Computational Modeling and Analysis of Heavy Fuel Feasibility in Direct Injection Spark Ignition Engine

SunilUdaya Simha Moda

Wright State University

Follow this and additional works at: https://corescholar.libraries.wright.edu/etd_all

Part of the Mechanical Engineering Commons

Repository Citation
https://corescholar.libraries.wright.edu/etd_all/774

This Thesis is brought to you for free and open access by the Theses and Dissertations at CORE Scholar. It has been accepted for inclusion in Browse all Theses and Dissertations by an authorized administrator of CORE Scholar. For more information, please contact corescholar@www.libraries.wright.edu, library-corescholar@wright.edu.
COMPUTATIONAL MODELING AND ANALYSIS OF HEAVY FUEL FEASIBILITY IN DIRECT INJECTION SPARK IGNITION ENGINE

A thesis submitted in partial fulfillment
of the requirements for the degree of
Master of Science in Engineering

By

SUNIL UDAYA SIMHA MODA
B.Tech, Acharya Nagarjuna University, 2008

2011
Wright State University
I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Sunil Udaya Simha Moda ENTITLED Computational Modeling and Analysis of Heavy Fuel Feasibility In Direct Injection Spark Ignition Engine, BE ACCEPTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF Master of Science in Engineering

Committee on Final Examination

__________________________  ___________________________
Haibo Dong, Ph.D.           Haibo Dong, Ph.D.
Thesis Director

__________________________  ___________________________
Greg Minickweicz, Ph.D.     George Huang, Ph.D.
Chair, Department of Mechanical and Materials Engineering

__________________________
Hui Wan, Ph.D.

__________________________
Andrew T. Hsu, Ph.D.
Dean, School of Graduate Studies
ABSTRACT


Direct Injection spark ignition (DISI) technology is helpful for the present day engine to increase the fuel efficiency. Power output and the choice of fuel as the demand and scarcity for fossil fuels are increasing. As a new technology DISI engines are employed in some commercial cars like the Pontiac Solstice using gasoline fuel. The advantages of DISI engines like use of different fuels, reduced compression ratio, reduced injection pressure and reduced operating pressures in DISI engines has not been widely tested. As a new technology DISI engines lack experimental results and combustion co-relations that can be directly used as in the case of conventional engines. The experimental analysis of this technique is very expensive as it involves large number of parameters to be changed each and every time the experiment is done. This makes the experimental analysis of DISI engine a costly and time consuming task. Computational fluid dynamics on the other hand can simulate the combustion process and let researches visualize the process of combustion inside the cylinder.

The ability of DISI engine to work on different fuels in the engine is successfully tested. Two engines, reciprocating and rotary, equipped with DISI technology is simulated in FLUENT. The engines are selected in such a way that they represent the major part of engine family to show that DISI technology is feasible in any type of IC engines. Because of unavailable experimental data on DISI diesel engines, the models used in the thesis are validated with a gasoline DISI reciprocating engine. The validated model is used for parametric study of diesel fuel in DISI engine. It was found that the
engine parameters need to be tuned to avoid the undesired effects of diesel fuel. After several parametric changes, combustion and power output which is identical to the experimental validated case are obtained. Hence it has been proved that the diesel fuel can be successfully utilized in DISI engine. This technology is then applied to the rotary engine. Because of the change in geometry and many other specifications, the parameters used for reciprocating engine are not feasible for rotary engine. Therefore a parametric study on rotary engine is carried out to obtain a good combustion and power output.

It is proven successfully in this thesis that DISI technology can be applied to any engine and can use any kind of fuel. However, each and every engine needs to be tuned according to its specifications and geometrical constrains to obtain the maximum fuel to air mixture and therefore the maximum power output. The thesis explains the influence of parameters on the power output considering the important properties of fuel such as cold start ability, flash point detonation, volatility and density. The behavior of fuel and flow physics inside the cylinder is visually explained. The fuel air interaction, which is very important to have a good air fuel mixture formation, is extensively studied and the methods are developed to time the injector depending on the air turbulence inside the cylinder. The conclusions in this thesis demand the importance of further studies of this technology. The results of the thesis show that this technology can be used as a more energy efficient and echo friendly technology. However, further studies on this technology are essential to build a flawless more efficient technology in the field of IC engines.
# TABLE OF CONTENTS

CHAPTER 1: INTRODUCTION

1.1 Direct Injection Engine ................................................... 1
1.2 Why Direct Injection Engine ............................................. 3
1.3 Fuel injection in direct injection engine .............................. 5
1.4 Computational Fluid dynamics for DISI engine .................... 5
1.5 Literature review ....................................................... 6
1.6 Combustion and equivalence ratio .................................... 12
1.7 Motivation for the Thesis ............................................. 13
1.8 Research Objectives .................................................... 14
1.9 Thesis Outline ......................................................... 14

CHAPTER 2 MODELING AND MESHING ........................................ 16

2.1 Reciprocating engine ................................................. 17
2.1.1 Modeling and Meshing ........................................... 17
2.1.2 Dynamic mesh motion .......................................... 20
2.2 Rotary engine ........................................................ 21
2.2.1 Modeling and Meshing .......................................... 21
2.2.2 Dynamic mesh motion .......................................... 22

CHAPTER 3 NUMERICAL METHODS ........................................... 26

3.1 Turbulence Modeling ................................................ 26
3.1.1 The spalart- Allamaras model .................................. 26
3.1.2 Standard K-ε model ............................................. 27
3.1.3 RNG K-ε model ................................................................. 27
3.1.4 Realizable K-ε model ...................................................... 28
3.2 Combustion models ............................................................ 28
3.3 Fuel Injection parameters .................................................. 29
3.4 Solver settings ................................................................... 30

CHAPTER 4 EXPERIMENTAL VALIDATION AND DIESEL FUEL TEST ...... 31
4.1 Validation ............................................................................ 31
  4.1.1 Geometric validation .................................................... 31
  4.1.2 Validation of boundary conditions and parameters .......... 33
  4.1.3 Validation of output ...................................................... 35
4.2 Study of Validation case ..................................................... 36
  4.2.1 Air flow in the chamber ............................................... 36
  4.2.2. Fuel air interaction and vaporization ............................. 38
  4.2.3. Fuel geometry interaction ......................................... 41
  4.2.4. Excess fuel .............................................................. 43
  4.2.5 Combustion ............................................................... 44
4.3 Diesel fuel in validation case .............................................. 46
  4.3.1. Air flow in the chamber and Excess fuel ...................... 47
  4.3.2. Fuel air interaction and vaporization ......................... 47
  4.3.3. Fuel geometry interaction ....................................... 49
  4.3.4. Combustion ............................................................ 49

CHAPTER 5 PARAMETRIC STUDY OF DIESEL FUEL .................... 52
  5.1 Parametric study on reciprocating engine with diesel fuel ....... 52
5.1.1. Injection velocity .......................................................... 53
5.1.2. Injection time ............................................................... 55
5.1.3. Particle size ................................................................. 57
5.1.4. Multiple plume injection ............................................... 59
  5.1.4.1 Injection modeling ................................................... 59
  5.1.4.2 Parametric study ....................................................... 60
5.1.5. Conclusions from parametric study ............................... 64
5.2 Results ............................................................................... 64
5.3 DISI diesel fuel Rotary engine ........................................... 67
  5.3.1 Modeling and Meshing .................................................. 67
  5.3.2 Cold flow study ............................................................ 68
    5.3.2.1. Turbulence of air in the chamber ............................ 68
    5.3.2.2. Maximum temperature and pressure at TDC ............ 70
    5.3.2.3. Injection location and orientation of injection .......... 71
5.4 Simulations of Rotary engine with diesel fuel DISI technology ... 72
  5.4.1 DISIR_M ................................................................. 73
    5.4.1.1. Air fuel interaction and fuel vaporization .......... 74
    5.4.1.2. Combustion ................................................. 77
    5.4.1.3. Power output .................................................. 77
  5.4.2 Comparison between DISIR_M, DISIR_MP and DISIR_MPL ... 78
    5.4.2.1. Fuel distribution ............................................. 79
    5.4.2.2. Combustion .................................................. 80
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>17</td>
</tr>
<tr>
<td>2.2</td>
<td>18</td>
</tr>
<tr>
<td>2.2(a)</td>
<td>19</td>
</tr>
<tr>
<td>2.3</td>
<td>20</td>
</tr>
<tr>
<td>2.4</td>
<td>23</td>
</tr>
<tr>
<td>2.5</td>
<td>24</td>
</tr>
<tr>
<td>4.1</td>
<td>32</td>
</tr>
<tr>
<td>4.2</td>
<td>33</td>
</tr>
<tr>
<td>4.3</td>
<td>33,34</td>
</tr>
<tr>
<td>4.4</td>
<td>35</td>
</tr>
<tr>
<td>4.5</td>
<td>37</td>
</tr>
<tr>
<td>4.6</td>
<td>38</td>
</tr>
<tr>
<td>4.7</td>
<td>40</td>
</tr>
<tr>
<td>4.8</td>
<td>41</td>
</tr>
<tr>
<td>Section</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>4.9</td>
<td>Wall Wetting due to left over fuel in the cylinder</td>
</tr>
<tr>
<td>4.10</td>
<td>Left over fuel in the cylinder with available oxygen</td>
</tr>
<tr>
<td>4.11</td>
<td>Combustion process in the validation simulation</td>
</tr>
<tr>
<td>4.12</td>
<td>Difference in vaporization of diesel and Gasoline</td>
</tr>
<tr>
<td>4.13</td>
<td>Difference in fuel air interaction for diesel and gasoline</td>
</tr>
<tr>
<td>4.14</td>
<td>Diesel fuel concentration on the piston showing the wall wetting…</td>
</tr>
<tr>
<td>4.15</td>
<td>Detonation in the case of diesel</td>
</tr>
<tr>
<td>4.16</td>
<td>Pressures of diesel and gasoline simulation</td>
</tr>
<tr>
<td>5.1</td>
<td>Parameters used for studying effect of injection velocity</td>
</tr>
<tr>
<td>5.2</td>
<td>Mass fraction of CO2 at the end of power stroke</td>
</tr>
<tr>
<td>5.3</td>
<td>Combustion process of the reciprocating engine using diesel</td>
</tr>
<tr>
<td>5.4</td>
<td>Difference in particle vaporization</td>
</tr>
<tr>
<td>5.5</td>
<td>Difference in vaporization with decrease in particle size</td>
</tr>
<tr>
<td>5.6</td>
<td>Dual plume and four plume injection models</td>
</tr>
<tr>
<td>5.7</td>
<td>Parameters used in parametric study of multiple plume injectors…..</td>
</tr>
<tr>
<td>5.8</td>
<td>Pictures showing the simulation of single orifice injector</td>
</tr>
<tr>
<td>Section</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>5.9</td>
<td>Pictures showing the simulation case 2</td>
</tr>
<tr>
<td>5.10</td>
<td>Pictures showing the simulation case 3</td>
</tr>
<tr>
<td>5.11</td>
<td>Production of CO2 during combustion cycle</td>
</tr>
<tr>
<td>5.12</td>
<td>Pressure history for the three cases</td>
</tr>
<tr>
<td>5.13</td>
<td>Pictures showing the disappearance of turbulence at 90 bTDC</td>
</tr>
<tr>
<td>5.14</td>
<td>Maximum pressure and temperature at TDC due to compression</td>
</tr>
<tr>
<td>5.15</td>
<td>Fuel injected opposite to the direction of flow</td>
</tr>
<tr>
<td>5.16</td>
<td>Fuel air interaction</td>
</tr>
<tr>
<td>5.17</td>
<td>Vaporization of fuel</td>
</tr>
<tr>
<td>5.18</td>
<td>Combustion in DISI diesel rotary engine with one injector</td>
</tr>
<tr>
<td>5.19</td>
<td>Power produced with one main injector</td>
</tr>
<tr>
<td>5.20</td>
<td>Difference between the cases with and without pilot injection</td>
</tr>
<tr>
<td>5.21</td>
<td>Fuel distribution with and without lead injector</td>
</tr>
<tr>
<td>5.22</td>
<td>Difference in combustion</td>
</tr>
<tr>
<td>5.23</td>
<td>Power outputs for the simulation in the study</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

I would like to express my sincere and profound appreciation to my honorific advisor, Dr. Haibo Dong, who was abundantly helpful and offered invaluable assistance, support and guidance with his experience and knowledge. Without his motivation, encouragement and immense patience, this thesis would not have been completed or written. I am thankful to my parents for all their support throughout my life. I am also greatly thankful to Dr Hui Wan for his assistance, generosity and advice throughout this study.

