

# Static analysis of a novel valve design for internal combustion engines

Dr. S. SUDHAKAR BABU

Professor and Principal

Geethanjali College Of Engineering & Technology

Nannuru (Vill), Orvakallu (Man), Kurnool(Dist).

[Sudhakar4099@gmail.com](mailto:Sudhakar4099@gmail.com)

Prof. K. Rajgopal

Vice Chancellor, SK University, Anantapuram

## Abstract

*In this work CFD was used to simulate the flow through a novel valve design for internal combustion engines. CFD is numerical method for simulating the behaviour of systems involving flow processes. A FEM was used for solving the equations.*

*Literature on the topic was studied to gain an understanding of the performance limiters on the Internal combustion engine. This understanding was used to set up models that better would mimic physical phenomena compared to previous studies. The models gave plausible results as to fluid velocities and in-cylinder flow patterns.*

## 1. Background

In today's society cars, motorcycles, trucks, buses and many other types of machines have come to play a major part in how we live our lives. Many of these utilise an Internal Combustion Engine (ICE) for their means of propulsion. Considering the fact that the infrastructure around the world has been built around the use of such engines they are very likely to continue to be important.

The ICE, in its two major forms, were invented by Nicolaus A. Otto (1832-1891) in 1876 and by Rudolf Diesel (1858-1913) in 1892 respectively. Even though well over a century has passed since then, and a lot of development has gone into the field of ICE's, the fundamental design of them remains the same[1].

In a very broad sense the ICE works by inducting a charge into the combustion chamber, the charge is then ignited by a spark in a spark ignition (SI), or Otto, engine or by compression in a compression ignition (CI), or Diesel, engine. The expansion in the charge that results from the combustion then exerts a work on mechanical parts inside the engine [1].

The research that has gone into the field of ICE has resulted mostly in development in the choice of materials, and of the design of the combustion chambers and fuels. Even though the poppet valve, the by far most common type of valve, has many disadvantages when it comes to Volumetric Efficiency (see below) very little has been done to come up with a better concept. In fact, the poppet valve has been modified in different ways to improve the combustion or fuel conversion efficiency, modifications that decrease the volumetric efficiency[1]. Awkwardly, very few have thought of novel concepts that would benefit both the volumetric efficiency and the combustion process up until now [6-8].

Dafab AB has developed a new concept for ICE valves that should improve both the volumetric and fuel conversion efficiencies [4].

### 1.1. Problem statement

The purpose of this project is to make flow analysis computations, i.e. Computational Fluid Dynamics (CFD) computations of the flow over the valve design to determine how it affects the volumetric and fuel conversion efficiencies.

## Background

The full agreement between the author, his tutor and Dafab can be found in Appendix 1.

### 1.2. Limitations

To carry out the calculations CAD geometry supplied by Dafab will be used. No effort will be made to verify the correctness of the geometry.

## 2. Method

Below is described which method was used to complete the project. The outcome of the different steps are described in detail in chapters 3 to 6.

### 2.1. Literature study

The first step of this project was to study the available literature on engine design and performance. This study mainly focused on reference item [1], but also on later articles and books. See the references section for details.

### 2.2. Critical scrutinization of FS Dynamics report

CFD simulations have already been carried out on the valve design by FS Dynamics, Gothenburg. However, since these simulations were made, physical tests have been made with a prototype. The result from these tests strongly contradicts the set up of the FS Dynamics computations.

### 2.3. New computations based on the above

New computations were made based upon the outcome of the two prior steps. A commercial code, Comsol Multiphysics 4.1, will be used for these computations.

## 3. Theoretical references

Below, the theoretical references that form the base for the thesis are introduced. First an overview of the performance limiters of the ICE. Further onto how the poppet valve influences the performance limiters, and how Dafab's design works around these problems. Lastly something about the mathematics used to describe the problem.

### 3.1. Performance Limiters on the Internal Combustion Engine

The amount of power,  $P$ , that can be produced by a four-stroke SI engine can be described using the following equation

$$P = \frac{\eta_f \eta_v N V_d Q_{HV} \rho_{a,i} (F/A)}{2} \quad (3.1)$$

where  $\eta_f$  is the fuel conversion efficiency,  $\eta_v$  is the volumetric efficiency,  $N$  is the crank shaft rpm,  $V_d$  is the displaced volume,  $Q_{HV}$  is the fuel heating value,  $\rho_{a,i}$  is the air density at inlet conditions and  $(F/A)$  is the fuel/air ratio.

It can thus be said that for any engine of given displacement, the amount of power that can be produced is proportional to the fuel conversion and volumetric efficiencies of that particular engine. The other parameters such as the inlet pressure and air/fuel ratio certainly plays an important part, however it is outside the scope of this study, and they can hence be considered to be constant.

The volumetric efficiency can be calculated as

$$\eta_v = \frac{m_a}{\rho_{a,i} V_d} \quad (3.2)$$

This shows that the volumetric efficiency is proportional to the amount of air/fuel mixture (henceforth referred to as air for short) that is induced into the cylinder.

The fuel conversion efficiency is given by

$$\eta_f = \frac{W_c}{m_f Q_{HV}} \quad (3.3)$$

Where  $W_c$  can be calculated as

$$W_c = \oint p dV \quad (3.4)$$

in a  $p$ - $V$  diagram.

All the above equations are taken from Heywood [1].

It can then be derived from equations 3.2-4 that both the volumetric and fuel conversion efficiencies increase with increased mass of air induced in the cylinder, since the pressure is proportional to the mass.

It has also been shown that the fuel conversion efficiency increases further by increased in-cylinder air motion velocities. Traditionally this has been achieved by creation of swirl during the induction process, and squish during the compression stroke. The term swirl refers to the rotating motion of air around the cylinder axis, and squish to the

inward/outward motion perpendicular to the cylinder axis [1].

