \) emission for high generator loads compared to diesel, whereas \( \text{CO} \) and \( \text{CO}_2 \) emissions were slightly higher than those of diesel as the load decreased.

Moreover, numerous experimental works have been done by using optical measurement equipment to understand combustion phenomena better. Those studies looked at the structure of turbulence and flame, and the interaction between turbulence and combustion. For example, Kaminiski et al. [12, 13] used a high speed PLIF
technique to record the evolution of turbulence in real time. Hult et al. [14] employed a
time resolved PLIF and instantaneous PIV measurements to study the dynamics of
turbulent/chemistry interaction in real time. Chen et al. [15] showed experimental results
with Rayleigh scattering and laser-induced fluorescence simultaneously using a high
speed premixed hydrocarbon flames. They found a linear relationship between the
measured CH signal intensity and the calculated CH concentration within a maximum
30% uncertainty range.

However, there are few published papers which report on the study of swirl-
promoted combustion of SVO from the experimental point of view. Cavarzere et al. [16]
formulated and tested a variety of straight vegetable oils including rapeseed, sunflower,
soybean, rapeseed-diesel blends, sunflower-diesel blends, and soybean–diesel blends in
a gas turbine. The study reveals that rapeseed SVO is the most preferable since it cannot
be used in the food industry, while soybean SVO is the least preferable. Bhimani et al.
[4] studied the combustion characteristics of emulsified canola oil in a swirl-promoted
environment. They found that increased vorticity at higher swirl angle led to better
mixing and lower emissions, including lower NO\textsubscript{x}, CO and unburned HC emissions than
pure canola oil. Savant et al. [3] conducted combustion experiments using corn oil-based
blends in a swirl-promoted combustion chamber. They concluded that NO\textsubscript{x} emissions
increases when used-corn oil is burnt at high swirl number. Furthermore, they found out
that all corn oil-based blends showed a generally decreasing CO trend at different swirl
numbers under very lean conditions.
2.2. Literature review of numerical studies on swirl-promoted turbulent flow in combustion chambers

The study of swirl-promoted turbulent combustion requires proper simulation of the turbulent flow field within the confines of a typical combustion chamber. There are three numerical approaches to simulate turbulent flow especially when a swirler is used, including Reynolds-averaged Navier-Stokes equations (RANS), Direct Numerical Simulation (DNS), and Large Eddy Simulation (LES) [17].

In RANS, a two-equation model is used to solve the continuity, momentum and energy equations using the $k$-$\varepsilon$ approach, which takes into account the turbulent kinetic energy $k$ and dissipation rate of kinetic energy $\varepsilon$ [18, 19]. To validate the applicability of $k$-$\varepsilon$ approach in the simulation of flows in combustion chambers, studies that used similar configurations should be considered. Amano [20] used the $k$-$\varepsilon$ model to simulate the turbulent flow downstream of an abrupt pipe expansion, and compared it with experimental measurements. The simulation results showed generally encouraging agreement with the experimental measurements. Zin et al. [21] investigated the influence of length-to-diameter of a pipe expansion on the fluid flow characteristic inside a vortex tube. They concluded that the $k$-$\varepsilon$ model needs to be improved due to the disagreement of the measured and simulated pressure values along the tube. Therefore, $k$-$\varepsilon$ approach should be complemented with another numerical scheme. Furthermore, the $k$-$\varepsilon$ model cannot precisely take into account flow instabilities.

DNS is probably the most advanced algorithm for flow simulations; however, it is computationally expensive. It involves solving the Navier-Stokes equations
numerically without the use of any turbulent model [22]. Luo et al. [23] analyzed the turbulent flow field of three-dimensional n-heptane premixed flame in a swirl combustor using DNS. The gas combustion kinetics were represented using an adaptive one-step irreversible reaction scheme. The variable-density, low-Mach number Navier–Stokes equations were solved using a fully conservative and kinetic energy using a finite difference scheme in cylindrical coordinates with Swirl Number (SN) of 1.0. Tanaka et al. [24] applied DNS to study the gas dynamic behaviors, characteristics of flame structures and heat transfer properties of hydrogen–air turbulent premixed flame in swirling flows. To investigate swirling effects on the flow and combustion fields, DNS of both cold and reactive flow was carried out for two SNs of 0.6 and 1.2. However, DNS requires a huge amount of computational resources especially for complicated geometry [25]. Therefore, a compromise between resolution and computational time is needed to make the best use of computational resources while coming out with sufficient information of the simulated flow field.

LES is another approach used to simulate large eddies found in turbulent flow. In LES, small eddies are filtered in the spectral space and taken into account through modeling. Huang et al. [26] published a review paper and concluded that although the RANS approach currently remains the main workhorse for combustor design analysis, LES is considered to be the next-generation analytical design tool for gas turbine combustors. Therefore, many researchers have reported their research on the flow characteristics of combustion chamber by LES, especially in swirl-promoted flow. Wang et al. [27] presented a comprehensive study on confined swirling flows in an operational
gas turbine injector. The calculated mean velocities and turbulence properties showed good agreement with experimental data. Wang et al. [28] employed LES to study a confined isothermal turbulent swirling flow in a model dump combustor, and reported that the large eddy simulation technique was able to depict the details of the turbulent structure. Kim et al. [29] investigated a swirl-stabilized flow in a gas turbine combustor using large eddy simulation. They observed a complex vortex shedding pattern with significant azimuthal structures in the chamber, and concluded that LES could be useful in the study of the characteristics of swirling flow. Grinstein et al. [30] employed a large eddy simulation to study the non-reacting and reacting-flow in a lean premixed gas turbine combustor with a swirler. The results showed a reasonable agreement with the experimental data. Lu et al. [31] conducted a large eddy simulation analysis with emphasis placed on flow development and its interaction with acoustic waves.

2.3. Literature review of large eddy simulation on swirl-promoted turbulent combustion

LES technique is not only applied to study the fluid dynamics characteristics of turbulent flow, but also used to study the characteristics of turbulent combustion on gaseous and liquid fuels. Huang et al. [32] conducted a numerical study in a gas-turbine combustor. The analysis took into account the finite rate chemical reactions and variable thermophysical properties of methane gas, and employed the LES technique to investigate the combustion dynamics. Wu et al. [33] studied the combustion of methane and propane in meso-scale vortex chambers, and explored flame stabilization mechanism
and chemical efficiency of non-premixed combustion by the LES technique. Stone and Menon [34, 35] investigated the effects of swirl and equivalence ratio on flame dynamics on a swirl-stabilized, lean-premixed gas turbine combustor using LES. The use of a premixed flamelet model, unsteady vortex flame and acoustic flame interactions were captured numerically. The response of the combustion process to explicit swirl modulation was investigated by using the acoustic approach [36].

Furthermore, many researchers have reported that large eddy simulation is capable to couple with the other models when liquid fuels are used. Liu et al. [37] predicted the droplet size distribution in sprays when using air-blast atomizers by using stochastic break-up models. They reported that the nonlinear relationship between the mean droplet diameter and droplet size distribution of the air-blast atomization process can be predicted exactly by this model. Xu et al. [38] employed LES to simulate the fluid flow while the Lagrangian method was used to simulate the motion of particles. They concluded that the LES model can definitely improve the accuracy of the prediction on the mean diameter of droplets. Apte et al. [39] applied the stochastic modeling of secondary breakup to simulate atomizing spray in large eddy simulations. They concluded that LES is computationally efficient and captures the complex fragmentary processes of liquid atomization and fuel droplet motion by coupling with stochastic break-up models.

Moreover, large eddy simulation has been used to capture the droplet distribution of liquid fuels in a swirl-promoted environment. Sankaran and Menon [40, 41] presented the velocity profiles and droplet distribution for General Electric DACRS combustor
using LES. They observed that increasing swirl number accelerated the breakdown of turbulent structures due to the stretch effect, which led to enhanced fuel–air mixing.

Jones et al. [42, 43] studied the droplet laden turbulent mixing layer and the influence of shear force on transportation of dispersed phase using large eddy simulation. They also used a stochastic model to simulate the effects of continuous phase on particle motion with LES. Salewski et al. [44] studied the combustion of sprays in a swirl-stabilized gas turbine burner using LES. They found that the low density of droplets in the recirculation zone in the case of reacting flow allowed the fluid to push the recirculation zone downstream, and the low density of droplets resulted in a reduction of the spray dispersion.

Due to the capability of LES technique on coupling with other models to predict the behavior and distribution of droplets, many studies have used large eddy simulation to simulate the turbulent combustion on liquid fuels. Jones et al. [45] investigated swirling kerosene spray flames with LES, and applied the probability density function (PDF) methods to overcome closure problems in turbulent reacting flow simulation. Zhou et al. [46] studied the spray and combustion characteristics in diesel engine using improved LES technique. They concluded that LES technique was capable of predicting the velocity distribution, droplet and vapor phase distribution and combustion characteristics. Sankaran et al. [47] investigated the unsteady interactions between diesel spray dispersion, vaporization, fuel-air mixing and heat release in a realistic combustor using LES. They concluded that large-scale organized structures were subjected to
complex stretch effects due to a combination of streamwise and azimuthal vortex motions.

Compared with conventional gaseous and liquid fuels, straight vegetable oils have complex chemical compositions. In order to improve the simulation accuracy, thermophysical properties need to be studied carefully. Many researchers have used the Kay’s mixing rule or group contribution methods to predict the physical properties of fuel blends for combustion modeling. Yuan et al. [48] applied the group contribution method to predict key physical properties of biodiesel, including critical properties, vapor pressure, latent heat of vaporization, density, surface tension, and liquid viscosity. They compared the predicted results with published data and obtained a good agreement. Ceriani et al. [49] proposed a group contribution method to estimate the heat capacities and heats of vaporization of biofuels. Benjumea et al. [50] studied basic properties of the blends of palm oil biodiesel and diesel. They used Kay’s mixing rule to predict density, heating value, cloud point, and applied Arrhenius mixing rule to predict viscosity of blends. They reported that the mixing rules they used were suitable to calculate properties of fuel blends.

The formulating chemical reaction mechanisms for combustion is very complicated. Therefore, numerical models of combustors which consider two- or three-dimensional geometry cannot currently include all the kinetic mechanisms [51]. Many researchers use a simplified reaction mechanism in computational fluid dynamics (CFD) codes. Yuan et al. [52] applied a one-step irreversible Arrhenius kinetic model to investigate the spray, ignition and combustion of biodiesel in a diesel engine using a
numerical approach. They found the simulation results by the kinetic model are reasonably accurate. Westbrook and Dryer [53] studied the chemical kinetic modeling of n-butane, and proposed a two-steps reaction mechanism. Yasin et al. [54] used a two-steps heptane reaction mechanism to study the biodiesel spray combustion. Irannejad et al. [55] employed a global chemical kinetic mechanisms when they simulated the high speed evaporation and combustion of n-heptane sprays using LES. They claimed that the simulation results showed a good agreement with the experimental results in term of temperature profile.

As summarized above the combustion of straight vegetable oil has been studied by experimental approach mainly. The combustion of conventional fuel or biodiesel, which has a relatively low viscosity, has been discussed by numerical studies. Moreover, the Large Eddy Simulation technique has been proven to be able to simulate the interactions of turbulent flow and droplet spray as well as the turbulent combustion using a simplified chemical reaction mechanism. However, few researchers have studied swirl-promoted combustion of straight vegetable oil following a numerical approach, capable of providing adequate results which could be used to analyze the details of the turbulent flow field, and the profiles of temperature and emissions during the combustion process. Therefore, the objective of current research is to study the effects of swirl number and equivalence ratio on emulsified canola oil combustion by relying on the Large Eddy Simulation technique by considering applicable modeling schemes.
3. METHODOLOGY OF THE STUDY

A comprehensive and reliable numerical approach has been selected, and used to help understand the effects of combusting SVO with different swirler conditions when mixed with air at varying flowrates. As discussed in chapter II, LES is the most suitable numerical approach to simulate the swirl-promoted turbulent flow in a combustion chamber, as the one described below. The LES technique chosen for simulation was coupled with kinetic chemical reaction models to simulate the overall combustion process of biofuels taking into account the thermodynamics properties of multicomponent mixtures, which include straight vegetable oil. In this chapter, the geometrical structure of a combustion chamber is presented in terms of a computational domain including its boundary conditions. The thermophysical properties of emulsified canola oil are also introduced. Specific details about the numerical approach are explained which fully accounts for turbulent flow, the droplets spray, and the applicable chemical reactions.

3.1. Description of combustion chamber geometry with boundary conditions

The geometry of a relatively common combustion chamber is based on a small scale combustion chamber located in the Coal and Biomass Energy Laboratory (CBEL) at Texas A&M University. The combustion chamber is rated at 30 kW and was initially built by Dr. Annamalai’s former students, and was modified by Dr. Alvarado’s former student Shreyas Bhimani who used it for the combustion of vegetable oil fuels [56]. The
A schematic representation of the combustion chamber is shown in Fig. 2, which was generated using CAD software (SolidWorks 2014, SolidWorks Corporation). It consists of a short guiding 50.8 mm pipe, a long 101.6 mm guiding pipe, a metal cone in the long guiding pipe, a swirl generator, a twin fluid atomizer, and a 152.4 mm combustion chamber. The total height of combustion chamber located in the CBEL is 2.4 meters. Figure 2 shows a section of the whole combustion chamber, or one quarter of the whole system.

![Fig. 2. Schematic of combustion chamber](image)

The twin fluid atomizer is an air-blasted type, which has a 0.5 mm inner diameter orifice at its bottom-center. The atomizer is placed such that the tip of nozzle is positioned approximately 5 mm above the bottom edge of swirl generator. The swirl
generator consists of eight flat vanes with an inclination angle of 51° or 60° as shown in Fig. 3. The two different vane inclination angles were used to generate different levels of vorticity within the combustion chamber. It has a hub at the center of swirler, which has a diameter of 59.4 mm. The outer diameter of swirler is 100.4 mm, and the height of swirler is 25.4 mm. The vane inclination angle in combination with the diameters of the swirler are represented by the swirl number, and defined as the ratio of angular momentum to the axial momentum of air flow inside the combustion chamber [57].

Fig. 3. (a) Front view of swirler, (b) Assembly drawing of swirler in guiding pipe

The swirl number of an annular swirler with constant vane angle is given by Equation 1:

\[
SN = \left( \frac{2}{3} \right) \left[ \frac{1 - \left( \frac{R_h}{R} \right)^3}{1 - \left( \frac{R_h}{R} \right)^2} \right] (Tan(\alpha))
\]  

(1)
Where, \( SN \) is swirl number; \( R_h \) is the outer radius of the hub; \( R \) is the inner radius of the hub; \( \alpha \) is the vane angle. In this study, two swirl numbers (\( SN \)) of 1.0 and 1.4 are considered during the simulation process.