I would like to thank all the professors who taught me. To complete my course work this laid strong fundamentals for the completion of my thesis work.

I am also thankful to Dr. Greg Minikiewicz for spending his valuable time to review my thesis and participate as defense committee member. He was instrumental in starting the rotary engine effort and getting Wright State University involved with direct injection modeling.

I am thankful for my group “FLOW SIMULATION RESEARCH GROUP”, AFRL, LKI and ISSI for funding me. I would like to thank Mr. Zongxiang Liang for his user defined function for running rotary engine simulations. I would like to thank Asela A Benthara for his help to me in teaching FLUENT and GAMBIT and also the preliminary knowledge on simulations. I would like to thank Brad Pollock, Tim Smith, Brain Nicholson and William McCuddy for their help in my research.
### NOMENCLATURE

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Full Form</th>
</tr>
</thead>
<tbody>
<tr>
<td>PFI</td>
<td>port-fuel-injected</td>
</tr>
<tr>
<td>CIDI</td>
<td>compression ignition direct injection</td>
</tr>
<tr>
<td>SI</td>
<td>Spark ignition</td>
</tr>
<tr>
<td>CI</td>
<td>Compression ignition</td>
</tr>
<tr>
<td>BSFC</td>
<td>brake-specific fuel consumption</td>
</tr>
<tr>
<td>DISI</td>
<td>Direct Injection Engine spark ignition</td>
</tr>
<tr>
<td>Φ</td>
<td>Equivalence ratio</td>
</tr>
<tr>
<td>UBHC</td>
<td>Un-burnt hydro carbons</td>
</tr>
<tr>
<td>GDI</td>
<td>Gasoline Direct Injection</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>HCCI</td>
<td>Homogeneous charge compression ignition</td>
</tr>
<tr>
<td>SCCI</td>
<td>Stratified charge Compression ignition</td>
</tr>
<tr>
<td>CO2</td>
<td>Carbon-di-oxide</td>
</tr>
<tr>
<td>C_{10}H_{22}</td>
<td>Diesel</td>
</tr>
<tr>
<td>C_{8}H_{18}</td>
<td>Gasoline</td>
</tr>
<tr>
<td>MDM</td>
<td>Moving Dynamic mesh</td>
</tr>
<tr>
<td>bTDC</td>
<td>Before top dead centre</td>
</tr>
<tr>
<td>aTDC</td>
<td>After top dead centre</td>
</tr>
<tr>
<td>TDC</td>
<td>Top dead centre</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Direct Injection Engine

Direct Injection Engine is the new trend in the internal combustion engines. It is a type of engine in which the fuel is injected into the combustion chamber and is spark ignited. Over the decades the port-fuel-injected (PFI) spark ignition gasoline engines and the compression ignition direct injection (CIDI) diesel engines served the automotive field. Because of the increasing scarcity of conventional fuels (gasoline and diesel) there is an increasing emphasis to achieve fuel economy. The CIDI engines have more fuel economy when compared to PFI engines due to high compression ratio coupled with un-throttled operation. CIDI engines on the other side have some disadvantages over SI engines like higher noise, cold start problems, lower speed range, and larger emission of NO\textsubscript{x} and soot formation. Research is going on to combine the best features of CI and SI engines which can give brake-specific fuel consumption (BSFC) approaching CI engine maintaining the good qualities of SI engine like silent operation, high speed and lower pollutant emissions. A solution for the above requirement is Direct Injection spark ignition engine.

In this type of engine fuel is injected into the cylinder directly with the help of high pressure injectors. A homogeneous mixture is forced to form inside the cylinder with an ignitable composition of fuel-air mixture near the spark plug at the time of spark. This type of engine can vary power output by varying the amount of fuel injected into the chamber like in CIDI diesel engine hence saving fuel by controlling the power output as needed. It is un-throttled hence it saves the work done to maintain the throttle operation.
in SI engines. It has spark plug so the fuel injected inside the cylinder does not need to have the inherent auto-ignition characteristics like the fuels used in CIDI engines. This feature allows the use of different varieties of fuel including the bio-fuels in the cylinder. DISI engines also have the advantage of working under ultra lean conditions.

Direct injection spark ignition engines can run in three different modes based on the necessity of power. The modes are classified depending upon the equivalence ratio of the mixture (ϕ). Mixtures with ϕ more than 1 are called rich mixtures, less than 1 are called lean mixtures and with 1 are called stoichiometric mixtures. Rich mixture has more fuel than required to consume all the oxygen available and for lean mixture it is a vice versa.

Injection in the DISI is a very critical operation. The mixture formation inside the cylinder in DISI is the major factor to have a good combustion. The air-fuel mixture formation is dependent on swirl of air from the intake port, geometry of the pocket and the spray properties. As the air swirl and spray properties are limited by the geometry the next most important factor to get a good air fuel mixture is injection. Efficient high pressure fuel injections which have different spray patterns are being manufactured. Research is going on to optimize the spray pattern and an injector to output a most optimal spray and also have maximum spread in minimum amount of time. Technologies like multi orifice injection are being implemented in the engines to increase the mixture properties in the chamber. The computational and experimental studies on injection velocity, angle of injection, multiple orifice injections are being conducted to minimize the experimental costs and give the experimental validation a good start.
1.2 Why Direct Injection Engine

Direct injection engine is a hybrid engine built with the hallmarks of PFI and CIDI engines. It combines the high power output with lower fuel consumption and less soot and NOx formation. Injecting fuel directly into the engine cylinder totally avoids the problem of wall wetting in the port which is seen in PFI engines. In PFI engines the fuel is injected in the valve when the inlet port is closed. This causes a transient film, or puddle, of liquid in the intake valve area of the port. This causes fuel delivery delay and an associated metering error due to partial vaporization, making it necessary to use more fuel that needed during cold start. It often causes the problems of miss fire and partial burn and emission of UBHC for the first 10 strokes in a PFI engine. The DI engines eliminate this problem in the other hand by directly injecting fuel into the chamber. These engines have more control of the metered fuel for each combustion event, as well as a reduction in the fuel transport time. The actual mass of fuel entering the chamber can be controlled better in DI engines than in PFI engines.

Because of the high operating pressure in the DI engines the fuel sprayed into chamber is much more atomized when compared to PFI so the chances of wall wetting is much more reduced and DI engines can be operated at ultra lean burn when much minimum power output is needed. DI engines also prevent the UBHC emissions during cold start and the engine transient response is enhanced. PFI needs at least 10 cycles to obtain a steady state oscillatory film i.e., to start any engine the fuel for 10 cycles is wasted to overcome the cold flow where as the DI engines need two cycles to overcome the cold start[1]. The higher pressure used in GDI injection systems as compared to PFI fuel systems increase both the fuel atomization and the fuel vaporization rate which helps
the DI engine to achieve stable combustion from first or second injection cycle without supplying excess fuel. The theoretical advantages of DI engine over PFI engine are listed as follows [1].

- **Improved fuel economy** (up to 25% potential improvement), depending on test cycle resulting from:
  - Less pumping loss (un-throttled, stratified mode);
  - Less heat losses (un-throttled, stratified mode);
  - Higher compression ratio (charge cooling with injection during induction);
  - Lower octane requirement (charge cooling with injection during induction);
  - Increased volumetric efficiency (charge cooling with injection during induction);
  - Fuel cutoff during vehicle deceleration (no manifold film).

- **Improved transient response.**
  - Less acceleration-enrichment required (no manifold film).

- **More precise air–fuel ratio control.**
  - more rapid starting;
  - Less cold-start over-fueling required.

- **Extended EGR tolerance limit** (to minimize the use of throttling).

- **Selective emissions advantages.**
  - Reduced cold-start UBHC emissions;
  - Reduced CO2 emissions.

- **Enhanced potential for system optimization.**

Because of all these advantages DI engines is very important technology for the present day engines. The combined effect of high efficiency and low fuel consumption make
these engines a very important trend in the present day automotive industry.

1.3 Fuel injection in direct injection engine

In DI engines the fuel is directly injected into the chamber and is mixed with air by virtue of injection properties and the air swirl inside the chamber. Fuel injection is critical in a DI engine. Many researchers are working on various fuel injection mechanisms to optimize the atomization, avoid wall wetting and to obtain a maximum possible homogeneous mixture in the combustion chamber. Fuel injection in DI engine is a high pressure fuel injection system which typically operates in the range from 4 to 13Mpa. This high pressure injection gives the better atomized and vaporized fuel. As the fuel is directly injected into the chamber the fuel systems are generally capable of injection of wide variety of fuels like gasoline, diesel, bio-fuels and even natural gas.

1.4 Computational Fluid Dynamics for DISI Engines

Combustion is a fast and complicated process. It is fast and hard to utilize. CFD provides an option to visualize the combustion computationally. The parametric study on the engine to increase the performance of the engine is cost and time effectiveness when done in computational domain than experimental as it takes much more time and cost to do a parametric study. It also gives an information about where to put the fuel injectors inside the cylinder and it gives information about the spray angle to be used. DISI diesel fuel technology is new and it is hard to find references or examples to do any further study, CFD analysis of DISI will provide references of any study involving much less cost and time. It will provide an initial idea of how things work before investing lots of time and money into experimental analysis.
CFD analysis also helps to understand the flow pattern inside the cylinder which is important to decide some of the important factors of a good combustion like injection location, injection timing and Injection orientation.

1.5 Literature Review

Spark Ignited Direct Injection (SIDI) engine has been a subject of research over decades because of the importance of fuel economy it is expected to have the ability to use any type of fuel and fuel mixtures. A literature review is done to get enough exposure on spray pattern modeling and the current trends available. This information was very useful in thesis work which showed me where to start and the type of results I should expect. The details of the literature review are presented below.

A complete study of direct injection engine is done by F.Zhao [1]. His paper gives all the information on direct injection engine such as how it is used as a technology in present day automobiles. It briefly explains all the concepts involved in direct injection and it gives a good overview of the technology.

The combined study of properties of spray and mixture formation based on flash boiling is done in this paper[2]. A new fuel is designed using diesel and a fuel with low flash point such as gasoline or gas fuel to obtain a new mixture which enhances the mixture formation as well as decreases the flash point.

Johan et al[3] in their publication described modeling of stratified combustion in a DISI engine. This paper gives an idea of how a combustion model in CFD works and how to calculate laminar flame speed. They published both HCCI and SCCI models which gives us the significant difference between them and an idea of how to get actual
combustion close to HCCI using SCCI.

Derek Bradley et al[4] explained the method to determine the auto ignition delay times for non-primary reference fuels and primary reference fuels. The information given in the paper is very useful for simulations using both kinds of fuels. Some experimental data are published which can be directly used in simulations.

Semin et al[5] studied multiple hole injection. They used 5 simulations in cosmos to simulate one through five holes of injections. They concluded that multiple hole injections are better than single orifice injection. Their study is more on mixture formation in port injector but this concept can be used in DISI engines as well.

Wakisaka et al[6] in their paper described the numerical analysis of cone sprays. They developed a droplet break up model which uses Rietz model. They studied the droplet behavior on the wall impingement and droplet behavior inside the chamber. Using these models they analyzed the mixture behavior in the chamber.

Werner Hentschel[7] in his paper gave a detailed visualized explanation of how the internal cylinder pressures effect the spray pattern. He also showed the variation of spray with change in injection rail pressure. This paper is a good visual reference for simulations. He also explained the difference between spray guided, air guided and wall guided DISI engines.

Choi et al[8] in their paper developed a CFD model for stratified combustion. This paper explains the effect of mixture formation on secondary burn flame and laminar flame velocity. They used a 3D model to achieve more realistic results.

Drake et al[9] performed experimental, computational and visualization study on spray patterns in SIGI engines. They found out that the spray spark plug interactions are
more important than of spray wall interactions. They explained how the spray causes the electrode wetting and decrease in combustion.

Zhao.H et al[10] in their study used different blends of gasoline with hydrogen. They found out that the combustion stability increased with increase in hydrogen. They also noted that hydrogen also increases the speeds of combustion due to high burning velocity. This paper is good example for multiple fuel injection strategies.

Wang.Z et al[11] did visualization study on DISI engines. They mentioned that the DISI process can be divided into three stages. Spark induction stage, flame propagation stage and compression ignition stage. They also mentioned that the high pressures are good for good combustion.

Wang.J et al[12] in their study described that with the increase of hydrogen in the hydrogen fuel mixture the combustion is more stabilized and it increases the rate of combustion. They did a visual study of how combustion varies with increase in concentration of hydrogen in fuel hydrogen mixture.

Lee et al[13] best described the effect of parameters like cone angle, ambient pressure and common rail pressure in the spray pattern. They emphasized more on spray penetration changes with change in the parameters. This can be used as a reference for spray guided DISI engines where penetration of spray is more important.