Given that the fuel conversion efficiency is proportional to the in-cylinder pressure, increased compression would improve this and also the overall performance of the engine. However, as the compression increases the charge in the cylinder becomes more sensitive to pre-ignition and knock. Two phenomena that are dependent on the pressure and heat, or local hot spots, in the cylinder [1], [9].

Finally, to achieve increased performance from a given engine using the same fuel under the same circumstances we can deduce that it is necessary to make changes to the engine's design that improves the flow of air into the combustion chamber.

### 3.2. The setbacks of the poppet valve

Evidently, the performance of an ICE can be limited if the flow of air into the cylinder is restricted. In internal flows the flow can be restricted if it has to pass through narrow passages or travel around sharp bends. With the poppet valve this is exactly what happens. The flow has to pass through very narrow passages while the valve opens and closes, as well as sharply change directions as it hits the top of the valve [1].

When the flow passes the valve seat in the port and also around the edge of the valve separation zones occur which further restricts the flow into the cylinder.

During the intake stroke the inlet valve gets cooled by the cool air being induced into the cylinder. However, the same thing is not true for the exhaust valve which is constantly heated by the hot exhausts passing out from the cylinder into the exhaust main fold. It is common for exhaust valves to operate at temperatures of up to 850 °C [1].

As the pressure in an air/fuel mixture increases, the flash point decreases. Eventually it will pass below the temperature of the hot exhaust valve. This phenomenon limits the compression ratio that can be used in a particular engine design. When exceeding these values one starts to experience knock and pre-ignition which reduces engine performance and wears heavily on the engine. For standard naturally aspirated engines the maximum compression ratio lies between 8-10:1 for SI engines and 17-23:1 for CI engines [1].

### 3.3. Novel valve design by Dafab

Dafab AB has developed a new type of rotating valve that theoretically possesses many advantages over the traditional poppet valve.

The valve consist of a cylinder with a slit

through it. The valve is placed in the cylinder head with its axis perpendicular to the combustion cylinder's axis and it rotates around its own axis with a mean ratio of 1:4 against the crank shaft rpm. The inlet and exhaust ports are then placed on top of the valve meaning that there is in effect only one valve serving both the intake and exhaust of air.

The motion of the valve is not strictly 1:4 against the crank shaft. The valve is accelerated during opening and closing

and then slowed down when fully open.

This means that there is a free flow of

air into the cylinder when the valve is fully open as opposed to the poppet valve where the actual valve represents an obstacle to the air. Theoretically this should mean that the rotational valve would give a much better volumetric efficiency

The fact that the valve serves as both the exhaust and intake, means that it will alternately be heated by the exhausts and cooled by the fresh air. This means that the rotation valve will keep a much lower working temperature, which in turn will enable higher compression [4].

### 3.4. Physical tests

Dafab AB has carried out physical tests on a prototype of the valve fitted to a small engine, where the air-pump effect was measured as the engine was motored. These tests showed that the rotation valve indeed improved the volumetric efficiency compared to standard poppet valves with about 50% [4].

### 3.5. Mathematics

Turbulent flows are described using the Navier-Stokes equation. For a compressible Newtonian fluid in a static case, these can be written [5]

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla \cdot p \mathbf{I} + \nabla \cdot (\mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) - \nabla \left( \frac{2}{3} \mu \nabla \cdot \mathbf{u} \right) \quad (3.5)$$

and

$$\nabla \cdot (\rho \mathbf{u}) = 0 \quad (3.6)$$

with no body forces and where  $\mathbf{u}$  is the velocity field.

Comsol Multiphysics 4.1 uses Reynolds Averaged Navier-Stokes (RANS)

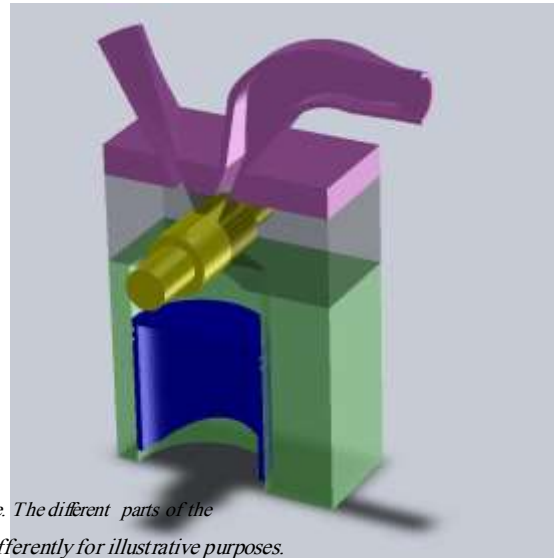


Figure 3.3.1: The valve. The different parts of the design are coloured differently for illustrative purposes.

to model the turbulence. The equations then read [2]

$$\rho(\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla \cdot p \mathbf{I} + \nabla \cdot ((\mu + \mu_T)(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) - \nabla \cdot \left( \frac{2}{3} (\mu + \mu_T) \nabla \cdot \mathbf{u} \right) - \nabla \cdot \left( \frac{2}{3} \rho k \mathbf{I} \right) \quad (3.7)$$

$$\rho(\mathbf{u} \cdot \nabla) k = \nabla \cdot \left( (\mu + \frac{\mu_T}{\sigma_k}) \nabla k \right) + P_k - \rho \epsilon \quad (3.8)$$

$$\rho(\mathbf{u} \cdot \nabla) \epsilon = \nabla \cdot \left( (\mu + \frac{\mu_T}{\sigma_\epsilon}) \nabla \epsilon \right) + C_{\epsilon 1} \frac{\rho k \epsilon}{k} - C_{\epsilon 2} \rho \frac{\epsilon^2}{k} \quad (3.9)$$

$$\mu_T = \rho C_\mu \frac{k}{\epsilon} \quad (3.10)$$

is the turbulent viscosity and

$$P_k = \mu_T (\nabla \mathbf{u} : (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) - \frac{2}{3} (\nabla \cdot \mathbf{u})^2 - \frac{2}{3} \rho k \nabla \cdot \mathbf{u} \quad (3.11)$$

a production term where

$$C_\mu = 0.09$$

$$C_{\epsilon 1} = 1.44$$

$$C_{\epsilon 2} = 1.92$$

$$\sigma_k = 1.0$$

and

$$\sigma_\epsilon = 1.3.$$

Comsol uses a Finite Element Method (FEM) to numerically solve these equations.

