EVALUATION OF NUMERICAL METHODS FOR MODELING FLOW FIELD DURING THE INTAKE STROKE OF SPARK-IGNITED ENGINES

Carlos Lozano Hontecillas

Key words: stratified charge
spark-ignited direct-injection engines
tumble number
computational fluid dynamic
steady-state simulation
transient simulation

Abstract

Recent changes in the exhaust emission legislations obliged the automotive industry to reduce the emissions and improve the fuel saving. Stratified charge direct-injection engines were indicated as a convenient short-term solution to adapt spark-ignited engines to these requirements. In this context, the objective of this thesis was to evaluate the effectiveness of the intake system for a stratified charge spark-ignited direct-injection engines by means of a computational fluid dynamic tool. The effectiveness of the intake system in such engines is assessed by the tumble number. Therefore, the first objective of the project was to compare differences between transient and steady-state simulations in calculating the tumble number. Steady-state simulations are mostly used by companies since they require less computational power than transient simulations. However, the differences of the model between both types of simulations speak in favor of a clear advantage for transient simulation to calculate an approximation of the tumble number. The second objective of the project was to perform an enhancement of the mesh corresponding to steady-state model. This was performed by refining the mesh through different programs and studying the impact of each refinement criteria on the quality of the results.
## Table of contents

Abstract ................................................................................................................................... I
Table of contents .................................................................................................................. II
List of figures ...................................................................................................................... IV
List of tables ....................................................................................................................... VII
List of the applied abbreviations ...................................................................................... VIII
List of applied symbols ....................................................................................................... IX

1  Introduction .................................................................................................................... 1
2  General concepts ............................................................................................................ 3
   2.1 Comparing SIDI engine and PFI engine ................................................................. 3
   2.2 SIDI operating mode in stratified engines .............................................................. 4
   2.3 Air flow structure ................................................................................................... 6

3  Computational Fluid Dynamics (CFD) simulations ...................................................... 8
   3.1 Turbulence models .................................................................................................... 8
      3.1.1 Introduction .................................................................................................... 8
      3.1.2 k-ε Models .................................................................................................. 11
      3.1.3 k-ζ-f model.................................................................................................. 12
   3.2 Meshing ................................................................................................................... 13
      3.2.1 Connectivity of a mesh .................................................................................. 13
      3.2.2 Type of elements ......................................................................................... 14
      3.2.3 Refinements ................................................................................................. 15

4  Model presentation ....................................................................................................... 17
   4.1 Describing the engine.............................................................................................. 17
      4.1.1 Steady-state simulations: .............................................................................. 17
      4.1.2 Transient simulations: .................................................................................. 18
   4.2 Describing the mesh .............................................................................................. 20
      4.2.1 Steady-state simulations: .............................................................................. 20
      4.2.2 Transient simulations: .................................................................................. 22
   4.3 Valve lift .................................................................................................................. 23
   4.4 Tumble number ....................................................................................................... 24

5  Results .......................................................................................................................... 27
   5.1 Analyses and comparisons of the steady-state and transient results ................. 27
5.1.1 Steady-state results ........................................................................................................ 28
5.1.2 Transient results......................................................................................................... 34
5.1.3 Comparison of steady-state and transient simulation ................................................ 45
5.2 Analyses and comparisons of the refinement in FIRE and FLUENT ......................... 46
  5.2.1 FIRE refinement ....................................................................................................... 46
  5.2.2 FLUENT refinement ............................................................................................... 48
  5.2.3 Comparison of refinements .................................................................................... 54
6 Conclusions ....................................................................................................................... 56
7 References ........................................................................................................................ 57
8 Attachments ........................................................................................................................ 60
  8.1 Formula for calculating tumble number in transient simulations ......................... 60
  8.2 Economic impact ......................................................................................................... 61
     8.2.1 GANTT of the thesis ....................................................................................... 61
     8.2.2 Investment .......................................................................................................... 62
  8.3 Environmental impact ................................................................................................. 64
**List of figures**

Figure 1. Annual light-duty vehicle sales by technology type [1]
Figure 2. Comparison of the PFI (left) and SIDI (right) mixture preparation system [4]
Figure 3. a SIDI stratified engine operating map [4]
Figure 4. Homogeneous mix cycle [10]
Figure 5. Stratified mix cycle [10]
Figure 6. swirl-based system (up) and tumble-based system (down) [5]
Figure 7. Laminar flow of a cigarette becoming turbulent [14]
Figure 8. Structured Quadrilateral Mesh for an Airfoil [23]
Figure 9. Unstructured Quadrilateral Mesh [24]
Figure 10. Hybrid Triangular Quadrilateral Mesh [26]
Figure 11. Hybrid tree structure [26]
Figure 12. 2D elements: quadrilateral and triangle [23]
Figure 13. 3D elements: hexahedra, tetrahedra, square pyramid and extruded triangle [23]
Figure 14. Example of a steady-state mesh
Figure 15. Steady-state mesh mesh
Figure 16. Boundary and initial temperatures
Figure 17. Boundary and initial pressures
Figure 18. boundary selections: inlet (blue), Outlet (yellow) and symetry (brown)
Figure 19. Adaptation for a steady-state simulation [27]
Figure 20. Intake valve closed – Exhaust valve open.
Figure 21. Intake valve open – Exhaust valve open.
Figure 22. Intake valve open – Exhaust vale closed
Figure 23. Intake valve closed – Exhaust valve closed
Figure 24. 10 mm valve lift
Figure 25. 6 mm valve lift
Figure 26. 2 mm valve lift
Figure 27. Exhaust and intake valve lift curves
Figure 28. Tumble area (brown) in a steady-state mesh
Figure 29. Inlet and cylinder pressure in transient and inlet and cylinder pressure in steady-state
Figure 30. 10 mm steady mesh – Flow velocity
Figure 31. 10 mm steady mesh – vector Flow velocity
Figure 32. 10 mm steady mesh – Pressure
Figure 33. 10 mm steady mesh – TKE
Figure 34. 10 mm steady mesh – Tumble number
Figure 35. 6 mm steady mesh – Flow velocity
Figure 36. 6 mm steady mesh – vector Flow velocity
Figure 37. 6 mm steady mesh – Pressure
Figure 38. 6 mm steady mesh – TKE
Figure 39. 6 mm steady mesh – Tumble number.
Figure 40. 2 mm steady mesh – Flow velocity
Figure 41. 2 mm steady mesh – vector Flow velocity
Figure 42. 2 mm steady mesh – Pressure
Figure 43. 2 mm steady mesh – TKE
Figure 44. 2 mm steady mesh – Tumble number
Figure 45. (a) 3 mm, (b) 5 mm and (c) 8 mm (valve opening) - Transient mesh – Pressure
Figure 46. 10 mm (maximum gap) - Transient mesh – Pressure
Figure 47. (a) 6 mm and (b) 2 mm (valve closing) - Transient mesh – Pressure
Figure 48. 3 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 49. 5 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 50. 8 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 51. 10 mm (maximum gap) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 52. 6 mm (valve closing) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 53. 2 mm (valve closing) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 54. (a) 8 mm (valve opening) and (b) 10 mm (maximum gap) - Transient mesh – Temperature
Figure 55. (a) 3 mm and (b) 5 mm (valve opening) - Transient mesh – Temperature
Figure 56. (a) 6 mm and (b) 2 mm (valve closing) - Transient mesh – Temperature
Figure 57. (a) 6 mm and (b) 2 mm (valve closing) - Transient mesh – TKE
Figure 58. (a) 3 mm and (b) 5 mm (valve opening) - Transient mesh – TKE
Figure 59. (a) 8 mm (valve opening) and (b) 10 mm (maximum gap) - Transient mesh – TKE
Figure 60. 10 mm - transient mesh – Tumble number
Figure 61. 6 mm transient mesh – Tumble number
Figure 62. 2 mm transient mesh – Tumble number
Figure 63. Area of refinement by Momentum error
Figure 64. 6 mm steady refined mesh – Flow velocity
Figure 65. 6 mm steady refined mesh – vector Flow velocity
Figure 66. 6 mm steady refined mesh – Pressure
Figure 67. 6 mm steady refined mesh – TKE
Figure 68. 6 mm steady refined mesh – Tumble number
Figure 69. 6 mm Fluent mesh – Original mesh
Figure 70. 6 mm Fluent mesh – Flow velocity
Figure 71. 6 mm Fluent mesh – vector Flow velocity
Figure 72. 6 mm Fluent mesh – Pressure
Figure 73. 6 mm Fluent mesh – TKE
Figure 74. 6 mm Fluent mesh – Refined mesh
Figure 75. 6 mm Fluent refined mesh – Flow velocity
Figure 76. 6 mm Fluent refined mesh – vector Flow velocity
Figure 77. 6 mm Fluent refined mesh – Pressure
Figure 78. 6 mm Fluent refined mesh – TKE
List of tables

Table 1. Geometry of steady-state mesh.
Table 2. Operating point and boundary conditions of steady-state mesh.
Table 3. Engine data of transient mesh.
Table 4. Wall temperatures of transient mesh.
Table 5. Z-coordinate of planes to calculate an average tumble number for steady state simulations.
Table 6. Tumble number for each lift and type of simulation.
Table 7. Tumble number for refined and original mesh in FIRE and FLUENT programs.
Table 8. GANTT of thesis.
Table 9. Staff costs.
Table 10. Computer equipment costs.
Table 11. Total investment.
## List of the applied abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BDC</td>
<td>Bottom dead center</td>
</tr>
<tr>
<td>CA</td>
<td>Crank angle</td>
</tr>
<tr>
<td>CAFE</td>
<td>Corporate average fuel economy</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CNG</td>
<td>Compressed natural gas</td>
</tr>
<tr>
<td>CO</td>
<td>Carbon Monoxide</td>
</tr>
<tr>
<td>CO2</td>
<td>Carbon Dioxide</td>
</tr>
<tr>
<td>EGC</td>
<td>Exhaust gas content</td>
</tr>
<tr>
<td>HC</td>
<td>Hydrocarbons</td>
</tr>
<tr>
<td>IEA</td>
<td>International energy agency</td>
</tr>
<tr>
<td>IVC</td>
<td>Intake valve closed</td>
</tr>
<tr>
<td>IVO</td>
<td>Intake valve open</td>
</tr>
<tr>
<td>LPG</td>
<td>Liquid petroleum gas</td>
</tr>
<tr>
<td>NOx</td>
<td>Nitrogen oxides</td>
</tr>
<tr>
<td>PDE</td>
<td>Partial differential equation</td>
</tr>
<tr>
<td>PFI</td>
<td>Port fuel injected</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>RKE</td>
<td>Realizable k-ε model</td>
</tr>
<tr>
<td>RNG</td>
<td>Renormalization k-ε model</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds stress model</td>
</tr>
<tr>
<td>SI</td>
<td>Spark Ignition</td>
</tr>
<tr>
<td>SIDI</td>
<td>Spark-ignited direct-injection</td>
</tr>
<tr>
<td>SSF</td>
<td>Survey solution file</td>
</tr>
<tr>
<td>SULEV</td>
<td>Super ultra-low emission vehicle</td>
</tr>
<tr>
<td>TDC</td>
<td>Top dead center</td>
</tr>
<tr>
<td>TKE</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>TU</td>
<td>Tumble number</td>
</tr>
<tr>
<td>ULEV</td>
<td>Ultra-low emission vehicle</td>
</tr>
</tbody>
</table>
### List of applied symbols