Kim et al[14] in their paper proved that the air swirl from the intake port can aid the formation of homogenous mixture in a combustion chamber than the tumble motion given by motion of the piston. In other words they say that air guided DISI engines have more homogeneous mixture than wall guided DISI engine. They also discussed the effects of parameters like pressure and temperature on fuel distribution.
Canacki et al[15] did experimental analysis on optimization of HCCI DISI engine using split injectors. This experimental study is an addition to the field of multi hole injection technology. They included parameters such as spark timing, amount of fuel injected, mass flow rate of fuel and split injection parameters for their experimental study. This experimental study is a complete study of spray guided direct injection engines.

The importance of using fuel mixtures in the direct injection engine and the capability of DISI technology to use mixed fuels is demonstrated by sato et al[16]. The issue of fuel causing rapid combustion in combustion chamber is taken into consideration and the use of mixed fuels with one less energy fuel and another regular fuel. They were able to get results for lean burn cases using appropriate mixture of methane and n-butane.

A visual spray pattern analysis using spectroscopic, imaging and pressure based analysis is done by Fansler et al[17]. In their paper they did imaging of spray distribution inside the chamber. They mentioned some of the important results about the effect of pocket and air swirl on the fuel distribution inside the chamber. They also mentioned the importance of rich mixture near the spark plug during ignition. The ideal equivalence ratio at spark during spark is specified as 1.5.

Papageorgakis and Dennis in their paper [18] mentioned about RNG based K-epsilon model to optimize the gaseous air fuel mixture inside a computational fluid dynamics model. The K-epsilon model is used in FLUENT commercial software so this paper provides a brief overview of the k-epsilon model. They also explained how the model effects the mixture conditions this is very important as the mixture preparation is
of more significance in DISI engines. They explored parameters like glow plug presence, direction of injection, injection timing, number of holes in the injector and initial swirl.

Ra et al[19] developed a droplet vaporization model in an attempt to solve the problem of vaporization of droplets numerically. It’s a very important factor and a challenge in present day numerical simulation method to differentiate between vapor and liquid in the continuum. Significance of multi-component model, ambient temperature and pressure, surface temperature on the effects of vaporization is best described in this paper.

Christoph Garth et al [20] explained the effects of tumble and swirl motion of air inside the combustion chamber. They explained the method to extract the data of swirl and tumble motion from a simulation. The visualization technique described in this paper is very helpful to actually show the swirl and tumble in the model. They mentioned that recirculation zones are important to decide the injection orientation.

Kim et al [21] in their paper conducted research on ignition delay times of dimethyl ether and diesel. They explained how the ambient temperature and pressure effects the ignition delay time of fuel and they visually proved their results. This study is important to know the characteristics of fuel in the combustion chamber and to estimate the detonation time and hence tune the engine to use the detonation in expansion stroke.

Keiya Nishida et al [22] in their paper on “An experimental and numerical study on sprays injected from two-hole nozzles for DISI engines” gave an interesting study on the importance of multi orifice injection. Two injectors with one and two orifice respectively are compared. They demonstrated the importance of two orifice nozzles. The merits of
two orifice nozzle or injector such as quick vaporization, higher injection penetration, lower real cone angles and reduced pressure requirement are demonstrated in this paper.

Ryoji Kwaga and others from Mazda motor corp. [23] in their paper on “A study direct injection stratification charge in rotary engine” gave every detail of DISI technology in rotary engine. They demonstrated the importance of ignition chamber, upstream injection and pilot injector. The utilization of air charge for fuel air mixture formation is well explained and the injection time for main injector and pilot injector to form a good air fuel ratio is specified. The paper mentions the importance of time required for the fuel to get vaporized, the fuel needs to be given time to vaporize and that time is derived from the selection of injection position and injection orientation. The paper demonstrates how enough time is achieved for injection using up stream injection. It also gives an idea of wall wetting due to wrong injection timing when the experimental downstream injection is done. This is a very good reference paper for diesel DISI rotary engine study.

Ted Pipe [24] is a book which explains the basic concepts of rotary engine. It explains the theory behind the combustion and the methods to calculate power output torque and other theoretical details of rotary engine.

J. Abraham et al [25] in their paper studied the modified geometries to improve combustion in stratified-charge rotary engines. The different rotor pocket shapes have been tested and respective conclusions have been drawn. The effect of geometry on the fuel distribution is demonstrated using different pocket shapes. The surface heating and vaporization is demonstrated. The paper states that the geometry of the pocket should be modeled in such a way that it increases the distance of travel of injected fuel stream and increase the contact area of the rotor face with the fuel so that vaporization is quickly done.
Chao Wang, Asela Anuruddika, Zongxian Liang and Haibo Dong, Computational Analysis of a Rotary Engine, Wright State University, Final report. [26] This paper explains every aspect of simulation using 2D geometry. It explains the modeling and meshing, performing simulations and calculating power output. The parametric study is done in this report which is very useful guide for the simulations carried out in this study.

T. I. P. Shih. [27], a numerical simulation of the flow field in a 2D wankel engine has been carried out. The main objective of this study was to numerically investigate effects of certain engine parameters on the fluid flow and fuel-air mixing inside the combustion chamber during the intake and compression cycles. It has been concluded that engine speed and the speed of the fuel jet have a significant effect on the fluid flow.

Muroki et al [28] in their paper derived the relations between flame propagation and combustion chamber shape. This paper studies the influences of the recess shape and spark plug location on flame propagation. J.Abraham and F.V.Bracco, [29], investigated different ignition strategies to improve combustion in a rotary engine. Multiple ignition sources have been used to achieve better combustion. The number of orifices in the fuel injectors have also been changed and tested. The paper shows that the increase in number of plumes in the injector vaporizes the fuel faster and it is very suitable for rotary engine as the time available for rotary engine is much less. The paper also explains the suitable position of spark and importance of multiple sparks in a rotary engine.

1.6 Combustion equation and Equivalence ratio

**Chemical Equation for the combustion**

The chemical equation for the combustion is:

\[ C_{10}H_{22} + 15.5 \ O_2 \ \rightarrow \ 10 \ CO_2 + 11 \ H_2O \]

This is the equation which guides the combustion for diesel.
From this equation we can calculate the stoichiometric ratio for mass of fuel and mass of oxygen which is very important to calculate the equivalence ratio.

**Stoichiometric ratio for mass of fuel to mass of oxygen:**

Mass of fuel = mass number for $\text{C}_{10}\text{H}_{22}$

Mass of Oxygen = $15.5 \times$ mass number for $\text{O}_2$

Stoichiometric ratio for mass of fuel to mass of oxygen = 0.286

**Equivalence ratio**

$$\phi = \frac{\frac{\text{fuel to oxidizer ratio}}{\text{fuel to oxidizer ratio}_{\text{st}}}}{\frac{\text{mass of fuel}}{\text{mass of air}_{\text{st}}}}$$

Using these equations equivalence ratio is calculated for supplied amount of fuel.

There are two kinds of equivalence ratio in this study:

1. Local equivalence ratio --- Just mentioned as equivalence ratio in the report
2. Total equivalence ratio --- Mentioned as total equivalence ratio.

Note: Unless specified equivalence ratio in the report refers to Local equivalence ratio.

1.7 Motivation for the Thesis

Reduction in availability of fossil fuels is increasing the demand for more fuel efficient and alternative technologies. The DOD wants to use single battle space fuel in the engines as it is much easier to store and it is easily available. The transformation of engines from fossil fuels to alternative fuels like bio fuels cannot be achieved over night. So there is a necessity to develop an engine which currently uses fossil fuel with much more efficiency and has an efficiency to use alternative fuels such as bio fuels. Such kind
of engine will be useful to make the inevitable transformation longer and smoother. DISI technology is the technology which can successfully handle the above specified task. As a new technology DISI does not have any prior practical data so a detailed computational analysis should be done on DISI engines to prove that it can be used in any kind of engine with any type of fuel. Computational analysis is cost effective and is the best choice when there is not much experimental data available. The above explanation motivates the computational analysis of DISI technology in reciprocating and rotary engine.

1.8 Research Objectives

Computational analysis of DISI reciprocating and rotary engine is one of the most advanced research area. As the new engine computational analysis of DISI engine is very important to give a starting point to the experimental setup. The computational analysis helps in optimization of DISI engine performance. The research objective in this study is to perform computational fluid dynamics analysis on DISI diesel operated reciprocating and rotary engine to demonstrate the efficiency of DISI engine and also the capability of DISI technology to use any kind of fuel in an engine.

1.9 Thesis Outline

In this study two types of engines reciprocating and rotary are considered. Both the engines are modeled and meshed to use in simulations. These engine models are equivalent computational models for Pontiac Solstice engine and Wankel rotary engine. These two engines represent major kind of engines in present day IC engine world. The models are validated with the available experimental results of DISI gasoline reciprocating engine to make sure that the models used produce good results. The
validated reciprocating engine model is then subjected to diesel fuel to check the performance of the model with new fuel. A parametric study using diesel fuel on DISI reciprocating engine is carried out to obtain the optimum power output. The results of the parametric study suggest that the DISI technology with diesel fuel can be used on reciprocating engine. The similar technique is applied to rotary engine and the parametric study on the rotary engine is done. The results show that DISI technology can be employed on both kinds of engines.
Chapter 2

This chapter includes the modeling and meshing of the computational models used in the simulation. This study involves two geometrical models:

1. Reciprocating engine
2. Rotary engine

The model of reciprocating engine is validated geometrically with a real Pontiac Solstice engine that is currently available in the market.

A Solidworks model based on the real Pontiac Solstice engine, as shown in Table 2.1 (a) and (b) was provided by ISSI for building the computational model for meshing. In this work, GAMBIT 2.4 has been used for modeling and meshing the complex fluid volumes. In order to make the high-quality meshes that represent the engine geometry, dynamic layering and local remeshing techniques have been utilized.

In details, the Solidworks model was imported into GAMBIT as shown in Table 2.2 (a), and refined to allow for meshing. Here, unstructured meshes will be used to mesh the working volumes. New faces or volumes may need to be created. Each face and volume is meshed and refined until the max cell skewness is under acceptable limits. Boundary conditions and continuum types are added before the mesh is exported. Table 2.3 shows the summary of the computational Solstice engine model. Deforming and rigid zones are listed, too. Since the piston is moving all the time during the stroke, Dynamic mesh model is used to handle the moving parts so that when the parts move into a new position at each time step of the calculation, the grid is automatically rebuilt. In this work, the mesh motion is through the setting of dynamics mesh and events in FLUENT. A completed dynamic mesh motion for four strokes is shown in Table 2.4. The quality of
the mesh is given primary importance as it is directly related to the results. Dynamic mesh rebuilt the new mesh based on the quality of static mesh. The shape of mesh is tetrahedron because of its accuracy.

2.1 Reciprocating engine

2.1.1 Modeling and Meshing

The Solidworks model was imported into GAMBIT. As it is not ready to mesh because of the complications in the geometry, the Solidworks model is refined to a much simpler geometry. The whole geometry is divided into suitable volumes and volume meshing is applied to the different volumes. The complex volume is split into many volumes depending on the boundary conditions the volumes are going to take and the expected work done by the volume in the simulation and each volume is meshed independently. The quality of mesh is maintained in all the volumes so that it does not affect the quality of complete mesh when joined together. The method of meshing is briefly explained with figures in the following few pages.
Solid works model imported to Gambit.

The different volumes are meshed separately and then joined together to get one complete structure.

Completed mesh in GAMBIT

Computational Solsticce engine model

Table 2.2 Completely meshed Solstice engine model
Table 2.2 shows the mesh file imported from GAMBIT. To obtain dynamic mesh motion the behavior of each and every mesh should be predefined. This part is partially done in GAMBIT, where we specify the names of each mesh and specify the property of the mesh.

![Diagram of mesh with labels:]

This is a cylinder Wall and it is stationary
This is a pressure inlet and it is deforming
This is piston which is a surface and it is deforming

<table>
<thead>
<tr>
<th>Mesh Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total No. of Elements: 115761</td>
</tr>
<tr>
<td>Maximum Skewness: 0.901531</td>
</tr>
<tr>
<td>Element: Tet / Hybrid</td>
</tr>
<tr>
<td>Type: TGrid</td>
</tr>
</tbody>
</table>

Each and every layer of the mesh is defined, named and exported to FLUENT. Dynamic mesh motion is set up in FLUENT as show in Table 2.2(a). Every portion of the mesh is pre given the motion conditions to perform. The stationary mesh remains
stationary all over the simulation and the dynamic mesh changes the mesh and also generated new mesh if necessary.

