

# INTERNAL COMBUSTION ENGINE

Himanshu Kumar<sup>1</sup>, Er Umesh Verma<sup>2</sup>, Anurag Singh<sup>3</sup>

<sup>1</sup>Mechanical Engineering, I.E.T, Dr. Rammanohar Lohia Avadh University, Ayodhya, Faizabad, Uttar Pradesh, India.

<sup>2</sup>Assistant Professor, Dept. of Mechanical Engineering, I.E.T, Dr. R M L Avadh University, Ayodhya, Uttar Pradesh, India

<sup>3</sup>Assistant Professor, Dept. of Mechanical Engineering, I.E.T, Dr. R M L Avadh University, Ayodhya, Uttar Pradesh, India

**Abstract**— This research paper strategy was previously applied to simplified geometries as well as real engine geometries. The purpose of this work is to show its validity for real and complex engine geometry, covering a longer simulation interval than previously. After an initial description of the mesh creation process, studies of the in-cylinder charge motion, created during the intake stroke, were made to investigate the presence of a tumble motion. The results show strong tumble motion is not generated. This could also be because of numerical diffusion caused by the mesh structure. Furthermore, investigations were made to see why the present engine has a tendency to knock, possible increasing the knock risk close to the exhaust valves. Problem calculated turbulence intensity that was too low for proper flame propagation were apparent, leading to inconsistency between the calculated cylinder pressure of the CFD calculation and the calibrated 1D calculation.

**Keywords**— simulation interval, in-cylinder charge motion, CFD calculation, numerical diffusion, engine geometry.

## 1. INTRODUCTION

The Internal Combustion Engine Group collaborates with many industrial partners and the work of this thesis was carried out in collaboration with MV Agusta S.P.A which is a company based in Varese outside Milan that makes exclusive high performance motor cycles. The thesis concerns the CFD modeling of a four stroke, four cylinder, spark-ignited engine which is mounted on the motorcycle models based on the F4 series made by MV Agusta. The CFD simulations were done on one of the cylinders during the intake, compression and expansion phases, including combustion simulation, taking into account the valve and piston motion.

CFD simulations can be a powerful engineering tool that can help to understand the thermo-physical and chemical processes that take place in an IC engine without being forced to use possibly complicated and expensive measurement techniques. Some things can be difficult to measure in a three-dimensional space and a one-dimensional model does not account for all geometric topologies, making CFD simulation an important complement. It can also be a valuable tool to run numerous tests and investigations so far as an engine without having to perform.

CFD research has unfortunately not yet reached a state where it completely describes all the processes that take place in an IC engine. This is mainly due to the fact that it is a highly complex mechanical device which incorporates many, simultaneously interacting, thermo fluid and chemical processes. Because of the IC engine's reciprocating nature it features extreme deformation of the solution domain due to the moving piston and valves, and the shape of the piston and cylinder head, which are key features for the engine design, are usually very complex. Aside from the geometric complexity there are also the simultaneously interacting thermo fluid processes such as; non-stationary turbulent flow, heat and mass transfer, injection, atomization, dispersion and vaporization of the liquid fuel, ignition, combustion and the consequent formation of harmful pollutants, to mention a few. Adding all this together makes a complete modeling of the IC engine one of the most challenging tasks in the area of CFD research.