#### Latin symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>area</td>
<td>[m²]</td>
</tr>
<tr>
<td>cp</td>
<td>specific heat</td>
<td>[J/Kg·K]</td>
</tr>
<tr>
<td>D</td>
<td>bore</td>
<td>[mm]</td>
</tr>
<tr>
<td>g</td>
<td>gravitational acceleration</td>
<td>[m/s²]</td>
</tr>
<tr>
<td>K</td>
<td>conductivity</td>
<td>[W/m·K]</td>
</tr>
<tr>
<td>L</td>
<td>characteristic linear dimension</td>
<td>[m]</td>
</tr>
<tr>
<td>q′′′</td>
<td>rate of internal heat generation</td>
<td>[W/m³]</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
<td>[s]</td>
</tr>
<tr>
<td>T</td>
<td>absolute temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>U</td>
<td>time-averaged velocity in x-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>Ũ</td>
<td>instantaneous velocity in x-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>U′</td>
<td>fluctuating velocity in x-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>u_i,u_j</td>
<td>Reynolds stress tensor</td>
<td>[m²/s²]</td>
</tr>
<tr>
<td>P</td>
<td>pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>q</td>
<td>flux</td>
<td>[m³/s]</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy</td>
<td>[m²/s²]</td>
</tr>
<tr>
<td>C1, C2, C3</td>
<td>constants in the model ε equation</td>
<td>[ ]</td>
</tr>
<tr>
<td>Gk, Gb</td>
<td>generation of turbulent kinetic energy</td>
<td>[kg/m²/s²]</td>
</tr>
<tr>
<td>YM</td>
<td>contribution of fluctuating dilatation</td>
<td>[kg/m²/s²]</td>
</tr>
<tr>
<td>x</td>
<td>X-coordinate</td>
<td>[ ]</td>
</tr>
<tr>
<td>y</td>
<td>Y-coordinate</td>
<td>[ ]</td>
</tr>
<tr>
<td>z</td>
<td>Z-coordinate</td>
<td>[ ]</td>
</tr>
<tr>
<td>u</td>
<td>velocity component in X-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>w</td>
<td>velocity component in Z-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>v</td>
<td>velocity component in Y-direction</td>
<td>[m/s]</td>
</tr>
<tr>
<td>V</td>
<td>volumetric flow rate</td>
<td>[m³/s]</td>
</tr>
</tbody>
</table>

#### Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>β</td>
<td>thermal expansion coefficient of air</td>
<td>[1/K]</td>
</tr>
<tr>
<td>δ</td>
<td>boundary layer thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>ε</td>
<td>dissipation</td>
<td>[m³/s²]</td>
</tr>
<tr>
<td>μ</td>
<td>dynamic viscosity</td>
<td>[Pa·s]</td>
</tr>
<tr>
<td>μt</td>
<td>dynamic turbulent viscosity</td>
<td>[Pa·s]</td>
</tr>
<tr>
<td>σk, σε</td>
<td>turbulent Prandtl number</td>
<td>[ ]</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>ν</td>
<td>kinematic viscosity</td>
<td>[Pa·s]</td>
</tr>
</tbody>
</table>
1 Introduction

The automotive engines are constantly improving, especially in terms of fuel economy and exhaust emissions. In this context, one of the raising topics is the research for alternative resources of energy like electricity or hydrogen. However, none of these alternative energy sources is yet mature enough to displace the classic internal combustion engines in all varieties of transportation means. Therefore, it is mandatory to keep improving the performance of internal combustion engines as they will still be the dominant propulsion units in the next decades.

A common indicator of the future trend in engine development is the proportion of each type of car in the total amount of cars sold, as well as the future sales plans. A global market study of sales of previous years run by IEA, shows that gasoline vehicles were the most sold type of vehicles followed by Diesel vehicles [1]. This situation is supposed to be permanent for the next 15 years (see Figure 1). During this time, important advances in the direction of reducing the emissions are expected, as long as companies are aware about the importance of improving their engines for fulfilling international legislation. Exhaust emissions from engines are a significant cause of pollution in the world and normative of emissions is becoming considerably more restrictive for upcoming combustion engines.

![Figure 1. Annual light-duty vehicle sales by technology type [1]](image)

Within internal combustion engines there are two main types of engine for the operation of cars. These are Spark-ignited (SI) engines and diesel engines. Regarding the emissions, SI engines represent an advantage over diesel engines, since diesel engines may produce on the average four times more carbon particles than SI engines [2]. On the other hand, diesel engines have a better thermal efficiency than SI engines. Nevertheless, both diesel and gasoline engines had in the last years important advances on their design reducing significantly their exhaust emissions, although there is still room for reducing them.

For this reason, the current trend is to reach a compromise between both types of engines, trying to apply the benefits of one of them into the other. A clear example is the evolution of SI engines from port-fuel-injection to direct injection, similar to diesel engines [3]. This change offers wide possibilities of improvement in emissions and fuel consumption for SI engines. Furthermore, direct injection SI (SIDI) engines can operate with homogeneous...
charge in the entire operating range or they can operate with homogeneous charge at higher loads and speeds, whereas at low loads and low speeds they operate with stratified charge. This brings an essential benefit of reducing gas exchange loses of the engines that are not equipped with fully variable valve trains. In the case of SIDI engines, air and fuel have to be accurately introduced into the cylinder to obtain a stoichiometric mixture near the spark plug to guarantee an appropriate ignition. This can easily be achieved when engine is working with homogeneous charge at full load, but much harder at part loads if engine is working with stratified charge. In order to solve this problem a method is needed to control the air induction into the cylinder, helping fuel and air to reach stoichiometric mixture near spark plug, thus allowing correct ignition even with excess of air. SIDI stratified engines use the tumble of the air vortex generated in the combustion chamber during the intake stroke. This tumble is the rotational component with a perpendicular axis to the axis of the cylinder. This is used to supply fuel after tumble of air is created in the cylinder, so mixture could reach the stoichiometric conditions near the spark plug, required for an appropriate combustion. However, having the stratified mixture ignited by a spark plug adds extra complexity to the use of this technology.

There are significant advantages of using SIDI engines with stratified charge. With this technology the amount of air introduced is higher than with homogeneous charge. As a result, the throttle is more open which means less friction of the air resulting in lower throttling losses. Besides, the temperature in the cylinder decreases so that it directly reduces exhaust emissions like NOx and lowers the heat transfer in the combustion chamber walls. Consequently, better fuel economy is achieved and thereby lower exhaust emissions. However, all these benefits can only be achieved if charge motion support generation of a stratified change with near stoichiometric air-fuel mixture at the spark plug.

In order to improve the performance of SIDI stratified engines, the control of the tumble represents the main goal. This is why the tumble number is used as a reference to assess the correct behavior of the charge motion in the cylinder. Understanding tumble number is crucial to use simulation programs. Creation of prototypes is expensive and leaves less room for small adjustments once the prototype is built. Using simulation programs to build and test prototypes overcome these limitations by significantly reducing the vast space of possible solution to only a limited number of prototypes that are afterwards actually tested experimentally. These programs are based on software called Computational Fluid Dynamics (CFD).

The aim of this research is to study the feasibility of the stratified charge of fuel in SIDI engines at partial load conditions. For this aim, steady-state and transient simulations are analyzed to explore their characteristics. Steady-state simulations are faster and commonly used in industry but they might be not accurate enough. On the other hand, transient simulations are more accurate but higher time and resource consuming.

For steady-state and transient simulations, the program used is CFD AVL Fire. In addition, in steady-state there is a refinement of the model mesh made by two different CFD programs to enable wider basis for comparison of the results.
2 General concepts

2.1 Comparing SIDI engine and PFI engine

Traditionally SI engines had an indirect fuel injection (PFI) where fuel is injected in the intake port. However, due to the aim of reducing emissions and fuel consumption the system was improved by using a direct injection (DI) (see Figure 2).

![Figure 2. Comparison of the PFI (left) and SIDI (right) mixture preparation system [4]](image)

In the PFI engine, fuel is injected into the intake port of each cylinder. However, there is an associated time lag between the injection and the induction of the fuel and air mixture into the cylinder. Most of current automotive PFI engines utilize timed fuel injection near intake valve when this gets closed. During cold starting, a transient film, or puddle, of liquid fuel gets formed in the intake valve area of the port [5]. This causes a delay in the fuel delivery and a measurement error due to partial vaporization, making necessary to supply an amount of fuel that significantly exceeds the required amount for the ideal stoichiometric ratio. As a result, this puddling and time lag may cause the engine to either misfire or experience a partial burn on the first 4–10 cycles, with an associated increase in the unburned hydrocarbons emissions [5]. A report from Nissan company states that the cold-start unburned hydrocarbons emissions obtained with the Nissan prototype gasoline SIDI engine are approximately 30% lower than the ones obtained with an optimized PFI engine under comparable conditions [6]. Alternatively, injecting fuel directly into the engine cylinder totally avoids the problems associated with fuel wall wetting in the port, while providing enhanced control of the metered fuel for each cycle, and a reduction of the fuel transportation time. The actual mass of fuel entering the cylinder on a cycle can be more accurately controlled in SIDI than in PFI. As a result of the higher operating fuel pressure of the SIDI system, the fuel entering the cylinder is better atomized than the PFI system. This better atomization decreases the temperature into the cylinder. Thereby, due to the lower temperature the compression ratio is higher and the result of that is higher thermal efficiency. Furthermore, it is worth to notice that a greater thermal efficiency allows a greater specific power, since more air and fuel can be supplied.
PFI engines have some limited advantages over SIDI engines due to the fact that the intake system acts as a pre-vaporizing chamber. When fuel is injected directly into the engine cylinder, the available time for mixture preparation is reduced significantly. As a result, the atomization of the fuel spray must be fine enough to permit fuel evaporation in the limited time available between injection and ignition. Unlike PFI system, SIDI atomization relies exclusively to the effectiveness of the injector, in case the system has any small failure fuel droplets that are not evaporated are very likely to participate in diffusion burning, or to exit the engine as unburned hydrocarbons emissions. These factors can contribute to levels of unburned hydrocarbons and particulate emissions, which can be easily higher than an optimized PFI engine [4]. Another advantage of PFI engines is the low-pressure fuel-system hardware.

Future emission regulations such as the ultra-low-emission-vehicle (ULEV), the super ultra-low-emission-vehicle (SULEV), and corporate average fuel economy (CAFE) requirements [7] give an important reason to develop SIDI engines considerable in a near future.

In summary, the potential advantages of the SIDI concept are too significant to receive a priority status. The concept offers many opportunities for achieving significant improvements in engine fuel consumption, while simultaneously causing large reductions in the emissions of engine unburned hydrocarbons. The current high technology PFI engine, although highly evolved, has nearly reached the limit of the potential of a system based on throttling and a port fuel film.

### 2.2 SIDI operating mode in stratified engines

In SIDI engines, when the engine is working at full load, the throttle is fully opened and the fuel is injected during the intake stroke to provide a homogeneous mixture. However, partial load can be processed differently using two charge modes, stratified or homogeneous charge [8] (Figure 3). During homogeneous charge, the adjustment of the load is done by throttling, while with stratified charge the throttle is less closed and the load is adjusted by fuel/air equivalence ratio. The benefit of SIDI engines with stratified charge is the improvement in the efficiency as a result of reduced pumping losses and higher compression ratios for the same amount of fuel.

SIDI engines with stratified charge require a precise definition of the necessary time for preparing the air/fuel mixture. At partial load conditions, fuel is injected during the compression stroke, approximately 75º before TDC, with a determined cylindrical turbulence (tumble effect) of the air at the end of the compression phase. This ensures that stoichiometric mixture is prepared in the vortex area that should be located in the vicinity of the spark plug, whereas this mixture is surrounded with the air (Figure 5) resulting in the overall excess air ratio (lambda) of 2 or 3 [9].