2.1.2 Dynamic mesh

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>(b)</td>
</tr>
<tr>
<td>TDC</td>
<td>190 bTDC</td>
</tr>
<tr>
<td>(c)</td>
<td>(d)</td>
</tr>
<tr>
<td>120 bTDC</td>
<td>TDC</td>
</tr>
<tr>
<td>(e)</td>
<td></td>
</tr>
<tr>
<td>TDC</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.3 reciprocating engine dynamic mesh motion
Table 2.3 gives the pictures of mesh motion in a reciprocating engine. Table 2.3 (a) shows the starting position of the mesh motion. (b) Shows the mesh at 190 bTDC. The value motion can be observed; the exhaust port is closed and the intake port is open. Table 2.3 (c) shows the values are closed in the compression stoke. Table 2.3(d) shows the TDC position of the engine at combustion while figure (e) shows the TDC position after exhaust.

All the motion aspects of the simulation are captured in the mesh motion which makes this mesh ready to use for simulations. The rotary engine is also modeled in a similar way. Mesh motion is developed in the rotary engine by way of a user defined function to calculate the shift of rotor as a result of the eccentricity of the shaft. The next section explains the modeling and meshing of rotary engine.

### 2.2 Rotary engine

#### 2.2.1 Modeling and Meshing

A Solidworks model is taken as the starting point of modeling the simulation. The Solidworks model was imported in the modeling and meshing software GAMBIT to create a mesh which can be used in FLUENT for simulations. The model from Solidworks is subjected to geometric cleanup to make the model and meshing as simple as possible. This helps to decrease the computational time. Figure (a) shown in Table 2.3 is the Solidworks model after completion of geometry cleanup in GAMBIT. After the geometry clean up is done, the whole model is meshed as shown in Table 2.4. The fluid properties, surface names and continuum types are assigned to the model in the next step. The maximum skewness of the mesh is decreased as much as possible maintaining the mesh size. A cell skewness of 0.901 and the mesh size ‘3’ is obtained. This is the best
combination of skewness and mesh size obtained without reducing the size (hence the calculation duration) and maintaining the skewness. The final mesh is exported which is ready for simulations.

The mesh in the rotary engine needs to be very fine and not skewed. This is because, when compared to reciprocating engine the mesh is not generated in rotary engine during dynamic mesh motion because of the complexity of mesh motion. So the elements in the mesh are subjected to high amount of skew.

2.2.2 Dynamic Mesh Motion

The mesh is exported from GAMBIT as a complete mesh with assigned surface names, Boundary conditions and continuum conditions. Before a simulation is performed on the mesh a successful rotor motion is to be achieved. FLUENT supports piston motion readily but in order to get a rotor motion a “cycloid” user defined function has to be used.

The model is divided into two divisions the housing and the rotor. A relative motion between rotor and housing is established using a user defined function. The apex in the rotor that enables smooth rotation of rotor inside the housing was developed in GAMBIT with a plate connecting rotor and housing. This increases the reality of the model and also helps to avoid any possible leaks in-between the sections of the engine. The user defined function enables the rotor motion for any crank angle. The required crank angle provided in FLUENT control panel stops the rotor motion. Using the rotor and user defined function, the mesh motion is achieved which is an important part for simulation. Some of the positions of mesh motions are captured and presented in the Table 2.5
Table 2.4 Modeling and meshing of rotary engine

Mesh Properties

Total No. of Elements: 115761

Maximum Skewness: 0.901531

Element: Tet / Hybrid

Type: TGrid
Table 2.5 Dynamic mesh motion of Rotary engine

<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>340 bTDC</td>
<td>(b)</td>
<td>270 bTDC</td>
<td></td>
</tr>
<tr>
<td>(c)</td>
<td>150 bTDC</td>
<td>(d)</td>
<td>110 bTDC</td>
<td></td>
</tr>
<tr>
<td>(e)</td>
<td>10 aTDC</td>
<td>(e)</td>
<td>90 aTDC</td>
<td></td>
</tr>
</tbody>
</table>
Table 2.5 explains the dynamic mesh motion of the rotary engine. The UDF controls the mesh motion and the maximum skewness is always less than ‘1’ throughout the whole simulation. These moving parts in this dynamic mesh are the rotor and housing while the stationary parts are inlet and outlet ports. This division helps to reduce the computational time as stationary mesh takes much less time to perform simulation than dynamic mesh.

The tip of the rotor is another challenge when meshing. Because it is a very small geometry, care need to be taken when meshing it so that it does not have a negative cell volume. The tetrahedral mesh used in the meshing process solves this problem. The shape of the rotor is successfully meshed including the minute geometrical details of original engine.

This section explains how the two geometries are successfully meshed and mesh motion is achieved for both the models. These are very important meshes as they are the first step in the simulation. The next step is a cold flow study of these meshes. A cold flow study explains the flow physics in the chamber so that the important factors of injection, such as injection timing, position and orientation, can be decided. As the mesh motion is achieved suitable boundary conditions are applied to these meshes.
Chapter 3

3.1 Turbulence modeling

It is well known that level of turbulence is essential in the fuel-air mixing process. For DIE operated in a homogeneous charge combustion mode (an early injection mode) where fuel is injected early in the intake stroke, the combination of high turbulence intensity and low mean velocity at the spark gap is desirable to produce a homogeneous air-fuel mixture. For DIE using stratified charge combustion mode (a late injection mode) where fuel-injection at the end of the compression stroke, a flow field with elevated mean velocity and reduced turbulence level is preferred [18]. Based on the setup of experiment, the Ecotec engine is operated in early injection mode in our case. Therefore, the focus is on homogeneous turbulence modeling in this phase.

The choice of turbulence model depends upon considerations such as

- Physics encompassed in the flow
- The established practice for a specific class of problem
- Level of accuracy required
- Available computational resources
- Amount of time available for simulation

The different types of turbulence models in FLUENT are listed in the following.

3.1.1 The Spalart-Allmaras model

This model can be used for all the Low-Reynolds-number flows. This is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy viscosity. This might be a good choice for relatively crude simulations where accurate turbulent flow computations are not critical. There are a few drawbacks such as
- It cannot be relied upon to predict the decay of homogenous, isotropic flows.
- One-equation models are often not capable of capturing rapid changes in the length scale, which might be necessary when flow changes from wall-bounded to free shear flow.

3.1.2 Standard k-ε model

This is one of the simplest two-equation turbulence model in which the solution of two transport equations determines the turbulent velocity and length scales independently. It is a semi-empirical model that is based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). The model equation for k was derived from the exact equation whereas the model equation for ε was obtained using physical reasoning. It is well-known for its robustness, economy and reasonable accuracy for a wide range of turbulent flows. The assumptions made in this model are: Flow is fully turbulent; Effects of molecular viscosity are negligible.

3.1.3 RNG k-ε model

This model is derived using a rigorous statistical technique called the renormalization group theory. It is similar in form to the standard k-ε models. The following refinements are included in RNG k-ε models

- It has an additional term in its equation for ε that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in RNG model enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard model uses user-specified, constant values.
While the standard model is a high Reynolds number model, the RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low Re-effects.

### 3.1.4 Realizable k-ε model

**Realizable k-ε model is used in our simulation[18].** The realizable k-ε model includes new formulation for the turbulent viscosity and the dissipation rate. The term *realizable* means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Realizable k-ε model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard k-ε model. Also it is likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

This model differs from the standard model in two important ways:

- The realizable model contains new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ε, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

### 3.2 Combustion Models

There are five different types of models provided in FLUENT. They are:

- (a) Species transport
- (b) Non-premixed combustion model
- (c) Premixed combustion model
- (d) Partially premixed combustion model
- (e) Composition PDF transport model

Species transport and partially premixed combustion model have been used in this research project. These models are most suitable for this type of research involving
discrete phase combustion process [10]. Species transport model, as the name suggests, involves the modeling of the mixing and transport of chemical species by solving the conservation equations involving convection, diffusion and reaction sources for each component species. Non-premixed combustion model is a model in which the fuel and air are not mixed before injection. They are separately injected and mixed inside the combustion chamber. In the premixed combustion model, the fuel and air is mixed prior to injection and then this mixture is injected into the combustion chamber. Partially-premixed combustion model is a simple combination of the Non-premixed combustion model and premixed combustion model. Non-premixed combustion model can also be used to perform the tests, but it does not provide the option of spark ignition. Combustion can be achieved using flamelet models in non-premixed combustion model.

In this work, partially premixed combustion model was successful in achieving combustion. The entire cases list below is computed by using this model.

3.3 Fuel injector parameters

- Droplet size & distribution
  - Minimum Diameter : 20 µm
  - Maximum Diameter : 50 µm
  - Mean Diameter : 40 µm
  - Number of Diameters : 10

- Spray model used
  - DPM and spray model
    - Unsteady particle tracking
    - TAB breakup model
Single injection particle

3.4 Solver setting

- Unsteady pressure-based PISO solver
- Green-Gauss cell based option for gradient
- Standard scheme for pressure, and first order upwind scheme for all other equations
Chapter 4

4.1 Validation

Before performing the computational study the model is validated using experimental results. Validation is done to verify if the model matches the experimental set up in geometry, ambient conditions and output. The reciprocating engine, which has available experimental data, is used to validate the simulation method that is being used. The results show that the computational method and the commercial software used in this study are good at handling the complicated combustion problems. So, this simulation is validated and used for further analysis.

The validation is carried out in three phases.

1. Geometric validation
2. Validation of the boundary conditions and parameters
3. Validation of the output

This three stage validation gives greater accuracy and it saves time as the second step is only carried out after validating the first step instead of doing every step all over again.

4.1.1 Geometric validation

The actual geometry of the engine is used to model the computational engine model used in simulations. This provides the first step of validation as the real engine is modeled into a computational engine which has exactly the same geometric specifications of the real engine. Table 4.1 shows the experimental design parameters used to model the computational engine model in GAMBIT. Fig (a) and (b) in the Table are the pictures of original engine. Fig (c) shows the whole computational model resembling the actual engine. Fig (d) shows how the pocket shape is restored in the
computational model which is very important for a direct injection engine. Table 4.2 show the physical parameters used to design the engine.

Table 4.1 Geometric validation of Reciprocating Engine

Fig (a)  Fig (b)  Fig (c)  Fig (d)

The data used for the construction of the model is provided in the Table 4.2 this data is useful for modeling the computational engine and is also used to input very important input values to the FLUENT software in order to define the mesh motion of the model. This data provided will be used by FLUENT for mesh generation during mesh motion. The valve lift data provided is used to define the valve motion in the valve profile used to control the valves in the computational model. The connecting rod length provided will define the length through which the mesh should generate. The
compression ratio provided is used to estimate the pressure developed inside the chamber during compression with this the extent of combustion and power output can be known.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore × Stoke</td>
<td>86.4 mm × 86.4 mm</td>
</tr>
<tr>
<td>Displacement</td>
<td>0.5 liter/cylinder</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>9.2:1</td>
</tr>
<tr>
<td>Connecting rod length</td>
<td>144.2 mm</td>
</tr>
<tr>
<td>Valve Lift at intake</td>
<td>10.3 mm</td>
</tr>
<tr>
<td>Valve Lift at exhaust</td>
<td>10.3 mm</td>
</tr>
</tbody>
</table>

Table 4.2 Geometric specifications for a computational engine model

These values are the dimensions of the real engine which are exactly used in developing the computational model. The modeling is done in GAMBIT as discussed in Chapter 2 and is double precision so the model developed is highly accurate and exactly resembles the engine in computational domain. Hence it is geometrically well validated.