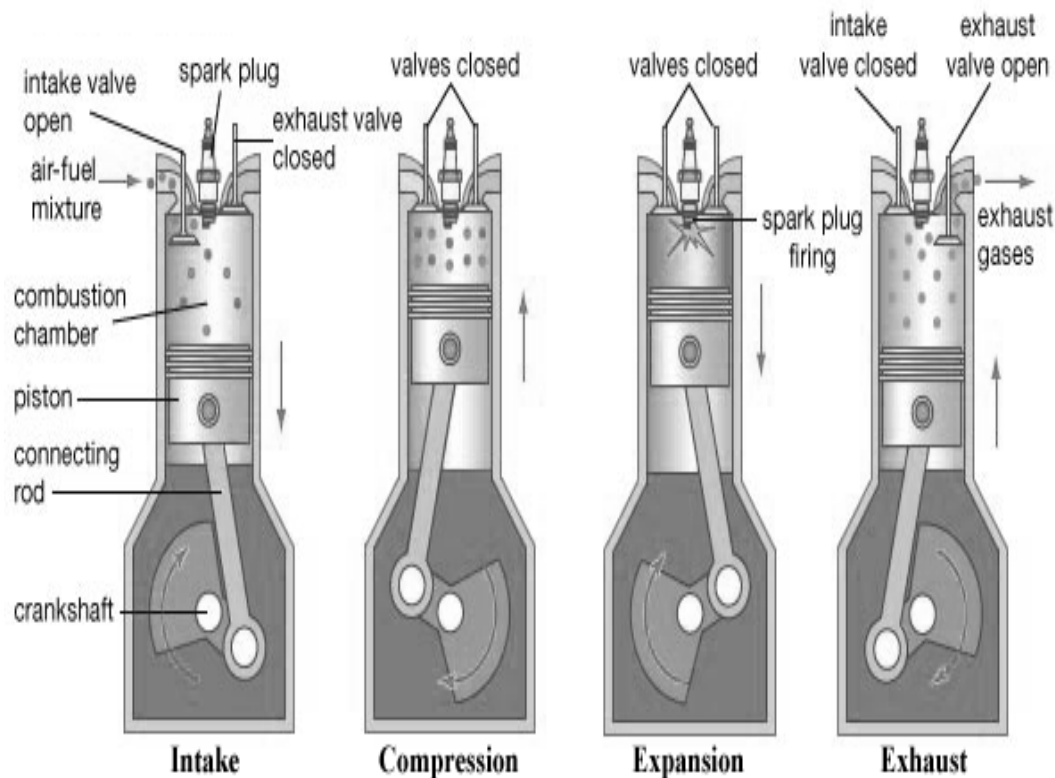
In this section a brief introduction to the Internal Combustion Engine (ICE) will be presented as well as the purpose of this thesis. Lastly a brief summary of all the chapters of the thesis is presented.

## 2. The Internal Combustion Engine

The internal combustion (IC) engine is today the most widely used energy source in the automotive and naval industry. It has therefore become increasingly important to improve the efficiency of the engines to reduce fuel consumption, emission levels and noise pollution to decrease the negative effect it has on the health of the human being as well as its environment. Since the introduction of the IC engine in the late 19<sup>th</sup> century its development has resulted in a constant reduction of fuel consumption and emission levels, while the power output per cylinder volume has continued to increase. The IC engine comes in many different types and sizes but can be divided into the category of the two stroke cycle or the four stroke cycle which can either be run by the Otto or the Diesel principle. For Otto engines the pre-mixed air-fuel mixture is ignited by a spark from a

sparkplug while for the Diesel principle the air is compressed beforehand in the cylinder and the incoming fuel spray is ignited by the high pressure and temperature. In this chapter a brief explanation of the four stroke, spark-ignited (SI/Otto) engine will be presented, since this is the type of engine that the work of the thesis is performed on.

The four stroke cycle starts with the piston positioned in TDC (Top Dead Center) and as the piston travels down towards BDC (Bottom Dead Center) the intake valve opens, letting the fresh charge of air-fuel mixture to enter the cylinder during the *intake stroke*, see Figure 1. With the piston at BDC the intake valve closes and as the piston travels towards TDC again the fresh charge is compressed during the *compression stroke*. As the piston reaches TDC the compressed air-fuel mixture is ignited by the spark plug and the chemical energy of the fuel is converted to heat during combustion. This increases the cylinder gas temperature and pressure there for adding work to the crank shaft during the power stroke or *expansion stroke*. When the piston reaches BDC again the exhaust valve opens and as the piston travels towards TDC the exhaust gas is pushed out from the cylinder during the *exhaust stroke*, see). When the piston has reached TDC the cycle is restarted again. In one power cycle the crankshaft has done two full revolutions (720 crank angle degrees or CAD) and the piston has traveled up and down the cylinder four times, therefore the name four stroke cycle.



i. **Figure 1.** The four phases of the four stroke IC engine: intake stroke (a), compression stroke (b), expansion or power stroke (c) and exhaust stroke (d).