**Table 1. Flow Conditions for \( SN \) of 1.0 and 1.4**

<table>
<thead>
<tr>
<th>Cases</th>
<th>Equivalent Ratio</th>
<th>( \dot{V} ) (l/min)</th>
<th>( D_l ) (mm)</th>
<th>( D_c ) (mm)</th>
<th>( Re_l )</th>
<th>( Re_c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( SN = 1.0 )</td>
<td>0.81</td>
<td>403</td>
<td>101.6</td>
<td>152.4</td>
<td>11,484</td>
<td>3,840</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.97</td>
<td>336</td>
<td>101.6</td>
<td>152.4</td>
<td>9,576</td>
<td>3,202</td>
</tr>
<tr>
<td>( SN = 1.4 )</td>
<td>0.81</td>
<td>403</td>
<td>101.6</td>
<td>152.4</td>
<td>11,484</td>
<td>3,845</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.97</td>
<td>336</td>
<td>101.6</td>
<td>152.4</td>
<td>9,576</td>
<td>3,202</td>
</tr>
</tbody>
</table>

Flow conditions for two \( SNs \) are shown in Table 1. Reynolds number is based on a volume flow rate and the corresponding local diameter \( D_l \) and \( D_c \), where subscripts \( l \) and \( c \) represent the short gilding pipe and combustion chamber, respectively. The volume flow rate of air \( \dot{V} \) is an adjustable parameter used to adjust the equivalence ratio (\( \phi \)), during the combustion simulations, which was calculated by the following equation.

\[
\phi = \left[ \frac{\left( \frac{\text{Fuel}}{\text{air}} \right)_{\text{Provided}}}{\left( \frac{\text{Fuel}}{\text{air}} \right)_{\text{Stoichiometric}}} \right]
\]  

In this study, the flow rate of primary air, which is used to generate the fuel droplets in atomizer, is 10 l/min. The flow rates of secondary air, which flows through
the guiding pipe, is 393 l/min and 326 l/min, respectively. Therefore, the two corresponding equivalent ratios are 0.81 and 0.97.

3.2. Description of fuel thermophysical properties

A 85-12.5 emulsion [12.5% methanol – in – 85% canola oil (w/w) emulsion with 2.5% surfactant] has been studied by Bhimani et al. [4] experimentally. The results of that study show that swirl number has a significant effect on emission levels. In order to understand the effect of swirl number on the overall combustion process, the chemical reactions should be simplified when conducting numerical simulations. During the combustion simulations, the effect of surfactant on combustion was neglected. Therefore, the mass ratio of emulsified biofuel blend was set to 87.2 – 12.8 emulsion [12.8% methanol – in – 87.2% canola oil (w/w) emulsion].

Pure canola oil of Crisco brand [58] and 99.9% concentrated methanol from EMD biosciences [59] was used for making the emulsions. The properties of pure canola oil are obtained by Bhimani’s experimental work [4] and estimated by the group contribution method [60]. The thermophysical properties of pure canola oil at 25 °C are shown in Table 2.

The thermophysical properties of emulsified canola oil have been obtained from Bhimani’s experimental work [4] and estimated by using the Kay’s mixing rule [61]. The thermophysical properties of emulsified canola oil at 25 °C are shown in Table 3.
<table>
<thead>
<tr>
<th>Property</th>
<th>Pure Canola Oil</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fatty acid of pure canola oil (%)</td>
<td></td>
</tr>
<tr>
<td>Palmitic acid (C 16:0)</td>
<td>4.93</td>
</tr>
<tr>
<td>Palmitoleic acid (C 16:1)</td>
<td>0.33</td>
</tr>
<tr>
<td>Stearic acid (C 18:0)</td>
<td>2.21</td>
</tr>
<tr>
<td>Oleic acid (C 18:1)</td>
<td>63.96</td>
</tr>
<tr>
<td>Linoleic acid (C 18:2)</td>
<td>19.17</td>
</tr>
<tr>
<td>Linolenic acid (C 18:3)</td>
<td>9.40</td>
</tr>
<tr>
<td>Dynamic viscosity* (cP)</td>
<td>55</td>
</tr>
<tr>
<td>Density* (kg/m³)</td>
<td>962</td>
</tr>
<tr>
<td>Surface tension* (N/m)</td>
<td>0.0278</td>
</tr>
<tr>
<td>HHV* (kJ/kg)</td>
<td>40,173</td>
</tr>
<tr>
<td>Specific heat (kJ/(kg-K))</td>
<td>1.94</td>
</tr>
<tr>
<td>Latent heat (kJ/mol)</td>
<td>59.28</td>
</tr>
<tr>
<td>Flash point (K)</td>
<td>452</td>
</tr>
<tr>
<td>Boiling point (K)</td>
<td>621</td>
</tr>
<tr>
<td>Thermal conductivity (W/(m-K))</td>
<td>0.20</td>
</tr>
</tbody>
</table>

* Bhimani, S. [4]
Table 3. Thermophysical properties of emulsified canola oil at 25 °C (% mass)

<table>
<thead>
<tr>
<th>Property</th>
<th>Pure Canola Oil</th>
</tr>
</thead>
<tbody>
<tr>
<td>SVO composition</td>
<td></td>
</tr>
<tr>
<td>Pure canola oil (%)</td>
<td>87.2</td>
</tr>
<tr>
<td>Methanol (%)</td>
<td>12.8</td>
</tr>
<tr>
<td>Dynamic viscosity* (cP)</td>
<td>41.2</td>
</tr>
<tr>
<td>Density* (kg/m³)</td>
<td>881.5</td>
</tr>
<tr>
<td>Surface tension* (N/m)</td>
<td>0.026</td>
</tr>
<tr>
<td>HHV* (kJ/kg)</td>
<td>37,506</td>
</tr>
<tr>
<td>Specific heat (kJ/(kg-K))</td>
<td>2.01</td>
</tr>
<tr>
<td>Latent heat (kJ/mol)</td>
<td>56.27</td>
</tr>
<tr>
<td>Flash point (K)</td>
<td>431</td>
</tr>
<tr>
<td>Boiling point (K)</td>
<td>585</td>
</tr>
<tr>
<td>Thermal conductivity (W/(m-K))</td>
<td>0.20</td>
</tr>
</tbody>
</table>

* Bhimani, S. [4]

The chemical formula of emulsified canola oil has been obtained from Bhimani’s experimental work [4]. Miscibility could have an effect on surface tension and droplet size; however, the mixing rule was used, which assumes complete miscibility or homogenous mixture without the use of surfactants. In fact, methanol is indeed miscible in canola oil up to 8% by volume as described by Bhimani [4]. Therefore, after
neglecting the components such as nitrogen and sulfur, the chemical formula of emulsified canola oil is $C_{6.65}H_{11.329}O_{0.8513}$, and the empirical formula is $CH_{1.704}O_{0.128}$.

3.3. Description of grid generation and numerical algorithm

The computation domain is based on the structure shown in Fig. 2. The computational mesh was generated by ICEM CFD (ANSYS, Inc.). The region of the metal cone and swirler was meshed using a tetrahedral grid, and the rest region of domain was meshed using a hexahedral grid, as shown in Fig. 4. The grid points are 4.8 million, and the resolution is 0.5 mm in the radial direction and 4 mm in the axial direction. The validation of mesh will be discussed in details in the next chapter.

![Fig. 4. Computational mesh of (a) Cone and Swirler, (b) Entire domain](image)

In this study, commercial program ANSYS Fluent 15 was employed to study the turbulent flow, fuel droplet spray and combustion numerically. The pressure-based segregated solver was selected, which is the SIMPLE algorithm. RANS was used as
turbulence viscous model firstly, which was used to obtain an initial solution of velocity flow field. Then the LES technique was employed to obtain detailed vortex structures of air-fuel turbulent flow. The numerical algorithms of RANS, LES and pressure-based solver are discussed below.

In the computational domain, the theoretical formulation is based on the full conservation equations of mass momentum, energy and species concentration in Cartesian coordinates as follows [62]:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0
\]  
(3)

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}
\]  
(4)

\[
\frac{\partial \rho E}{\partial t} + \frac{\partial [(\rho E + p) u_j]}{\partial x_j} = -\frac{\partial q_i}{\partial x_i} + \frac{\partial (u_i \tau_{ij})}{\partial x_j}
\]  
(5)

\[
\frac{\partial \rho Y_k}{\partial t} + \nabla \cdot (\rho \vec{v} Y_k) = -\nabla \cdot \vec{J}_k + R_k
\]  
(6)

Where \(i, j, \) and \(k\) are the spatial coordinate index, the dummy index to spatial coordinate, and the species index, respectively. \(Y_k\) and \(\vec{v}\) are mass fraction and diffusion velocities of species \(k\), respectively. \(R_k\) is the net rate of production of species \(k\) by chemical reaction. \(J_k\) is the diffusion flux of species \(k\). The viscous stress and heat flux vector are \(\tau_{ij}\) and \(q_j\), which are shown below.

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right)
\]  
(7)
\[ q_j = -\lambda \frac{\partial T}{\partial x_j} + \rho \sum_{k=1}^{N} h_k Y_k U_{k,j} \] (8)

Where \( \mu \) and \( \lambda \) are the coefficients of viscosity and heat conductivity, respectively.

The specific total \( E \) is given by

\[ E = e + \frac{u_j u_j}{2} \] (9)

\( e \) is the specific internal energy, which is obtained as:

\[ e = h - \frac{p}{\rho} \] (10)

\( h \) is the specific enthalpy of mixture, which is shown as following:

\[ h = \sum_{k=1}^{N} Y_k h_k = \sum_{k=1}^{N} Y_k \left( \Delta h_{j,k} + \int_{T_{ref}}^{T} C_{p,k}(T')dT' \right) \] (11)

Where \( C_{p,k} \) is the species specific heat at constant pressure.

In turbulent flow, the diffusion flux of species \( k \) is computed in the following form:

\[ \bar{J}_k = \left( \rho D_{T.k} + \frac{\mu_t}{Sc_t} \right) \nabla Y_k - D_{T.k} \frac{\nabla T}{T} \] (12)

Where \( Sc_t \) is the turbulent Schmidt number \( \left( \frac{\mu_t}{\rho D_t} \right) \) where \( \mu_t \) is the turbulent viscosity and \( D_T \) is the turbulent diffusivity). The default \( Sc_t \) is 0.7.

In the simulation of turbulent flow, the continuity, momentum and energy equations are solved using the \( k-\varepsilon \) model of the RANS approach, which takes into account the turbulent kinetic energy \( k \) and dissipation rate of kinetic energy \( \varepsilon \). In ANSYS
Fluent, the standard k-ε model is a model based on transport equation for the turbulence kinetic energy \((k)\) and its dissipation rate \((\varepsilon)\) as shown below [62].

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{13}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{14}
\]

Where, \(G_k\) represents the generation of turbulence kinetic energy due to the mean velocity gradients; \(G_b\) is the generation of turbulence kinetic energy due to buoyancy; \(Y_M\) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; \(C_{1\varepsilon}, C_{2\varepsilon}, \) and \(C_{3\varepsilon}\) are constant; \(\sigma_k\) and \(\sigma_\varepsilon\) are the turbulent Prandtl numbers for \(k\) and \(\varepsilon\), respectively; \(S_k\) and \(S_\varepsilon\) are user-defined source terms. In Fluent, the default values of constant \(C_{1\varepsilon}, C_{2\varepsilon}, \sigma_k\) and \(\sigma_\varepsilon\) are 1.44, 1.92, 1.0 and 1.3, respectively [62].

The RANS approach was applied firstly to initialize the flow field of the computational domain and to speed up the convergence of simulation. Then LES was then applied to obtain the detailed vortex structure of turbulent flow in the domain. In ANSYS Fluent, large eddies, which are larger than the subgrid-scale, are resolved directly, while small eddies are modeled numerically [62]. Therefore, the subgrid-scale stresses, which correspond to small eddies are filtered and modeled accordingly. When applying the Large Eddy Simulation (LES) technique during the simulation process, the following filtered Navier-Stokes equation is obtained:

\[
\frac{\partial \bar{\rho} \bar{u}_i}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{\partial \bar{p}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} \right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{15}
\]
Where $\sigma_{ij}$ is the stress tensor due to molecular viscosity of $\mu$. $\tau_{ij}$ is the subgrid-turbulent stress component, which is based on the Boussinesq hypothesis which is used by ANSYS Fluent. It can be calculated as follows [62]:

$$
\tau_{ij} = \frac{1}{3} \tau_{kk} \delta_{ij} - 2 \mu_t \bar{S}_{ij}
$$

Where $\mu_t$ is the subgrid-scale turbulent viscosity. The isotropic part of the subgrid-scale stresses $\tau_{kk}$ is not modeled, but added to the filtered static pressure term of $\bar{p}$. $\delta_{ij}$ is the Kronecker delta. $\bar{S}_{ij}$ is the rate-of-strain tensor for the resolved scale defined by

$$
\bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
$$

In ANSYS Fluent, the basic approach used to solve the subgrid-scale turbulent viscosity $\mu_t$ is based on the Smagorinsky-Lilly model, as follows:

$$
\mu_t = \rho L_s |\bar{S}|
$$

where $|\bar{S}| = \sqrt{2\bar{S}_{ij} \bar{S}_{ij}}$. $L_s$ is the mixing length for subgrid scales and is computed using $L_s = \min(\kappa d, C_s \Delta)$, where $\kappa$ is the von Karman constant of 0.41, and $d$ is the distance to the closest wall. $C_s$ is the Smagorinsky constant, which has the default value of 0.1 in Fluent. $\Delta$ is computed according to the volume of the computational cell using $\Delta = V^{1/3}$. The Smagorinsky-Lilly model assumes that the energy production and dissipation of the small scale are in equilibrium.

For liquid fuel droplet modeling, the Discrete Phase Model (DPM) [62] was used to generate the fuel droplets. The physical model for simulating the behavior of the
discrete phase was based on the stochastic collision model, which has been used in the past with liquid fuels [62]. The breakup of fuel droplets was considered by using the Taylor Analogy Breakup model (TAB) [62], which is based on the analogy between an oscillating and distorting droplet as a spring-mass system commonly used for fuel droplet break up [62]. The detailed numerical algorithms to generate the fuel droplets and to simulate the fuel droplet-gas interactions using ANSYS Fluent are explained below.

In ANSYS Fluent, the discrete phase model is based on the Lagrangian discrete phase model [62]. The fluid phase is treated as a continuum by solving the Navier-Stokes equation, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase, and the interactions between particles are neglected [62]. The trajectory of a discrete phase particle is predicted by integrating the force balance on each particle, and the force balance equation can be written as [62]

$$\frac{d\vec{u}_p}{dt} = F_D(\vec{u} - \vec{u}_p) + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} + \vec{F}$$

(19)

Where, $\vec{u}$ is the fluid phase velocity, $\vec{u}_p$ is the particle velocity, $\rho$ is the fluid density, and $\rho_p$ is the density of the particle. $\vec{F}$ is an additional acceleration (force/unit particle mass) term (inertia-based). $F_D(\vec{u} - \vec{u}_p)$ is the drag force per unit particle mass and

$$F_D = \frac{18 \mu}{\rho_p d_p^2} \frac{C_D \text{Re}}{24}$$

(20)
Where, \( \mu \) is the molecular viscosity of the fluid, \( d_p \) is the particle diameter, and \( C_D \) is the drag coefficient for a spherical droplet. \( Re \) is the relative Reynolds number, which is defined as

\[
Re = \frac{\rho d_p |u_p - u|}{\mu}
\]  
\[
(21)
\]

In ANSYS Fluent, the drag coefficient \( C_D \) for smooth particles or spherical droplets can be obtained as follows:

\[
C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2}
\]  
\[
(22)
\]

Where \( a_1, a_2, \) and \( a_3 \) are constants that apply over several ranges of \( Re \) as shown in the study of Morsi et al. [63].