As aforementioned, SIDI engines work with two types of charge depending on the load: stratified and homogeneous charge, which is analyzed below.
Evaluation Of Numerical Methods For Modeling Flow Field During The Intake Stroke Of Spark-Ignited Engines

In homogeneous charge, the intelligent control of injection facilitates a homogeneous mixture at the higher regime (when the power of the engine is required, Figure 3). With homogeneous mixture, the air is induced during the intake stroke (1) (see Figure 4). Then, the high pressure injector injects the fuel through a long jet with cone shape (2), trying to obtain dispersion in the cylinder. The cooling effect of this dispersion helps to avoid spontaneous ignitions in the cylinder that may occur when the engine has a high compression ratio (3) leading to the controlled ignition by the spark plug (4).

In stratified charge, the mixture applied in the engine is overall lean when the vehicle works at part loads. To be able to make the engine working with this lean mixture and still remaining with the flammability limits, fuel must be applied in stratified way. The air/fuel mixture gets concentrated around the spark plug at a strategic center position in the combustion chamber, whose peripheral area is practically an air layer [7]. This measure allows for large throttle opening and thus results in increased efficiency and this in improved fuel economy. This improvement in the fuel consumption is also a consequence of the heat transfer decrease. The concentrated air at the periphery of the combustion chamber, while ignition is produced in the central zone of the chamber, provides a thermal isolation.

As shown in Figure 5, during the intake (1) the air volume flowing through the vertical intake ports flows along the curved surface of the piston (2) and reflows upwards creating
a powerful rotating flow. In the compression stroke, the rotatable flow is decomposed into numerous small vortices. Then, in the last phase of the compression stroke, the high-pressure injector sprays the fuel (3) into the vortex. This movement of the flow, along with the elevate density of the compressed air and the small vortices, keeps the sprayed jet of fuel concentrated in the vicinity of the spark plug. In the stratified mode it is mandatory that the air to fuel mixture is close to stoichiometric in the center and lean in the periphery, where it should be no fuel. As result of this specific stratification of the load, lambda varies between 2 and 3 in the combustion area while around the spark plug the mixture has a lambda near to 1.

The last phase is the spark (4), which initiates the combustion. This is controlled by the spherical piston cavity that is spreading through a chain reaction. The resulting improvement of all this process is improvement reduction of 20% in the gasoline consumption [10].

![Figure 5. Stratified mix cycle [10]](image)

### 2.3 Air flow structure

In SIDI engines with stratified charge, turbulent air motion controls the efficiency in the combustion in contrast to homogeneous injections, where flow structures are less pronounced. The air entering the chamber combustion creates a rotating flow structure after contacting with the geometry of the cylinder. The rotational component that has an axis parallel to the axis of the cylinder is denoted as swirl (Figure 6), and the component with a perpendicular axis to the axis of the cylinder is denoted as tumble (Figure 6). The magnitudes of both, the swirl and tumble components, are highly dependent on the geometry of the intake port design, the intake valve geometry, the bore/stroke ratio, the shape of combustion and the engine speed [11].
Swirl is created by bringing the intake flow into the cylinder with an initial angular momentum. While some decay in swirl occurs during the engine cycle due to friction, the generated intake swirl usually persists through the compression, combustion, and expansion processes. In engine designs with bowl-in-piston combustion chambers, the rotational motion set up during the intake is modified during compression. Swirl is used to promote a more rapid mixing between the induted air charge and the injected fuel [12]. 

The tumble component of the flow is transformed into turbulence near TDC by tumble deformation and the associated velocity gradients. The fuel flow is deflected from a shaped cavity in the piston, and the vapor and liquid fuel are transported to the spark plug chamber. The tumble component of the motion tends to decay into large-scale secondary flow structures due to the effect of the curved cylinder wall. With respect to the turbulence generation, the presence of a significant tumble component is effective in enhancing the turbulence intensity at the end of the compression stroke, which is able to compensate the reduced flame speed of a lean stratified mixture.

The role of turbulent mixture motion becomes very important for lean operation of spark ignition engines where burning rates are much lower than in stoichiometric engines. Given its easier generation in the four-valve SI engine, the tumble motion has been used more often than swirl for the generation of turbulence in the cylinder. Since there is a strong correlation between tumble strength and turbulence levels at ignition, it has been suggested that a stronger tumble motion, achieved by modifying the inlet ports, may result in a faster burn rate [13].
3 Computational Fluid Dynamics (CFD) simulations

Nowadays, automotive companies spend more and more resources in research, trying to find new products or technologies, which require prototypes to test the new solutions. Construction of prototypes might be difficult, very expensive, and highly time consuming. Therefore, the appearance of computer programs able to simulate any kind of model required by the companies constitutes an important advance. These programs work with software based on computational fluid dynamics (CFD). CFD is a field of fluid mechanics that uses algorithms and numerical methods to solve problems of fluid flows. Computers are used to perform required calculations to simulate the interaction of gases and liquids with surfaces defined by boundary conditions. Since the field of CFD has made rapid progress, CFD technology has been increasingly applied to the research and development of engines. Engine simulations with CFD may be an invaluable tool to an engine designer, since it could provide insight into the highly complex physical and chemical processes in the engine. Novel design ideas could be introduced and analyzed faster and at lower cost than using conventional methods.

In all of these CFD codes the same basic procedure is as follow.

- The geometry of the model is defined.
- The model is divided into discrete cells (the mesh). The mesh may be structured or unstructured.
- Boundary conditions are defined.
- The simulation is started as a steady-state or transient.
- Finally in the postprocessor there is an analysis and visualization of the resulting solution.

There are several factors in CFD codes that have to be taken into account when simulating. For the present case, the two basic factors are the turbulence model and the meshing.

3.1 Turbulence models

3.1.1 Introduction

Turbulence is a type of fluid (gas or liquid) flow in which the fluid experiment irregular fluctuations, or mixing, in contrast to laminar flow, in which the fluid moves in smooth paths or layers. Figure 7 is an example of how a laminar fluid becomes turbulent. In turbulent flow the velocity vector features variation in both magnitude and direction. Turbulence can be visualized as consisting of irregular swirls of motions called eddies [14]. Usually turbulence consists of many different size eddies superimposed on each other.
A turbulent flow has a number of characteristic such as:

- Irregularity: Turbulent flow is irregular, random and chaotic. The flow consists of a spectrum of different scales (eddy sizes) where largest eddies are of the order of the flow geometry (i.e. boundary layer thickness, jet width, etc.).
- Diffusivity: In turbulent flow the diffusivity increases. This means that the spreading rate of boundary layers, jets, etc. increases as the flow becomes turbulent. The increased diffusivity also increases the resistance (wall friction) in internal flows [15].
- Large Reynolds Numbers: Turbulent flow occurs at high Reynolds number, which means the turbulent flow is dominated by inertial forces. The Reynolds number is defined to be the ratio of inertial forces to viscous forces and consequently quantifies the relative importance of these two types of forces for given flow conditions [16].

\[ Re = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{\rho v L}{\mu} = \frac{v L}{v} \]  

(3.1)

- Three-Dimensional: Turbulent flow is always three-dimensional. However, when the equations are time averaged, the flow can be treated as two-dimensional [15].
- Dissipation: Turbulent flow is dissipative, which means that kinetic energy in the small eddies are transformed into internal energy. The small eddies receive the kinetic energy from slightly larger eddies. The slightly larger eddies receive their energy from even larger eddies and so on. The largest eddies extract their energy from the mean flow. This process of transferred energy from the largest turbulent scales (eddies) to the smallest is called cascade process [15].
- Continuum: Even though there are small turbulent scales in the flow, they are much larger than the molecular scale and thus the flow can be treated as a continuum.
Fluctuations combine transported quantities such as momentum and energy which fluctuate as well. These fluctuations can be of a small scale and, in direct simulations, the smallest vortex needs to be resolved over multiple cells resulting in a huge number of cells for macroscopic problems, creating large computational expenses for calculations.

To reduce such computational expenses, the equations of motion used to describe the flow (Navier Stokes equations, i.e. continuity, momentum and energy equation) are averaged with respect to time so that these turbulent transport models predict the effect of turbulence on the time averaged mean motion [17]. Since all of the terms currently in the equations of motion are instantaneous values, they are decomposed to their mean value ($\langle U \rangle$) and fluctuating part ($U'$), see 3.2.

$$U = \langle U \rangle + U'$$ (3.2)

Therefore, a new set of equations of motion with the same form are obtained with the addition of this time-averaging.

Continuity:

$$\frac{d}{dx}(p\bar{u}) + \frac{d}{dy}(p\bar{v}) + \frac{d}{dz}(p\bar{w}) = 0$$ (3.3)

Momentum in x-direction:

$$\frac{d}{dt}(p\bar{u}) + \frac{d}{dx}(p\bar{u}\bar{u}) + \frac{d}{dy}(p\bar{v}\bar{u}) + \frac{d}{dz}(p\bar{w}\bar{u}) = -\frac{dp}{dx} + (\mu + \mu_t)\left(\frac{d^2\bar{u}}{dx^2} + \frac{d^2\bar{u}}{dy^2} + \frac{d^2\bar{u}}{dz^2}\right)$$ (3.4)

Momentum in y-direction:

$$\frac{d}{dt}(p\bar{v}) + \frac{d}{dx}(p\bar{u}\bar{v}) + \frac{d}{dy}(p\bar{v}\bar{v}) + \frac{d}{dz}(p\bar{w}\bar{v}) = -\frac{dp}{dy} + (\mu + \mu_t)\left(\frac{d^2\bar{v}}{dx^2} + \frac{d^2\bar{v}}{dy^2} + \frac{d^2\bar{v}}{dz^2}\right)$$ (3.5)

Momentum in z-direction:

$$\frac{d}{dt}(p\bar{w}) + \frac{d}{dx}(p\bar{u}\bar{w}) + \frac{d}{dy}(p\bar{v}\bar{w}) + \frac{d}{dz}(p\bar{w}\bar{w}) = -\frac{dp}{dz} + (\mu + \mu_t)\left(\frac{d^2\bar{w}}{dx^2} + \frac{d^2\bar{w}}{dy^2} + \frac{d^2\bar{w}}{dz^2}\right)$$ (3.6)

Energy:

$$\frac{d}{dt}(\rho c_p \bar{T}) + \frac{d}{dx}(\rho\bar{u}c_p \bar{T}) + \frac{d}{dy}(\rho\bar{v}c_p \bar{T}) + \frac{d}{dz}(\rho\bar{w}c_p \bar{T})$$

$$= K \left(\frac{d^2\bar{T}}{dx^2} + \frac{d^2\bar{T}}{dy^2} + \frac{d^2\bar{T}}{dz^2}\right) + \frac{d}{dx_i} \left(\frac{d\bar{T} c_p \mu_t}{dx_i \sigma_t}\right) + q''''$$ (3.7)

The consequence of time-averaging the equations of motion is the introduction of additional unknown variables due to the influence of turbulence on the mean values. For this reason new models are needed to solve these unknown variables. These models are called turbulent models and cannot be universally applied to all situations. The appropriate
turbulence model depends on the level of accuracy, physics encompassed in the flow and computation resources available.

In the industrial CFD codes the turbulence models mostly used are [17]:

- **k-ε Models**
  - Standard k-ε model
  - Renormalization k-ε model (RNG)
  - Realizable k-ε model (RKE)
- **k-ω models**
  - Standard k-ω model
  - Shear-stress transport k-ω model (SST)
- **k-ζ-f**
- **Reynolds stress model (RSM).**

Other models as PANS and large-eddy are also commonly used but they are characterized by more hardware capabilities and by longer computational times.