The combustion process in the SI engine can be divided into four stages:

1. **Spark ignition.** The air-fuel mixture is ignited by the spark discharge of the sparkplug. The sparkplug releases a high energy plasma with a temperature around 60 000 K for a very short period (typically in the order of 1 ns). The spark ignition usually occurs around 20-40 crank angle degrees before TDC.
2. **Early flame development.** The first phase of the combustion process is the laminar flame propagation. The flame front propagates in a sphere around the sparkplug and the flame front is considered smooth with a typical thickness of 0.1 mm and a velocity of around 0.5 m/s. The flame speed depends on the pressure, temperature, the air-fuel ratio and the amount of residual gases from the previous cycle. High temperature and a low air-fuel ratio results in a higher flame velocity while high pressure and residual gases results in a lower velocity.
3. **Turbulent flame propagation.** After a few crank angle degrees the laminar phase is transformed into a turbulent phase, caused by the turbulence created during the intake and compression stroke. The smooth laminar flame front is broken down by vortices into a turbulent flame front with a very irregular and wrinkled shape. The flame speed now reaches velocities of 10-50 m/s. The velocity still depends on pressure, temperature, the air-fuel ratio and amount of residual gases, but the turbulence intensity is the most important factor in this phase. Increasing the engine speed leads to more generated turbulence, this in turn results in higher flame velocity. This leads to a combustion time that is unchanged (expressed in crank angle degrees) allowing for high engine speeds for the SI engine.
4. **Flame extinction.** Lastly the flame reaches the cylinder liner causing flame extinction because of the liner walls relatively low temperature. The combustion phase is usually completed around 30 crank angle degrees after TDC.

This is a simplified explanation of the typical combustion process where the spark ignited flame moves steadily across the combustion chamber until the charge is consumed. However two types of abnormal combustion can be identified; *knock* and *surface ignition* which will be explained in the next section.

### 3. Abnormal Combustion Phenomena

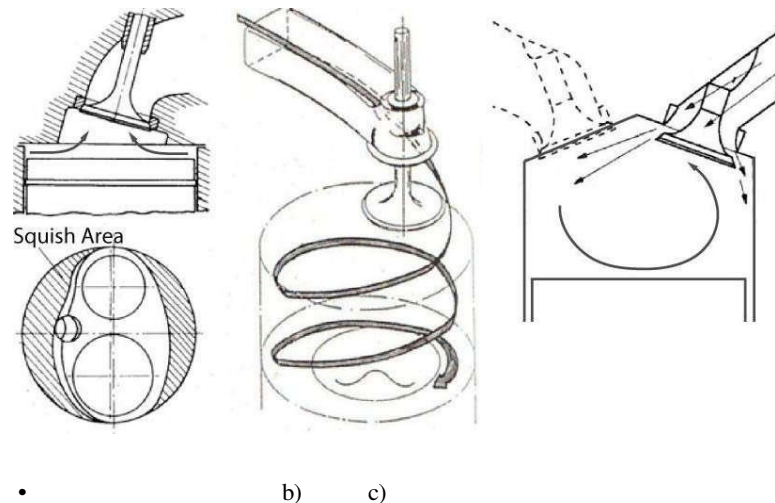
As the flame propagates across the combustion chamber the unburned mixture ahead of the flame (called the *end-gas*) is compressed, causing its pressure, temperature and density to increase. Some of the end-gas air-fuel mixture may then undergo chemical reactions prior to the normal combustion when the flame front reaches the end-gas. These chemical reactions may cause the end-gas to auto ignite, releasing a large part or all of their chemical energy. This causes the end-gas to burn very rapidly with flame velocities of up to 1000 m/s and it creates high-frequency pressure oscillations inside the cylinder that produce the sharp metallic noise called *knock* [1]. Knock is mainly caused by high pressure and temperature in the end-gas but depends also on the octane number of the fuel. Knock is common in engines with a high compression ratio, which leads to a high cylinder pressure and an increased risk of knock when the end-gas is compressed further by the propagating flame. The pressure waves that knock creates, lead to high mechanical load on the engine, causing damage to the material. Knock can be detected by knock sensors, for example with an accelerometer in the cylinder wall that detects high-frequency pressure oscillations. Knock can be avoided by adjusting the spark timing automatically when knock is detected in the engine. By retarding the time of spark discharge the maximum cylinder pressure is lowered with a decreased risk of knock as a result.