As explained above, a twin-fluid atomizer was used to inject fuel droplets into the combustion chamber as a spray of emulsified canola oil [4]. The atomizer is supplied with two fluids, with liquid fuel on one side and high pressure air on the other. The atomizer has a mixing chamber within it, where the air stream impinges on the liquid jet and forms a good quality liquid-gas mixture which is then sprayed as a very fine mist. The size of fuel droplets is determined by the pressure of air and liquid fuel in the atomizer.
In the numerical method, the air-blast atomization model or the discrete phase model has been selected, where a fuel liquid is formed into a sheet by the nozzle, and air is then directed against the sheet to promote atomization as shown in Fig. 5 [62]. The operating principles of atomizer model (i.e. discrete phase model) are the same as the atomizer used in the experiments.

According to ANSYS Fluent [62], the sheet breakup in the air-blast model is based on the formation of short waves around the fuel nozzle. The diameter of the ligament that leads to the formation of fuel droplets is assumed to be linearly proportional to the wavelength that breaks up the sheet, as follows:

\[ d_L = \frac{2\pi C_L}{K_s} \]  

(23)

Where \( C_L \) is the ligament constant of 0.5. \( K_s \) is the wave number, which can be calculated as follows:
\[ K_s = \frac{\rho_l U^2}{2\sigma} \] (24)

Where \( \rho_l \) is density of fuel, \( U \) is local relative velocity between fuel and air in the nozzle, and \( \sigma \) is the surface tension of fuel droplet.

The diameter of fuel droplets can be obtained as follows:

\[ d_0 = 1.88 d_L (1 + 3Oh)^{1/6} \] (25)

Where \( Oh \) is the Ohnesorge number, which is a combination of the Reynolds number and the Weber number.

Based on the explanation above, the size of the fuel droplets depends on the relative velocity between fuel and air in the simulated fuel nozzle. It is known that the relative velocity between both streams depends on the pressure of air and liquid fuel in the atomizer, which induces the necessary shear flow to form a thin liquid ring and eventually a fine dispersion of droplets.

In turbulent flow, ANSYS Fluent uses the mean fluid phase velocity \( \bar{u} \) to predict the trajectories of particles [62]. In the stochastic tracking approach, ANSYS Fluent predicts the turbulent dispersion of particles by integrating the trajectory equations of individual particles, using the instantaneous fluid velocity of \( u = \bar{u} + u'(t) \) along the particle path. By computing the trajectory of particles, the random effects of turbulence on the particle dispersion can be taken into account [62].

In order to take into account the effect of droplet breakup on sprays, the Taylor Analogy Breakup (TAB) model has been applied. In ANSYS Fluent, the TAB model is based on Taylor’s analogy between an oscillating and distorting droplet and a spring-
mass system. When the droplet oscillations reach a critical value, the “parent” droplet will break up into a number of smaller “child” droplets. The results of the TAB model equation can be used to determine the droplet oscillation and distortion at any time [62].

The equation governing a damped, forced oscillator is given by:

\[ F - kx - d \frac{dx}{dt} = m \frac{d^2x}{dt^2} \quad (26) \]

Where \( x \) is the displacement of the droplet from its spherical position. The coefficients in the equation, \( \frac{F}{m}, \frac{k}{m}, \frac{d}{m} \), are taken from Taylor’s analogy [62]:

\[ \frac{F}{m} = C_F \frac{\rho_i u^2}{\rho_i r} \quad (27) \]

\[ \frac{k}{m} = C_k \frac{\sigma}{\rho_i r^3} \quad (28) \]

\[ \frac{d}{m} = C_d \frac{\mu_i}{\rho_i r^2} \quad (29) \]

Where \( \rho_i \) and \( \rho_g \) are the discrete phase and continuous phase densities, respectively. \( u \) is the relative velocity of the droplet, \( r \) is the undisturbed droplet radius, \( \sigma \) is the droplet surface tension, and \( \mu_i \) is the droplet viscosity. The values of \( C_F, C_k \) and \( C_d \) are 1/3, 8 and 5, which represents the constant values of droplet drag force, surface tension forces, and droplet viscosity forces, respectively.

When the distortion reaches half of the droplet radius, the droplet is assumed to break up, as shown in Fig. 6.
The breakup requirement is given as follows [62]:

\[ x > C_b r \]  

(30)

Where \( C_b \) is a constant with a value of 0.5.

By setting \( y = \frac{x}{C_b r} \) and substituting the relationships in Equations 27-29 in Equation 26, the new nondimensionalized control equation is given as

\[
\frac{d^2 y}{d t^2} = \frac{C_F}{C_b} \frac{\rho_g u^2}{\rho_i r^2} - \frac{C_i \sigma}{\rho_i r^3} y - \frac{C_d \mu_i}{\rho_i r^2} \frac{dy}{dt}
\]

(31)

The solution of equation is shown as follows:

\[
y(t) = W e_c + e^{-\frac{t}{t_d}} \left[ \left( y_0 - W e_c \right) \cos(\omega t) + \frac{1}{\omega} \left( \frac{dy_0}{dt} - \frac{y_0 - W e_c}{t_d} \right) \sin(\omega t) \right]
\]

(32)

Where

\[
W e = \frac{\rho_g u^2 r}{\sigma}
\]

(33)
\[ We_c = \frac{C_F}{C_1C_b} We \]  
\[ y_0 = y(0) \]  
\[ \frac{dy_0}{dt} = \frac{dy}{dt}(0) \]  
\[ \frac{1}{t_d} = \frac{C_d}{2} \frac{\mu_i}{\rho r^2} \]  
\[ \omega^2 = C_k \frac{\sigma}{\rho r^3} - \frac{1}{t_d^2} \]  

If equation is solved for all droplets, those with \( y > 1 \) are assumed to break up.

In order to simulate the turbulent combustion process, the Eddy-Dissipation model [62] was applied. It takes into account the effects of chemical reaction and turbulent flow simultaneously. In ANSYS Fluent, the Eddy-dissipation model assumes reaction rates are controlled by the turbulent mixing. The model is based on the work of Magnussen and Hjertager [64]. The net rate of production of species \( i \) due to reaction \( r \) is given by the following equations

\[ R_{i,r} = v'_{i,r} M_{w,i} A \rho \frac{\varepsilon}{k} \min_r \left( \frac{Y_R}{v'_{r,r} M_{w,R}} \right) \]  
\[ R_{i,r} = v'_{i,r} M_{w,i} AB \rho \frac{\varepsilon}{k} \sum_{j} Y_p \frac{1}{\sum_{j'} v'_{j,r} M_{w,j}} \]  

Where,

\( v'_{i,r} = \) stoichiometric coefficient for reactant \( i \) in reaction \( r \)

\( v'_{r,r} = \) stoichiometric coefficient for reactant \( R \) in reaction \( r \)
\( \nu''_{j,r} \) = stoichiometric coefficient for product \( j \) in reaction \( r \)

\( M_{w,i} \) = the molecular weight of species \( i \)

\( M_{w,j} \) = the molecular weight of species \( j \)

\( M_{w,R} \) = the molecular weight of species \( R \)

\( Y_P \) = the mass fraction of any product species, \( P \)

\( Y_R \) = the mass fraction of a particular reactant, \( R \)

\( A \) = an empirical constant of 4.0

\( B \) = an empirical constant of 0.5

When the LES turbulence model is used, the turbulent mixing rate, \( \varepsilon/k \), in Equation 39 and 40, is replaced by the subgrid-scale mixing rate, which is calculated as [62]

\[
\tau_{sgs}^{-1} = \sqrt{2 S_{ij} S_{ij}} \quad (41)
\]

Where,

\( \tau_{sgs}^{-1} \) is subgrid-scale mixing rate (s\(^{-1}\))

\( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \) is strain rate tensor (s\(^{-1}\))

When considering a \( r_{th} \) reaction, the general form of the reaction can be written as follows:

\[
\sum_{i=1}^{N} \nu'_{i,r} \chi_i \Leftrightarrow \sum_{i=1}^{N} \nu''_{i,r} \chi_i \quad (42)
\]

Where, \( \chi_i \) is the chemical formula of species \( i \), \( k_{f,r} \) and \( k_{b,r} \) is the reaction rate constants of the forward and backward reactions, respectively. The reaction rate constant of forward reaction can be obtained as follows:
\[ k_{f,r} = Ae^{-E/RT} \]  

(43)

Where, \( A \) is the pre-exponential factor, \( E \) is the activation energy for the reaction (J/kmol). \( R \) is the universal gas constant (J/kmol-K).

The reaction rate constant of forward reaction is calculated as follows:

\[ k_{b,r} = \frac{k_{f,r}}{K_r} \]  

(44)

Where \( K_r \) is the equilibrium constant for the \( r \)th reaction, which is shown as follows:

\[ K_r = \exp \left( \frac{\Delta S_r}{R} - \frac{\Delta H_r}{RT} \right) \left( \frac{P_{\text{atm}}}{RT} \right)^{\sum (v_i^r - v_i^r)} \]  

(45)

Where \( P_{\text{atm}} \) is the atmospheric pressure. The terms within the exponential function can be calculated as follows:

\[ \frac{\Delta S_r}{R} = \sum_{i=1}^{N} \left( v_{i,r}^n - v_{i,r}^l \right) \frac{S_i}{R} \]  

(46)

\[ \frac{\Delta H_r}{RT} = \sum_{i=1}^{N} \left( v_{i,r}^n - v_{i,r}^l \right) \frac{h_i}{RT} \]  

(47)

Where \( S_i \) and \( h_i \) are the entropy and enthalpy of the \( i \)th species evaluated at temperature \( T \) and atmospheric pressure.

According to the studies of Westbrook and Dryer [53] and Yasina [54], a two-step reaction mechanism is selected as the chemical kinetic reaction mechanism. The stoichiometric combustion equations for the two-step mechanism is shown in Table 4.
Table 4. Two steps chemical reaction mechanism of emulsified canola oil

<table>
<thead>
<tr>
<th>Reaction</th>
<th>Pre-Exponential Factor (A)</th>
<th>Activation Energy (E) (J/kg-mol)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{CH}<em>{1.6305}\text{O}</em>{0.0806} + 0.867\text{O}_2 \rightarrow \text{CO} + 0.81525\text{H}_2\text{O}$</td>
<td>$5 \times 10^5$</td>
<td>$6.6 \times 10^7$</td>
</tr>
<tr>
<td>$\text{CO} + 0.5\text{O}_2 + \text{H}_2\text{O} \rightleftharpoons \text{CO}_2 + \text{H}_2\text{O}$</td>
<td>$3.98 \times 10^{11}$</td>
<td>$1.6736 \times 10^8$</td>
</tr>
</tbody>
</table>

During the combustion process, the formation of thermal $\text{NO}_x$ is determined by a set of highly temperature-dependent chemical reactions known as the Zeldovich mechanism. The principal reactions governing the formation of the thermal $\text{NO}_x$ are shown as follows [62]:

$$\text{O} + \text{N}_2 \rightleftharpoons \text{N} + \text{NO}$$

$$\text{N} + \text{O}_2 \rightleftharpoons \text{O} + \text{NO}$$

A third reaction has been shown to contribute to the formation of thermal $\text{NO}_x$, particularly at near conditions and in fuel-rich mixtures [62]:

$$\text{N} + \text{OH} \rightleftharpoons \text{N} + \text{NO}$$

The rate for these reactions can be expressed as follows [62]:

$$k_{f,1} = 1.8 \times 10^8 e^{-38370/T} \quad k_{r,1} = 3.8 \times 10^7 e^{-425/T}$$

$$k_{f,2} = 1.8 \times 10^4 e^{-4680/T} \quad k_{r,2} = 3.81 \times 10^3 e^{-20820/T}$$

$$k_{f,3} = 7.1 \times 10^7 e^{-450/T} \quad k_{r,3} = 1.7 \times 10^8 e^{-24560/T}$$

The net rate of formation of $\text{NO}$ is calculated as follows:
The reaction rate for NO is given by:

\[
\frac{d[NO]}{dt} = 2k_{r,1}[O][O_2] \left( 1 - \frac{k_{r,2}[NO]^2}{k_{f,1}[N_2]k_{f,2}[O_2]} \right) \left( \frac{1 - \frac{k_{r,1}[NO]}{k_{f,2}[O_2] + k_{f,3}[OH]}}{k_{f,1}[N_2]^2} \right) \text{ (mol/m}^3 \cdot \text{s)}
\] (48)

Table 5 shows all the models used during the simulations including their applicability and limitations.

<table>
<thead>
<tr>
<th>Model</th>
<th>Applicability</th>
<th>Limitation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Large Eddy Simulation</td>
<td>Turbulent Flow</td>
<td>Should not be used for complex geometry and very high Reynolds number because of the need for high computational resources</td>
</tr>
<tr>
<td>Discrete Phase Model</td>
<td>Discrete phase trajectory using a Lagrangian formulation for both steady and unsteady flows</td>
<td>Cannot be used for high volume fraction of discrete phase, usually larger than 12%</td>
</tr>
<tr>
<td>Stochastics Model</td>
<td>The dispersion of particles due to turbulence in the fluid phase</td>
<td>Cannot be included if the Spalart-Allmaras turbulence model is used</td>
</tr>
<tr>
<td>Taylor Analogy Breakup</td>
<td>Used to determine the droplet drag coefficient for low-speed sprays into a standard atmosphere</td>
<td>Cannot be used for high-Weber-number sprays</td>
</tr>
<tr>
<td>(TAB) Model</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Eddy-Dissipation Model</td>
<td>Turbulence-chemistry interaction reaction</td>
<td>Requires products to initiate reaction (LFR model)</td>
</tr>
</tbody>
</table>
Fig. 7. Overview of the pressure-based solver
In term of the computational algorithm, Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm was selected to solve the pressure-velocity coupling. According to ANASYS Fluent [62], the SIMPLE algorithm uses a relationship between velocity and pressure correlations to enforce mass conservation and to obtain the pressure field. The overview of this algorithm is shown in Fig. 7 [62].