### 3.1.2 k-ε Models

These models fall into the class of eddy viscosity models. Two transport equations are derived which describe transport of two scalars like the turbulent kinetic energy $k$ and its dissipation $\varepsilon$.

\[
\frac{d}{dt} (\rho \varepsilon) + \frac{d}{dx_i} (\rho \varepsilon U_i) = \frac{d}{dx_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{d \varepsilon}{dx_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (P_k + C_3 \varepsilon G_k) - C_2 \varepsilon \rho \frac{\varepsilon^2}{k} \tag{3.8}
\]

\[
\frac{d}{dt} (\rho k) + \frac{d}{dx_i} (\rho k U_i) = \frac{d}{dx_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{d k}{dx_j} \right] + G_k - \rho \varepsilon + P_k \tag{3.9}
\]

Where:

\[
P_k = \rho \bar{u}_i \bar{u}_j \frac{d}{dx_j} (\rho U_i) = \mu_t \left( \frac{d}{dx_j} (U_i) + \frac{d}{dx_i} (U_j) \right) \frac{d}{dx_j} (U_i) \tag{3.10}
\]

In these equations, $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as described in Modeling Turbulent Production in the k-ε Models. $C_{1\varepsilon}$, $C_2 \varepsilon$, and $C_3 \varepsilon$ are constants. $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$, respectively. $S_k$ and $S_\varepsilon$ are user-defined source terms.

The turbulent (or eddy) viscosity, $\mu_t$, is computed by combining $k$ and $\varepsilon$ as follows [18]:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3.11}
\]

where $C_\mu$ is a constant.
The model constants $C_1\varepsilon$, $C_2\varepsilon$, $C_\mu$, $\sigma_k$ and $\sigma_\varepsilon$ have the following default values:

$$C_1\varepsilon = 1.44, \quad C_2\varepsilon = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \quad (3.12)$$

### 3.1.2.1 Standard k-$\varepsilon$ Model

The standard k-$\varepsilon$ model is highly used in turbulence modeling. In this model the turbulence kinetic energy $k$ and its rate of dissipation $\varepsilon$ are obtained from the following transport equations [19]:

$$\frac{d}{dt}(\rho k) + \frac{d}{dx_i}(\rho k u_i) = \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_\tau}{\sigma_k}\right)\frac{dk}{dx_j}\right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (3.13)$$

and

$$\frac{d}{dt}(\rho \varepsilon) + \frac{d}{dx_i}(\rho \varepsilon u_i) = \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_\tau}{\sigma_\varepsilon}\right)\frac{d\varepsilon}{dx_j}\right] + C_1\varepsilon(G_k + C_3\varepsilon G_b) - C_2\rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (3.14)$$

$G_b$ is the generation of turbulence kinetic energy due to buoyancy. $Y_M$ represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.

Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism [19].

### 3.1.3 k-$\zeta$-f model

The k-$\zeta$-f model has been developed by Hanjalic, Popovac and Hadziabdic (2004). The authors propose a version of an eddy viscosity model based on Durbin’s elliptic relaxation concept [20]. The aim is to improve numerical stability of the original $\nu^2$–f model, which is similar to Standard k-$\varepsilon$ although it incorporates also some near-wall turbulence anisotropy as well as non-local pressure-strain effects. The $\nu^2$–f model introduces the wall boundary condition for the elliptic relaxation function $f$ (3.16) proportional to $1/y^4$ (y is a dimensionless wall distance) making computations more sensitive near wall cells [20].

The transport equation for $\nu^2$ is shown as [21]:

$$\frac{d\nu^2}{dt} + U_j \frac{d\nu^2}{dx_j} = k f - \frac{\nu^2}{k} \varepsilon + \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_\tau}{\nu^2}\right)\frac{d\nu^2}{dx_j}\right] \quad (3.15)$$

And the elliptic equation for the relaxation function $f$ is [21]:

$$L^2 \nabla^2 f - f = \frac{C_1 - 1}{\gamma} \left(1 - \frac{2}{3}\right) - C_2 \frac{p_k}{\varepsilon} \quad (3.16)$$

Hanjalic proposed an eddy viscosity model, which solves a transport equation for the velocity scale ratio $\nu^2/k$ instead of $\nu^2$ (3.15). With that, a more robust wall boundary condition for $f$-equation is introduced, this time $f$ is proportional to $1/y^2$ [20]. On
average, the computing time is increased by up to 15% [20] when compared with the computing time needed for the k-ε model calculations. Nevertheless, the model is usable for a relatively coarse mesh next to the wall.

This turbulence model has been chosen for simulations as it is sufficiently robust to be used for computations involving grids with moving boundaries and highly compressible flows as it is the case in internal combustion engines.

3.2 Meshing

The partial differential equations that govern fluid flow and heat transfer (3.13 and 3.14) can generally be solved analytically only for specific simplified cases, whereas in general they cannot be solved analytically. Therefore, in order to analyze fluid flows, flow domains are split into small cells. The governing equations are then discretized and solved inside each of these subdomains. The subdomains are often called elements or cells, and the collection of all elements or cells creates a mesh. The process of obtaining an appropriate mesh is named mesh generation.

As CFD has developed, better algorithms and more computational power are necessary to CFD analysts. One of the direct results of this development is the expansion of available mesh elements and mesh connectivity. The easiest classifications of meshes are based upon the connectivity of a mesh or on the type of elements present [22].

3.2.1 Connectivity of a mesh

The connectivity represents how cells are connected to one another. There are three ways to classify the connectivity of a mesh.

3.2.1.1 Structured Meshes:

A structured mesh can be recognized by all interior nodes of the mesh having an equal number of adjacent elements (Figure 8). The mesh generated by a structured grid generator is typically all quadrilaterals in 2D or hexahedral in 3D.

![Structured Quadrilateral Mesh for an Airfoil](image)

Figure 8. Structured Quadrilateral Mesh for an Airfoil [23]

3.2.1.2 Unstructured Meshes:

Unstructured mesh generation use any possible element that a solver might be able to use. Triangle and Tetrahedral meshes are most commonly (Figure 9), although quadrilateral and hexahedral meshes can also be unstructured. While there is certainly some overlap between
structured and unstructured mesh generation technologies, the main feature which distinguish the two fields are the unique iterative smoothing algorithms employed by structured grid generators [24].

3.2.1.3 **Hybrid Meshes:**

A hybrid mesh is a mesh consisting of structured portions and unstructured portions. Hybrid meshes rely on the observation that most of the advantage of irregular meshes can be obtained by interspersing irregular operations in a preponderance of regular refinements [25]. The Figure 10 and Figure 11 are examples of hybrid structures.

3.2.2 **Type of elements**

Depending upon the analysis type and solver requirements, meshes generated could be 2-dimensional (2D) or 3-dimensional (3D) [23]. Common elements in 2D are triangles or rectangles (Figure 12).
Most popular 3D mesh elements are hexahedra, tetrahedron, square pyramids and extruded triangles (Figure 13).

Figure 13. 3D elements: hexahedra, tetrahedra, square pyramid and extruded triangle [23]

3.2.3 Refinements

Once the basic mesh is made, it is possible to enhance some parts of the mesh depending on the relevance or the interest in results. Thus, the aim is to improve the cells dividing them into smaller cells. Consequently, the accuracy of the results is enhanced and discretization is better.

Refine is a technique that gives users freedom to choose where to perform and how much. There are two main ways.

3.2.3.1 Selecting zones

This is a simple technique to perform refinement. Selecting zones means the user selects exactly the parts of the mesh which need to be improved. In that case if some part of the mesh has a special interest, the refinement would be performed just selecting this part and setting up the desired size of the cells.

It is very common to perform the refinement at the same time that the mesh is being made. In that case the meshing program would make a basic mesh and in those selected parts, the program would make the refinement previously set up. This is a fast method to obtain a good mesh when the user already knows which points would be important for the results of the simulation. However, the main disadvantage is the high number of cells refined that may not result in the improvement of the results but make the model more complex.

3.2.3.2 Momentum error

By this method, first the model has to be simulated to obtain a steady-state data where the momentum error has a defined value. When this value is known, the simulation is restarted from a point where stable values are already achieved (convergence state). In this new simulation is added the information of the maximum momentum error value (a value of 2 in this case) where all cells with higher value are refined.

Therefore, once the simulation starts, after a determined number of iterations (200 for example), the mesh is refined in all those cells with higher momentum error than the limit imposed. Afterwards, during the next iterations, the new refined mesh obtains new values of momentum error in its cells. Consequently, 200 iterations later the cells are evaluated and refined again in those with a value still over the limit. This process of refinement is performed as many times as desired until the mesh acquires a suitable number of cells.
As a result of this method, the refinement is located in cells where relatively high momentum error exists; this usually happens near the valve and thus where the results need to be more accurate.

### 3.2.3.3 Turbulence intensity

Refining by turbulence intensity means the mesh is denser where cells have high turbulence intensity. However, refinement based on solely turbulence intensity turned out not to be sufficient in the analyzed case as regions around the valves are not refined sufficiently. Therefore, an alternative problem specific approach was developed, where refinement is performed based on the turbulence intensity divided by absolute value of the Z-coordinate as the regions refined are in the negative and positive region of Z-coordinate. The higher turbulence intensity and closer to a Z-coordinate of zero (where the region is near the valves), the higher parameter there is in a cell, meaning that a refinement needs to be done. To avoid discontinuity in the equation there is a very small term (0.01) added to the value of |z|. The equation is as follow:

\[
\text{Parameter} = \frac{\text{Turbulence intensity}}{|z| + 0.01} \tag{3.17}
\]
4 Model presentation

The characteristic of the model depends on the type of simulation. The model required for steady-state simulations is different for the one required for transient simulations. In both types of simulations, the model needs to include an inlet and an outlet that represent the input and output of the air in the cylinder. For the particular case of steady-state simulations the model is represented at a determined crank angle, so the flow is constant in time. In order to set the inlet and the outlet, inlet is located in the intake port. However, since in this type of simulation the engine is represented during intake stroke so that exhaust port is cut off, the outlet must be located at the bottom of the cylinder (Figure 14).

On the other hand, in transient simulations the model does not have this constrain, since the model changes along the crank angle. Therefore, inlet and outlet are not required to be in the model at the same time as in the case of steady-state. In transient there is an inlet represented by the intake port and the outlet by the exhaust port, same as in real engines.

These variations in the location of the outlet may affect directly in the accuracy of the results. The adaptation done in steady-state simulations avoid the possibility of the creation of a real model, as a result, the behavior of the air is influenced and the results might not be precise enough. However, even though transient simulations use a model exactly as a real one, the time required to obtain the results is far higher than steady-state simulations.

4.1 Describing the engine

4.1.1 Steady-state simulations:

4.1.1.1 Geometry:

The steady-state model is a half-model engine with a symmetry plane that permits to simulate the model as if the whole engine was modeled (Figure 15 and Table 1).

Table 1. Geometry of steady-state mesh.

<table>
<thead>
<tr>
<th>Geometric data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>84 mm</td>
</tr>
<tr>
<td>Cylinder height</td>
<td>252 mm</td>
</tr>
</tbody>
</table>
For steady-state simulations, the boundary conditions are an important input as the simulation starts iterating from those defined conditions (Table 2) and they remain constant during the entire iterative process.

Table 2. Operating point and boundary conditions of steady-state mesh.

<table>
<thead>
<tr>
<th>Operating point and boundary conditions</th>
<th>Mixture (inlet and cylinder)</th>
<th>Total pressure at inlet</th>
<th>Static pressure at outlet</th>
<th>Temperature at inlet</th>
<th>Wall temperatures</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mixture (inlet and cylinder)</td>
<td>Air</td>
<td>1 bar</td>
<td>0.9 bar</td>
<td>293 K</td>
<td>293 K</td>
</tr>
</tbody>
</table>

4.1.2 Transient simulations:

4.1.2.1 Main engine data:

Transient model has the same geometry as steady-state model; the difference is in all the considerations that have to be defined (Table 3) because piston and valves move along crank angle (Figure 27).
Table 3. Engine data of transient mesh.

| Data             |  
|------------------|------------------|
| Bore             | 84 mm            |
| Stroke           | 89.6 mm          |
| Connecting rode  | 144.35 mm        |
| Compression ratio| 10.2 /10.24      |
| Engine speed     | 2000 rpm         |
| Ex-Valve open    | 154 deg (0.2 mm) |
| Ex-Valve close   | 402 deg (0.2 mm) |
| In-Valve open    | 338 deg (0.2 mm) |
| In-Valve close   | 588 deg (0.2 mm) |

Table 4 shows the temperatures of the walls that remain constant during the simulation.