The other abnormal combustion phenomenon is *surface ignition*. This phenomenon is caused when the air-fuel mixture is ignited by an overheated surface in the engine, like the valves or the sparkplug. It is defined as ignition by any other source than the spark of the sparkplug. It can occur before the spark ignition (preignition) or after (postignition). Either way it means that the combustion process is no longer controlled by the ignition of the spark. When surface ignition results in knock in the end-gas it is called *knocking surface ignition* [1].

Knock can be considered a bigger problem than surface ignition, since surface ignition can be avoided with proper engine design. Knock on the other hand is an inherent constraint on engine performance and efficiency since it limits the maximum compression ratio that can be used with any given fuel [1].

#### 4. Charge Motion Classifications

As explained above, the turbulence level in the cylinder is very important for the turbulent flame propagation and the efficiency of the combustion. There are many ways in which the turbulence level in the cylinder can be increased by different constructions of the intake ducts. The flow motion of the incoming fresh charge can vary dramatically depending on the design of the intake ducts, valves and combustion chamber. The structure of the flow can be divided in three main motion categories; *squish*, *swirl* and *tumble*, see Figure 2. Squish is defined as the flow created when the piston reaches TDC and the outer part of the piston crown approaches the cylinder head closely, causing a flow motion directed from the cylinder liner towards the center of the combustion chamber, see Figure 2(a). Swirl is the rotational flow motion around the cylinder axis and is created by the shape of the intake duct, see Figure 2(b). By bringing the intake flow into the cylinder with an initial momentum a swirl motion is created, usually persisting through the whole intake, compression and expansion stroke. Tumble is the rotational flow around the axis perpendicular to the cylinder axis, see Figure 2(c). The tumble motion is also created by the shape of the intake duct and the angle of the incoming jets of fresh charge.



**Figure 2.** Charge motion classification for three typical flow structures; squish (a), swirl (b) and tumble (c).

While all three of these charge motions can help to improve the turbulence level in the SI engine, the tumble motion is instead the most important. Tumble is generated during the intake stroke and intensified during the compression stroke where the large scale tumble motion keeps its angular momentum but increases its intensity because of the shrinking cylinder volume. As the piston reaches TDC and the combustion chamber becomes “flat”, the large scale tumble motion breaks up in small scale turbulence, creating a high level of turbulence intensity in the cylinder. To create a strong tumble motion it is not just important to accelerate the incoming flow, but also to direct it mainly towards the area of the exhaust valve. In a traditional intake duct the incoming charge is distributed quite uniformly around the valve causing a flow that counteracts itself and hinders the creation of a main tumble motion. By directing a bigger part of the flow towards one side of the valve, preferably towards the exhaust valve, a strong tumble motion can be created by this main flow jet that is not counteracted by the jets created on other sides of the valve [2]. A strong tumble motion creates high turbulence which in turn leads to a high flame velocity. This increases the efficiency of the combustion process, but the fast combustion also leads to hard engine noise. A strong tumble motion is positive for high performance and racing engines but for normal engines a compromise between engine noise levels and performance has to be made.

#### 5. Purpose of the research paper

The purpose of this research paper can be divided into two parts. The first part is related to the strategies of mesh management and CFD simulation, and the second part is related to the analysis of the results produced for the specific engine used for these simulations.