A FEM is a way to discretize a partial differential equation using piecewise polynomials. The domain(s) are divided into a finite number of sub-domains or elements with nodes placed in the corners<sup>1</sup>. Numerical values are computed in the nodes through the use of direct methods for linear problems or iterative methods for non-linear problems such as this. A continuous solution is obtained by interpolation between the nodes [2].

## 4. Problem

Below is a description of the problem that needs to be solved.

### 4.1. What do we want to know?

From above it is known that the amount of air that is induced into the cylinder and the flow pattern inside the cylinder strongly affects the performance of the engine. It is therefore interesting to study the velocities and flow patterns during the intake and exhaust phases.

#### 4.2. How well did FS Dynamics deliver this results?

The previous calculations that were carried out was attempted at simulating a blow test of the valve configuration. According to Heywood [1], physical blow tests were used by engine makers to determine the amount of swirl a certain valve and port generated. In these tests a cylinder with valves but no piston were used, and a measuring device fitted in the bottom of the cylinder measured the amount of torque the air generated and hence the amount of swirl could be determined.

Considering the design of the port and valve in the FS Dynamics model, and the fact that static computations were made it is not likely that any swirl will occur at all, and thus this approach could be questioned.

Further, the boundary conditions used for the calculations were based on a few assumptions. For the two intake simulations a mass flow inlet boundary condition was used. According to the report the mass flow has somehow been calculated, but it doesn't tell how this was calculated. It doesn't tell from which data the calculation(s) were done. The mass flow was then scaled based on the assumption that the flow would be constant during 80% of the intake stroke. As the rotation valve was an entirely new concept at the time of these calculations it would have been impossible for FS Dynamics to have any data to base such an assumption on. They further assumed the volumetric efficiency to be 100%, yet another assumption lacking supporting data. From the physical tests the volumetric efficiency was shown to be 186% at 3000 rpm [4].

A pressure outlet boundary condition was then placed at the top of the piston. However, it is evident from looking at the pictures of the results from the computations that no attempt was done to modify the geometry to mirror the different volume the cylinder has at different crank angles. Also the pressure boundary condition doesn't capture in-cylinder flow patterns. Rather it lets the air pass straight through instead of forcing it to bounce off the cylinder walls and piston top and tumble around.

For the exhaust simulations the boundary conditions were simply reversed, which is consistent with the inlet boundary conditions, but still poorly motivated.

Since the domain is symmetric, FS Dynamics chose to split it in half and only compute on one of the halves. There is nothing that says that the actual airflow is symmetric. By computing on one half of the domain only, the flow will however, be forced to be symmetric in the simulations by the use of a symmetry



boundary condition.

#### 4.3. How can the computations be done better?

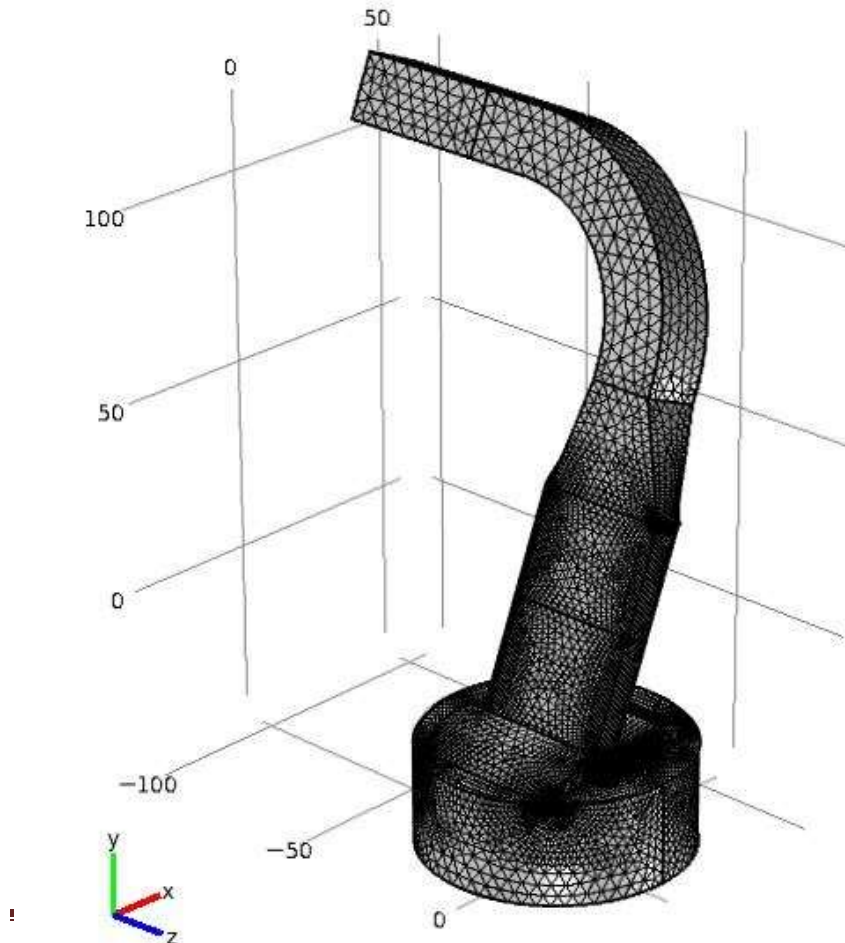
The overall performance and behaviour of an engine equipped with a rotation valve cannot be assumed known. It is therefore important to model the problem as close to reality as possible and to make as few simplifications as possible.