4.1.2 Validation of boundary conditions and parameters

After the geometry validation parameters used for computational model are validated. It is made sure that the parameters in computational and experimental environments are exactly the same. The conditions used for experimental and computational models are tabulated in the Table 4.3

<table>
<thead>
<tr>
<th>Input Parameters</th>
<th>CFD</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fuel used</td>
<td>80 Octane</td>
<td>80 Octane</td>
</tr>
<tr>
<td>Composition of fuel</td>
<td>80%Octane and 20%Heptane</td>
<td>80%Octane and 20%Heptane</td>
</tr>
<tr>
<td>fuel flow rate</td>
<td>0.021265Kg/s</td>
<td>0.021265Kg/s</td>
</tr>
<tr>
<td>Velocity of injection</td>
<td>120 m/s</td>
<td>120 m/s</td>
</tr>
<tr>
<td>Radius of injector</td>
<td>1 mm</td>
<td></td>
</tr>
<tr>
<td>Spark</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-----------------------------------</td>
<td>-------</td>
<td>-------</td>
</tr>
<tr>
<td>Initial Fuel temperature</td>
<td>314 K</td>
<td>314 K</td>
</tr>
<tr>
<td>Density of the fuel</td>
<td>702 kg/m³</td>
<td>685 kg/m³</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Dynamic mesh set up</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spark radius</td>
<td>0.0026 m</td>
<td>0.0026 m</td>
</tr>
<tr>
<td>Spark energy</td>
<td>0.1 J</td>
<td>0.1 J</td>
</tr>
<tr>
<td>Spark start angle</td>
<td>705</td>
<td>705</td>
</tr>
<tr>
<td>Spark duration/spark dissipation</td>
<td>0.001 s/0.001 s</td>
<td>0.001 s/0.001 s</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameters</th>
<th>In Cylinder</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine Speed</td>
<td>1901 rpm</td>
<td>1901 rpm</td>
</tr>
<tr>
<td>Start Crank Angle</td>
<td>360</td>
<td>n/a</td>
</tr>
<tr>
<td>Crank Period</td>
<td>720</td>
<td>720</td>
</tr>
<tr>
<td>Crank Step Size</td>
<td>0.5°</td>
<td>n/a</td>
</tr>
<tr>
<td>Compression Ratio</td>
<td>9.2:1</td>
<td>9.2:1</td>
</tr>
</tbody>
</table>

| Connecting Rod                    | 0.144145 m | 0.144145 m |
| Bore ×Stroke                      | 86.86 mm × 86.86 mm | 86.86 mm × 86.86 mm |

<table>
<thead>
<tr>
<th>Boundary Conditions</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient pressure (used as pressure for walls)</td>
<td>100179 Pascal</td>
<td>100179 Pascal</td>
</tr>
<tr>
<td>Fuel Temperature (used as initial fuel temperature)</td>
<td>314 K</td>
<td>314 K</td>
</tr>
<tr>
<td>Super charge Pressure (used as pressure at intake)</td>
<td>140835 Pascal</td>
<td>140835 Pascal</td>
</tr>
<tr>
<td>Super charge temperature (used as temperature at intake)</td>
<td>323 K</td>
<td>323 K</td>
</tr>
</tbody>
</table>

Table 4.3 Validation of computational data provided to the model
4.1.3 Validating the output

The computational model, that is validated both geometrically and computationally, is subjected to calculation. A numerical calculation is done on the computational model for one whole cycle. The results of the computational model are compared to experimental results. The time history of pressure produced inside the single cylinder is plotted for experimental and computational setups. The Table 4.4(a) shows the comparison. It shows that the pressure produced from the computational model under the same operating conditions follows the exact trend of the original engine. Though the pressure peaks vary, the computational results are very close to the experimental. The computational does not account for the pressure losses which are inevitable in mechanical systems. It also does not consider for limitations of pressure sensors that are used to measure the pressure inside the chamber. Considering the mechanical difficulties in measuring the pressure and limitations of numerical calculation to predict the exact behavior of real systems, the results obtained are considered to be very acceptable.

<table>
<thead>
<tr>
<th>Experimental and computational gasoline pressure plots</th>
</tr>
</thead>
<tbody>
<tr>
<td>Crank angle</td>
</tr>
<tr>
<td>Pressure in Mega pascal</td>
</tr>
<tr>
<td>360</td>
</tr>
<tr>
<td>0</td>
</tr>
</tbody>
</table>

Table 4.4 Validation of computational results with the experimental results
However, the error calculation is done between the computational and experimental data. The average percentage error of computational results when compared to experimental results is 4.89%. This is close to the experimental results and the same process is applied to the simulations in this study as it is well validated and supported by experimental results.

4.2 Study of validation case

Having validated computational results with experimental results, a detailed study on the computational model will be helpful to explain the combustion process in the engine. The advantage of computational fluid dynamic simulation is the ability to look into the model and understand the whole process and apply modifications as needed. To understand the process of combustion the following study is considered.

1. Air flow in the chamber
2. Fuel air interaction and vaporization
3. Fuel geometry interaction
4. Excess fuel
5. Combustion

The above parameters are important to study as each and every aspect of above cases is cause for fuel air distribution in the chamber.

4.2.1 Air flow in the cylinder

The air is taken into the cylinder from the two inlet ports. The turbulence of the incoming air causes two kinds of motions, swirl and tumble [7, 9, and 14]. Tumble is motion of air with the axis along ‘Z’ direction in figure (a) of Table 4.5. Swirl motion is motion of air with the axis along ‘Y’ direction. Figure (a) and figure (b) in Table 4.5
shows swirl and tumble motion respectively [14, 20]. Swirl motion of the air promotes air fuel mixture formation and the tumble motion is good to have the charge lifted up against the spark plug. Both these charges are significant at different points of the engine cycle. Tumble motion provides a good air fuel mixture so it is considered significant from the start of injection to near the TDC [14, 20]. Near the TDC tumble disappears and the motion then considered is swirl which helps the mixture to get up from the geometry towards the spark plug. In the experimental case, as it is from the engine which is being used in the field, we can see that the tumble and swirl of air can be clearly seen. This causes a good fuel air mixture in the cylinder and also holds the rich mixture near the spark during ignition. Figure (b) in Table 4.5 shows the rich mixture at the spark plug surrounded by lean mixture at TDC which indicates a very good distribution of fuel inside the cylinder. This is an important aspect of the study as it clearly gives an idea of the expected mixture characteristics near TDC. The tumble motion is very significant as it helps the primary mixing of the fuel. Most of the tumble motion disappears as the pressure in the cylinder builds up so the fuel is injected when the tumble motion is active.

<table>
<thead>
<tr>
<th>Table 4.5 Tumble and swirl motions of the air</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig (a) Tumble motion of the air</td>
</tr>
<tr>
<td>Fig (b) Air Swirl</td>
</tr>
</tbody>
</table>

![Fig (a) Tumble motion of the air](image1)

![Fig (b) Air Swirl](image2)
4.2.2 Fuel air interaction and vaporization

The fuel injected into the cylinder is interacted with the tumble motion as well as swirl motion of the air coming from the intake port [2]. The design of the experimental model is cleverly done in such a way the fuel is injected towards the intake port so that the fuel stream has readily available air turbulence to start mixing and vaporizing. The motion of fuel in the chamber is studied using particle tracks. The particle tracks of fuel (liquid) are tracked in the chamber to explain the fuel distribution pattern. As the liquid property is tracked it also gives information about fuel vaporization [19]. In Table 4.6 the figures (a) and (b) shows how the fuel is influenced by air turbulence to form a air fuel mixture. Table 4.6 shows the role of air tumble in fuel-air mixture formation. The turbulence also helps to increase the atomization of fuel droplets in the cylinder. At high pressure air offers resistance to the particles injected into the cylinder and the turbulence aides the shearing of fuel droplets which tear them into tiny particles which are easy to vaporize.

Table 4.6 Fuel air mixing

<table>
<thead>
<tr>
<th></th>
<th>Fuel air mixing at 240 bTDC</th>
<th>Fuel air mixing at 180 bTDC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig (a)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fig (b)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Along with the mixture formation vaporization is also an important phenomenon. Vaporization is improved in this commercial engine by using a droplet size of mean diameter 40 microns. This minute droplets should vaporize before the ignition [19]. Even though much care is taken to atomize the fuel during injection some large droplets, which are large in size compared to mean diameter, will exist. The temperature and pressure plays a very important part in vaporizing these large droplets before ignition point. There are still chances for existence of fuel droplets which are not vaporized after all the vaporization treatment. These droplets can cause wall wetting and they remain in the cylinder un-burnt or partially burnt. The experimental case does have this situation. A detailed particle track study of liquid particles will explain the remaining liquid droplets in the chamber. It is not possible in reality to get complete vaporization in the cylinder [2, 19]. The experimental case is quite successful in getting a good vapor before ignition. Table 4.7 shows the series of figures which explain the vaporization in the cylinder.

Table 4.7 (f) shows the local equivalency ratio increasing lower in the cylinder. This is because the fuel is not vaporized at that location. The fuel here needs much more time to get combusted and it may remain in the cylinder for several cycles or come out from the outlet as un-burnt hydro-carbon which pollutes the atmosphere. The part of fuel shown in the figure is only 2.23% of the whole fuel and is thus negligible. This study shows that the real engine is very efficient and can vaporize most of the fuel. But the effect of the left over fuel is important and it is essential to study what happens to the left over fuel after combustion. This is explained in the later sections.
Table 4.7 Vaporization of fuel in the cylinder
4.2.3 Fuel geometry interaction

Fuel geometry interaction is one major design parameter to be considered. This aspect has both advantages and disadvantages. The advantage is if the fuel is vaporized before it gets into contact with the geometry or if the fuel is atomized so that it vaporizes just after touching the wall because of wall temperature then the fuel interaction with the geometry is helpful for mixture formation [6]. On the other hand if the fuel is still large liquid droplets, the droplets will stick to the wall, reduce the wall temperature and never vaporize. So, the optimal use of this aspect gives a good air fuel mixture and hence good combustion. In the experimental case described in this chapter the fuel partly uses geometry to form a good air fuel mixture. Table 4.8 shows the pictures of fuel distribution where fuel hitting the piston and bouncing back into the cylinder can be seen.

<table>
<thead>
<tr>
<th>Table 4.8 Fuel hitting piston and mixing with air</th>
</tr>
</thead>
</table>

The advantage of this simulation is the droplet size is maintained not more than 50 microns as suggested in the experimental data. So, it is very important to have good atomization of fuel and to use the fuel geometry interaction for fuel distribution. It is
better to avoid this aspect if droplet size is huge. However, in the experimental case, as explained in section 4.2.3, there is a part of fuel which is concentrated at one position in the cylinder and not vaporized as shown in Table 4.7 (f). A thorough observation on this part of the engine is made to check the behavior of left over fuel. The fuel left over in this case is fairly small but this can be used to study the behavior of left over fuel. Table 4.9 shows the behavior of left over fuel in the cylinder. The phenomenon in which the fuel that is not vaporized in the cylinder, settles on the wall without getting combusted is called wall wetting. This is an important drawback in engine as it quenches the engine surfaces and reduces the internal temperature and also causes un-burnt hydro-carbons.

Table 4.9 Wall Wetting due to left over fuel in the cylinder
Two positions in the simulation are considered to explain this aspect. One is before TDC and other is after TDC. These positions are selected so that the behavior of left over fuel can be shown. It can be noticed that in 4.9 (a) and (b) the fuel is left over and is at highest concentration. After the combustion it is still there on the wall and the piston without getting combusted. This is a typical case of wall wetting. Though in this particular case it is not significant as the percentage of fuel left over is negligible but in real time engines this is the major problem where large amount of fuel is left over and cannot be combusted for many number of cycles. This is an important factor under consideration in this study.

4.2.4 Excess fuel

When the engine is running under rich condition the oxygen available is not sufficient for the fuel, this causes left over fuel in the cylinder and leads to emission of un-burnt hydrocarbons. The commercially available engines have this complexity but the trend of energy efficient engines demands the reduction of excess fuel. Most of the engines, even the present day engines, use rich mixture to get more power output, hence resulting in more pollution emission. As smart engines it’s the new challenge for direct injection spark ignition technology to reduce the emission of un-burnt hydro carbons. The experimental case being an eco-tech engine that operates on lean and stoichiometric conditions. Though it is capable of giving higher power outputs using rich mixture, it uses the technique of rich mixture surrounded by lean mixture to obtain maximum power and maximum efficiency, which has been theoretical before the invention of this technique. Therefore there is no excess fuel in the simulation and also the experimental set up. However, there is some fuel left over (2.23%) on the cylinder because of wall
wetting. This is negligible and the engine is very efficient compared to the engines that use port injection versus direct injection technology. This is the reason the direct injection spark ignition technology is considered as the eco-friendly technology for its capability to burn any kind of fuel as well as less emission of UBHC’s. Table 4.10 shows the left over fuel and also the left over available oxygen to explain that the fuel left over is not excess fuel and it still has oxygen available for combustion.