Since the moving piston and valves deform the solution domain extremely during a whole engine cycle, the computational grid will at some point become too distorted, leading to numerical errors and abortion of

the calculations. A strategy to solve this was previously developed by the Internal Combustion Engine Group at Politecnico di Milano which uses a number of computational grid some to cover the whole engine cycle. Each mesh is deformed for a certain interval and when the quality of the mesh becomes too low, the solution is interpolated onto a new mesh. This strategy is called the MUMMI approach (Multiple Mesh Motion and Mesh to Mesh Interpolation) and it had previously been employed on a simplified engine geometry [10] and later on a real engine geometry [12]. In the last case the calculations of the whole engine cycle intended to be simulated was not actually performed due to the fact that no real boundary conditions were used. Thus, the purpose of this thesis was to use the same strategy of how each mesh is constructed and generated as in the previous cases, but this time employed on the real and highly complex engine geometry of the F4 engine, including the use of realistic boundary and initial conditions for temperature, pressure and mass flow. The simulation was intended to cover the intake, compression and expansion stroke starting from the opening of the intake valve and ending at the completion of the combustion process. The purpose was also to study a combustion model developed by Weller [3], which is pre-implemented in the OpenFOAM code, to investigate if a realistic combustion process could be simulated for the F4 engine.

The second part of the purpose of this thesis is strictly engine related. First it was of interest to make an investigation of the structure of the charge motion created during the intake and compression stroke to see if a strong tumble motion is created. As mentioned in Chapter 1.1.2, tumble is important for the performance of the SI engine and is mostly created by the design of the intake port and the intake valves. Furthermore, the F4 engine has a tendency to knock at full load for the engine speed that produces the maximum amount of torque, and it was of importance to try to make an investigation to why and where the knock occurs. As mentioned in section 2

the knock risk depends on the temperature and pressure of the end gas and a study had to be made to see if, for example, the charge motion could create a temperature distribution in the cylinder that is not uniform, causing a higher knock risk in some parts of the cylinder.

As a sub purpose it was also of interest to see if the whole process of mesh generation and simulation could be done within a reasonable amount of time to make it valid as an engineering tool.