The spatial discretization method used has a second-order accuracy based on a central-differencing scheme for the momentum, energy and species equations. A control-volume-based technique was used to convert the general scalar transport equation to an algebraic system [62]. The control volume technique consists of integrating the transport equation about each control volume, yielding a discrete equation that expresses the conservation law on a control-volume basis. The discretization of the governing equations are illustrated by considering the unsteady conservation equation for transport of a scalar quantity \( \phi \). The following equation written in integral form for an arbitrary control volume \( V \) as follows:

\[
\int_V \frac{\partial \rho \phi}{\partial t} \, dV + \oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi \, dV \tag{49}
\]

Where

\( \rho \) is the density

\( \vec{v} \) is the velocity vector

\( \vec{A} \) is the surface area vector

\( \Gamma_\phi \) is the diffusion coefficient for \( \phi \)

\( \nabla \phi \) is the gradient of \( \phi \)
\( S_\phi \) is the source of \( \phi \) per unit volume

Equation 41 can be applied to a control volume or cell in the computation domain. For a given cell, equation 41 can be shown as follows:

\[
\frac{\partial \rho \phi}{\partial t} V + \sum_{f} \rho_f \vec{v}_f \cdot \vec{A}_f = \sum_{f} \Gamma_\phi \nabla \phi \cdot \vec{A}_f + S_\phi V
\]

(50)

Where

\( N_{\text{faces}} \) is the number of faces enclosing the cell
\( \phi_f \) is the value of \( \phi \) converted through face \( f \)
\( \rho_f \vec{v}_f \cdot \vec{A}_f \) is mass flux through the face
\( \vec{A}_f \) is area of face \( f \)
\( \nabla \phi_f \) is the gradient of \( \phi \) at face \( f \)
\( V \) is the cell volume

For the second-order discretization, \( \frac{\partial \rho \phi}{\partial t} V \) in Equation 42 is defined as follows:

\[
\frac{\partial \phi}{\partial t} = F(\phi) = \frac{3\phi^{n+1} - 4\phi^n + \phi^{n-1}}{2\Delta t}
\]

(51)

Where

\( \phi \) is a scalar quantity
\( n+1 \) is the value at the next time level, \( t + \Delta t \)
\( n \) is the value at the current time level, \( t \)
\( n-1 \) is the value at the previous time level, \( t - \Delta t \)

In ANSYS Fluent [62], a second-order-accurate central-differencing discretization scheme is available for the momentum equations when the LES turbulence
model is used. This scheme provides improved accuracy for LES calculations. The central-differencing scheme calculates the face value for a variable \( \phi_f \), as follows:

\[
\phi_f = \frac{1}{2} (\phi_0 + \phi_1) + \frac{1}{2} \left( \nabla \phi_0 \cdot \vec{r}_0 + \nabla \phi_1 \cdot \vec{r}_1 \right)
\]  

(52)

Where the indices 0 and 1 refer to the cells that share face \( f \). \( \nabla \phi_{r,0} \) and \( \nabla \phi_{r,1} \) are the reconstructed gradients at cells 0 and 1, respectively, and \( \vec{r} \) is the vector directed from the cell centroid toward the face centroid.

The PREssure STaggering Option (PRESTO!) scheme was used for pressure interpolation. In ANSYS Fluent [62], the PRESTO! Scheme uses the discrete continuity equation to calculate the pressure field on a mesh that is geometrically shifted so that the new cell centers are where the faces of the ordinary mesh are located. This means that the pressures on the faces can be determined [62]. Fig. 8 shows a structured grid for the PRESTO! Scheme, where the shade area illustrates the staggered grids.

![Fig. 8. Schematic diagram of the PRESTO! Scheme](image-url)
In order to analyze the size distribution of vortices, the Fast Fourier Transform (FFT) method was used using MATLAB. FFT is a mathematical algorithm which is used to calculate the Discrete Fourier Transform (DFT) [65]. DFT is based on a mathematical operation such as the Fourier transform, which converts a signal into its constituent frequencies. Once Fourier transform is done, all the signals on the time-domain are computed into signals on the frequency domain. According to the study by Cooley and Tukey [65], a given vectors of length N can be transformed by FFT as follows:

$$X(k) = \sum_{j=1}^{N} x(j) \omega_N^{(j-1)(k-1)}$$

Where, $\omega_N = e^{(-2\pi i)/N}$, which is an $N_{th}$ root of unity.

Once the frequency spectrum of each flow structure has been determined, it is used to understand the effect of Swirl number and flowrate on the distribution of eddy size in the flow, as shown in Chapter 5.
4. LARGE EDDY SIMULATION OF SWIRL PROMOTED TURBULENT COLD FLOW IN A COMBUSTION CHAMBER

The distribution and combustion of fuel droplets is highly dependent on the flow field in any combustion chamber. For droplets entrained in a gas stream, understanding the evolution of turbulent flow is the key in the study of combustion processes and emission generation in a combustion chamber. In this chapter, the simulation of turbulent flow field without combustion is validated using the chosen numerical approach, as outlined in the previous chapter. The velocity profile, streamlines and vorticity distribution of cold flow in a combustion chamber are shown and discussed. Furthermore, the simulation of fuel droplets distribution is validated by comparing with Jones’ experimental results \[66\]. The effects of flow field on fuel droplets distribution have been studied as described below.

4.1. Validation of numerical approach on turbulent flow field

To validate the numerical approach for simulating swirling flow in the prescribed combustion chamber, the first step is to simulate and study the turbulent flow without the swirler. The simulation of a sudden expanding flow is based on the work by Wessman \[67\], which was used to validate the current simulation results in terms of velocity profile and experimental data shown in Wessman \[67\].

The experimental setup used by Wessman \[67\] is shown in Fig. 9(a). It consists of a long guiding pipe, and a sudden expansion chamber. The inlet bulk velocity was set
to 1.59 m/s. Coarse and fine meshed grids with 2.1 million and 4.6 million cells, respectively, were generated using ANSYS ICEM CFD as shown in Fig. 9 (a) and (b), respectively.

**Fig. 9.** (a) Schematic of a chamber with an inlet duct. Dimensions are in mm, (b) a coarse grid mesh of sudden expansion chamber used for validation, (c) a fine grid mesh of sudden expansion chamber used for validation.

The numerical simulation was performed using ANSYS Fluent 15.0. The simulation was based on the Reynolds Averaged Navier Stokes (RANS) equation with the k-ε turbulence model first. The Large Eddy Simulation (LES) technique was used when the solution of RANS converged. The Semi-Implicit Method for Pressure-Linked
Equations (SIMPLE) algorithm was used for pressure-velocity coupling scheme, and the second order wind was used for the spatial discretization of pressure, momentum, and the other parameters. The time interval was set to $10^{-5}$ s.

The simulation results of two cases compared with the analytical solution of pressure drop, which was calculated using the Bernoulli’s Equation as shown below.

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + Z_1 - h_{\text{loss1}} - h_{\text{expansion}} - h_{\text{loss2}} = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + Z_2$$  \hspace{1cm} (54)

Where,

$$h_{\text{loss}} = f \frac{L_i}{D_i} \frac{V_i^2}{2g} \hspace{0.5cm} (f = 0.0176 \text{ from Moody chart})$$  \hspace{1cm} (55)

$$h_{\text{expansion}} = \left(1 - \frac{A_i}{A_2}\right)^2 \frac{V_1^2}{2g}$$  \hspace{1cm} (56)

$$Z_1 = Z_2 = 0$$  \hspace{1cm} (57)

The comparison between the two cases and their simulation results with analytical solution are shown in Table 6.

According to Table 6, the fine grid case gives a small error when compared to the analytical solution. The fine grid with a resolution is 0.5 mm in the radial direction and 4 mm in the axial direction was chosen for simulation purposes. Fig. 10 shows the comparison of experiment data [67] with simulation results obtained using the grid shown in Fig. 9(b) and (c), at three axial locations. Time averaged axial velocity, $\langle u \rangle$, and bulk velocity at inlet of guiding pipe, $U_b$; the radial position, $r$, and the radial of guiding pipe, $R$; were used as physical variables during the simulation process. The non-
dimensional axial location shown in Fig. 10 is the ratio of axial position, $z$, to the step height, $H$.

**Table 6.** Comparison of simulation results with analytical solution for the chamber without swirler

<table>
<thead>
<tr>
<th>Cases</th>
<th>Number of grids</th>
<th>Type of grid</th>
<th>simulation value of pressure drop (Pa)</th>
<th>Analytical value of pressure drop (Pa)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse Grid</td>
<td>2.1 million</td>
<td>Hexahedron</td>
<td>35.2</td>
<td>38.3</td>
<td>8.1</td>
</tr>
<tr>
<td>Fine Grid</td>
<td>4.6 million</td>
<td>Hexahedron</td>
<td>37.1</td>
<td>38.3</td>
<td>3.1</td>
</tr>
</tbody>
</table>

In Fig. 10, the solid line depicts the simulation result, which is based on the fine grid and the numerical algorithm explained above. The data points represent the experimental results shown in Wessman [67]. As Fig. 10 shows, the implemented approach can accurately simulate turbulence flow for the case described above. The other simulations were based on the same resolution of mesh and numerical algorithm.
In order to validate the numerical methods used for simulating turbulent flow with a swirler, the simulation with the swirler was compared first with the experimental tests conducted by Favaloro [68]. The experimental setup is shown in Fig. 11. It consists of a long guiding pipe, a swirl generator, and a sudden expansion chamber. The inlet bulk velocity is 19.2 m/s. The swirler with 12 circular inlet guide vanes is located at 50.8 mm upstream of the dump plane. The swirl number is 0.5.
Computational meshes were generated using ANSYS ICEM CFD. Three meshes were generated, including a coarse mesh with tetrahedron grid elements (1.9 million), a fine mesh with tetrahedron grid elements (3 million), and a fine mesh with a combination grid (4.1 million), which is made of tetrahedron grid elements for the region around the swirler and hexahedron grid elements for the rest of region of domain, respectively. All three meshes are based on the same approach and resolution used for the validation of turbulent flow without swirler. The three meshes are shown in Fig. 12 (a), (b), and (c), respectively.
The numerical algorithm is the same as the one used for the validation cases without a swirler. The simulation was based on the Reynolds Averaged Navier Stokes (RANS) equation and the k-ε turbulence. LES was used when the solution of RANS converged. The time interval was set to $10^{-5}$ s. Simulation data are collected for about 3 flow-through times (66 ms) to obtain statistically meaningful turbulent properties. The simulation results were compared using radial distributions of time-mean normalized velocity profile at a location $(z/H)$ of as shown in Fig. 13.

Fig. 13 shows that the combination grid gives the best agreement with experimental data when compared with the other two meshes. Therefore, the simulation of turbulent flow for the combustion chamber in the CBEL is based on the combination mesh, Fig 12c. Fig 14 shows the radial distribution of the time-mean axial, radial and azimuthal normalized velocity components at three locations $(z/H)$ of 0.38 and 4.0,
respectively. The combination mesh was used, and the simulations were compared with the experimental data of Favaloro [68]. After completing the validation process, the CBEL swirler was meshed and used accordingly, as described below.

![Graph showing the comparison of three simulation cases and experimental data for the chamber with swirler at axial location (z/H) of 0.38](image)

**Fig. 13.** Comparison of three simulation cases and experimental data for the chamber with swirler at axial location (z/H) of 0.38

According to Fig. 14, the simulation results agree reasonably well with the experimental data in the three directions of velocity. Therefore, the approach used for mesh generation and the numerical algorithm can be used for the simulation of turbulent flow in the combustion chamber described above.
Fig. 14. Radial distribution of mean velocity components at two axial locations
4.2. **Large eddy simulations of swirl-promoted turbulent cold low in combustion chamber.**

In order to understand the combustion processes and corresponding emissions in swirl promoted flows, obtaining realistic turbulent flow field is the key to be able to analyze the effects of swirl number and fuel properties on combustion. This section discusses the effects of swirl number on turbulent flow field without combustion by analyzing velocity profiles and turbulent kinetic energy in the combustion chamber. It also presents the evolution of flow vortices in the combustion chamber, which have been analyzed using streamlines and the Q-criterion of vortices.

For the combustion studies, the simulation domain and schematic of the geometry of the CBEL swirler are shown in Fig. 2 and Fig. 15, respectively. The flow conditions are in Table 1 including a flow rate of 393 l/min with $SN$ of 1.0 and 1.4. The computational mesh as shown in Fig. 16 was generated using ANSYS ICEM CFD following the same approach as explained above. ANSYS Fluent was used as the mathematical solver to obtain the corresponding numerical solutions. The initial time interval was set to be $10^{-5}$ s, and then increased to $5 \times 10^{-5}$ s in order to reduce the simulation time. The Courant number of 0.6 is based on the time interval of $5 \times 10^{-5}$ s.
Fig. 15. Schematic of domain for swirl promoted turbulent simulation. Dimensions is in mm.

Fig. 16. Combination mesh for swirl promoted turbulent simulation

4.2.1. Effects of swirl number on recirculation zones within the combustion chamber

Swirlers are used in combustion chambers [69] to produce Central toroidal Recirculation Zones (CRZ) to enhance the mixing of fuel and oxygen. The characteristic of CRZ is affected by swirl number notably. In this section, the profiles of normalized
time mean velocity and turbulent kinetic energy are discussed in terms of three axial locations. The shape of recirculation zone is shown using iso-surface of mean axial velocity on $<u_z> = 0$. The types of recirculation zone and the corresponding evolution process are considered.

Fig. 17(a) – (c) shows radial distributions of the normalized time averaged axial, tangential, and radial velocity components at three measurement locations ($z/H$) of 0.38, 1.0, and 4.0 for swirl number (SN) of 1.0 and 1.4, respectively. Fig. 17(a) shows that the axial velocity is negative between the inner wall of the swirler and the wall of the chamber. It implies that reverse flow is produced in the chamber, which is part of the recirculation zone. Comparing the cases of SN of 1.0 and 1.4, it is observed that the range of the negative normalized velocity is larger for SN of 1.4 than for SN of 1.0. This implies that a high swirl number there is an overall expansion of the recirculation zone. This is due to the centrifugal effect, which spreads the flow in the radial direction gradually. The location of the peak positive axial velocity is close to the wall, when a high swirl number is used. The effect of swirl number diminishes along the axial direction and converges to a similar axial velocity profile at the location ($z/H$) of 4.

The tangential velocity profile is shown in Fig. 17(b). The swirler with a high swirl number produces greater tangential velocity, and the largest tangential velocity is created near the outlet region of the swirler, where the radial position ($r/R$) varies from 0.39 to 0.67. The tangential velocity decreases along the axial direction; however, its average value decreases faster with low swirl number.
Fig. 17. Radial distribution of time-averaged normalized velocity components at (a) axial direction, (b) radial direction, (c) tangential direction, respectively.
Both axial and tangential velocity reduce along the axial direction; however, the radial velocity increases in the downstream region as shown in Fig. 17(c). Within the outlet region of the swirler, the energy and momentum remain almost the same, and neither dissipate by turbulent diffusion or viscous dissipation. Therefore, the momentum which is generated by the axial and tangential velocities is converted to radial momentum, which leads to an increase in radial velocity. In addition, the radial velocity magnitude for $SN$ of 1.4 is also greater than for $SN$ of 1.0 due to greater centrifugal effect.