4.2.5 Combustion

Combustion is the most important study in the engine. This is the source of output and demands a good mixture preparation and a rich mixture at the time of injection. In

<table>
<thead>
<tr>
<th>Table 4.10 Left over fuel in the cylinder with available oxygen</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) Fuel at 180 aTDC</td>
</tr>
<tr>
<td>(b) Oxygen at 180 aTDC</td>
</tr>
</tbody>
</table>

the simulation of experimental set up 97.77% of fuel is combusted and most of the available oxygen is utilized. This resulted in the pressure plot presented in the validation section Table 4.4 (a). As a result the computational and experimental models are in agreement with each other. The combustion in the simulation showed, with the help of series of figures in Table 4.11, shows the combustion process going on in the real experimental engine which cannot be seen. This study explains very well about how the
The experimental engine is able to achieve good combustion with a good air fuel mixture, minimum wall wetting and no excess fuel. These aspects are proven to be utilized by the Carbon-di-oxide and Oxygen at different angles:

<table>
<thead>
<tr>
<th>Carbon-di-oxide</th>
<th>Oxygen</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="a" alt="Image" /> 30 aTDC</td>
<td><img src="b" alt="Image" /> 30 aTDC</td>
</tr>
<tr>
<td><img src="c" alt="Image" /> 90 aTDC</td>
<td><img src="d" alt="Image" /> 90 aTDC</td>
</tr>
<tr>
<td><img src="e" alt="Image" /> 180 aTDC</td>
<td><img src="f" alt="Image" /> 180 aTDC</td>
</tr>
</tbody>
</table>

4.11 Combustion process in the validation simulation

At 180 aTDC the outlet is open and some CO2 is exhausted.
engines using direct injection spark ignition technology and are used as references for the further study shown in this work. Table 4.11 shows the series of pictures explaining the combustion. The extent of combustion can be described by the amount of carbon-dioxide produced and the amount of oxygen utilized.

4.3 Diesel fuel in validation case

Diesel fuel is used in the simulation as it represents the family of heavy fuels. The direct injection spark ignition technology is capable of using any type of fuel and diesel has its own benefits over gasoline.

1. Lower cost
2. High energy content
3. Used in most of the heavy motor vehicles
4. Low maintenance cost
5. More energy efficient

Because of the above advantages diesel fuel is considered to be used in direct injection spark ignition technology. It is a new topic in present engine development and lot of research is going on in this field to successfully apply the DISI technology to diesel fuel.

The diesel fuel is replaced with gasoline and the behavior of diesel fuel in the direct injection spark ignition engine is studied. The comparisons are made between gasoline and fuel behavior based on the same aspects considered for the study of validation case.

1. Air flow in the chamber and excess fuel
2. Fuel air interaction and vaporization
3. Fuel geometry interaction
4. Combustion
4.3.1 Air flow in the chamber and excess fuel

Air flow in the cylinder is same as the validation case as all the parameters are maintained constant other than the change in fuel. So the air flow in the chamber is not studied as this is considered as the same explained in section 4.2.1 and as explained in section 4.2.4 there is no excess fuel in the simulation [14, 20]. The simulation is run in lean burn condition so there is enough oxygen available in the cylinder to burn the fuel.

4.3.2 Fuel air interaction and vaporization

Similar study on fuel air interaction and vaporization is carried out. It is observed that Diesel does not vaporize as quickly as gasoline. It is shown in Table 4.12 using two figures 4.12 (a) and (b) [14, 20]. These are the pictures taken at 30 bTDC for the gasoline and diesel validation cases. It shows the diesel is very slow in vaporization process.

<table>
<thead>
<tr>
<th>Gasoline</th>
<th>Diesel</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="link" alt="Gasoline Image" /></td>
<td><img src="link" alt="Diesel Image" /></td>
</tr>
</tbody>
</table>

Table 4.12 Difference in vaporization of diesel and Gasoline

As the physical and chemical properties of diesel differ from gasoline, the air fuel interaction is also different. Diesel fuel is heavy and after injection it directly settles downs on the piston without combusting. So the particle size of the injection should be
small compared to gasoline. Table 4.13 shows the difference in fuel air interaction in the cylinder for diesel and gasoline.

Table 4.13 explains how the diesel settles down on the surface of the piston without getting vaporized in the cylinder. This can be avoided by decreasing the particle size at the time of injection. The decrease in particle size reduces the time required for vaporization. This will improve the air mixture formation and improve the combustion.

<table>
<thead>
<tr>
<th>Gasoline</th>
<th>Diesel</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) 240 bTDC</td>
<td>(b) 240 bTDC</td>
</tr>
<tr>
<td>(c) 60 bTDC</td>
<td>(d) 60 bTDC</td>
</tr>
</tbody>
</table>

Table 4.13 Difference in fuel air interaction for diesel and gasoline
4.3.3 Fuel geometry interaction

As the diesel fuel is not vaporized in this simulation, all the fuel is settled down on the piston. This is a wall wetting problem where the fuel is attached to the wall and cannot be combusted [6]. A large amount of diesel liquid is settled down on the piston. Wall wetting is observed in the model because of poor vaporization and fuel air mixing. Table 4.14 shows the wall wetting in simulation with diesel fuel.

<table>
<thead>
<tr>
<th>Contours of Mass fraction of c10h22:oils (Time=2.0322e-02)</th>
<th>Crank Angle=660.000(deg)</th>
<th>ANSYS FLUENT 12.0 (3d, dp, pbase, dynamic2, pd2d5, rie, transient)</th>
</tr>
</thead>
</table>

Table 4.14 Diesel fuel concentration on the piston showing the wall wetting

4.3.4 Combustion

Diesel is heavy fuel and the fuel is injected at 270 bTDC during the air intake is started this makes diesel detonate or self ignite before starting the actual ignition process. This is called detonation of fuel which is not accepted as a good phenomenon [4, 21]. Table 4.15 shows the pictures of in cylinder detonation. The Table shows the difference between fuel distribution of gasoline and diesel. It can be observed that before top dead center the fuel in the diesel case is not visible. This is due to detonation where all the fuel is already combusted before reaching the top dead center. In experimental case this kind
of combustion will lead to a noisy engine and even can damage the engine itself. This should be avoided in the case of diesel fuel DISI engines.

The pressure peaks comparison between gasoline case and diesel case is done. The pressure peak in the diesel case is completely different from the gasoline and the pressure peak of the diesel engine simulation before the TDC explains the detonation.

<table>
<thead>
<tr>
<th>Gasoline</th>
<th>Diesel</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Gasoline" /></td>
<td><img src="image2.png" alt="Diesel" /></td>
</tr>
<tr>
<td><img src="pressure_plot.png" alt="Pressure plot for gasoline" /></td>
<td><img src="pressure_plot.png" alt="pressure plot for gasoline" /></td>
</tr>
</tbody>
</table>

(a) 30 bTDC  
(b) 30 bTDC

Table 4.15 Detonation in the case of diesel
All the above discussion implies that diesel fuel cannot be directly used in direct injection spark ignition engine with same set up as for gasoline. The major problems experienced during this simulation were

1. Improper mixing and wall wetting
2. Detonation

\[
\begin{array}{c}
\text{Pressure peak for diesel} \\
\end{array}
\]

\[
\begin{array}{c}
\text{(a) Pressure started building before spark} \\
\text{Table 4.16 Pressure of diesel and gasoline simulation.} \\
\end{array}
\]

These two are the important aspects to be considered, the parametric study on the droplet size is given higher priority. This is because the droplet size in the chamber plays an important role of eliminating the improper mixing of air and fuel in the chamber. It also decreases the time of vaporization and hence decreases the wall wetting. The average droplet size is reduced to 20 microns. Some other factors like injection velocity and injection time are also considered as factors for the parametric study of diesel fuel in reciprocating DISI engine. The complete parametric study of reciprocating engine and the application of this knowledge to a new kind of engine are explained in the next chapter.
5. Parametric Study

The parametric study is conducted on the reciprocating and rotary engines. This study shows the sensitivity of various parameters on the air fuel mixing inside the chamber and also shows how a good co-relation between the parameters increases the power output. Both the engines are different. Reciprocating engines have a greater period of time for the fuel to get mixed with the air. The fuel injected into reciprocating engine is exposed to longer durations of air current and heat to vaporize where as in rotary engines the fuel is to be mixed and vaporized in a very short time. They are completely different engines but direct injection can be used on any sort of engine and any sort of fuel can be injected using this injection strategy. The focus in this section is on the parametric study of fuel injection parameters, on fuel mixing, and how these requirements change with engines, and how the direct injection technology manages to provide all the engines the required kind of fuel injection.

5.1 Parametric study on reciprocating engine with diesel fuel

As stated earlier reciprocating engines have more time for mixture preparation. The pocket of Solstice engine is designed for three purposes. It provides a cavity during ignition i.e. more air for ignition which increases power. It guides air streams inside the chamber to mix with fuel. It also helps fuel distribution by spreading the injected fuel all over the chamber. Though there are three advantages the engines are named as air guided, wall guided and spray guided, depending upon the advantage they use. A collective use of these advantages will improve the air fuel mixture. From the results in the section 4.3
about the test of diesel fuel in the validation case we found certain parameters influence the whole process. The main parameters under consideration are

1. Injection velocity
2. Injection time
3. Particle size
4. Multiple plume injection
5. Conclusions from parametric study

Each parameter is carefully considered and the effects of these parameters on combustion are studied.

5.1.1 Injection velocity

The parameters used in studying the injection velocity effect are listed in Table 5.1, and resulted mass fraction of C_{10}H_{22} and CO_{2} are shown in Table 5.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection Begin</td>
<td>417°</td>
<td>417°</td>
</tr>
<tr>
<td>Injection end</td>
<td>473°</td>
<td>473°</td>
</tr>
<tr>
<td><strong>Injection Velocity</strong></td>
<td><strong>60 m/s</strong></td>
<td><strong>100 m/s</strong></td>
</tr>
<tr>
<td>Fuel Mass flow rate</td>
<td>0.0216 kg/s</td>
<td>0.0216 kg/s</td>
</tr>
<tr>
<td>Spark Start</td>
<td>719°</td>
<td>719°</td>
</tr>
<tr>
<td>Initial radius of spark</td>
<td>0.0017m</td>
<td>0.0017m</td>
</tr>
<tr>
<td>Energy/duration/dissipation</td>
<td>0.1/0.0001/0.0001</td>
<td>0.1/0.0001/0.0001</td>
</tr>
</tbody>
</table>

Table 5.1 Parameters used for studying effect of injection velocity

The air-fuel mixture is critical for good combustion. The comparison of fuel distribution is shown in figure 5.1 for velocity of injection 60 and 100m/s. Clearly the high injection velocity generates a much better fuel distribution over the chamber. Figure 5.2 presents
the combustion result at the end of power stroke by showing the mass fraction of CO$_2$. Almost no combustion is seen in the case of 60m/s injection velocity; while a uniform distribution of CO$_2$ is shown as injection velocity is 100m/s, indicating a good combustion in the chamber. Thus we can see the change in velocity of injection is a major parameter to get good combustion. The velocity of injection needs to be much higher than velocity of piston, which depends on the stroke and rpm. At 5000 rpm the velocity of piston is 14.6 m/s. To get a good air-fuel mixture, the injection velocity should be at least 8 times higher to make sure the fuel stream hits the pocket, bounce
back into the chamber, and mixes with air. We can also see that pocket shape plays an important role in re-distributing and bouncing back the fuel stream [6].

5.1.2 Injection Time

The fuel used in the simulation in section 4.3 had detonation. This is due to longer exposure of heavy fuel in the cylinder, to higher temperatures, and pressures. Diesel being the heavy fuel has a limited time to be exposed. This limited time of exposure is called ignition delay. The cylinder can reach a temperature of 900K and the maximum pressure reached is 6000000 Pascal due to compression (without combustion). At this temperature and pressure the maximum ignition delay time is calculated using the information in the paper [4] and this delay time is introduced into the FLUENT using the knock model in FLUENT governed by the following equation.

\[ T = 0.01768 \left[ \frac{\text{Octane Number of the Fuel used}}{100} \right]^{3.402} p^{-1.7} \exp \left[ \frac{3800}{T} \right] \]

Once the maximum temperature and pressure are reached in the cylinder the diesel particles in the cylinder will have 0.15ms. From many number of simulations it is calculated that the earliest the fuel can be injected is 180bTDC. This means the fuel must be injected into the cylinder in the compression stroke as it is good to use the tumble motion of air while the fuel is injected from 180 bTDC to 30 bTDC and the velocity increase information from the section 5.1.1 aids in quick air fuel mixing. Combining the increase in injection velocity and changing the injection time eliminated detonation and a good combustion can be observed as shown in Table 5.3. Though the combustion is achieved the simulation is still not good when compared to validation case because the diesel is not vaporized as well as gasoline before top dead center. Because of volatility of
Table 5.3 Combustion process of the reciprocating engine using diesel
diesel it is hard to evaporate. Table 5.4 shows the difference between the evaporation of gasoline and the present case.

The change in the injection time and injection angle was helpful in reducing the detonation, however, diesel being heavy fuel is hard to get vaporized. In Table 5.4 it is clearly seen that the concentration of fuel at 30 bTDC is much more when compared to that of diesel case. In order to increase the vaporization a parametric study on vaporization is carried out.