In this study the full model has been used to make sure no asymmetric turbulence phenomenon would be missed. A moving wall boundary condition has been put on the top of the piston to simulate the upward or downward motion of the piston, and pressure boundary conditions on the inlet and outlet openings.

### 5. The models

Below follows a short description of the four models used to solve the problem.

The models consists of three domains, inlet or exhaust, valve and cylinder. They are meshed with between 270617 and 333012 tetrahedral elements depending on the configuration and the size of the cylinder. Linear basis functions were used everywhere.



### 5.1. Intake fully open

The geometry was first mirrored to give a full model. Secondly, the volume of the cylinder was adjusted to the actual volume of the cylinder at 45° crank angle from top dead centre when the valve is fully open to the intake port. These values were based on data supplied by Dafab AB [4].

A downward velocity of 7.07 m/s was applied to the top of the piston. This velocity was derived from strictly geometrical relationships and the full computations can be found in Appendix 2. A pressure boundary condition of 1 atmosphere was put on the inlet boundary. Slip wall functions were used on all other boundaries. Batchelor [5] concludes that the thickness of the boundary layer shrinks considerably with increased Reynolds number. Since the Reynolds number in these computations is in the range of 150000 and upwards, the boundary layers' impact on the flow would be negligible so the omission of wall functions can be justified. Further, this approach removes the need for boundary layer meshes which keeps down the size of the models.

Compressible air was used and turbulence was modelled using the RANS model.

This model would not converge, and it would eventually turn out that the geometry given by the CAD model was bad and probably “leaked” somewhere.

At this point it was decided that new geometry had to be created. Since the actual geometry used in the physical tests doesn't look exactly as the geometry used in the first computations, the new geometry should correspond better to the physical geometry. That way it would be easier to compare the calculations and the results from the physical tests.

After the new geometry was created the same boundary conditions were used. These calculations had huge trouble to converge too. It was concluded that the inlet boundary condition was a source of instability [2], and instead a boundary that only allowed normal velocities was used, and the computation converged without any problem.

The mesh was then refined to obtain a mesh twice as fine, and the computation was rerun, albeit with the first solution as initial condition. This computation also converged without any problem and gave roughly the same result.

### The models

It could be argued that the normal velocity boundary condition would give a somewhat misleading flow pattern. Since it is fairly uninteresting to look at the flow pattern at the inlet end of the model this was neglected. For good measure an extension was added to the inlet to counteract any too artificial flow patterns.

#### 5.2. Exhaust fully open

The same geometry as for the “Inlet fully open” computation was used but with the valve rotated 45° to the exhaust port.

The velocity for the moving wall boundary condition on the piston top was reversed and a pressure outlet boundary condition of 1 atmosphere was used on the outlet boundary.

In every other sense the computation was set up identically as in the first computation.

The computation converged without any problems. The mesh was again refined and the computation rerun. It converged and gave the same result.

#### 5.3. Intake part open

This model was built like the “Intake fully open”, but with the volume of the cylinder changed and the valve rotated to leave a 4 mm gap between the valve wall and the intake port wall. This occurs at 140° after top dead centre.

The velocity for the piston top wall was modified to match the velocity of the piston and was set to -3.61 m/s.

The computation converged. The same procedure with mesh refinement was applied, and the solution gave the same result.

#### 5.4. Exhaust part open

This model was built like the “Exhaust fully open”, but with the volume of the cylinder changed and the valve rotated to leave a 4 mm gap between the valve wall and the exhaust port wall. This occurs at 55° before top dead centre.

As with “Intake part open” the velocity for the piston top wall was modified to match the velocity of the piston and was set to 6.69 m/s.

The computation converged. The same procedure with mesh refinement was applied, and the solution gave the same result.

## 6. Results

The result of the computations are shown in the picture sets below.