Fig. 18 shows the turbulent kinetic energy at three locations. It is observed that it reaches a peak at the location $(z/H)$ of 0.38. It implies that the shear layer which is a result of the Kelvin-Helmholtz instability, dominates the flow at this location. A shear layer is induced by the helical vortex flow and recirculation taking place within the zone between the swirler and the radial plane located $(z/H)$ at 0.38. This layer can be categorized as a buffer zone between the core of the recirculation zone and the helical vortex flow that goes around it. Fig. 18 also shows that at a $SN$ of 1.4 leads to a wider center recirculation zone than for a $SN$ of 1.0. Additionally, the development of the flow structure expands radially and becomes more uniform in the downstream region. The smooth distribution of turbulent kinetic energy in the downstream region shows that turbulence tends to become isotropic under the effects of turbulent dispersion and viscous dissipation.
**Fig. 18.** Radial distribution of normalized turbulent kinetic energy

Fig. 19(a) and (b) show the shape of recirculation zones using iso-surface of time-averaged axial velocity on $\langle u_z \rangle = 0$ for SN of 1.0 and 1.4, respectively. The black solid lines are two reference lines to identify the size of the center of recirculation zone in the spanwise direction. In Fig. 19, the green surface shows a time-averaged axial velocity of zero. The region, where the axial velocity is less than zero, is inside the green surface. Hence, the green surface in the center of chamber represents the shape of center recirculation zone, and the green surface, which is near the inlet of combustion chamber, is the corner of the recirculation zone. As shown in Fig. 19, for a SN of 1.4 not only
leads to a wider center of recirculation zone than for a SN of 1.0 in the spanwise direction near the swirler, but also produces a longer center of recirculation zone in the streamwise direction, which validates the findings seen in Fig. 17 and 18. Therefore, the center recirculation zone for high swirl number is much greater than that of a low swirl number.

Fig. 19. Iso-Surface of time-averaged axial velocity on \(<u_z> = 0\) for (a) SN of 1.0 and (b) SN of 1.4

4.2.2. Effects of swirl number on flow structure within the combustion chamber

The flow structure is relevant to this study because of its dominant influence in determining the fuel droplet distribution and the mixing of fuel and oxygen, which is relevant in combustion kinetics. In this section, a snapshot of a vorticity profile on the z-
plane and the streamline on the \( z-r \) and \( r-\theta \) plane are presented and analyzed to understand the flow structure in the chamber. The effects of swirl number on flow structure are discussed as well.

Fig. 20 is a snapshot of vorticity profile at the \( z-r \) plane for \( SN \) of 1.0 and \( SN \) of 1.4 at flow time of 500 ms. It shows that the greatest size of vortices, represented by the colors on the scale, are generated near the outlet region of the swirler for both \( SN \) of 1.0 and 1.4 due to the dump plane and the wake and helical flow induced by vane. The vortices break down along the axial direction due to the presence of counter-rotating vortices within the buffer zone next to the core recirculation zone as it is shown below. In terms of the effects of swirl number, \( SN \) of 1.4 generates larger sizes of vortices based on the strength of the vortices depicted in color in Fig. 20. In the next chapter, Fast Fourier Transfer analysis indicate that the size of the vortices are greater for \( SN \) of 1.4. It can also be inferred that the speed of turbulent diffusion and dissipation is slower than for a \( SN \) of 1.0 since it has greater angular momentum.
Fig. 20. Snapshot of vorticity profile at the z-r plane on SN of 1.0 and SN of 1.4 at flow time of 500 ms

To facilitate the visualization of the vortices, the streamlines at the r-θ planes at axial locations (z/H) of 0.38, 1, and 4 for SN of 1.0 and 1.4 are presented in Fig. 21(a) - (b), respectively. The three solid vertical lines shown in Fig. 20 represent the locations of the r-θ planes.
In turbulent flow, the vortices are convected downstream where their strength diminishes and directionally converged within the core region [17]. This leads to the generations of the processing vortex core. In Fig. 21(a) - (b), two kinds of processing vortex cores are observed, which are the spiral mode and double-helix mode. At the location \((z/H) = 0.38\), two vortex cores are generated for both \(SN = 1.0\) and 1.4 due to the combined effect of the induced helical flow and wake flow. At the location \((z/H) = 1\), two vortex cores are found spiraling around the centerline, which means the helical

![Streamlines at the r-θ planes of z/H= 0.38, z/H=1, and z/H=4 on (a) SN of 1.0](image)

![Streamlines at the r-θ planes of z/H= 0.38, z/H=1, and z/H=4 on (b) SN of 1.4](image)

**Fig. 21.** Streamlines at the \(r-\theta\) planes of \(z/H= 0.38\), \(z/H=1\), and \(z/H=4\) on (a) \(SN = 1.0\) (b) \(SN = 1.4\)
flow and wake flow can still be found in the fluid flow, but the effect of wake flow starts to reduce due to the shear friction between the two vortex cores. Because the swirler on a high swirl number generates greater angular momentum, the two cores for SN of 1.0 are closer to each other than at a SN of 1.4. At the location (z/H) of 4, the two vortex cores disappear, and a single vortex core appears as a single spiral around the center line due to the diminished effect of the wake flow. The same behavior can be seen in Fig. 19, which shows that the diameter of the center of recirculation zone suddenly reduces near z/H of 4. The next section provides additional visual representation of the vortices by using the Q-criterion approach.

Fig. 22. Instantaneous streamlines at the r-θ planes (z/H) of 1 when SN is 1.4 at time of (a) 450 ms, (b) 500 ms, (c) 550 ms, (d) 600 ms, (e) 650 ms, (f) 700 ms
Fig. 22(a)-(f) show the instantaneous streamlines when SN is 1.4 at the $r$-$\theta$ planes ($z/H$) of 1 at times of 450 ms to 700 ms. The double-helix mode can be observed at time of 500 ms, 550 ms, and 650 ms. It shows that the wake flow affects the $r$-$\theta$ planes ($z/H$) of 1 alternatively. Fig. 23(a)-(f) show the instantaneous streamlines when SN is 1.4 at the $r$-$\theta$ planes ($z/H$) of 4 at times of 450 ms to 700 ms. Only spiral mode vortex can be observed due to the diminished effect of the wake flow. The results are consistent with the findings seen in Fig. 21.

Fig. 23. Instantaneous streamlines at the $r$-$\theta$ planes ($z/H$) of 4 when SN is 1.4 at time of (a) 450 ms, (b) 500 ms, (c) 550 ms, (d) 600 ms, (e) 650 ms, (f) 700 ms
4.2.3. **Effects of swirl number on vorticity profile**

Turbulent flow is very complicated because it includes momentum diffusion and convection, rapid variation of pressure and velocity in space and time. All of these factors generate very complex vortices. Vorticity has a direct effect on the flow field of air and the fuel droplet distribution. Therefore, it is very important to understand the induced vorticity profile to understand its effects on combustion.

In general, there is no universal threshold to fully identify vorticity. Several criteria have been developed to identify vorticity within turbulent flow. These include $Q$, $\lambda_2$ and $\Delta$ criteria. Compared to others, $Q$-criterion is used to identify vortices within the flow regions by using the positive second invariant of $\nabla v$, as follows [70]:

$$Q = \frac{1}{2} \left( \|\Omega\|^2 - \|S\|^2 \right)$$  \hspace{1cm} (58)

Where

$$\|\Omega\| = tr\left[\Omega\Omega^T\right]^{1/2}$$  \hspace{1cm} (59)

$$\|S\| = tr\left[S S^T\right]^{1/2}$$  \hspace{1cm} (60)

$$\Omega = \frac{1}{2} \left( \nabla v - (\nabla v)^T \right)$$  \hspace{1cm} (61)

$$S = \frac{1}{2} \left( \nabla v + (\nabla v)^T \right)$$  \hspace{1cm} (62)

The $Q$-criterion is a local measure of the excess rotation rate relative to the strain rate [70]. It provides an approach to visualize vortices by extracting specific information from the flow field.
Fig. 24. Snapshot of the iso-surface based on Q-criterion \((Q = 10000 \text{ s}^2)\) at flow time of 500 ms

Fig. 24 shows a snapshot of the iso-surface based on the Q-criterion at \(10000 \text{ s}^2\) at flow time of 500 ms. It can be seen that there is a greater number of vortices with high intensity \((Q = 10000 \text{ s}^2)\) downstream from the swirler when \(SN\) is set at 1.4 than when \(SN\) is 1.0, especially in the upper layer of the downstream region. According to the velocity and turbulent kinetic energy profiles, the values of velocity and turbulent kinetic energy for \(SN\) of 1.4 are greater than for \(SN\) of 1.0. This implies that the intensity of the vortices for \(SN\) of 1.4 is greater than for \(SN\) of 1.0. As shown in Fig. 24, the distribution
of vortices with high intensity is wider when SN is 1.4. Furthermore, the vortices in the downstream appear to have greater curvature when SN is set at 1.4. This is a reflection of the reverse flow present within the region of interest. The direction and intensity of vortices also have an effect the distribution of fuel droplets. This will be discussed in the next section.

Fig. 25(a)-(d) shows a snapshot of the iso-surface based on the Q-criterion at 10000 s$^{-2}$ when SN is 1.4 at flow time of 300 ms to 600 ms. It shows that the vortices with high intensity develop on the downstream with time. It also shows that the vortices in the downstream region appear to have greater curvature at different flow times.
Fig. 25. Snapshot of the iso-surface based on Q-criterion (Q = 10000 s$^{-2}$) when SN is 1.4 at flow time of (a) 300 ms, (b) 400 ms, (c) 500 ms, and (d) 600 ms.
4.3. Fuel droplet distribution in combustion chamber

4.3.1. Validation of fuel droplet distribution in combustion chamber

In this study, a combustion chamber where liquid hydrocarbon fuel is introduced via a fuel nozzle has been chosen for simulation and analysis. In convectional arrangements, the liquid fuel exits the injector in the form of a thin sheet and then breaks up into droplets. The distribution of droplets depends on the fuel properties, type of atomizer, and the turbulence of the surrounding gas phase.

In order to validate the numerical algorithm described in Chapters 2 and 3 for fuel droplet distribution with swirl promoted flow, an experimental data set based on the fuel droplet distribution of kerosene by Jones et al. [66] was used for validation purposes.

The experimental setup used by Jones et al. [66] is shown in Fig. 26. It consists of a guiding pipe, a swirl generator, a pressure jet atomizer, and a sudden expansion chamber. The inlet bulk velocity is 15.6 m/s. The swirling flow is generated by the swirler with 20 equally spaced vanes, and the discharge or inclination angle set at 30° with respect to the axis.

The fuel injector is located in an annular duct 50 mm downstream of the front plate of the swirler, and the diameter of atomizer is 0.25 mm. The fuel flow rate is 0.000951 g/s and the air-to-fuel ratio is 27.88. The kerosene droplets are injected into the chamber at a spray angle of 74°.
A computational mesh for the combustion chamber used by Jones et al. [66] was generated by ANSYS ICEM CFD following the same approach as explained above using a combination mesh, as shown in Fig 27. The numerical solution was obtained using ANSYS Fluent 15.0. The turbulent flow simulation procedure is the same as described in section 4.1. The Discrete Phase model was used to generate the fuel droplet when the LES model was applied. The Stochastic and TAB models were used to take into account the dispersion and breakup of droplets. For the computational solution methods, the PRESTO scheme was used for taking into account and solving pressure-velocity coupling, using a second order upwind scheme. Spatial discretization of pressure, momentum, energy, and species were undertaken as prescribed by the algorithm. The time interval was $10^{-6}$ s, and simulation data were collected for $20\times10^{-3}$ s.
Fig. 27. Computational mesh for droplet spray validation

Fig. 28 shows the simulation and experimental results of droplet size-velocity correlation at the crossing surface $z$ of 50 mm in the combustion chamber based on Jones et al. study [66]. The shaded region represents the area where all the experimental data points of Jones et al. [66] can be found. The simulation results, which were obtained following the steps outlined above, show a good agreement with the experimental data. Therefore, the numerical approach can be applied to simulate the behavior of fuel droplets in a combustion chamber.
4.3.2. **Emulsified canola oil droplet distribution within swirl-promoted turbulent cold flow in combustion chamber**

In order to understand the combustion process and emissions of emulsified canola oil, the effects of swirl-promoted turbulent flow on fuel droplet distribution needs to be studied carefully. In this section, the numerical results for the effects of swirl number and equivalence ratio ($\phi$) on droplet distribution are analyzed by taking into account droplet size, spatial and velocity distribution in the combustion chamber, respectively.

For the numerical studies, the simulation domain is shown in Fig. 2. The flow condition has been shown in Table 1 with $\phi$ of 0.81 and 0.97 on $SN$ of 1.0 and 1.4, respectively. The thermophysical properties of emulsified canola oil are shown in Table...
3. The computational mesh as shown in Fig. 16, was generated by ANSYS ICEM CFD. The numerical approach for simulation is the same approach as explained above.

Fig. 29(a)-(c) shows fuel droplets spatial distribution for $SN$ of 1.4 and $\phi$ of 0.81 at time of 40 ms to 60 ms, respectively. It can be observed that the shape of droplets distribution at different time is similar. When time increases, the droplets become more dispersed downstream in the combustion chamber. The distribution of fuel droplets highly depends on the distribution of vortices, which is also observed by Jones et al. [39]. In Fig. 24 and 25, the distribution of vortices show an asymmetrical profile in the downstream side, which leads to the fuel droplets spatial distribution in Fig. 29.
Fig. 29. Fuel droplets spatial distribution for $SN$ of 1.4 and $\phi$ of 0.81 at time of (a) 40 ms, (b) 50 ms, (c) 60 ms
As explained in Fig. 20, the vortices breakdown on the downstream side of the combustion chamber. According to the study of P. Billant et al. [71], the instabilities of breakdown may result from the streamwise shear stress (Kelvin-Helmholtz-like instabilities) or from centrifugal instabilities due to the azimuthal motion, or more probably from a combination of both. Kelvin-Helmholtz-like instabilities induce roll-up of the helix in the streamwise direction, thereby leading to a disorganization of the whole pattern farther downstream [71]. Moreover, since the swirl-promoted flow in the combustion chamber is rotating, it is also subjected to centrifugal instabilities. This azimuthal motion can be viewed as a loss of symmetry. Based on the Rayleigh’s criterion, centrifugal flow can become unstable if $1 + C_\theta < 0$, where $C_\theta = \frac{du_\theta}{u_\theta} \frac{dr}{d\theta}$.

Therefore, centrifugal flows are expected to become unstable when the tangential velocity decreases with respect to the radius of rotation [72]. As shown in Fig. 17(b), the tangential velocity profile shows this trend in the outside region of the core area of the combustion chamber. Therefore, the combination or competition between Kelvin–Helmholtz-like instability and the generalized centrifugal instability leads to the asymmetrical distribution of vortices and the corresponding droplets distribution.
Fig. 30. Fuel droplets spatial distribution on \( SN \) of 1.0 in 60 ms (a) \( \phi \) of 0.81, (b) \( \phi \) of 0.97

Fig. 30(a) and (b) shows the numerical results of fuel droplet spatial distribution for \( \phi \) of 0.81 and 0.97 for \( SN \) of 1.0 after 60 ms without combustion, respectively. When the droplet particles were injected out of nozzle, the shape of the spray shows a cone-like distribution in the upstream side of the chamber. Then, the fuel droplets tend to
accumulate in the upper layer on the downstream side of the chamber. According to Table 1, the equivalence ratio is based on air and fuel flow rates, but since the fuel flow rate is fixed, lower equivalence ratios represent higher air flow rate. Therefore, lower equivalence ratios imply greater turbulence within the combustion chamber. Due to the greater intensity of turbulence when equivalence ratio is 0.81, more fuel droplets accumulated in the upper layer of chamber at that equivalence ratio. In Fig. 30, the scale illustrates the axial location of fuel droplet. By comparing the value of axial location in Fig. 30(a) and (b), it can be found that the effects of equivalence ratio on droplet distribution in the streamwise direction is not significant when SN is 1.0.