<table>
<thead>
<tr>
<th>Gasoline</th>
<th>Diesel</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) 30 bTDC</td>
<td>(b) 30 bTDC</td>
</tr>
</tbody>
</table>

Table 5.4 difference in particle vaporization

5.1.3 Droplet size

Droplet size is a crucial parameter in the vaporization of fuel. The behavior of droplets in the engine is very well explained in the papers [12]. In the simulation with gasoline fuel the droplet size was 50 microns which is good for gasoline as the diesel is a less volatile fuel it takes much more time to vaporize. To aid the vaporization of diesel the droplet size need to be reduced. Parametric study on droplet size is done to know the optimum droplet size for diesel. Simulations with droplet sizes 50 microns, 40 microns,
30 microns and 20 microns are conducted and it is found that the lesser the droplet size the greater the amount of vaporization. As it is not practically feasible to go less than 20 microns droplet size without increasing the injection pressure, It is concluded that 20 microns is the right size for the droplets. So the mean diameter of the particles in the simulations is maintained 20 microns and it can vary from 10 to 30 microns in any injection. Table 5.5 shows the difference in vaporization with change in particle size.

<table>
<thead>
<tr>
<th>50 Microns</th>
<th>40 Microns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>(a) 30 bTDC</td>
<td>(b) 30 bTDC</td>
</tr>
<tr>
<td>30 Microns</td>
<td>20 Microns</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>(c) 30 bTDC</td>
<td>(d) 30 bTDC</td>
</tr>
</tbody>
</table>

Table 5.5 Difference in vaporization with decrease in particle size
5.1.4 Multiple orifice injection

In order to minimize the contact of diesel fuel with the wall and avoid wall wetting, a better air fuel distribution is needed in the cylinder. Utilizing the air charge and the multiple orifice injection is the best way to achieve the good air fuel distribution [5, 15]. The design and analysis of multiple orifice injectors is carried out for reciprocating engine to demonstrate its importance. This portion of the study is helpful to improve the diesel fuel performance in DISI engine and optimum use of gasoline. The following sections will explain the different types of injections under consideration and how they are modeled and simulated.

5.1.4.1 Injection Modeling

The injections are modeled using GAMBIT and three types of injections are selected to demonstrate the importance of multi plume injection. The three injections are

1. Single Orifice injection
2. Dual Orifice injection
3. Four orifice injection

![Image of injections](image)

Table 5.6 dual plume and four plume injection models

The models are tested and as expected the results show that, with increase in number of orifice the distribution of fuel in the chamber improves and the mixture is close to
homogeneous. All other parameters of spray modeling are kept constant to emphasize the
effect of multiple orifice injection strategy.

5.1.4.2 Parametric study

Three simulations using the simulation data shown below are modeled using
diesel fuel. The fuel is injected from 180 bTDC to 120 bTDC and the velocity of the
injector is taken from the parametric study done for the diesel fuel DISI reciprocating
engine. The parameters in common for all the three simulation are listed in Table 5.7

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection start</td>
<td>180 bTDC</td>
</tr>
<tr>
<td>Injection End</td>
<td>120 bTDC</td>
</tr>
<tr>
<td>Calculated homogeneous equivalence ratio of the injected fuel</td>
<td>0.97</td>
</tr>
<tr>
<td>Net mass flow rate of fuel</td>
<td>0.01 kg/s</td>
</tr>
</tbody>
</table>

Table 5.7 Parameters used in parametric study of multiple plume injectors

Few slides are selected to show the process in the three cases. The first slide in each
case shows the injection of fuel in the chamber at 140 bTDC close to the end of injection.
The second slide shows the distribution of fuel at TDC. The third slide shows the carbon-
di-oxide formed due to combustion which basically shows the extent of combustion.

1. Single Orifice Injection

In this case the single orifice injector is tested and the fuel distribution from this case is
compared to other cases. The details about this case are explained with the help of
pictures in Table 5.8. The single plume can be clearly seen from Table 5.8(a). The fuel
distribution on the TDC can be seen in Table 5.8 (b) the mixture is not well formed and
the large amount of fuel is settled in one side of the engine. This should be improve and
is possible by adding another plume so that it increases the distribution of fuel during injection and also increases the utilization of air charge in less amount of time.

It can be seen the fuel distribution at TDC is not homogeneous and from the picture Table 5.8 (c) the poor combustion due to poor fuel distributions can be observed. In order to improve the fuel distribution the single orifice is split into two orifices.

(a) particles of fuel showing the single orifice injection at 140 bTDC

(b) Contours of equivalence ratio at TDC

(c) Contours of mass fraction of CO\textsubscript{2} produced due to combustion

Table 5.8 Pictures showing the simulation of single orifice injector
2. Duel Orifice Injection

In this case the effect of dual orifice injector is studied and compared to single orifice injection. The results of this case are shown in Table 5.9. It can be observed from Table 5.8 and 5.9 that the distribution of fuel in dual plume injector case is much better than the single plume injector case. Also the combustion in dual plume injector case is improved because of better fuel distribution.

![Particle Tracer Colored by phi](image1)
(a) particles of fuel showing the dual orifice injection at 140bTDC

![Contours of equivalence ratio at TDC](image2)
(b) Contours of equivalence ratio at TDC

![Contours of Mass fraction of CO₂ produced due to combustion](image3)
(c) Contours of mass fraction of CO₂ produced due to combustion

<table>
<thead>
<tr>
<th>Table 5.9 Pictures showing the simulation case2</th>
</tr>
</thead>
</table>

The two orifice injection shows better results than the single orifice. The amount of CO₂ produced in case 2 are much more than the amount of CO₂ produced in case 1. This means the combustion is better in case 2 than case 1. In the next case the fuel is
injected from 4 orifices. Note that the amount of fuel considered in all the cases mentioned in section 5.4.1.2

3. Four-Orifice Injection

In this case 4 orifices are used in the injection. The fuel is coming out of the injector from four orifices so it has more spread and more atomization of the fuel. Table 5.10 explains the results of case 3. It can be clearly seen that the distribution is improved with increase in number of orifices. The orientation of orifice is also important but in the abstract only the effect of number of orifice is shown as a preliminary result. A detailed optimized injection strategy is the goal for this work.

![Images](a) particles of fuel showing the single orifice injection at 290bTDC  
(b) Contours of equivalence ratio at TDC  
(d) Contours of mass fraction of CO\textsubscript{2} produced due to combustion  

Table 5.10 Pictures showing the simulation case3
Table 5.10 clearly shows with multiple orifice injection the distribution of fuel can be improved and hence the combustion and the power output can be improved. The mixture in the simulations is nearly homogeneous when the injection starts at the intake stroke like in port injection. The fuel has more time to mix with air, but if a lean mixture is considered, the fuel is injected just before the spark event. This injection pattern provides lesser time for fuel air mixture formation. So we need to look for maximum spread in less time which is possible when multiple orifice injection is used.

5.1.5 Conclusions of parametric study

From the parametric study the following conclusions are derived.

1. Injection velocity should be in the range 100 m/s to get the optimal distribution
2. Injection should start after 180 bTDC to avoid detonation
3. Droplet size of injection should be in order of 20 microns to aid the diesel vaporization
4. Multiple orifice injection aids the mixing and vaporizing process.

Using these conclusions, parameters of the simulation are changed and a simulation is set up with the new parameters in consideration. The new case had a complete combustion and it is summarized in the next section.

5.2 Results

The information in the parametric study is used in one single simulation and diesel combustion in DISI reciprocating engine is successfully achieved. Table 5.12 shows the pressure plots for all the simulations experimental gasoline, computational gasoline and computational diesel. These pressure plots show the difference between the pressure produced in diesel and gasoline. Table 5.11 shows the combustion in this
simulation which shows that there is a good combustion in the chamber and the amount of fuel left over is 4.25% of the total fuel.

Table 5.11 Production of CO$_2$ during combustion cycle
The pressure plots are plotted for three result sets.

1. Experimental pressure history
2. Reciprocating gasoline engine pressure history
3. Reciprocating diesel engine pressure history

The Table 5.12 explains that diesel can be used in the DISI technology. As diesel is a heavy fuel the injection and other parameters need to be optimized to get a good power output. This optimization can be achieved with the help of parametric study. The parametric study involves controlled detonation, vaporization and controlled injection
which is only possible with direct injection spark ignition technology. The pressure produced by gasoline is a little higher but for a new technology with much research still to be done diesel fuel with DISI technology is proved efficient and with much more research this performance can be improved and can be applied to other fuels like bio fuel and many more heavy fuels.

5.3. Conclusions of parametric study

Having successfully demonstrated the use of gasoline in DISI reciprocating engine, this technology is now applied to the rotary engine. Rotary engines are capable of producing high power output but they lack efficiency. As a smart engine technique it is the challenge for DISI technology to make the rotary engine more efficient while maintaining the high power output of the rotary engine.

Application of DISI technology is again done in phases but with the knowledge of the computational analysis of DISI reciprocating engine with diesel fuel.

1. Modeling and meshing
2. Cold flow study
3. Simulation

5.3.1 Modeling and Meshing

Modeling and meshing of the models used in the simulation are explained in the chapter 2. This is because modeling and meshing of any model follow same order and it is convenient to present them in the same chapter. After modeling and meshing, the mesh is imported into the FLUENT and a mesh motion is achieved which is very important for further simulations. Mesh motion in rotary engine require a UDF because of the complexity of the motion of the rotor. The mesh motion of the rotor is also explained in
Chapter 2 of the study. The next step of the process is cold flow study which is very important to understand the flow physics. This is explained in the next section of the study.

5.3.2 Cold Flow Study

Before doing simulations a detailed study of flow physics is carried out to understand the air charge and to verify the injection position. Cold flow study helps to understand the tumble motion of air inside the chamber and therefore we can estimate the mixture formation. The maximum pressure and temperature can be observed hence the maximum time for the diesel fuel detonation can be estimated so that the fuel injection can be made to avoid detonation. In cold flow study the following aspects are considered.

1. Turbulence of air in the chamber

2. Maximum temperature and pressure at TDC

3. Injection location and orientation of injection

5.3.2.1 Turbulence of air in the chamber

Turbulence of the air in the cylinder is a very crucial for formation of fuel and air mixture. The study of turbulence in the cylinder will explain the behavior of air charge in the cylinder. The main aspect studied under this section is that the turbulence in the cylinder starts with the air inflow at 180 bTDC. So the fuel should be injected in between 180 bTDC to 90 bTDC to utilize the air charge [24, 26, and 27]. This aspect is very important as it decides the injection duration and injection start. The duration of injection can be changed but it should be in between the crank angle 180 bTDC and 90 bTDC. Table 6.1 shows the air charge inside the cylinder. It shows how the turbulence
disappears after 90 bTDC. The Table 5.13 shows that the turbulence is good at 180 bTDC and it is getting reduced up to 90 bTDC and at 90 bTDC there is almost no turbulence. This gives an idea of injection start timing and injection duration.
5.3.2.2 Maximum temperature and pressure at TDC

These details are very important to calculate the ignition delay of diesel in rotary engine. The maximum pressure is recorded as 561e+04 Pascal and the maximum temperature is recorded as 1100 K. Using these values the ignition delay time after the fuel reaches maximum temperature is 0.15 ms. Normally the ignition delay varies from 0.25 to 0.4 ms which gives us a time of 100 degrees at 8000 RPM engine shaft speed.
before detonation. These values are taken from the papers [23, 24, and 26]. Table 6.4 shows the contours of pressure and temperature at TDC where highest values are recorded.

The pressure shown above is obtained purely due to combustion without any chemical reaction in it. This is the maximum temperature and pressure where the detonation should be avoided. If it is avoided at this position the detonation will happen after TDC which gives maximum pressure in power stroke which is very useful.

5.3.2.3 Injection location and Orientation of injector

The most important injection timing is decided using the concepts of detonation and the cold flow study. Also the other interesting factors are decided for the simulation using the cold flow study including injection location and orientation [13, 22]. These are important as they influence the flow of fuel and the formation of fuel air mixture. Studies in this field show that the fuel injected opposite to the air flow in the cylinder is well mixed and vaporized because of the droplet shear of air charge inside the cylinder. This droplet shear acts as a catalyst to the vaporization process hence enhancing the air fuel mixture as vapors are better at mixing than liquids. The location of the injector is decided in such a way that it can spray into a space without hitting the walls starting at 180 bTDC. There is only one position which can use the merits mentioned and the position and direction of injection are show in Table 5.15. It can be observed that the fuel is avoiding the walls and is moving through the opposite air stream. The air stream shears the fluid particles and makes the fuel vaporize faster. The motion of air causes the fuel to move towards the leading edge which is also a very important factor to be
considered. Using the experience of Solstic diesel DISI engine parametric study and the cold flow study of rotary engine a diesel DISI rotary engine simulation is designed. The next section explains the details of this simulation.