Table 4. Wall temperatures of transient mesh.

<table>
<thead>
<tr>
<th>Temperatures [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Intake Port</td>
</tr>
<tr>
<td>Intake Valve</td>
</tr>
<tr>
<td>Exhaust Valve</td>
</tr>
<tr>
<td>Piston</td>
</tr>
<tr>
<td>Liner</td>
</tr>
<tr>
<td>Dome</td>
</tr>
<tr>
<td>Exhaust Port</td>
</tr>
<tr>
<td>Intake Seats</td>
</tr>
<tr>
<td>Exhaust Seats</td>
</tr>
</tbody>
</table>

Since transient mesh changes along with CA, for the entire period of calculation it is necessary to specify the boundary conditions. Boundary conditions can be fixed (Table 4) or can vary with time. Figure 16 and Figure 17 show the evolution of the boundary
condition of temperature and pressure in the intake and exhaust ports along with the results of the calculated in-cylinder pressure. The simulation was started at 154° CA and thus values of in-cylinder pressure and temperature at 154° CA in Figure 16 and Figure 17 correspond to initial conditions for the cylinder domain. Boundary conditions of both ports in Figure 16 and Figure 17 are specified only during their opening.

![Initial Temperature](image)

*Figure 16. Boundary and initial temperatures*

![Initial Pressure](image)

*Figure 17. Boundary and initial pressures*

4.2 Describing the mesh

4.2.1 Steady-state simulations:

In this kind of simulations the mesh is the same during the simulation, this means there is no change in the shape along iterations, and only one steady-state solution is sought. Therefore, the simulation is computationally less demanding and thus for example the duration of a simulation was around 2 days on a Core 2 Quad processor with 2.40 GHz and 8 GB RAM. A transient simulation can last for more than a week. For this reason, companies usually use this kind of simulations to be able to try different configurations and
models in a short period of time. Those companies use a number of computers to help calculating the same simulation even faster.

As it was explained previously, in steady-state the flow is constant in time, as if the mass flow entering the system is the same as the mass flow leaving the system. Consequently, the model needs an inlet and an outlet. For this case the inlet is situated in the intake port where the air flow is supplied. However, the outlet cannot be in the exhaust port because in such CA configuration simulated the exhaust valve is closed and it is represented directly cutting of the exhaust port. As a result, the only outlet possible is where the piston is supposed to be in the model (Figure 18).

![Figure 18. boundary selections: inlet (blue), Outlet (yellow) and symmetry (brown)](image)

Nevertheless, if the outlet is too near the valve, the turbulences of the flow may affect as there is no space to develop all turbulent movements. Therefore, an extrusion of the cylinder is required (Figure 19).

![Figure 19. Adaptation for a steady-state simulation [27]](image)

After the final mesh is generated, it is necessary to select the settings of the simulation and the conditions of the mesh. Some of this information introduced for a steady-state simulation is directly given by the AVL company [27]. This information is introduced in the Solver steering file (SSF). Here the simulation is selected as steady-state, the turbulence model is K-ε-f, the fluid is selected as compressible and other data about specifications of the motion equations are included.
Since it is not possible to solve the entire system matrix directly, an iteration procedure is applied to solve the system matrix. In steady-state simulations after a high number of iterations the results are considered converged (see Figure 39). In this case the maximum number of iterations is 3000, whereas solution can converge faster.

4.2.2 Transient simulations:
In the case of transient simulations the mesh changes along crank angle as intake and exhaust valves are open or closed.

In this study, the simulated process and thus also the mesh starts in a position of 154° of crank angle (26° before BDC). This is the moment when exhaust valve opens to allow the outflow of exhaust gases from the cylinder and just before the BDC (180 CA) when the exhaust stroke starts. The simulation calculates an entire cycle, which means a movement
of 720° of the crank, so until 874° of crank angle. During these 720° the mesh has 4 positions depending on the valves opened or closed. These positions are Intake valve closed – Exhaust valve open (Figure 20), Intake valve open – Exhaust valve open (Figure 21), Intake valve open – Exhaust vale closed (Figure 22) and Intake valve closed – Exhaust valve closed (Figure 23). The way to represent a valve closed is closing the corresponding port, like it does not exist.

To simulate the alternation of 4 different meshes in 1, it is necessary to join those meshes in a dynamic mesh that changes its shape along with the crank angle. In this study, the CFD program includes a tool which create this dynamic mesh after specifying the moving parts of the model and the crank angle were those parts are introduced to perform the simulation (Table 3). In order to coordinate the piston with the variation of crank angles, it is introduced a linear equation. However, for the movement of the valves a valve lift table is necessary (Figure 27). Once the mesh is completely created, there is the same basic procedure as in steady-state. In the SSF boundary conditions (Table 4) and other settings of the numerical solver [28] are introduced, similar to steady-state but in this case the simulation is selected as variable by crank-angle.

### 4.3 Valve lift

To be able to compare both kinds of simulations, in the case of steady-state simulation, the simulations are made in three specific position of the intake valve. These positions are at 10 mm (Figure 24), 6 mm (Figure 25) and 2 mm (Figure 26) valve lift.

![Figure 24. 10 mm valve lift](image1)
![Figure 25. 6 mm valve lift](image2)
![Figure 26. 2 mm valve lift](image3)

Therefore, results of transient simulation are compared to those of the steady-state simulation at exactly these valves opening that correspond to the following crank angle positions 460° CA (80° before BDC), 522° CA (18° before BDC) and 555° CA (15° after BDC) respectively (Figure 27).
4.4 Tumble number

Tumble number is a dimensionless indicator. The result of this number is a reference to compare and reason discrepancies between the tumble number calculated by steady-state and transient simulation.

In order to calculate the tumble number for steady-state simulations, five horizontal cross-sectional planes should be inserted in the geometry. These are used to define a volume-averaged quantity where the tumble axis is parallel to the X-axis. The planes have the following Z-coordinate:

<table>
<thead>
<tr>
<th>TU1</th>
<th>-32 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>TU2</td>
<td>-37 mm</td>
</tr>
<tr>
<td>TU3</td>
<td>-42 mm</td>
</tr>
<tr>
<td>TU4</td>
<td>-47 mm</td>
</tr>
<tr>
<td>TU5</td>
<td>-52 mm</td>
</tr>
</tbody>
</table>

On each plane $n$ (Table 5) the momentum around the X-axis should be calculated as:

$$M_{x,n} = \sum_{i=1}^{N} \rho_i A_{i,n} (Y_i \cdot w_i) \; \text{abs}(w_i)$$  
(4.1)
Where

- $\rho_i$: Static density of air at face area $i$.
- $A_{in}$: Face area $i$ on tumble plane $n$.
- $Y_i$: Y-coordinate of center of face $i$.
- $w_i$: Velocity component in Z-direction at center of face $i$.

The tumble number on each plane $n$ is given as:

$$TU_n = \frac{D}{2\rho Vz} M_{x,n}$$

(4.2)

Where

- $D$: Bore.
- $\rho$: Area averaged static density of air at outlet.
- $Vz$: Volumetric flow rate at the outlet.

The tumble number is then the average of the five planes.

$$TU = \frac{\sum_{n=1}^{5} TU_n}{5}$$

(4.3)

The resulting tumble area defined by the planes is shown in Figure 28.

![Figure 28. Tumble area (brown) in a steady-state mesh](image)

The methodology to calculate tumble number in transient simulations needs a modification regarding the steady-state method. The variation of the area inside the cylinder due to the movement of the piston prevents the possibility of creating a fixed number of plans where tumble number is calculated. Instead, the formula needs to be adapted to the remaining
area. For this reason an algorithm is developed to calculate tumble number with similar
definition as in steady-state simulation but with the conditions of the transient simulation.
Such algorithm can be found in the attachment 8.1 at the end of this thesis. Unlike at
steady-state simulations, in transient simulation the area where tumble number is
calculated consists of the entire area of the cylinder at each CA.
5 Results

In order to analyze the simulations results (besides the tumble number), we analyzed the data obtained on flow velocity, pressure, temperature and turbulence kinetic energy. Also, this data was plotted to explain the differences between methods. All of them take influence on tumble number, which is the reference taken for stratified engines.

Flow rotations can be observed through the velocity. A vortex is detected by circular areas with no velocity surrounded by higher velocities. Since this is not always enough to find vortexes, plotting the direction of certain points within cylinder gives an estimation of the global movement of the flow. Thus, an approach of the tumble effect can be represented. In addition, the Z-component of the velocity affects considerably the tumble number as it has been shown previously in the equations 4.1, 4.2 and 4.3.

When a valve first opens, air flow moves from the high pressure regions to the low pressure regions. Therefore, the pressure in the combustion chamber could indicate how the flow can react and then predict the flow’s trend, which is useful to explain backflows and other singularities of the flow. A vortex is recognizable by the low pressure zone in the center of it. Another important note is that a change of pressure affects directly to the temperature of the fluid.

Other parameters to take into account are the temperature and the turbulence kinetic energy (TKE). When temperature is too high, it may cause a change in the composition of the fluid. Such change may affect the natural behave of that fluid. The intensity of the turbulence is directly related to the TKE. TKE shows the regions where turbulence intensity is low, characteristic to detect boundary layers, or high. The vortex that stratified charge engines use; needs a high TKE region in the periphery of the vortex so that such vortex contains the sufficient energy to last until fuel is injected.

5.1 Analyses and comparisons of the steady-state and transient results

As transient simulation is continuous in crank angle, only three positions are selected to compare transient and steady-state simulations in this thesis. These positions are when the valve gap is 2, 6 and 10 mm. In transient simulation those gaps are situated in crank angles of 555° (2 mm), 522° (6 mm) and 460° (9.7 mm), when intake valve is closing. Therefore, this phase takes place just before fuel is applied into the engine, given that the engine is working at part load with stratified charge (late injection).

The reason why in transient simulation the choosen position is 9.7 mm instead of 10 mm is because it corresponds to the lift-curve limit of the valve in this mesh. However, in the steady-state simulation the mesh was directly given with 10 mm gap. In any case, the small difference may be considered negligible for the purpose of this analysis.

Before starting to compare both simulations, it is important to show basic information of the pressures at inlet and at a central point of the cylinder for both simulations (see Figure 29). It should be noticed that as transient mesh is changing according to the crank angle,
inlet pressure is only plotted when intake valve opens, this is from 338° CA (22° before TDC) to 588° CA (48° after BTD). Another important data to point out is that, as shows Table 2, the pressure at the inlet in steady-state is a constant value of 1 bar.

![Comparing pressures](image)

**Figure 29. Inlet and cylinder pressure in transient and inlet and cylinder pressure in steady-state**

Figure 29 shows the fluctuation of pressures in transient simulations where there are backflow periods that are analyzed in detail later. On the other hand, in the steady-state simulations there are not such periods. It is worth to note the decrease of pressure at the inlet due to compensation of pressures when intake valve opens (IVO) and the increase of pressure at both cylinder and inlet when exhaust valve closes at 402° CA (see Figure 27). On the other hand, steady-state pressures remain constant at all valve lifts. Consequently, the gradient of velocities remains high and there are no backflow periods. However, in order to completely determinate the behavior of the flow, other parameters need to be plotted.