Fig. 31 shows the fuel droplet spatial distribution for $\phi$ of 0.81 and 0.97 for SN of 1.4 after 60 ms without combustion. The effects of equivalence ratio on droplet spatial distribution is more significant when SN is 1.4. Due to the greater angular momentum at $\phi$ of 0.81, the sprays in the streamwise direction is shorter than when the equivalence ratio is set at 0.97, and most of the fuel droplets tend to accumulate in the upper layer of the chamber. Moreover, the effects of swirl number on fuel droplets spatial distribution can be studied by comparing Fig. 30(a) and Fig. 31(a). In Fig. 31(a), the maximum scale is 460 mm, but the maximum scale is 510 mm as shown in Fig. 30(a). This implies that the droplets are less dispersed in the streamwise directions when SN is 1.4. In section 4.2.3, it is concluded that the fluid flow when SN is set at 1.4 generates more reverse flow from downstream side to upstream side. Hence, droplets in the downstream side have a higher probability of getting sucked back into the upstream side, which makes the spatial distribution of fuel droplets more concentrated at SN of 1.4.
Fig. 31. Fuel droplets spatial distribution on $SN$ of $1.4$ in $60$ ms (a) $\phi$ of $0.81$, (b) $\phi$ of $0.97$. 
Fig. 32. Histogram of droplet size distribution on $\phi$ of 0.81 and 0.97 when (a) swirl number is 1.0, (b) swirl number is 1.4
Fig. 32(a) and (b) display the histograms of droplets size distribution for $\phi$ of 0.81 and 0.97 when $SN$ of 1.0 and 1.4, respectively. Fig. 32(a) and (b) show that the air flow at $\phi$ of 0.81 is capable of generating smaller size of fuel droplets when $SN$ is 1.0 and 1.4, but the effect of equivalence ratio on droplets size distribution is not significant. As shown in Fig. 32(a) and (b), the size of droplets for $SN$ of 1.4 is smaller than that on $SN$ of 1.0, and the distribution of droplets size for $SN$ of 1.4 is more concentrated than that for $SN$ of 1.0. According to the discussion in the section 4.2.3, the intensity of turbulence for $SN$ of 1.4 is greater than for $SN$ of 1.0. Hence, the forces which act on the fuel droplet are greater when $SN$ is 1.4 than the forces when $SN$ is 1.0. The greater forces, specifically drag force, facilitate the breakup of big droplets into small droplets easily, which leads to a finer droplet size distribution at $SN$ of 1.4.

Fig. 33(a) and (b) illustrate the time-averaged droplet axial velocity profiles for $\phi$ of 0.81 and 0.97 for $SN$ of 1.0 at axial locations of 100 mm, 300 mm, and 500 mm, respectively. The axial locations have been shown in Fig. 30. Fig. 33(a) and (b) show that the droplet axial velocity for an equivalence ratio of 0.81 is not as stable as for a $\phi$ of 0.97. The high flow rate of air at $\phi$ of 0.81 leads to variations in the axial velocity of fuel droplets due to greater intensity of turbulence flow as seen in Fig. 33(a). Moreover, the droplets distribution in the spanwise direction is wider for $\phi$ of 0.81 than that for an equivalence ratio of 0.97.
Fig. 3. Time-averaged droplet axial velocity profiles of SN of 1.0 at three axial locations for (a) $\phi$ of 0.81, (b) $\phi$ of 0.97
Fig. 34. Time-averaged droplet radial velocity profiles of SN of 1.0 at three axial locations on (a) $\phi$ of 0.81, (b) $\phi$ of 0.97.

Fig. 34(a) and (b) display the time-averaged droplet radial velocity profiles for $\phi$ of 0.81 and 0.97 at a SN of 1.0 at three axial locations, respectively. It shows that the droplets at a $\phi$ of 0.81 exhibit much higher radial velocity in the middle of combustion.
chamber, and the distribution of droplet is wider when equivalence ratio is 0.81. This suggests that at higher flow rate, there is a tendency for droplets to flow radially, which lead to a stronger reverse flow within the combustion chamber.

Fig. 35 and 36 show the time-averaged droplet axial and radial velocity profiles for SN of 1.0 and 1.4 at a $\phi$ of 0.81 at three axial locations of 100 mm, 300 mm, and 400 mm, respectively. The axial locations have been shown in Fig. 30 and 31. As shown in the figures, the time-averaged axial and radial velocity of droplets at SN of 1.4 is smaller than that for SN of 1.0, especially in the radial direction. This can be explained by the fact that the droplet sizes at SN of 1.4 are smaller than at SN of 1.0. Conceptually, small droplets lose their momentum more rapidly than larger droplets in a turbulence flow field [65], which should lead to low droplet velocity at SN of 1.4. Furthermore, fuel droplets at SN of 1.4 shows a stronger tendency to accumulate in the upper layer on the downstream side of the chamber, which can be observed at the axial location of 400 mm. Also, the velocity fluctuation at SN of 1.4 is greater than that at SN of 1.0.
Fig. 35. Time-averaged droplet axial velocity profiles of $\phi$ of 0.81 at three axial locations on (a) $SN$ of 1.0, (b) $SN$ of 1.4
**Fig. 36.** Time-averaged droplet radial velocity profiles of $\phi$ of 0.81 at three axial locations on (a) $SN$ of 1.0, (b) $SN$ of 1.4
5. FLOW AND COMBUSTION OF SWIRL-PROMOTED FLOW OF EMULSIFIED CANOLA OIL USING LES-BASED NUMERICAL SCHEME

When combustion takes place in a combustion chamber, the chemical reactions will generate different types of emissions, including CO and NO\textsubscript{x}. Furthermore, the chemical reactions lead to high velocity of combustion products, which affects the profile of turbulent flow field significantly. Therefore, the structure of the turbulent flow field and the corresponding vortices profile during the combustion should be analyzed to understand the effects of both physical mechanisms on the generation and distribution of emissions within the combustion chamber. In this chapter, the chosen numerical approach for combustion simulation was validated using experimental results from Jones et al. [47]. The turbulence kinetic energy and vortices profiles during the combustion process are discussed by considering SN of 1.0 and 1.4 and the corresponding \( \phi \) of 0.81 and 0.97. The profiles of temperature and emissions are discussed by taking into account the effects of turbulent vortex flow.

5.1. Validation of numerically-based combustion scheme

In this study, the experimental work by Jones et al. [43] was selected to validate the numerical algorithm used to simulate liquid fuel combustion. The simulation results are compared with experimental results in terms of temperature and CO\textsubscript{2} profiles at a crossing plane with a fixed axial location.
The domain for the numerical approach is shown in Fig. 37, which is the same experimental setup used for the validation of droplet distribution, as described in the previous chapter. The fuel flow rate of kerosene was set at 0.000951 g/s and the equivalence ratio was 0.64. The inlet air temperature in combustion chamber was maintained at 1000 K.

Fig. 37. Jones et al. experimental setup [43]

The computational mesh for the domain is shown in Fig. 38, which is based on the combination meshing technique explained in Chapter 3. The solution was solved using ANSYS Fluent 15.0. The numerical algorithms for turbulent flow and fuel droplet injection are the same as described in section 4.1 and 4.3, respectively. The mixture was ignited using a constant power of 0.1 W at the center of nozzle outlet for $5 \times 10^{-5}$ s (0.05 ms), which combined with the initial combustion chamber temperature of 1000 K ensures complete ignition of the fuel mixture. The eddy dissipation model was used to
take into account the interaction between the chemical reaction of kerosene and the turbulent air flow. The global kinetic reaction mechanism was applied to simulate the chemical reaction in the combustion chamber. The chemical reaction is as follows:

\[
\text{C}_{12}\text{H}_{23} + 17.75\text{O}_2 \Rightarrow 12\text{CO}_2 + 11.5\text{H}_2\text{O}
\]

The pre-exponential constant \( A \) is \( 2.587 \times 10^9 \), and the activation energy \( E \) is \( 1.257 \times 10^5 \) J/kg-mole [62], which are used in the combustion simulation as explained in Chapter 3.

Spatial discretization of pressure, momentum, energy, and species were undertaken as prescribed in Chapter 3. The interval of each time step was set at \( 10^{-6} \) s (1 \( \mu \)s), and the simulation data were collected within \( 200 \times 10^{-3} \) s (200 ms).

\[\text{Fig. 38. Computational mesh used for combustion validation}\]
Fig. 39 and 40 show the time-averaged temperature and $CO_2$ profiles of the simulation and experimental results at the axial location of 200 mm in the combustion chamber, respectively. In the experiments of Jones et al. [43], heat losses led to lower temperature profiles when compared with the simulation results, as shown in Fig. 39. Furthermore, Jones et al. [43] used a 4-step chemical reaction mechanism, while the current study was based on a global (1-step) chemical reaction mechanism. The simplified approach contributes somewhat to the difference between the simulation results of Jones et al. [43] with the current study, as shown in Fig.40. However, the simulation results show a reasonable agreement with the trend of the experimental and simulation data of Jones et al. [43]. Therefore, the numerical approach described above can be applied to simulate the combustion process of biofuels in the described combustion chamber.

**Fig. 39.** Comparison of simulation results and experimental data [43] of the time-averaged temperature at an axial location of $z/R=2.0$
In order to study the emission characteristics of emulsified canola oil, understanding the effects of swirl number and equivalence ratio on the profile of turbulent vortices and temperature, is necessary establish relationships between operating parameters and emission generation. In this section, simulated swirl-promoted turbulent flow during the combustion is discussed and analyzed in terms of the frequency spectrum of pressure oscillations, turbulent kinetic energy, and the shape of recirculation zone. The temperature profile in the combustion chamber was studied by comparing it with the distribution of vortices with high intensity. The profiles of emissions for CO and thermal NO$_x$ were analyzed by comparing their concentration to corresponding temperature profiles. The simulated overall concentrations of CO and NO$_x$ were compared with Bhimani’s [4] experimental data as well.
The time interval for the simulation was set at $1 \times 10^{-6}$ s (1 μs). Simulation data were collected for about 3 flow-through times ($60 \times 10^{-3}$ s or 60 ms) to obtain statistically meaningful turbulent properties. The numerical models used for combustion simulation are shown in Table 5 in Chapter 3, and have been validated as explain above.

5.2.1. Swirl-promoted turbulent flow during combustion

Due to the chemical reaction between air and emulsified canola oil, a high temperature zone is produced by the flame, which leads to intensified turbulent flow field within the combustion chamber. Therefore, it is important to understand the effects of swirl number and equivalent ratio on the turbulent flow field during the combustion process. In this section, the effects of swirl number and equivalent ratio on the turbulent flow field with combustion are discussed in terms of frequency spectrum of pressure oscillations, the profile of turbulent kinetic energy, and the shape of recirculation zone in the combustion chamber.

According to the statistical approach used to describe turbulence, the frequency spectrum of pressure oscillation can be used to estimate the size of vortices in the flow field. The high value of frequency indicates small size of vortex, and vice versa [25]. In order to obtain the distribution of turbulent eddies during the combustion process, four simulation probes were used to register pressure oscillation within the simulated combustion chamber. In the discussion of cold flow, a corner recirculation flow was observed, hence the simulation probe, $P1$ at the axial location ($z/H$) of 0.01 was selected in the corner of the combustion chamber. Furthermore, the most intense turbulent flow is
produced near the outlet region of swiler. Therefore, two simulation probes, namely $P2$ and $P3$ were selected within this region at axial locations ($z/H$) of 0.38 and 1. In order to understand the effects of chemical reaction on pressure oscillation, $P4$ near the center line of the combustion chamber was also selected at an axial location ($z/H$) of 4. The locations of four probes are shown in Fig. 41.

![Simulation pressure probes within the simulation domain](image)

**Fig. 41.** Simulation pressure probes within the simulation domain

The collected data in time domain form has been converted to the frequency domain using by Fast Fourier Transform (FFT). Fig 42 (a-d) and (e-h) show the frequency spectrum for $\phi$ of 0.81 and 0.97 at a $SN$ of 1.0, respectively. Fig 42(a-d) and (e-h) show the frequency spectrum for $SN$ values of 1.0 and 1.4 at $\phi$ of 0.81, respectively. As shown in Fig. 42 and 43, the locations of peak values (i.e. frequencies) for all the four probe points are the same, however, the peak values (i.e. amplitudes) are different. It can be concluded that the turbulent structure or turbulent mode is dominated by the specific conditions given by swirl number and equivalence ratio. The imposed conditions affect the turbulent structure significantly, especially when considering different swirl number.
Fig. 42(a-d) and (e-h) show the effect of equivalence ratio on the size distribution of vortices. In Fig. 43(a-d), the highest peak or amplitude can be observed when the frequency is very small. This peak represents the helical vortex generated by swirler, which indicates that the swirl-promoted flow is prevalent in the flow structure within an axial distance \((z/H)\) range from 0 to 4.0. When the frequency is between 0 and \(4 \times 10^4\) Hz, several peaks can be found, and there is no peak when the frequency is larger than \(4 \times 10^4\) Hz. This means that the flow for \(SN\) of 1.0 and \(\phi\) of 0.81 is dominated by large size vortices. In Fig. 42(e-h), the low-frequency peak can be observed as well, but the value of peak is lower than the value seen in Fig. 42(a-d), which indicates that the magnitude of helical vortex generated by the flow for \(\phi\) of 0.97 is smaller than the flow for \(\phi\) of 0.81. Moreover, two peaks can be found when the frequency is larger than \(4 \times 10^4\) Hz, and the value of those peaks are much larger than the peaks between 0 and \(4 \times 10^4\) Hz. This indicates that the flow for \(SN\) of 1.0 and \(\phi\) of 0.97 is able to produce more small size vortices in the combustion chamber.
Fig. 42. Frequency spectrum of pressure oscillation at locations of $P1$, $P2$, $P3$, and $P4$ for $SN$ of 1.0 at (a)–(d) $\phi$ of 0.81, (e)–(h) $\phi$ of 0.97
Fig. 43(a-d) and (e-h) shows the effect of swirl number on the size distribution of vortices. By comparing the value of the low-frequency peak in Fig. 43(a-d) and (e-h), the peak value for $SN$ of 1.4 in Fig. 43(e-h) is much greater than the value for $SN$ of 1.0 in Fig. 43(a-d), which means that the intensity of helical flow at high swirl number is much greater. In Fig. 43(e-h), two peaks can be observed clearly, including the low-frequency peak and the peak near the $4 \times 10^4$ Hz. However, it is hard to identify another dominant peaks in Fig. 43(a-d) except the low-frequency peak. As discussed in Chapter 4, there are two types of vortex flow in the combustion chamber, including the helical flow and the wake flow, which is generated by the swirler and vane, respectively. Therefore, it can be concluded that the low-frequency peak is induced by the helical flow as generated by swirler, and peak near the $4 \times 10^4$ Hz is induced by wake flow caused by the vanes. When the $SN$ is 1.4, the turbulent flow is dominated by these two types of flow, and leads to a strong center of recirculation zone. When $SN$ is 1.0, the two types of vortex flow in the combustion chamber are close to each other as shown in Fig. 21(a), which leads to a strong interaction between the helical and wake flows. This process is able to generate different sizes of vortices and reduces the size and intensity of the center of the recirculation zone.
Fig. 43. Frequency spectrum of pressure oscillation at locations of $P1$, $P2$, $P3$, and $P4$ for $\phi$ of 0.81 at (a)-(d) $SN$ of 1.0, (e)-(h) $SN$ of 1.4
Fig. 44 shows the axial locations of three crossing plane, including $z/H=0.38$, $z/H=1$, and $z/H=4$, respectively. The three locations are important in the analysis of combustion and turbulent flow. Fig. 45(a) and (b) shows the mean temperature profile when the mean temperature is set at 2000 K for SN of 1.0 and 1.4, respectively, which represents the location of the flame within the combustion chamber. As Fig 45 shows, swirl number has a direct effect on the extent of the flame within the combustion chamber. For SN of 1.0, the flame can extend to a $z/H$ of 6, while for SN of 1.4, it only extends to $z/H$ of 3.25.
Fig. 45. The mean temperature profile at 2000 K for (a) SN of 1.0, (b) SN of 1.4