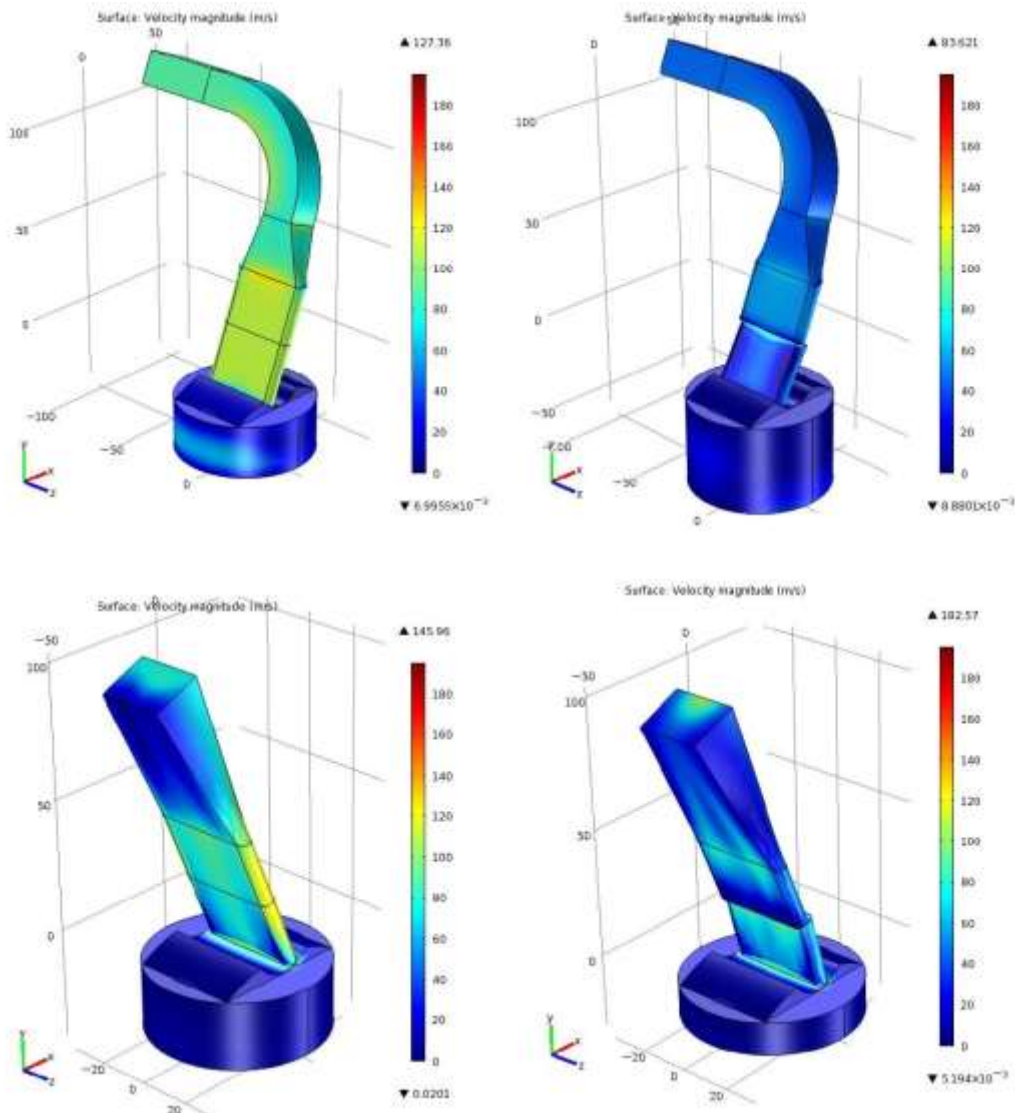


Figure 6.1: Surface plot of velocity magnitude (m/s).

Results

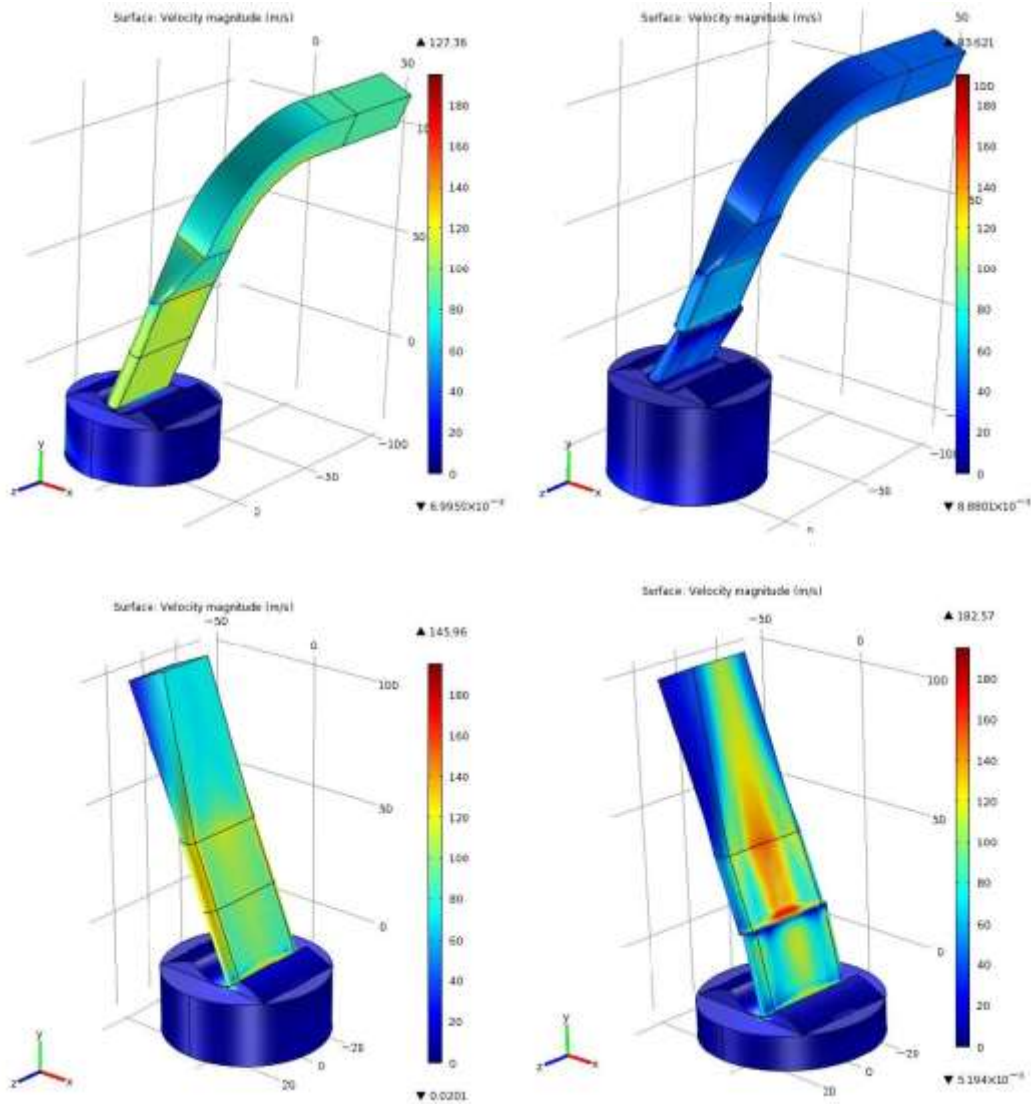


Figure 6.2: Surface plot of velocity magnitude (m/s).