## 6. RESULT

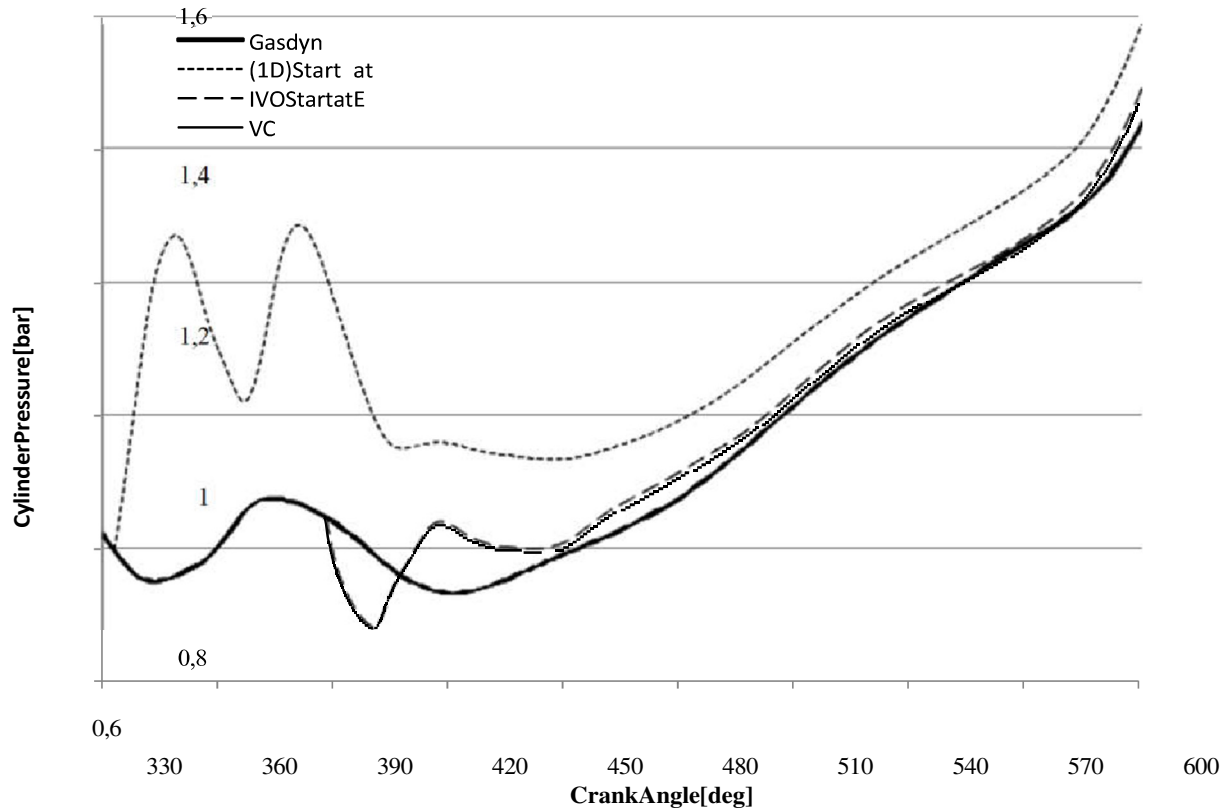
### 6.1 Simulation from EVC

1. Simulation starts at EVC (388 CAD) instead of IVO (333 CAD)
2. Change of Air-Fuel (A/F) ratio to 12.4
3. Correction of compression ratio from IVC (600 CAD)

The overestimation of volumetric efficiency is most likely because of the fact that the exhaust valve was kept closed during the valve overlap period, and therefore an amount of extra mass was trapped in the cylinder. In reality there will be a certain mass that exits the cylinder through the exhaust valve during valve overlap period, and this was not taken into account for. To solve this problem the simulation was set to start at exhaust valve closure times instead. New initial conditions at EVC (388 CAD) for temperature and pressure was assigned from the Gasdync calculations. The original boundary condition for mass flow at the intake duct was also used, see Figure 3.14.

The initial case had an Air-Fuel ratio corresponding to a stoichiometric mixture (14.7) which is not the case of the F4 engine at full load and 8000 rpm. Instead the real A/F ratio should be 12.4 which corresponds to a  $\lambda$  value of 0.84. A lower A/F mixture will lead to less air in the cylinder and therefore lower cylinder pressure because of a smaller volume of the charge (gasoline has higher density than air). To implement the correct A/F ratio is obviously important also for the future combustion simulation.

The intake stroke was then re-calculated with the first two changes (point 1 and 2). First a case was calculated only with the new starting time at EVC. Then another case was simulated with both the new starting time at EVC and the new A/F ratio to compare them both against the initial case described in Chapter 4.1.1. Note that the compression ratio correction was not made to the geometry until intake valve closure time (600 CAD) since the difference in cylinder pressure will be negligible before the intake valve closes and the compression stroke starts. The result of the calculations for the first two changes can be seen in Figure 3.



**Figure 3.** Cylinder pressure during intake stroke for Gasdyn (1D) and OpenFOAM (CFD) calculations, comparing the difference of starting the simulation from IVO and EVC, and changing  $\lambda$  from 1 to the correct value of 0.84.

Figure 3 shows that the biggest improvement for the consistency between the 1D and CFD calculations was from the change of simulation starting time. As suspected the overestimation of volumetric efficiency was mainly because of the neglected valve overlap period. A small improvement was also achieved by decreasing the air-fuel ratio, and therefore a smaller gas volume of the trapped charge in the cylinder, resulting in a lower cylinder pressure.