When the fuel droplets are injected into the combustion chamber, the fuel droplets respond to the imposed high temperature of the combustion chamber by
absorbing energy, which leads to partial to full evaporation of the droplets. As a result, a rich fuel/air mixture is produced in the vicinity of the nozzle [47]. This process leads to combustion, where a flame is generated within the center of chamber. Fig. 46 shows the mean temperature profile for $SN$ of 1.0 and $\phi$ of 0.81 at the crossing plane ($z/H$) of 4. In Fig. 46, the highest temperature is in the center of chamber. According to the study by Menon and Sankaran [73], the turbulence fluctuation peak is also near the center line of the combustion chamber, where heat is released due to combustion reactions taking place. Hence, it significantly contributes to the generation of turbulent kinetic energy at the center of the chamber.

![Graph showing mean temperature profile](image)

**Fig. 46.** Mean temperature profile for $SN$ of 1.0 and $\phi$ of 0.81 at the crossing plane of $z/H=4$
Fig. 47(a) and (b) show the time-averaged normalized turbulent kinetic energy (TKE) for $\phi$ of 0.81 and 0.97 for $SN$ of 1.0 at three axial locations, respectively. As explained above, the peak value of TKE can be observed in the center line of the combustion chamber. Due to the great concentration of fuel droplets and the reaction between air and the fuel, the time-averaged normalized TKE increases in the region of the nozzle outlet, which leads to a maximum value at a $z/H$ of 1. When the combustion spreads downstream of the chamber, the fuel droplets and reaction products lose their initial mass and momentum. Therefore, the time-averaged normalized TKE decreases at a location of $z/H$ of 4.

By comparing the Fig. 18 and Fig. 47 (a), it is obvious that the combustion of the fuel leads to much greater total kinetic energy under the same flow conditions (i.e. flowrate). Fig. 47(a) and (b) shows that the intensity of turbulence for $\phi$ of 0.97 is greater than that for $\phi$ of 0.81 in the vicinity of nozzle outlet ($0 < z/H < 1.0$); however, the turbulent intensity for $\phi$ of 0.97 reduces rapidly in the downstream direction. It should be kept in mind that the flowrate of air associated with $\phi$ of 0.81 is 18% greater than when $\phi$ is 0.97, so a decay in the turbulence intensity at lower air flowrate is expected. Nevertheless, the adiabatic flame temperature within the nozzle outlet region ($0 < z/H < 1.0$) is greater for $\phi$ of 0.97 than for $\phi$ of 0.87. This is consistent with previous findings that show that the adiabatic flame temperature reaches its greatest value near stoichiometric conditions, which leads to higher turbulent intensity [74]. As discussed in Chapter 4, the angular momentum is able to make turbulent flow stronger and wider.
Fig. 47. Time-averaged normalized turbulent kinetic energy (TKE) for SN of 1.0 at three axial locations at (a) $\phi$ of 0.81, (b) $\phi$ of 0.97
Fig. 48. Time-averaged normalized turbulent kinetic energy (TKE) for $\phi$ of 0.81 at three axial locations when (a) $SN$ is 1.0, (b) $SN$ is 1.4

Fig. 48(a) and (b) show the time-averaged normalized turbulent kinetic energy (TKE) for $SN$ of 1.0 and 1.4 when $\phi$ is 0.81 at three axial locations, respectively. In the flame region (between $z/H=0.38$ and $z/H=1$), the TKE values for $SN$ of 1.4 are generally higher than those at $SN$ of 1.0. At a location of $z/H=0.38$, it is evident that TKE is considerably greater at $SN = 1.4$. On the other hand, the difference between TKE values at $z/H=0.38$ and $z/H=1.0$ for $SN$ of 1.4 is much smaller than that for $SN$ of 1.0. This
implies that the turbulence flow field is relatively stable within a certain axial distance on the upstream side when the SN is set at 1.4, which in turn is able to enhance the stability of flame. At the location of z/H=4, TKE for SN of 1.4 is very small, which is consistent with Fig. 45, since it is outside of the region of the flame.

Fig. 49. Center recirculation zones for SN of 1.0 (a) when \( \phi = 0.81 \), (b) when \( \phi = 0.97 \)

Fig. 49(a) and (b) show the shape of center of the recirculation zones using iso-surface of time-averaged axial velocity \( \langle u_z \rangle = 0 \) for \( \phi = 0.81 \) and 0.97 at SN of 1.0, respectively. By comparing Fig. 49 with Fig. 19, it is evident that the center recirculation zones (CRZ) with combustion are not completely enclosed by the iso-surfaces seen in Fig. 19. The shape of the recirculation zones on the upstream side is similar to the shape in the cold flow case. However, the recirculation zone on the downstream side is not enclosed, since the shape of the zone is affected by the interaction between the swirl-
promoted air flow and the flame itself. This can explain the shift of the distribution of CRZ in the spanwise direction, as seen in Fig. 49. However, when the swirl number is kept the same, the effect of equivalence ratio on the shape of the recirculation zone is not significant.

Fig. 50 (a) and (b) illustrate the shape of the center recirculation zones using iso-surface of time-averaged axial velocity ($\langle u_z \rangle = 0$) for SN of 1.0 and 1.4, when $\phi$ is 0.81, respectively. It shows that the center recirculation zone at SN of 1.4 is wider and longer than that at SN of 1.0. It implies that the fluid flow at SN of 1.4 has a stronger recirculation zone when combustion takes place.

Fig. 50. Center recirculation zones for $\phi$ of 0.81 (a) when SN is 1.0, (b) when SN is 1.4
5.2.2. Effects of swirl number and equivalence ratio on combustion characteristics

In this section, the effects of swirl number and equivalence ratio on the profiles of temperature, $CO$, and $NO_x$ emissions in the combustion chamber are discussed. The relationship between the turbulent flow and the combustion characteristics are discussed and analyzed.

Fig. 51. Mean temperature profile for SN of 1.0 when (a) $\phi$ is 0.81, (b) $\phi$ is 0.97
Fig. 51(a) and (b) show the mean temperature profile for $SN$ of 1.0 when $\phi$ is 0.81 and 0.97, respectively. When the liquid fuel droplets are injected into the combustion chamber at a temperature of $1000$ K, the droplets start to evaporate and a fuel vapor is created. The air surrounds the droplets spray, and mix with the fuel vapor which is at the front of the spray. When air-to-fuel ratio is nearly the stoichiometric ratio at the chamber temperature of $1000$ K, the flame is generated. In Fig. 51, the temperature profile for $\phi$ of 0.97 shows to be a more concentrated distribution when compared with the temperature profile for $\phi$ of 0.81. As explained above, the swirler is able to create a center recirculation zone. The reverse flow present in the CRZ helps to deliver oxygen to the flame, which enhances the combustion inside the fuel spray. This leads to the dispersion of the temperature profile at $\phi$ of 0.81. However, the effect of equivalence ratio on flame length is not significant when the swirl number is the same.

Fig. 52(a) and (b) illustrates the mean temperature profiles for $\phi$ of 0.81 when $SN$ is 1.0 and 1.4, respectively. It can be observed that the length of the flame for $SN$ of 1.4 is much shorter than that for $SN$ of 1.0. In Fig. 50, a wider and longer recirculation zone is generated by the flow when $SN$ is 1.4 when compared with the flow at a $SN$ of 1.0. As the figure shows, higher swirl number leads to greater confinement of the flame region caused by the recirculation zone. It is well know that the recirculation zone constitutes a well-mixed region, where vigorous heat and mass transfers take place. Furthermore, it helps to recirculate fresh combustible mixture of air and fuel [75] from the downstream region. Therefore, the combustion at $SN$ of 1.4 is more intense, which leads to the formation of a concentrated high temperature region as shown in Fig. 52(b).
Fig. 52. Mean temperature profile for $\phi$ of 0.81 when (a) $SN$ is 1.0, (b) $SN$ is 1.4

Fig. 53(a) and (b) show the iso-surface of mean temperature of 2000 K after 60 ms for $SN$ of 1.0 and 1.4 when $\phi$ is 0.81, respectively. It illustrates the shape of flame in the combustion chamber. As shown in Fig. 53, the flame when $SN$ is 1.4 concentrates in the upstream region of the combustion chamber. Moreover, a significant amount of reverse flow at $SN$ of 1.4 leads to a backward motion of the flame in the vicinity of nozzle ($0 < z/H < 1.0$), which is characterized by a wider flame around the nozzle. When $SN$ is 1.0, the flame region increases gradually along the downstream direction as shown in Fig. 53(a). However, this process cannot be observed in Fig. 53(b). This implies that the
distribution of fuel vapor on the upstream side ($1.0 < z/H < 4.0$) is much wider in the spanwise direction and more concentrated in the streamwise direction when $SN$ is 1.4.

![Image of iso-surface of mean temperature](image_url)

**Fig. 53.** Iso-surface of mean temperature of 2000 K at 60 ms for $\phi$ of 0.81 when (a) $SN$ is 1.0, (b) $SN$ is 1.4

Fig. 54(a) and (b) show the iso-surface of mean temperature of 2000 K combined with the surface representing a Q-criterion value of 40000 s$^{-2}$ after 60 ms at $\phi$ of 0.81 when $SN$ is 1.0 and 1.4, respectively.
Fig. 54. Snapshot of iso-surface of mean temperature of 2000 K with Q-criteria value of 40000 s$^{-2}$ at 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4

It can be observed that the vortices profiles correspond to the shape of flame somewhat, and high intensity of vortices are generated at the front of flame for both swirl numbers. However, more vortices with high intensity are produced near the outlet
region of swirler when SN is 1.4 as shown in the black boxes of Fig. 54(a) and (b), and these vortices are outside of the high temperature region as illustrated by the red surface. In Fig. 39, the greatest amplitude of helical flow, wake flow, and corner recirculation flow are observed at the location of \( P_1 (z/H=0.01) \) for SN of 1.4 as shown in Fig. 43(e). This implies that the flow when SN is 1.4 induces the greatest level of turbulence, which leads to a great evaporation of fuel droplets in the vicinity of the nozzle. In Fig. 48(b), the TKE for SN of 1.4 at the location of \( z/H=0.38 \) is much higher than that for SN of 1.0, which reflects the level of intense combustion near the nozzle region.

Fig. 55(a) and (b) show the distribution of emulsified canola oil on a mass fraction basis after 60 ms for \( \phi \) of 0.81 when SN is 1.0 and 1.4, respectively. According to the value of the scale in Fig. 55, the concentration of fuel in the vicinity of nozzle when SN is 1.4 is much higher than that when SN is 1.0, which is attributed to high level of turbulent intensity and recirculation zone of SN of 1.4 as explained above.
Fig. 5. Distribution of emulsified canola oil on a mass basis after 60 ms for \( \phi \) of 0.81 when (a) SN is 1.0, (b) SN is 1.4

Fig. 56(a) and (b) show the distribution of \( CO \) after 60 ms for \( SN \) of 1.0 when \( \phi \) is 0.83 and 1.00, respectively. Comparing with the contour in Fig. 54(a) and Fig. 56(a), it can be observed that the distribution of \( CO \) corresponds with the distributions of temperature and vortices. This implies that the generation of \( CO \) depends on the corresponding temperature and vortices profiles significantly. Due to the small effect of
equivalence ratio on temperature and vortices as discussed above, the distribution of CO for both equivalence ratios has a similar range in the streamwise and spanwise directions, as shown in Fig. 56(a) and (b). On the downstream side of the combustion chamber, the temperature profile for $\phi$ of 0.97 is less dispersed as shown in Fig. 51(b). It implies that less fresh air flows into the flame region, which leads to a richer fuel/air condition when $\phi$ is 0.97. Therefore, the combustion at $\phi$ of 0.97 generates more CO in the center of downstream side in the combustion chamber. Furthermore, Fig. 56 shows the overall CO emissions in the combustion chamber after 60 ms. As seen in Fig. 56, the combustion at $\phi$ of 0.97 produces more CO than that at $\phi$ of 0.81 due to the high equivalence ratio (less oxygen) and lower turbulent intensity.
Fig. 56. Distribution of CO after 60 ms for SN of 1.0 when (a) \( \phi \) is 0.81, (b) \( \phi \) is 0.97.

Fig. 57(a) and (b) show the distribution of CO after 60 ms for \( \phi \) of 0.83 when SN is 1.0 and 1.4, respectively. The effect of the swirl number on the distribution of CO in the combustion chamber is significant. As shown in Fig. 57(a) and (b), the distribution of CO for SN of 1.4 is much concentrated than that for SN of 1.0. Most of CO at SN of 1.4 is produced on the upstream side of the combustion chamber, and the distribution of CO is accumulated in the upper layer of the combustion chamber as shown in Fig. 57(b). Moreover, high concentration of CO is generated near the center of the combustion chamber when SN is 1.4. As shown in Fig. 58, a large area of CRZ is observed on the bottom side of the combustion chamber when SN is 1.4 as illustrated by the green
surface. The CRZ induces the backward motion of the air/fuel mixture and reaction products, which leads to a lower concentration of oxygen in this region as shown by contour in Fig. 58. Therefore, high concentration of CO is observed near the center of the combustion chamber. Furthermore, Fig. 57 shows the overall CO emissions in the combustion chamber after 60 ms. As seen in Fig. 57, the combustion process for SN of 1.0 produces more CO than that at SN of 1.4. Although there are some relatively high concentration areas of CO that can be found in the combustion chamber when SN is 1.4, the strong CRZ confines the combustion to a small region. This in part leads to small CO emissions in the combustion chamber when SN is 1.4.