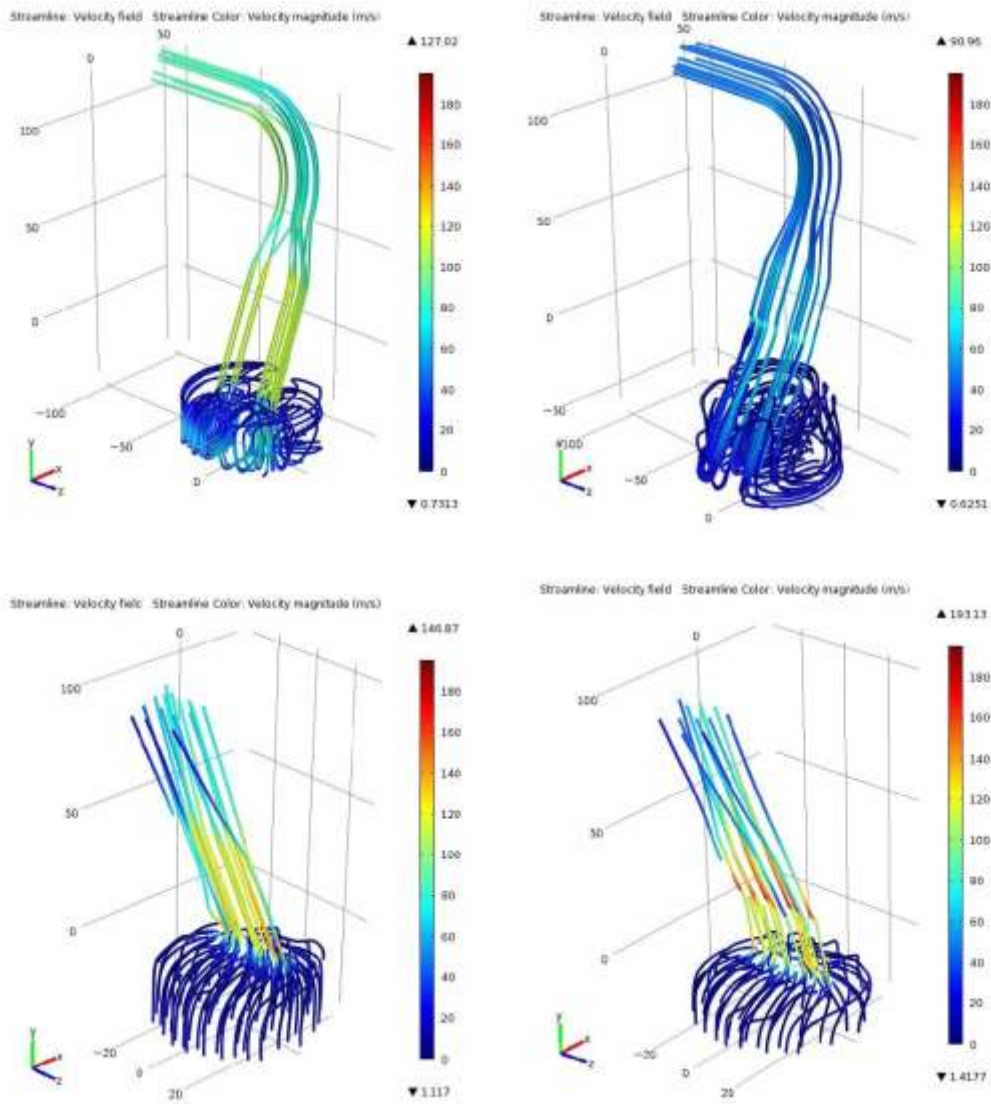


Figure 6.3: Streamline plots of velocity field.  
Colour according to velocity magnitude (m/s).

Results

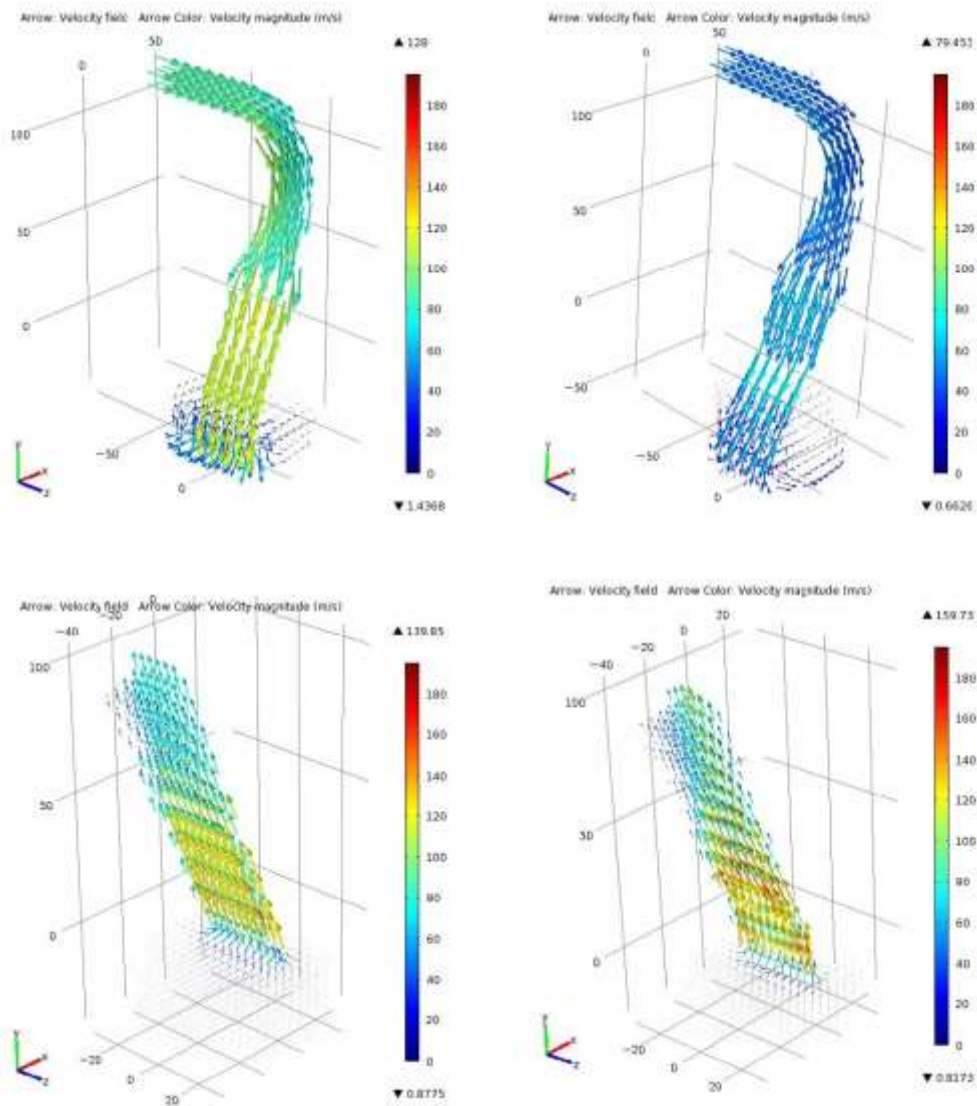


Figure 6.4: Arrow plots of velocity magnitude (m/s).