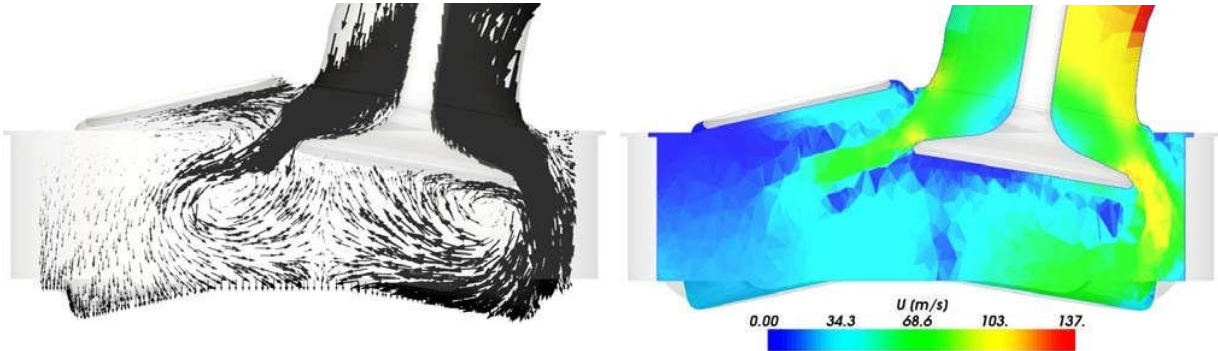
It was discovered that the engine model had a compression ratio  $\epsilon$  that was wrong compared to the real engine. The correct compression ratio is 13 while the engine model had a ratio of 14.1. The reasons for this is probably because the engine model is slightly simplified, for example are the crevices above the first piston ring not present. There are most likely also some additional crevices in the cylinder head, close to the valves and spark plug that are not present either in the engine model. Since the error in compression ratio will have a negligible effect before the valve closes and compression stroke starts at 600 CAD it was decided to only re-make the meshes from 600 – 780 CAD. An investigation was made to confirm this by also re-making the meshes for the two intervals before 600 CAD which did not show any difference in cylinder pressure.

## 6.2 Results from Final Intake and Compression Stroke Simulation

The results of the calculations can be post-processed with a software called ParaView. This software offers a graphical user interface and many options to display the calculated field values in a number of different ways. ParaView is started by importing an OpenFOAM case to it with the utility paraFoam. This chapter will deal with some results of the field variables calculated from the final simulation of the intake and compression stroke, which was started at EVC, has a corrected compression ratio and the correct air-fuel mixture.

## 6.3 Flow Field

Starting from a flow field that is zero over the whole domain, an initial flow field is established because of the pressure difference between the intake duct and cylinder as well as the imposed boundary condition for the mass flow at the intake duct. After the first few crank angle degrees of simulation where a flow field is established, the flow structure of the intake stroke is starting to become visible at a round 424 CAD, see Figure 4. The results are represented on a cutting plane made through the middle of the intake valve and parallel to the symmetry plane.



**Figure 4.** Flow field (a) and velocity magnitude (b) at 424 CAD. Results are shown for a plane cut through the middle of the intake valve and parallel to the symmetry plane.

It is visible in Figure 4. That the two incoming jets on each side of the valve create two strong vortices, one on the left side of the valve and one under the valve plate since the flow is restricted by the liner wall.

## 7. CONCLUSION

This research paper simulations were carried out from the start of the intake stroke (IVO) until around the end of the combustion process, thus covering the intake, compression and part of the expansion phases. A significant amount of the total time spent on the work was dedicated to the mesh creation. Before the initial template mesh could be created, the geometry had to be cleaned and divided into sub-volumes which, together with the mesh creation itself, required a lot of time. This process could probably be made a lot faster with more experience and/or mesh software that allowed for easier manipulation of the geometry and more options for mesh generation than Gambit possesses. The power of Gambit is in the Journal function, with which the rest of the meshes were created very fast and easily, taking only around 10 min each compared to these several hours needed to create the meshes by hand. In total, 18 different meshes were used to cover the whole simulation interval and the mesh strategy of the MUMMI approach was shown to be successfully employed also for this complex engine geometry.