**5.1.1 Steady-state results**

To proceed with the analysis of the results in steady-state, the configuration of the model is at a valve lift of 2 mm, 6 mm and 10 mm. As in steady-state simulations there is not time history, this type of simulation does not differ between the valve opening or closing, which is not the case in transient simulations.

In this section the velocity, the pressure and the TKE for each valve gap are considered to describe the behavior of the flow within the cylinder and thus the tumble number. A basic consideration previous to the following analysis is that the configuration of the intake valve at the cylinder gives asymmetry to the flow field and thus other parameters. The air inflows from the left and the valve disperse it towards the walls. Besides, the orientation of the intake port adds extra orientation to the main flow that tends to the right side of the cylinder.
5.1.1.1 10 mm valve lift

Figure 30. 10 mm steady mesh – Flow velocity

Figure 31. 10 mm steady mesh – vector Flow velocity

Figure 32. 10 mm steady mesh – Pressure

Figure 33. 10 mm steady mesh – TKE
Figure 32 shows a high gradient of pressures between the combustion chamber and the intake port which results in high velocities near the valve (Figure 30). In the right part of the cylinder, the geometry of the valve and the channel impose main flow direction that is inclined for approximately 45º (Figure 31). After the flow reaches the wall it repels and thus contributes to the generation of the vortex below the rightmost edge of the valve (Figure 31). This high velocity region is also characterized by the high intensity of TKE (Figure 33). Besides, below the leftmost edge of the valve, the vortex below the rightmost edge of the valve of the flow repelled from the wall and the inflowing air in this region create a small vortex (Figure 31). Observing at the tumble area (Figure 28), the flow field shows a high velocity flow going down in the right side of the cylinder (Figure 31) and at the left side there is another flow in opposite direction. As a result, looking at the momentum component (4.1) of the tumble number equation (4.2), the $Y_i \cdot w_i$ factor is negative for both flows and contains a high value. Moreover, the high gradient of velocities in the area raises the tumble number value due to the component $abs(w_i)$ of the momentum equation (4.1). Consequently, the global tumble number at 10mm valve lift has a value of -1.731 (Figure 34).

5.1.1.2 6 mm valve lift

Figure 35. 6 mm steady mesh – Flow velocity

Figure 36. 6 mm steady mesh – vector Flow velocity
Figure 37. 6 mm steady mesh – Pressure

Figure 38. 6 mm steady mesh – TKE

Figure 39. 6 mm steady mesh – Tumble number.
The difference in pressure between inlet and cylinder (Figure 37) is similar to 10mm valve lift (Figure 32), which means also high velocities near the valve (Figure 35). The reduction of the valve gap also reduces the inclination of the main flow in the right side of the cylinder (Figure 36) that reaches the wall and is repelled. This flow originates a vortex below the rightmost edge of the valve closer to the valve than at 10mm valve lift (Figure 36). The smaller valve gap obstructs the inflowing air to the left side of the cylinder that, along with the vortex at the right side, leads the air to flow along the wall (Figure 36). Those gradients of velocity near the wall explain the high intensity of the TKE in those regions (Figure 38), although the lower TKE comparing with 10mm is a result of a lower velocity in the cylinder. Observing the flow field, it is possible to see the boundary layer along the wall of the left side where the flow direction is modified and the vortex creates a significant flow in a perpendicular direction to z-component. This horizontal flow is located in the tumble area (Figure 28) and all those cells in that region contain a zero value of the velocity component that tumble number formula uses. As a consequence, the flow in the tumble area is not appropriate to maximize the \( Y_i \cdot w_i \) factor or \( \text{abs}(w_i) \) component. As a result the tumble number at 6 mm valve lift has a lower value which is -0.995 (Figure 39).

5.1.1.3 2 mm valve lift

![Figure 40. 2 mm steady mesh – Flow velocity](image1)

![Figure 41. 2 mm steady mesh – vector Flow velocity](image2)
As happens in 10mm and 6mm valve lift, pressure between inlet and cylinder is considerably similar (Figure 42). However, there is influence of boundary layer around the valve due to the small gap and it results in a substantial reduction of the inflowing velocity (Figure 40), the mean flow inclination (Figure 41) and the intensity of the TKE (Figure 43). Consequently, at the right side the flow is slightly repelled from the wall creating a small vortex (Figure 41). This small vortex fills the cylinder significantly less than at the other valve lifts, allowing the inflowing air at the left side to create another small vortex.
below the leftmost edge of the valve. Those vortexes originate a vertical flux in the center of the cylinder in the tumble area (Figure 28). Observing the momentum component (equation 4.1) of the tumble number equation (4.2), the component $Y_i \cdot w_i$ is not relevant in the center of the cylinder due to opposition of sign in those cells. However, it is relevant in the right side near the wall. On the other hand, the component $abs(w_i)$ contains a meaningful value around all the tumble area (Figure 28). As a result, the tumble number at 2mm valve lift reaches a high value which is -3.268 (Figure 44).

5.1.2 Transient results

In order to explain air behaviour, the transient simulation is studied at several configuration of the valve lift where the graphs of some parameters help for the explanation of the flow field and thus the tumble number. Moreover, there is an essential relation in each configuration with the previous ones. For this reason, at every parameter there is a graph of significant configurations of the valve. Those are at the gap of 3 mm, 5 mm and 8 mm when the valve is opening; at 10 mm, which is maximum gap; and 6 mm and 2 mm when the valve is closing.

The parameters to explain the tumble number are pressure, velocity, temperature and turbulence kinetic energy.

5.1.2.1 Pressure

![Figure 45. (a) 3 mm, (b) 5 mm and (c) 8 mm (valve opening) - Transient mesh – Pressure](image)
Figure 46. 10 mm (maximum gap) - Transient mesh – Pressure

Figure 47. (a) 6 mm and (b) 2 mm (valve closing) - Transient mesh – Pressure
The pressure is a strong indicator to predict backflows of the air. In Figure 29 there is a basic representation of the relation of the pressure between inlet and the cylinder. This graph shows a direct relation between them. When the intake valve opens, the exhaust valve is still opened and pressure in the exhaust channel is higher than the pressure in the cylinder (Figure 45a), which is higher than the pressure in the intake channel (Figure 45a). Due to the geometry of the combustion chamber this results in the backflow of exhaust gasses from the exhaust into the intake (Figure 48). Afterwards, exhaust valve closes and the piston moves toward the BDC thus lowering the pressure in the cylinder (Figure 45b) resulting in the positive flow direction through the intake valve (Figure 49). As the air mass inflowing into the cylinder increases, the pressure in the cylinder increases too compensating the depression created by the piston. Later in the cycle, when the movement of the piston is slower around BDC, the depression is also lower and the pressure in the cylinder is increased by the air mass flowing inside resulting in higher in-cylinder pressure compared to the one in the inlet channel. Therefore, there is backflow again. This backflow is produced until the valve gap is so small that the air mass of the backflow and the air mass of the inlet increase the pressure at the inlet port more than the ascendant movement of the piston during the compression stroke increases the pressure of the cylinder.

Backflows have a significant effect on pressure field that in turn again drives the mass flows. However, pressure also indicates the origin of a vortex. The motion of the air mass in a vortex creates a dynamic pressure that is lowest in the center of this vortex. Figure 45c shows a low pressure center below the rightmost edge of the intake valve at 8 mm valve lift. In Figure 46 the low pressure center seen at 8mm valve lift seems to be considerably bigger below the intake valve. In Figure 47a and Figure 47b the region with lowest pressure is at the center of the cylinder as a result of a vortex that fills the cylinder.

5.1.2.2 Velocity

![Figure 48. 3 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity](image-url)
Figure 49. 5 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity

Figure 50. 8 mm (valve opening) - Transient mesh – Flow Velocity and vector Flow velocity

Figure 51. 10 mm (maximum gap) - Transient mesh – Flow Velocity and vector Flow velocity
Figure 52. 6 mm (valve closing) - Transient mesh – Flow Velocity and vector Flow velocity

Figure 53. 2 mm (valve closing) - Transient mesh – Flow Velocity and vector Flow velocity
As explained in section 4.4, tumble number is highly dependent on the component $Y_1$ and the velocity with component $w_1$ of the cells. Therefore, flow field is an essential indicator of this number. However, for this type of simulation the tumble area includes the whole domain of the cylinder. Then it is necessary to explain the behavior of the flow in this entire region during the evolution of the flow field.

Observing Figure 48 the backflow at 3mm valve lift, previously explained by pressures (Figure 45a), is evident. The air flows from the region with highest pressure to the region with the lowest so that air from the exhaust port flows directly into the intake port. This backflow also generates a small vortex under the rightmost edge of the exhaust valve (Figure 48). The velocity of the flow in the cylinder is low that gives a hint of a low tumble number in the area. Later in the cycle, when the exhaust valve is almost closed and the intake valve is at 5 mm gap, the pressure field changes. The piston going toward the BDC creates depression along with the closure of the exhaust valve which generates a lower pressure in the cylinder compared to the one in the intake channel (Figure 45b). Therefore, the air starts flowing into the cylinder (Figure 49). The motion of the piston changes the direction of the main flow that is repelled at the right side of the wall, creating a vortex in this region (Figure 49). Besides, this change in the direction of the air flow prevents the flow to reach the left side of the cylinder, allowing the air flow in the opposite region to create another vortex below the leftmost edge of the intake valve (Figure 49). Both vortexes generate a vertical flow in the center of the cylinder. This flow contains low velocity that indicates a low tumble number because of the component $\text{abs}(w_1)$ in the equation 4.1.

It is at 8 mm valve lift when the maximum velocity of the air flow into the inlet is reached (Figure 50). The air mass inflowing towards the right part of the cylinder has a higher angle due to the valve gap. This inflowing air mass impacts against the air mass remained from the backflow preventing the air flow to reach the wall and flowing directly towards the piston (Figure 50). The geometry of the piston separates this flow in two parts creating a vortex near the wall and another vortex in the center of the cylinder (Figure 50). On the other hand, the shape of the valve and the effect of the wall create a third vortex near the wall in the left part of the cylinder (Figure 50), this vortex is weak because of the effect of the boundary layer near the wall that keeps a low velocity of the flow, favoring a higher gradient of velocities in the right part of the cylinder. At this configuration, tumble number might start to take a considerable value due to a high velocity into the cylinder and interesting Z-component of the flow. Nevertheless, so many vortexes may affect negatively reducing this number.

As the air mass enters the cylinder, in the maximum gap of the intake valve, even though pressure increases as well as the gap of the valve, the velocity decreases (Figure 51). The boundary layers from the valve and the one created by the right vortex put a resistance to the inflowing air mass. These differences of velocities along with the expansion created by the piston and its shape weaken this vortex and the left small vortex; and strengthen the vortex in the center of the cylinder (Figure 51). As a result, the air mass flows in vertical in...
the tumble area as an effect of this large vortex in the center of the cylinder, which doubtless affects considerably to the tumble number.

Once the intake valve starts to close, the air mass supplied and the motion of the piston remove the rightmost and leftmost vortexes from the cylinder. Leaving a unique large vortex centered on the cylinder (Figure 52) with still high velocities of the air mass near the right part of the wall (Figure 52). This large vortex has a significant influence on the tumble number. However, there is a large amount of air mass flowing perpendicular to the Z-component as a result of the effect of the boundary layer under the exhaust valve and at the left side of the intake valve which might be counterproductive to a high tumble number.

In Figure 53 it is seen that velocity increased in the central region of the combustion chamber. Besides, after substantial amount of air mass entered the cylinder, the effect of the pressures explained previously results in a backflow. This backflow diminishes the velocity of the air mass in both the cylinder and the intake port (Figure 53). At this configuration the direction of the air mass contains more Z-component which must improve the tumble number significantly although as the velocity is lower the tumble number is reduced significantly as well.