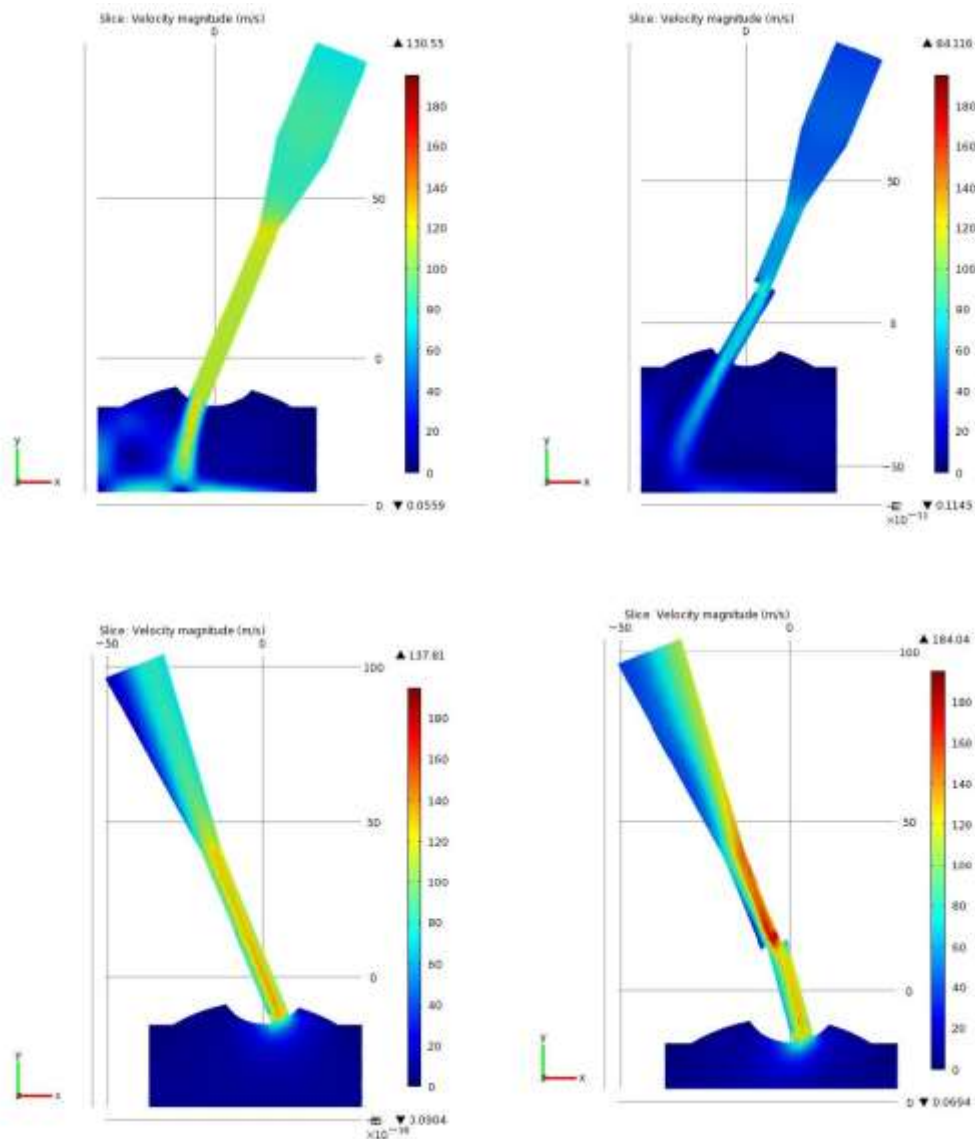


Figure 6.5: Velocity magnitude in cross section (m/s).