The initial calculations were started from IVO but without simulating the closure of the exhaust valve during the valve overlap period. This led to an overestimation of the volumetric efficiency since in reality a small part of the cylinder mass will be lost through the exhaust port during valve overlap. This resulted in a cylinder pressure that was higher than expected compared to the 1D calculation, which was not acceptable. The conclusion is that either the closure of the exhaust valve has to be simulated as well or the simulation has to be started from EVC. All the following calculations were therefore started from EVC.

An investigation was done using the *total pressure* boundary condition on an “infinite” volume attached to the intake port, thus simulating the atmosphere. This allows the incoming mass flow to be calculated freely without the restriction of a fixed boundary condition. The calculated cylinder pressure did not show satisfactory results at all during the intake stroke using this strategy, and it is clear how important it is with correct boundary conditions for the incoming mass flow. This is especially true for engines that use the pressure pulses in the intake system to create a ram effect before valve closure. This approach can be valid when there are no boundary conditions available or for simple intake systems, otherwise it is important with time-varying boundary conditions for the mass flow if the whole intake system is not simulated.

## REFERENCES

- [1] J.B.Heywood.*InternalCombustionEngineFundamentals*.McGraw-Hill,1988.
- [2] G. Ferrari.*Motoriacombustioneinterna*.EdizioniIlCapitello,Torino2005.
- [3] H.G. Weller. *The development of a new flame area combustion model using conditionalaveraging*.Thermo-FluidsSectionReportTF9307,ImperialCollegeofScience,TechnologyandMedicine,1993.
- [4] H.Jasak,Z.Tukovic.*AutomaticMeshMotionfortheUnstructuredFiniteVolumeMethod* .Journalof Computational Physics, 2004.
- [5] H. Jasak. *Error Analysis and estimation for the finite volume method with applications tofluid flows*. PhD thesis, Imperial College of Science, Tecnology and Medicine, London,1996.
- [6] H. Jasak. *Numerical Solution Algorithms for Compressible Flows*. Faculty of MechanicalEngineeringandNavalArchitectureUniversityofZagreb,Croatia,2006.
- [7] D. Leskovsek. *Prediction of Combustion in Spark-Ignition Engines*. PhD thesis, ImperialCollegeof Science,Tecnologyand Medicine,London, 1996.
- [8] L.Dahlén.*OnAppliedCFDandModelDevelopmentinCombustionSystemsDevelopme nt in DI Diesel Engines: Prediction of Soot Mediated Oil Thickening*. PhDthesis,Royal Institute ofTechnology, Stockholm,2002.
- [9] T.Lucchini,G.D’Errico,H.JasakandZ.Tukovic.*AutomaticMeshMotionwithTopologi calChangesforEngineSimulation*.SAEPaper,2007-01-0170,2007.
- [10] T. Lucchini, G. D’Errico, F. Brusiani and G.M. Bianchi. *A Finite-Element Based*



*MeshMotionTechniqueforInternalCombustionEngineSimulations*.SAEPaper,2007.

- [11] T.Lucchini,G.D'Errico,F.Brusiani,G.M.Bianchi,Z.Tukovic,H.Jasak.*Multi-dimensional modeling of the air/fuel mixture formation process in a PFI engine formotorcycleapplications*. SAEPaper,2009-24-0015,2009.
- [12] P. Neumeier. *In-cylinder flow modeling with OpenFOAM*. Term Paper, Department ofEnergy,Politecnico di Milano,2008.
- [13] P. Strålin. *Lagrangian CFD Modeling of Impinging Diesel Sprays for DI HCCI*. PhDthesis,Royal Institute ofTechnology, Stockholm, 2007.
- [14] OpenCFDLimited.*OpenFOAMUserGuidev.1.4*.OpenCFDLimited,2007.
- [15] FluentIncorporated.*GAMBIT2.3ModelingGuide*.FluentIncorporated,2006.
- [16] OpenFOAMwebsite.<http://www.openfoam.org>.OpenCFDLimited,2009.