5.1.2.3 Temperature

Figure 54. (a) 3 mm and (b) 5 mm (valve opening) - Transient mesh – Temperature

Figure 55. (a) 8 mm (valve opening) and (b) 10 mm (maximum gap) - Transient mesh – Temperature
Temperature is not a strong parameter to define tumble number but it is necessary to verify the composition of the air mass does not change during the intake stroke in case of anomalies. Moreover, the burned gases after combustion contains high temperatures, so it might be easy to understand the location of those gases and the influence to the inflowing air mass introduced. Thus, as Figure 54a shows, there is high amount of burned gases in the cylinder because of the backflow generated (Figure 48) by the difference of pressures between exhaust and intake channels (Figure 45a). In Figure 54b the burned gases start to disperse as the inflowing air flows into the cylinder (Figure 49). Later in the cycle, once the inflowing air flow is substantial, the burned gases are located near the exhaust valve (Figure 55a) as a result of the vortex near the right part of the wall (Figure 50). In the maximum valve lift the temperature of the air mass is reduced in the entire cylinder (Figure 55b) due to the high amount of clean air mass entered (Figure 51) and the depression generated by the piston (Figure 46). Further in the cycle the global temperature remains constant (Figure 56a) because of the low velocity of the piston near BDC and it is before the intake valve closes that the temperature in the intake port increases (Figure 56b) as a result of the backflow generated (Figure 53) by the difference of pressures due to the initialization of the compression stroke.
5.1.2.4 Turbulence Kinetic Energy

Figure 57. (a) 3 mm and (b) 5 mm (valve opening) - Transient mesh – TKE

Figure 58. (a) 8 mm (valve opening) and (b) 10 mm (maximum gap) - Transient mesh – TKE

Figure 59. (a) 6 mm and (b) 2 mm (valve closing) - Transient mesh – TKE
Once a flow is turbulent, the intensity of this turbulence tends to remain although it needs a continuous supply of energy. This energy is extracted from the kinetic energy of the turbulent motion that increases the internal energy. The turbulence requires a continuous transfer of energy to replace the viscous losses. If there is no supply of energy, the intensity of the turbulence decays. As a result of the motion of the air in the cylinder during the cycle, there is a variation in the TKE. Consequently, TKE is high in the regions where it is adverted to and in the regions where it is generated mainly through gradients of the mean velocity, turbulent viscosity and in the boundary layer (see section 3.1).

As a result at 3 mm opening a high velocity backflow through the nearly closed exhaust valve is a large generator of the TKE (Figure 49 and Figure 57a). The same is valid also for the 5 mm lift (Figure 50 and Figure 57b) with the addition of the high velocity gradient region above the right of the intake valve and above the piston. Due to the same mechanisms maximum intensity of TKE further increases at 8 and 10 mm valve lift (Figure 58a and Figure 58b). Afterwards, intensity of the TKE decreases at 6 and 2 mm (Figure 59a and Figure 59b) in the central part of the combustion chamber due to reduced generating mechanisms (Figure 52 and Figure 53) and dissipation of the TKE with exception of the near valve regions, where generation mechanisms are still pronounced. However, the latter contributions do not significantly influence the TKE level in the central part of the combustion chamber.

5.1.2.5 **Tumble number**

![Tumble Number](image)
Observing the momentum component of the equation of the tumble number (4.1) the main components that directly affect tumble number are $Y_i \cdot w_i$ and $abs(w_i)$. Therefore, for maximizing tumble number, $Y_i$ and $w_i$ should have the same sign. This effect is achieved with a central vortex which rotates clockwise. However, the tumble number takes a negative value. In terms of effectiveness, tumble number is treated as an absolute value. In Figure 60 is shown the first CA where tumble number takes a value of -1.535. This occurs when the valve gap is the maximum position. As explained previously, at 10 mm valve lift there is a vortex in the center of the cylinder that starts gaining importance relative to the other vortexes. There is a relative high velocity with evident Z-component in the cylinder comparing with previous points. During the intake valve opening, at 3 mm and 5 mm valve lift, due to backflows and boundary layers the velocity of the flow is considerably low, resulting in low tumble number hovering around 0.5. At 8 mm valve lift when the velocity is significantly higher, the tumble number must increase as well. However, the main flow flows along the center of the cylinder, where the Y-component is almost zero. This situation makes the tumble number to be around 0.5 too.

For the study of effectiveness of the stratified system in SIDI engines, it is at the end of the compression stroke when tumble number gains significant value. Figure 60 shows the tumble number at the maximum gap. This tumble number is supposed to be improved in the followings CA. However, as Figure 61 shows the improvement of the tumble number is
almost the same (-1.540) because of the reduction of the velocity that is compensated by the increase of flow in Z-component in faces with high Y-component.

On the other hand, in the configuration of the Figure 62, tumble number decreases (-1.310) because even though the vortex fills the cylinder with high Z-component in the flow in high Y-component faces, the velocity decreases into the cylinder due to the backflow. Once the backflow finishes and compression stroke starts, the tumble number increases again until its maximum.

5.1.3 Comparison of steady-state and transient simulation

In summary, Table 6 shows the values of the tumble number for each case.

*Table 6. Tumble number for each lift and type of simulation.*

<table>
<thead>
<tr>
<th>Valve gap</th>
<th>Tumble_transient</th>
<th>Tumble_steady-state</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 mm</td>
<td>-1.310</td>
<td>-3.269</td>
</tr>
<tr>
<td>6 mm</td>
<td>-1.540</td>
<td>-0.995</td>
</tr>
<tr>
<td>9.7 mm</td>
<td>-1.535</td>
<td>-1.731</td>
</tr>
</tbody>
</table>

Observing Figure 29, the fact that in steady-state simulation the inlet pressure is always higher than the cylinder pressure restricts those events caused such as backflow. Backflow can change completely the flow field. Besides, the extrusion of the cylinder prevents the creation of vortexes far from the valve and thus those results that appear far from the valve might be considered as not reliable, with the existence of the piston they would not exist. That absence of the influence of the piston is a high handicap for a real representation of the flow field. Piston motion supports the creation of the vortex on a correct position. Therefore, steady-state simulations are not accurate enough in those areas where the piston has relevant influence. Nevertheless, with appropriate boundary conditions it might be good indicator for areas near the valve were the piston does not affect that much.

However, for transient simulation the events of different valve lift along with the geometry of the model trigger a series of events that promote the creation of a main vortex. When intake valve starts to open and exhaust valve is still closing, the conditions in the cylinder create flow structures near the valve that do not help to create vortexes in the tumble area. This happens until the piston is down enough to induce a depression in the cylinder and thus the inflowing air mass starts flowing properly into it. Afterwards, when the valve gap increases and the piston are still moving down, the flow starts creating some vortexes, it is then when the geometry of the cylinder has the function to transform one of this vortexes
into a larger one. Once the vortex fills the whole cylinder, thanks to the previous events, this vortex has enough intensity and velocity to remain in the desired position until the fuel is injected. All this process is reflected in a constant value of the tumble number for many CA.

Summarizing, tumble values in steady-state simulations are higher in two valve lifts than in transient simulations, however, those values are only a result of the velocity of the flow in the tumble area and not in the shape of tumble of the flow that it is what a stratified system needs. The influence of the piston is determinant to create airflow that fulfills the necessities demanded. It is the vortex created by the geometry of the piston and the depression originated by it that at the end of the intake stroke is useful. Therefore, tumble number in transient results is much more reliable for studies on stratified engines.

5.2 Analyses and comparisons of the refinement in FIRE and FLUENT

In this section, the steady-state mesh previously analyzed (see section 5.1.1.2) is automatically refined aiming to improve results. Two different programs were used for this refinement: AVL FIRE 2013 and ANSYS FLUENT 13.0. The procedure followed to compare them was using 6 mm steady-state mesh, with 2.4 million cells, and applying the different refinements.

5.2.1 FIRE refinement

In AVL FIRE 2013 the refinement is done through momentum error method, previously explained in 3.2.3.2. The refinement area is shown in Figure 63.

This refinement follows the direction of the flow when approaching the valve and near the wall. This improvement may provide a better accuracy of the flow movement, thus observing changes in critical areas as the boundary layer of the valve, which may modify considerably the behavior of the flow. The final refined mesh has 3.4 millions of cells.
Figure 64. 6 mm steady refined mesh – Flow velocity

Figure 65. 6 mm steady refined mesh – vector Flow velocity

Figure 66. 6 mm steady refined mesh – Pressure

Figure 67. 6 mm steady refined mesh – TKE
Starting to analyze the pressure, Figure 66 shows a slightly lower pressure around the valve and near the walls of the cylinder than in Figure 37. This might increase the velocity of the inflowing air mass but in the left side of the valve, the refinement in that area shows a better representation of the effect of the boundary layer around the valve that slows the flow (Figure 64) in that region. As a result of this, the flow in the left side of the wall seen in Figure 35 has lower velocity after the refinement (Figure 64). The air mass in that left side of the cylinder creates small vortexes that changes the flow field in that region. Figure 65 shows an area of flow perpendicular to Z-component, in the refined mesh this area is smaller affected by vortexes in the left side and increase of the velocity of the vortex in the right side. However, those increments just improve the tumble number to -1.043 (Figure 68), which is similar to the -0.995 from the original mesh (Figure 39), because those difference are low.

5.2.2 FLUENT refinement

To compare results in ANSYS FLUENT 13.0, first the original mesh (Figure 69) has to be simulated in order to obtain a first glance about the difference between both programs.
5.2.2.1 Original mesh

Figure 70. 6 mm Fluent mesh – Flow velocity

Figure 71. 6 mm Fluent mesh – vector Flow velocity
**Figure 72. 6 mm Fluent mesh – Pressure**

**Figure 73. 6 mm Fluent mesh – TKE**
The first difference in the results with FLUENT from FIRE is the pressure. Pressure in Figure 72 is considerably lower than in Figure 37 except from the right side of the wall. Difference of pressure between the cylinder and the intake port is also lower (Figure 72) and so is the velocity of the air mass inflowing (Figure 70). However, the velocity in the center of the cylinder is slightly higher. As a result of the vortex in the center of the cylinder created by the inflowing air mass repelled from the wall (Figure 71), that increases the velocity in that region. This central vortex also modifies the flow from the left side, along with the shape of the valve and the boundary layer in it, there is a small vortex under the valve that strengthens at the same time the central vortex. The meaning of such difference in results, when comparing with FIRE steady-state simulations, is the location of the boundary layer near the wall. Observing Figure 38 the effect of the boundary layer is at the left side of the cylinder which does not affect the vortex of the right side. The flow field in the tumble area in FIRE simulation contains a vortex at the right side, while the flow field in FLUENT simulations results in a central vortex and a vortex below the intake valve (Figure 71). Even though the velocity at the flow field is lower, the Y-component is higher at the regions with maximum velocity (Figure 71). As a result, the tumble number in FLUENT is -1.15 comparing to -0.995 with FIRE, which is not a negligible difference since both meshes are equal with the same number of cells.

5.2.2.2 **Refined mesh**

The method of refinement in FLUENT is by turbulence intensity, explained in section 3.2.3.3. This method was applied ten times to obtain the simulation here presented. The refined mesh is shown in Figure 74.