![Diagram](image)

Table 5.15 Fuel injected opposite to the direction of flow

5.4 Simulations of Rotary engine with diesel fuel DISI technology

Based on the computational analysis experience from reciprocating engine parametric study and also from the information from the paper review in the first chapter of this study, it is known that rotary engines use more than one injector. This has a significance that is rotary engines have much less time for fuel air mixture to form. So it is a required and necessary condition to improve fuel distribution. Instead of one injector as shown in Table 6.5 two more injectors may be used [22, 23]. These injectors are called “pilot injector” and “lead injector”. Pilot injector supply a small amount of fuel near the spark plug at TDC to make the air fuel mixture rich during the time of spark at TDC. This is also proved to be achieved in reciprocating engine as shown in Table 4.5 Fig (b). Because of the less time available to form air fuel mixture in rotary engine, it is important
to have pilot injector to have a rich mixture near the spark just before TDC. On the other hand lead injector is very important to extend the power produced during power stroke and also to increase the concentration of fuel in the leading edge to increase the power output of the engine.

Considering these aspects as important to study and as the information from the parametric study can be readily used. Three cases are designed to verify the possibility of converting a rotary engine into DISI diesel rotary engine.

1. Rotary DISI diesel engine with main injector (DISIR_M)
2. Rotary DISI diesel engine with main and pilot injectors (DISIR_MP)
3. Rotary DISI diesel engine with main, pilot and lead injectors (DISIR_MPL)

Each case is considered and after making small changes to the parameters used combustion is achieved in each case. The parametric changes made here are not significant as they are very close to the parameters used for the diesel reciprocating engine DISI simulations and are not listed here. The main consideration is given to number of injectors as it is needed to check the amount of power increase by using the DISI technique and the multiple injection technique that is proved to be used for increase in efficiency of the engine.

Before doing a comparative study between the above listed cases it is important to understand the simulation of single injector DISI diesel rotary engine.

5.4.1 Rotary DISI diesel engine with main injector (DISIR_M)

This case, as the name says, uses only the main injector with the merits of injecting opposite to the air flow and injecting at 180 bTDC for duration of 90 degrees, and avoiding wall wetting. Table 6.6 shows the series of pictures which will explain the
combustion and the output of the simulation. The set of pictures with the analytical plots extracted from FLUENT are useful to calculate the power output which is a very easy comparison point. The torque of the rotor is an output from the simulation which can be used to calculate the power.

As done earlier in the case of reciprocating engine, the study of rotary engine is done in steps

1. Air fuel interaction and fuel vaporization
2. Combustion
3. Power output

5.4.1.1 Air fuel interaction and fuel vaporization
Table 5.16 Fuel air interaction

Table 5.16 explains how air charge is used to mix the fuel with the air. Injection starts at Table 5.16 (a) and at each and every stage fuel is injected against the stream. The air flow is pushing the fuel in the direction of rotor. This increases the concentration of fuel in the leading edge. Table 5.16 (f) is the mixture of fuel and air at TDC. This is the best mixture formed with single injector. Table 5.17 explains the vaporization in the cylinder. The injected fuel particles are vaporized because of the shear caused by the opposing air. Surprisingly, due to the utilization of facts learn from diesel DISI reciprocating engine, a well distributed and vaporized air fuel mixture is formed in the
Table 5.17 vaporization of fuel
combustion chamber before the spark. This leads to a controlled combustion as the detonation terms are taken care of. The next section explains the combustion in this simulation.

5.4.1.2 Combustion

Because of well vaporized and well mixed air fuel mixture a good combustion is achieved in this simulation [25, 28, and 29]. The fuel is almost completely utilized with 5% of fuel left over. The oxygen is utilized and there is a positive power output. Table 5.18 shows the combustion in the cylinder.

<table>
<thead>
<tr>
<th>Carbon-di-oxide</th>
<th>Oxygen</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
</tbody>
</table>

(a) Combustion at 90 aTDC  
(b) Oxygen utilized at 90 aTDC

Table 5.18 Combustion in DISI diesel rotary engine with one injector

From Table 5.18 it is evident that the simulation had successful combustion and the oxygen in the front half of the rotor is mostly utilized. But there is still some oxygen available in the leading edge and the fuel distribution can be still improved by introducing pilot and lead injectors explained in later sections.

5.4.1.3 Power output

Power output from the engine is calculated using the torque produced due to combustion. The power is calculated for 3 cycles and is averaged for one cycle. The
calculated power is in horse power and it is an easy way to check the extent of combustion and what can be expected from the engine. Table 5.19 shows the power output of the engine calculated.

Table 5.19 shows the power produced with one single injector but we can improve this power by using lead injector and pilot injector. We can also increase the oxygen consumption and power output. In the next section the differences between the three simulations are shown to explain the importance of pilot and leading edge injector.

5.4.2 Comparison between DISIR_M, DISIR_MP and DISIR_MPL

Three cases are considered. The first case is the same case explained in section 6.3.1 while the two other cases are one with pilot injector added and the other with pilot and lead injector added. The comparison is made in three steps.
1. Fuel distribution
2. Combustion
3. Power output

5.4.2.1 Fuel distribution

The fuel distribution differs between these cases. A lead injector increases the concentration of fuel in the leading edge where as the pilot injector increases the concentration of fuel at TDC. It is difficult to show exactly the same pictures taken at same position to show this difference as pilot injector injects near TDC where all the fuel from main injector is also present but the difference between the concentrations of fuel at spark can be shown. Table 5.20 shows the concentration difference between the simulations with pilot injector and without pilot injector. As explained earlier the pilot injector increases the concentration of fuel near TDC at the time of spark and it can be clearly seen in Table 5.20 (b). In Table 5.20 it can be seen that the case without pilot injector does not have much fuel concentration near TDC. When the pilot injector is added the concentration of fuel at TDC is increased and this is good for combustion. The

<table>
<thead>
<tr>
<th>Case</th>
<th>Concentration at TDC</th>
</tr>
</thead>
<tbody>
<tr>
<td>pilot injector</td>
<td>Increased</td>
</tr>
<tr>
<td>without pilot injector</td>
<td>Decreased</td>
</tr>
</tbody>
</table>

Table 5.20 Difference between the cases with and without pilot injection
concentration of fuel near TDC at the time of spark increases the laminar flame speed, which burns the fuel in very short time. The impact of the combustion will be more and hence the power output will be improved once the fuel is combusted in the chamber the combustion process will be completed. So the power output is less as it is only for less period of time. In order to increase the power output throughout the power stroke a lead injector is used. Also the lead injector increases the concentration of fuel in the leading edge. This improves the power output as all the power produced is because of the pressure produced due to combustion at leading edge. Table 5.21 shows the difference in fuel distribution between the case with main injector and pilot injector and the case with lead injector. After the study of fuel mixture in the cylinder the next step is to study the combustion. The combustion in each simulation is discussed in next section.

![Image of fuel distribution with and without lead injector](image)

**Table 5.21 Fuel distribution with and without lead injector (Difference cannot be seen as the lead injector starts at 60 aTDC)**

5.4.2.2 Combustion

All the three simulations have combustion. In the DISIR_M there is a good combustion. The only difference between these cases is addition of fuel. But the combustion varies from case to case. DISIR_M has the least combustion whereas the
combustion is fast and better in DISIR_MP. Combustion in DISIR_MPL is same as the DISIR_MP but the combustion process is prolonged until the exhaust gases are released supplying fuel to the combusting charge. Though there is change in number of injectors the total amount of fuel is kept constant so that these processes can be compared and a conclusion can be drawn. Table 5.22 explains the difference in combustion in these three simulations.
5.4.2.3 Power output

The final step in this study is the results of rotary engine DISI diesel simulations. The results primarily prove that this smart technology can be used for any kind of engine reciprocating, rotary, migrating combustion chamber engine, etc. It can be stated that direct injection spark ignition technology is fuel efficient and high power producing engines. Table 5.23 shows the power output produced in the three simulations in the study.
Power outputs for the simulations under study are

1. Power output for DISIR_M = 28.7 HP
2. Power output for DISI_MP = 31.1 HP
3. Power output for DISI_MPL = 38.2 HP

The power outputs show how the pilot and leading edge injectors are useful to increase the power output. In the Table 5.23 it can be seen that the DISI_MPL plot continues to have power output after 60 aTDC, which is the position where the fuel is injected from the lead injector and burning of the fuel started. This extended fuel supply gives extended power output instead of burning all the fuel at once. The extended power output increases the overall power output. So using the same amount of fuel in different injectors can increase the power of an engine.
Chapter 6

Conclusions

The computational analysis of direct injection spark ignition is done to check the feasibility of heavy fuel. The modeling and analysis of two kinds of engine which represent the major part of IC engine world are done and a heavy fuel (diesel) is injected to prove that these engines are capable to burn any kind of fuel and are more efficient. In order to do this the model is validated with an experimental set up and results and it is proved that the computational model is close to the experimental model. After validation the diesel fuel is injected with the same model settings used for gasoline and the model experienced detonation. It proves that diesel cannot be operated under same conditions as gasoline.

Investigation on optimal settings to use heavy fuel (diesel) is done to find out whether heavy fuels are feasible in DISI engines. After detailed parametric study on the flow physics inside the ignition chamber and behavior of injected fuel some important conclusions are derived.

1. Heavy fuels should be injected into the cylinder in such a way that they do not have time to detonate. i.e., the injected fuel should only have the ignition delay time to mix with the air to form air-fuel mixture.

2. The vaporization of heavy fuel takes much time than gasoline, but the heavy fuel cannot be injected earlier into the cylinder as it detonates. So the only way to decrease the vaporization time is to decrease the droplet size of the injection.
The above facts are proved to be important and using these studies a good matching between computational results of validation case with gasoline fuel and diesel fuel can be achieved. There by it is proved that heavy fuels can be used in DISI engines.

The other challenge is to prove if this technology can be applied to any kind of engine. A rotary engine is selected and DISI technology with diesel fuel is employed in it. The parametric studies done for reciprocating engine are used to develop a successful combustion model in this engine. As the rotary engine has much less time to get a good air fuel mixture the following important conclusions are made.

1. The air charge in the engine disappears after 90bTDC and fuel cannot be injected before 180 bTDC as it causes detonation. So the best injection time for fuel is between 180bTDC and 90bTDC

2. The single injector even with small droplet size cannot form a very good air fuel mixture because of the complexity in motion and geometry and also limited available time.

3. Pilot injector and lead injector increase the quality of mixture formed in the cylinder. They avoid wall wetting as the main injector does not have to penetrate as far to provide an acceptable air fuel mixture. The mixing work is divided in-between main, lead and pilot injectors which make mixing easier.

4. By injecting fuel into the leading edge after the combustion process is started provides extended power output which increase the power output of the whole cycle.

All the above results prove that the DISI technology can be employed to any engine with any kind of fuel in it.
Chapter 7

Recommendations and future work

As a new technology, much more research needs to be carried out in the field of DISI engine. Though there are commercial engines in the market, like the Pontiac Solstice, they use gasoline fuel. So the capability of this type of engine is not yet fully utilized. This technology is very powerful and is going to be the key to face the future scarcity of regular fuels and make the transition to alternate fuels smoother. The utilization of heavy fuel, even though proved to be possible in this study, has to be extensively studied. Such studies show the efficiency of this engine to use any type of fuel. Extensive parametric study and optimization of many parameters, like injection angle, injection timing, injection design, location and orientation, and also the design of geometry to cause the turbulence in the air, are very important. This technology should also be tested with more fuels and the operating conditions each fuel should be defined so that it provides an adequate starting point for the experimental research.

A continued progress in DISI engine research is going on in our group. Research using heavy fuel in a Rotax engine is going on to extend the DISI technology into the aerospace industry. Additionally a highly optimized study of injection parameters is done on heavy fuel rotary engine to get the best possible power output using least fuel, Optimizations of injection parameters to increase the fuel spread and optimization of number of injectors and number of holes in the injector, decreasing the radius of injector etc are some of the future works to be done to improve this technology.
REFERENCES


20. Christoph Garth, Robert S. Laramee, Xavier Tricoche, Jurgen Schneider, and Hans Hagen. “Extraction and Visualization of Swirl and Tumble Motion from Engine Simulation Data”.


26. Chao Wang, Asela Anuruddhika, Zongxian Liang, Haibo Dong, Computational Analysis of a Rotary Engine, Wright State University, Final report