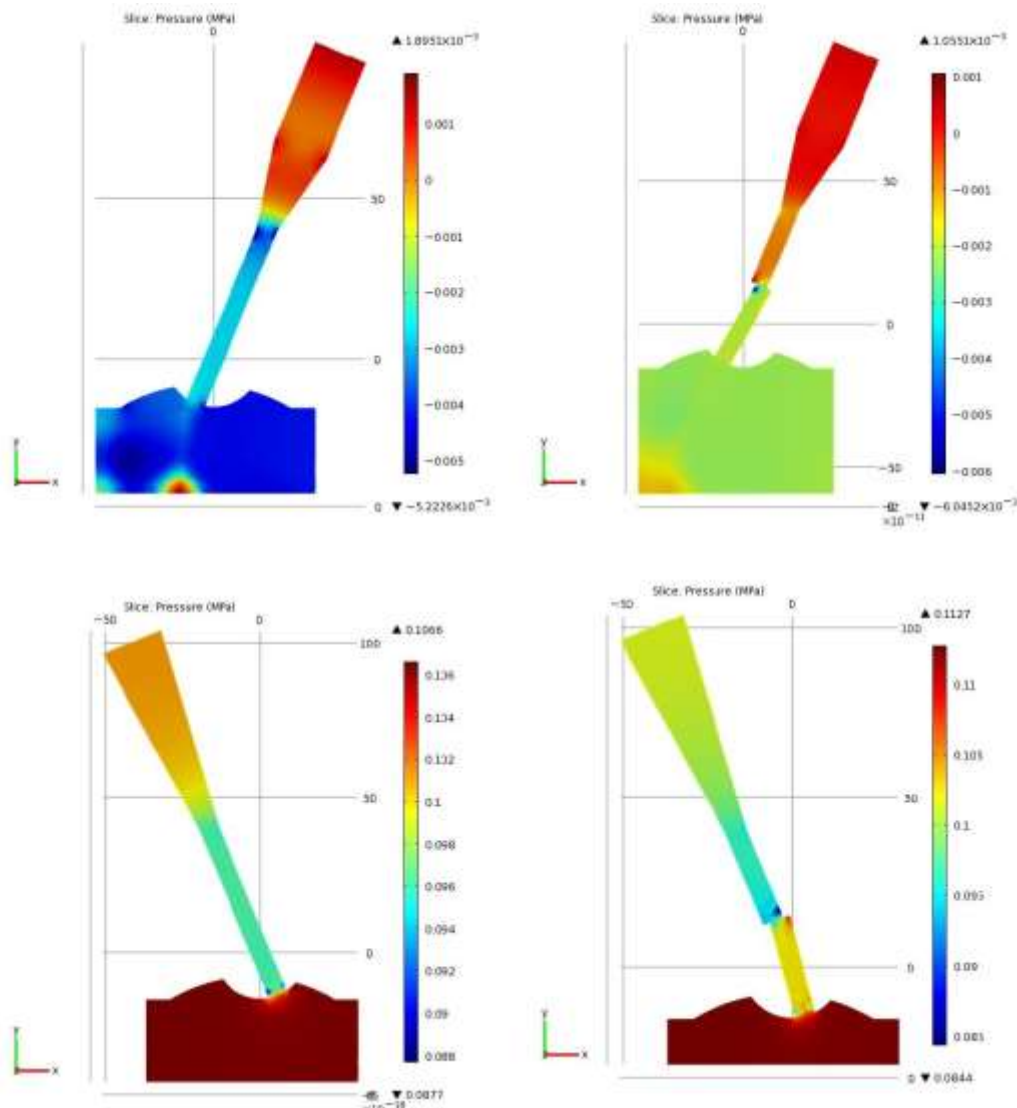


Figure 6.6: Pressure in cross section (MPa). Note the difference in range between the plots!

### 6.1. Analysis

From the pictures above, it can be seen that the rotation valve enables a very free flow in and out of the cylinder. Some separations zones occur during opening and closing which is inevitable.

The streamline and arrow plots suggest that there is plenty of in-cylinder air motion that would be beneficial for the fuel conversion efficiency.

## 7. Discussion

The breathing mechanism of an engine is indeed a very dynamic process and static analysis of a few instances or snap shots doesn't tell the whole truth.

Since the valve rotates, it is reasonable to suspect the in-cylinder flow pattern would look slightly different and perhaps tumble around more.

For the exhaust phase there would probably also be more complex in-cylinder flow considering that there would have been an explosion taking place just prior to the instance(s) depicted above. Whether or not this would have any impact on the flow through the valve and port is difficult to tell as this is an entirely new concept, and very little is known about it.

While comparing the result from this study with the result <sup>2</sup> from the study made by FS Dynamics there are clear differences in a couple of places. Firstly the studies by FS Dynamics didn't in any way capture the in-cylinder flow patterns. Secondly, the velocities in the different studies differ greatly which casts doubt over the choice of boundary condition used by FS Dynamics. The maximum velocity found in FS Dynamics studies is about 60 m/s and can be found during the intake phase, whereas in the present study it is about 190 m/s and is found as the valve is closing during the exhaust phase. Now, it can of course be argued that the initial condition of 1 atmosphere pressure in the cylinder at the start of the calculation is too low as there would have been an explosion just prior with a likely increase in pressure and temperature. However, this would yield an even higher velocity.

To achieve a better understanding of an engine's fitted with a rotation valve breathing mechanism it would be necessary to use transient computations.

## 8. Conclusions

This thesis aimed at producing better simulations of the intake and exhaust phases of a naturally aspirated four stroke internal combustion engine fitted with a rotation valve.

The results show that the valve enables a free flow of air in and out of the cylinder with the valve fully open, albeit with some inevitable separation zones occurring while the valve opens and closes. The flow patterns shown suggest that the rotation valve can be beneficial for the combustion process as well.

Since the breathing mechanism is very dynamic it is, however, still

recommended that transient studies with a moving valve and piston are made before any further conclusions can be drawn. For comparison it would be beneficial to make a simulation of the breathing for the same engine but with traditional poppet valves fitted.

## 9. References

- [1] Heywood, J.B. (1988). *Internal Combustion Engine Fundamentals*. International Edition, McGraw-Hill, Singapore, ISBN 0-07-100499-8
- [2] Comsol Multiphysics 4.1, Help
- [3] Felippa, C. *Introduction to Finite Element Methods*. 2011-06-12  
<http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch01.d/IFEM.Ch01.pdf>
- [4] Dahlborg, C. Dafab AB. (2011-02-01 to 2011-05-30) +46706860522
- [5] Batchelor, G.K. (2000). *An Introduction to Fluid Dynamics*. New York, Cambridge University Press. ISBN 978-0-521-66396-0
- [6] Coates, G.J, *Spherical rotary valve assembly for use in a rotary vavle internal combustion engine*. U.S. Patent 5,341,739, 1993-05-12
- [7] Zimmerman, M.A, *Rotary valve internal combustion engine*. U.S. Patent 3,871,340, 1972-10-03
- [8] Hagiwara, A. *Rotary Sleeve-valve Internal Combustion Engine*. U.S. Patent 5,191,863, 1993-03-09
- [9] Costa, R.C., Sodré, J.R., (2011) *Compression ration effects on an ethanol/gasoline fuelled engine performance*. Applied Thermal Engineering, 31, p. 278-283