![Figure 74. 6 mm Fluent mesh – Refined mesh](image)
Figure 75. 6 mm Fluent refined mesh – Flow velocity

Figure 76. 6 mm Fluent refined mesh – vector Flow velocity
Figure 77. 6 mm Fluent refined mesh – Pressure

Figure 78. 6 mm Fluent refined mesh – TKE
Analyzing the refinement, Figure 77 contains zones with higher pressures than the original mesh (Figure 72). The smooth mesh near the wall and around the valve gives a more accurate representation in the parameters. The difference of pressures (Figure 77) increases and the effect of the boundary layer around the valve is lower at this mesh. It is obvious then that the velocity of air mass inflowing is also increased (Figure 75) and thus the TKE (Figure 78). The consequence of this is a creation of a vortex below the leftmost edge of the valve (Figure 76). Moreover, the strengthening of the velocity of the air mass also affects to the left side of the cylinder. In the original mesh the flow in the left side was dominated by the vortex from the right side. After the refinement, the vortex at the left side happens to have enough momentum to not being modified by the other vortex, diverting the flow to the center of the cylinder (Figure 76). The flows deviated by both vortexes find each other with opposite directions but slightly ascendant. As a result, there is a unique flow going upwards. The difference of flow fields between the original mesh and the refined mesh is significant. A better Z-component in the flow and higher velocities makes an improvement of the tumble number from -1.15 in the original mesh to -1.48 in this refined mesh.

5.2.3 Comparison of refinements

Table 7. Tumble number for refined and original mesh in FIRE and FLUENT programs.

<table>
<thead>
<tr>
<th>Kind of mesh</th>
<th>Tumble number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady-state FIRE Original</td>
<td>-0.995</td>
</tr>
<tr>
<td>Steady-state FIRE Refined</td>
<td>-1.043</td>
</tr>
<tr>
<td>Steady-state FLUENT Original</td>
<td>-1.15</td>
</tr>
<tr>
<td>Steady-state FLUENT Refined</td>
<td>-1.48</td>
</tr>
</tbody>
</table>

Table 7 shows the set of tumble numbers obtained from steady-state simulations of every mesh. The refinement used in FLUENT (Section 3.2.3.3) seems to be more adequate that the refinement used in FIRE (Section 3.2.3.2). Turbulence intensity refinement creates smaller cells in the valve and the walls. This improvement is reflected in a better accuracy to represent the boundary layers in the cylinder. Those boundary layers are the reason of a significant change in the flow field. Vortexes into the cylinder have different positions and different strength. Therefore, the tumble number increases as the velocity is higher and the Z-component of the flow is better.

Momentum error refinement is focused more after the valve. And it is near the valve that needs to be improved to make a better representation of the boundary layers. The area refined by momentum error is an area with low influence in the rest of the flow in the
cylinder. The refinement in the left side of the valve is interesting but it is not the main side that affects to the flow field. On the right side the cells in the valve are poorly improved. As a result, the boundary layers of the valve and the walls contain small changes and so does the flow field, meaning a low change in the tumble number.
6 Conclusions

In this thesis, results of steady-state and transient simulations were compared in terms of determining a tumble number of a SIDI engine with stratified charge working at partial load. These analyses are supported by analyses of detailed 3D flow phenomena. In addition, steady-state results calculated with two different tools were compared to analyze tool specific differences in determining the tumble number and underlying flow phenomena. The main conclusions of this thesis are explained as follows:

- The fact that transient simulations use a dynamic mesh that adapts itself for each crank angle increases their fidelity as the movement of the piston is a vital component in a vortex generation process.
- SIDI engines with stratified charge are highly dependent on the flow structure in the cylinder, where a central vortex is needed to supply fuel in lean conditions. This flow structure appears in transient simulations and verifies the possibility of recreation of a real cycle in a transient simulation. Consequently, once the mesh performs the results desired, the engine might be optimized just modifying the mesh or parameters. Saving costs from prototypes and shorten time in research and improvement of this technology.
- Steady-state simulations do not represent truly a real engine due to its geometry and piston limitation. Nevertheless, they are a valid approach for calculations in the region near the valve where the piston motion does not influence the flow field significantly. This sort of simulations becomes relevant for short changes in meshes and fast references of results, but not for the explanation of the stratified charge in SIDI engines.
- The refinement done by momentum error improves slightly the quality of the steady-state mesh. Despite increasing considerable the number of cells, the location of those cells does not directly influence in the improvement of the critical cells that might enhance the accuracy of the flow field. As a result, the results after refinement are similar to the results of the original mesh whereas the time simulating increases.
- The application of the refinement by turbulence intensity in the steady-state contributes to an enhancement in the cells near the wall and around the valve. For this reason the results show higher accuracy of the flow field. Therefore, the tumble number improves and vortexes are better located for a more realistic study near the valve.
7 References


8 Attachments

8.1 Formula for calculating tumble number in transient simulations

*Global Formula Variables (accessible both from Initialization and Body)*

'\(tumble\) about center of gravity within cell selection

char mark[0];
#define SEL_x_y_z__tumble_vector_component 0

*Formula Initialization*

int i;
double r2omega, r2, mass, v[3], x[3], center[3], tumble;
double rot_dir[3] = 0.0;
double rpm = SSFDOUBLE("RM/speed");
if (init){
    resize (mark, NCELL);
    mark = 0;
    // rot_dir[SEL_x_y_z__tumble_vector_component] = 1.0;
} else {
    'at this point, all cells within selections are marked (1)
    'center of gravity
    rot_dir[SEL_x_y_z__tumble_vector_component] = 1.0;
    divisor = 0.0;
    center = 0.0;
    for (i = 0; i < NCELL-NUMBUF; i++){
        if (mark[i] == 1){
            mass = DEN[i] * VOL[i];
            x = XP[i];
            center += mass * x;
            divisor += mass;
        }
    }
    MPI_SUM_VEC_D(center, 3); MPI_SUM_D(divisor);
    if (divisor > 0.0) center /= divisor;
    'tumble component about rot_dir
    divisor = 0.0;
    sum = 0.0;
    for (i = 0; i < NCELL-NUMBUF; i++){
        if (mark[i] == 1){
            mass = DEN[i] * VOL[i];
            x = XP[i] - center;
            v = U[i];
            'angular speed times radius squared
            r2omega = (x ^ v) . rot_dir;
            'square of distance normal to rotation axis
            r2 = x . x - (x . rot_dir) * (x . rot_dir);
            'sum rot. inertia
            divisor += mass * r2;
        }
    }
}
Evaluation Of Numerical Methods For Modeling Flow Field During The Intake Stroke Of Spark-Ignited Engines

\begin{verbatim}
  \'sum angular momentum
  sum += mass * r2omega;

  MPI_SUM_D(sum); MPI_SUM_D(divisor); // MPI

  tumble = 60.0 / (2.0 * Pi * rpm) * sum / divisor;
  if (IAMPRO < 2){
    print "tumble within", name, "about axis", rot_dir, ":", tumble;
    print " center of gravity:", center, "[m]";
    print " angular inertia:", divisor, "[kgm2], momentum:", sum,
    "[kgm2/s]";
    print " engine speed", rpm, "[1/min]";
  }
  sum = tumble;
  divisor = 1.0;

  \end{verbatim}

Formula Body

\begin{verbatim}
  \'mark cells
  mark[index] = 1;
  return 0.0;

\end{verbatim}

8.2 Economic impact

Nowadays, automotive companies spend more and more resources in research, trying to find new products or technologies, which require prototypes to test the new solutions. Construction of prototypes might be difficult, very expensive, and highly time consuming. Therefore, the appearance of computer programs able to simulate any kind of model required by the companies constitutes an important advance.

Nevertheless, simulations have been a hard solution to trust, as long as there is a considerable difference between simulation and reality. However, as a result of new technologies and the effort of developing programs with higher accuracy, simulations as a solution is already considered an optimal choice.

This thesis has an economic impact in order to show to companies if there is a benefit of choosing computer programs instead of making prototypes themselves. The evaluation shown is an investment research associated to the project, temporally and materially.

8.2.1 GANTT of the thesis

In Table 8 is shown the time-sharing of the thesis (from February 2013 to March 2014) split in events and periods of the performance of it by an engineer. It is worth to notice that the number of hours spent in the event of simulations correspond to the time spent of an engineer at checking that each simulation is working properly or checking errors from
previous simulations. Some simulations last for a week, but the number of hours of it are not considered as a job done by an engineer, and thus, they are not paid to them.

Table 8. GANTT of thesis.

<table>
<thead>
<tr>
<th>Month</th>
<th>February</th>
<th>March</th>
<th>April</th>
<th>May</th>
<th>June</th>
<th>July</th>
<th>August</th>
<th>September</th>
<th>October</th>
<th>November</th>
<th>December</th>
<th>January</th>
<th>February</th>
<th>March</th>
<th>Hours</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>25</td>
</tr>
<tr>
<td>Learning program</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>125</td>
</tr>
<tr>
<td>Creating models</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>60</td>
</tr>
<tr>
<td>Simulations</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>20</td>
</tr>
<tr>
<td>Writing thesis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>300</td>
</tr>
<tr>
<td>Selecting Results</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10</td>
</tr>
<tr>
<td>Corrections of thesis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10</td>
</tr>
<tr>
<td>Total</td>
<td>650</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

8.2.2 Investment

The investment of the thesis includes all fixed costs from computer equipment and salaries.

8.2.2.1 Staff costs

A single engineer is considered in charge of the thesis to carry it out.

Table 9. Staff costs.

<table>
<thead>
<tr>
<th></th>
<th>Hours (h)</th>
<th>Salary/hours (€/h)</th>
<th>Staff cost (€)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engineer salary</td>
<td>650</td>
<td>15</td>
<td>9750</td>
</tr>
</tbody>
</table>

8.2.2.2 Computer equipment costs

This section is the strongest investment for the thesis. A couple of computers are required in order to simulate more efficiently due to the large period of time each simulation takes. On the other hand, as important as the computers are the licenses of each program used to simulate or writing the thesis.

Table 10. Computer equipment costs.

<table>
<thead>
<tr>
<th></th>
<th>Costs (€)</th>
</tr>
</thead>
<tbody>
<tr>
<td>License AVL FIRE 2013</td>
<td>10000</td>
</tr>
<tr>
<td>License ANSYS FLUENT v13.0</td>
<td>10000</td>
</tr>
<tr>
<td>2 Computers Core 2 Quad</td>
<td>1500</td>
</tr>
<tr>
<td>License Windows XP</td>
<td>360</td>
</tr>
<tr>
<td>License Office</td>
<td>720</td>
</tr>
<tr>
<td>Total Computer equipment costs</td>
<td>22580</td>
</tr>
</tbody>
</table>
8.2.2.3 Total investment

Considering all the costs during performance of the thesis, the total amount of money as investment is shown in the Table 11.

Table 11. Total investment.

<table>
<thead>
<tr>
<th></th>
<th>Investment (€)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Staff costs</td>
<td>9750</td>
</tr>
<tr>
<td>Computer equipment costs</td>
<td>22580</td>
</tr>
<tr>
<td><strong>Total investment</strong></td>
<td><strong>32330</strong></td>
</tr>
</tbody>
</table>
8.3 Environmental impact

The environmental impact of this thesis is highly related with the improvement of obtaining results from a computer instead of prototypes. Through simulations, there are several options to modify and test models just consuming electricity. However, the creation of prototypes means a high spending of material and, at the same time, there is a pollution associated to each test that is made at any engine-prototype.

Another important environmental impact achieved in this thesis is the reduction of fuel consumption and thus the pollution of future engines. As explained previously, SIDI engines have the benefits of Diesel engines regarding fuel consumption and pollution. Moreover, the stratified system applied to SIDI engines gives even more efficiency. The pollution from engines of ordinary cars is a significant and worrying issue that affects the worldwide and it is one of the main causes of pollution. There are considerable advances on creating non-fuel engines, but people is still highly dependent on those polluters engines, and the fact that this will still be a trend for a long period of time, all solutions able to reduce the pollution from fuel combustion can provide less environmental impact.