**Fig. 57.** Distribution of CO after 60 ms for $\phi$ of 0.81 when (a) SN is 1.0, (b) SN is 1.4
Fig. 58. Distribution of O$_2$ and center recirculation zone after 60 ms for $\phi$ of 0.81 and SN of 1.4

Fig. 59(a) and (b) show the distribution of NO$_x$ after 60 ms for SN of 1.0 when $\phi$ is 0.81 and 0.97, respectively. It can be observed that most of the NO$_x$ emissions are produced on the downstream side of the combustion chamber for both equivalence ratios at SN of 1.0. Due to the high level of turbulent intensity when $\phi$ is 0.81, the value and region of the temperature profile in the combustion chamber for $\phi$ of 0.81 is larger than that when $\phi$ is 0.97, as seen in Fig. 51. Therefore, overall NO$_x$ emissions at $\phi$ of 0.81 are higher, as expected.
Fig. 59. Distribution of NO\textsubscript{x} after 60 ms for SN of 1.0 when (a) $\phi$ is 0.81, (b) $\phi$ is 0.97

Fig. 60(a) and (b) show the distribution of NO\textsubscript{x} after 60 ms for $\phi$ of 0.81 when SN is 1.0 and 1.4, respectively. Due to the strong effect of CRZ at SN of 1.4, the distribution of NO\textsubscript{x} emissions is located mainly on the upstream side of the combustion chamber. It indicates that the recirculation size volume is directly related to reductions in NO\textsubscript{x} levels. On the other hand, the distribution of NO\textsubscript{x} at SN of 1.0 can be found farther downstream into the chamber, as shown in Fig. 60(a). Another key observation in the size of the NO\textsubscript{x} distribution region, which is smaller when SN of 1.4.
To further understand the distribution and levels of NO\textsubscript{x} present at different swirl numbers and equivalence ratios, the distributions of NO\textsubscript{x} with respect to temperature should be evaluated. Fig. 61(a-d) show the distribution of NO\textsubscript{x} after 60 ms for SN of 1.0 and 1.4 at \(\phi\) of 0.81 and 0.97, respectively. Fig. 61(a-d) also show the iso-surface of temperature at 2400 K as depicted by the red surface. It illustrates that the regions of temperature with the highest value can be found on the downstream side of the combustion chamber as well. As explained in Chapter 3, this study considers the thermal NO\textsubscript{x} mechanism only. Therefore, the emissions of NO\textsubscript{x} are dominated by the size and value of the high temperature region in the combustion chamber, which correspond fairly well with the distribution of NO\textsubscript{x} as shown in Fig. 61(a-d).
Fig. 6. Distribution of thermal $NO_x$ after 60 ms coupled with iso-surface of temperature at 2400 K for (a) $SN$ of 1.0 and $\phi$ of 0.81, (b) $SN$ of 1.0 and $\phi$ of 0.97, (c) $SN$ of 1.4 and $\phi$ of 0.81, (d) $SN$ of 1.4 and $\phi$ of 0.97

Looking closer at the relationship between temperature and $NO_x$ emission levels, it can be observed that even though the flame temperature at $SN$ of 1.4 is a little higher than that at $SN$ of 1.0 as seen in Fig. 52, the flame still is confined in a small region by
CRZ when \( SN \) is 1.4. As the results show, \( NO_x \) emissions at \( SN \) of 1.4 are much smaller than that at \( SN \) of 1.0, which suggests that flame confinement through a strong CRZ can lead to lower \( NO_x \) level overall.

Fig. 62 shows the comparison between the simulation work and the experimental work [4] in terms of \( CO \) emissions based on the same combustion chamber. The solid and dashed lines represent the simulation work and the experimental work of Bhimani et al. [4], respectively. According to the experimental results work, there were no \( CO \) emissions when the equivalence ratio was equal or smaller than 1.0. However, the simulations show \( CO \) emissions for different stoichiometric ratio. The major reason for the difference in the level of \( CO \) emissions is due to the location of the exhaust gas analyzer. In the experiments by Bhimani et al. [4], the gas analyzer was connected to the combustion chamber at a sample port located 1.85 m below the nozzle tip, and the temperature in the combustion chamber was higher than 1000 K. These conditions allow \( CO \) enough time and energy to react with \( O_2 \), which could lead to no \( CO \) emissions flowing into the exhaust gas analyzer. However, the average droplet size considered in the simulation was about 3 times as large as the calculated value reported by Bhimani et al. [4]. The effect of emulsified canola oil droplet size on \( CO \) level emissions should be investigated in the future.
Fig. 6. Comparison between simulation work and experimental work [4] in terms of CO emissions

Fig. 6 shows the comparison between the simulation work and experimental work [4] in terms of NO\textsubscript{x} emissions based on the same combustion chamber. The solid and dashed lines represent the simulation work and the experimental work of Bhimani et al. [4], respectively. In Fig. 5, the simulation results show the same trends as the experimental work, and the simulation results are close to the experimental results when \textit{SN} is 1.0. However, the combustion simulations under-estimate the emission levels when compared with the experimental results. This is also discussed in the section of combustion validation in Chapter 5, Section 5.1.

As shown in Fig. 6, due to the lower level of turbulent intensity of \textit{SN} of 1.0, the combustion process is not as complex as the combustion at \textit{SN} of 1.4. The numerical
scheme and the simplified reaction mechanisms are able to approach the real combustion behavior. Therefore, the errors of simulations are relatively small, especially in the case of SN of 1.0 and $\phi$ of 0.97. When SN is 1.4, the greater level of turbulent intensity makes the flow field very complicated, and leads to a complex turbulent combustion process. Moreover, only thermal NOx was considered in this study. The absence of prompt NOx in the chemical mechanism may lead to a slightly significant difference from experimental results of NOx. However, the application of the numerical models and simplified reaction mechanism could lead to a slight difference between the simulated combustion process and the real combustion behavior.

![Graph](image)

**Fig. 63.** Comparison between simulation work and experimental work [4] in terms of the concentration of NOx in the combustion chamber
The effects of external boundary conditions should also be considered when comparing the numerical simulations with the experimental results. In the current study, adiabatic conditions were assumed at the wall of the combustion chamber. However, the experiments were conducted in a chamber that experienced heat losses as shown in Table 7. Furthermore, all the measured experimental emission levels were between 800 and 1000 °C [4], which are much lower than the temperatures seen in the combustion simulation. Therefore, the effect of temperature on emission generation should be studied in the detail in the future.

<table>
<thead>
<tr>
<th>Equivalence Ratio</th>
<th>Adiabatic Flame Temperature (K)</th>
<th>Experimental Furnace Temperature at z/H of 10 for SN of 1.4 (K)</th>
<th>Experimental Furnace Temperature at z/H of 10 for SN of 1.0 (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.83</td>
<td>2166</td>
<td>1218</td>
<td>1208</td>
</tr>
<tr>
<td>1.00</td>
<td>2435</td>
<td>1240</td>
<td>1227</td>
</tr>
</tbody>
</table>

In summary, the results shown in the chapter highlight the importance of swirl number on emission levels, which confirm many of the experimental observations done with emulsified canola oil as combustion chamber fuel [4].
6. CONCLUSIONS

The primary objective of this study is to understand the effects of swirl number and equivalence ratio on the turbulent flow field, fuel droplets distribution, temperature profiles and combustion emissions of emulsified canola oil in a 30 kW furnace at CBEL by using a comprehensive numerical approach. To study the combustion phenomena of emulsified canola oil, a proper numerical approach has been chosen and validated. The numerical models including LES, DPM, Stochastics model, TAB, and Eddy-Dissipation model have been used to understand the swirl promoted turbulence flow, the fuel droplets distribution, and combustion. Based on the results of this study, the major conclusions of this research work are presented below.

6.1. Large eddy simulations of swirl-promoted turbulent cold flow in combustion chamber – Conclusions

From the simulation results of swirl-promoted turbulent cold flow, it can be said that,

- The combination between RANS and LES can be used successfully to capture the fluid structure of swirl-promoted turbulence based on validated experimental results
- The swirler is able to generate a center recirculation zone in the combustion chamber.
• The swirl-promoted cold flow at high swirl number produces a greater velocity in axial, radial and tangential direction, and the effect of swirl number diminishes along the axial direction in the combustion chamber.

• The swirl-promoted cold flow at high swirl number produces intense turbulence and a wider and stronger center of recirculation zone in the combustion chamber.

• Two types of processing vortex cores are generated by swirler, including spiral flow and double-helix wake flow.

• The flow at high swirl number enhances the range and intensity of the vortices distribution, and produces a significant reverse flow in the combustion chamber.

6.2. Fuel droplet distribution in combustion chamber – Conclusions

From the simulation results of droplet distribution in the cold flow, it can be said that,

• The used of LES, DPM, Stochastics and TAB models can be used successfully to capture the droplet dispersion within the fluid structure based on validated experimental results

• The effects of equivalence ratio on droplet distribution is not significant when swirl number is the same.

• The fuel droplets in the flow with high swirl number disperses less in the streamwise direction, and the droplets on the downstream side have a higher
probability of getting sucked back into flowstream, which makes the spatial
distribution of fuel droplets more concentrated at high swirl number.

- The flow with high swirl number produces smaller sizes of droplets, and
makes the distribution of droplets size more concentrated.

6.3. Swirl-promoted turbulent combustion of emulsified canola oil – Conclusions

From the simulation results of swirl-promoted turbulent combustion of emulsified canola oil, it can be said that,

- The flow at high equivalence ratio is able to produce more small sized vortices in the combustion chamber when the swirl number is the same.
- The flow at low swirl number produces a strong interaction between the helical flow and wake flow, which leads to different sizes of vortices and reduces the size and intensity of the center recirculation zone.
- The helical flow and wake flow dominate the turbulent flow in the combustion chamber at high swirl number, which leads to a strong center recirculation zone in the combustion chamber.
- The turbulent kinetic energy in the combustion chamber depends on the temperature of chemical reaction when combustion takes place.
- The effect of equivalence ratio on the distribution of temperature is not significant at the same swirl number.
- The combustion at high swirl number produces an intense and concentrated high temperature region.
• The distribution of temperature highly depends on the distribution of fuel concentration.

• The combustion at high swirl number produces a great concentration of evaporated fuel in the vicinity of nozzle.

• The combustion under lean condition produces high turbulent intensity and greater dispersion of combustion on the downstream side when the swirl number is the same, which leads to lower CO emissions.

• The effects of swirl number on CO emissions is significant. CO emissions at high swirl number are reduced sharply due to the confinement of the combustion region by the center recirculation zone.

• The combustion under lean condition produces more NO\textsubscript{x} emissions on the downstream side of the combustion chamber due to the high turbulent intensity in that region.

• The effects of swirl number on NO\textsubscript{x} emissions is significant. NO\textsubscript{x} emissions at high swirl number are reduced due to the confinement of the combustion region by the center recirculation zone.

• The simulation results show the same trends as the experimental work. The error associated with simulations increases when higher levels of turbulence intensity is considered.
7. FUTURE WORK

This study was attempted to simulate and identify the key features of swirl-promoted turbulent combustion when using emulsified canola oil as fuel. According to the findings from the simulation results, the effects of fuel viscosity and droplets size on the combustion characteristics should be studied further to understand the complex reaction mechanism that could lead to differences between simulation and experimental results. The following recommendations should be considered for further work in the near future:

1. Due to the lack of experimental information regarding the size distribution of droplets generated by a twin-fluid atomizer, future simulations should consider a wide range of droplet size distributions, especially for droplets sizes below than 100 μm. Moreover, the applicability of direct droplet injection should be studied by comparing with an air-blasted atomizer.

2. In this study, one type of emulsified canola oil was considered. In order to understand the effect of fuel viscosity on the combustion characteristics, a conventional fuel should also be considered to understand the effects of fuel properties on combustion.

3. Since the simulation error can be considerable when the turbulent intensity is high, a complex reaction mechanism is needed to reduce the associated error between the simulation and the experiments.

4. This study found that the size of the center of the recirculation zone affects the combustion characteristics significantly. However, there is a lack of an appropriate
methodology to evaluate the size of the center of the recirculation zone. Therefore, an analytical based approach should be formulated to precisely predict the size of recirculation zone in a myriad of applications. Furthermore, different combustion chamber configurations should be considered that could enhance CRZ.
REFERENCES


[56]. S. Bhimani, Experimental characterization of canola oil emulsion combustion in a modified furnace, M.S. Thesis, Texas A&M University, College Station, 2011.


APPENDIX A

The following equation represents the stoichiometric combustion reaction equation of emulsified canola oil as shown in Table 3.

\[ \text{CH}_{1.704}\text{O}_{0.128} + a\times (\text{O}_2 + 3.76\text{N}_2) \rightarrow \text{CO}_2 + b\times (\text{H}_2\text{O}) + c\times \text{N}_2 \]

After balancing the stoichiometric combustion equation on both sides of the equation, the value of stoichiometric coefficients are as shown in Table A-1.

<table>
<thead>
<tr>
<th>Stoichiometric coefficient</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>1.362</td>
</tr>
<tr>
<td>b</td>
<td>0.852</td>
</tr>
<tr>
<td>c</td>
<td>5.12112</td>
</tr>
</tbody>
</table>

Table A-1. Stoichiometric coefficients for complete combustion of emulsified canola oil

Using the stoichiometric coefficients in Table A, the molar fraction of \( \text{CO}_2 \) for complete combustion of emulsified canola oil can be obtained as follows:

\[ \chi_{\text{CO}_2} = \frac{1}{1 + b + c} = \frac{1}{1 + 0.852 + 5.12112} = 0.14 \]

Therefore, the molar fraction of \( \text{CO}_2 \) for complete combustion of emulsified canola oil is 14%.

Fig. A-1 (a)-(d) show the simulation results of molar fraction of \( \text{CO}_2 \) at the axial location \((z/H)\) of 0.38, 1, and 4 at 60 ms for \( SN\) of 1.0 and \( \phi \) of 0.81, \( SN\) of 1.0 and \( \phi \) of 0.97, \( SN\) of 1.4 and \( \phi \) of 0.81, and \( SN\) of 1.4 and \( \phi \) of 0.97, respectively.
Fig. A-1. Simulation results of molar fraction of $CO_2$ at the axial location ($z/H$) of 0.38, 1.0, and 4.0 at 60 ms (a) $SN$ of 1.0 and $\phi$ of 0.81, (b) $SN$ of 1.0 and $\phi$ of 0.97, (c) $SN$ of 1.4 and $\phi$ of 0.81, and (d) $SN$ of 1.4 and $\phi$ of 0.97

As shown in Fig. A-1 (a)-(d), the simulation results of molar fraction of $CO_2$ are a little smaller than the value of 14%. As shown in Table 4, a two-step chemical
reaction mechanism was used for this study. It considered the reaction from $CO_2$ to $CO$, which leads to a reduction of $CO_2$ in the simulation. However, Fig. A-1 (a)-(d) show that the $CO_2$ concentration or molar fraction reaches the values corresponding to complete combustion (14%) after 60 ms of simulation time. Therefore, a simulation time of 60 ms was long enough to ensure complete combustion.