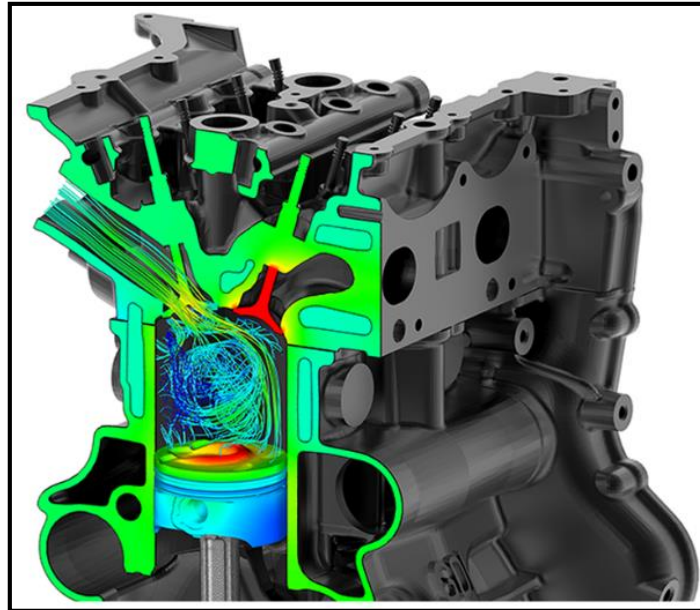


Tumble-Flow Development Through Port Geometry Optimisation and the Influence of Turbulence on GDI Engines Efficiency and Emissions.



Research Project Completed By:
YOUSOUF BOUKRA / 1700055

▪ Supervised by: Andrew Gibson

Bachelor's Degree Thesis
Submitted to the Faculty of Engineering
Automotive/Motorsport Department

April 2020



University of Wales Trinity Saint David

Contents

ABSTRACT 6

ACKNOWLEDGEMENTS 7

ABBREVIATIONS 8

Chapter 1: Introduction..... 9

1.1. Background 9

1.2. Motivation and Aims 10

Chapter 2: Literature Review..... 11

2.1. IC Engine Efficiency, Emissions and Related Pollution 11

2.2. IC Engine Development and GDI Technology 13

2.3. IN-Cylinder Charge Motion 16

Chapter 3: Methodology 19

3.1. Data Collection and Processing 20

3.2. CAE Design and Validation 23

3.2.1. Cross-sections Method: 23

3.2.2. Surface-knit Method: 24

3.2.3. CAD models Validation: 25

3.3. CFD Simulation and Flow Validation 27

3.3.1. Flow-bench Experiment: 28

3.3.2. CFD Set-up Procedures: 29

3.3.3. Turbulence Modelling: 31

3.3.4. Theoretical Verification: 33

Chapter 4: Results Discussion 34

4.1. CFD Model Validation 34

4.1.1. RNG K-ε Turbulence Model: 36

4.1.2. Realisable K-ε Turbulence Model: 37

4.1.3. Flow-Velocity Verification: 39

4.2. Base-Model Analysis 40

4.3.

Optimised-Design Analysis.....

43

4.3.1.

Design Parameters:

43

4.3.2.

Design Concept:.....

44

4.3.3.

Optimised-Design CFD Results:

46

4.4.

Designs Comparison

48

4.4.1.

Intake-port Flow-velocity:

48

4.4.2.

In-Cylinder Turbulence:.....

50

CONCLUSION

54

REFERENCES.....

55

APPENDIX

62

List of Figures

Figure 1 Charge Movement through port (Schogl et al. 2011) 16

Figure 2 In-Cylinder Flow-Motion (Kim et al. 2019) 16

Figure 3 Intake Port Layouts and Different flow motions (Schogl et al. 2011) 17

Figure 4 Project development plan (Yousouf, 2020) 20

Figure 5 Highlight of missing data from scanned files (Yousouf, 2020)..... 21

Figure 6 Surface File Repair (Yousouf, 2020) 21

Figure 7 Combustion Chamber Clean Surface File (Yousouf, 2020) 22

Figure 8 Intake port Clean Surface File (Yousouf, 2020) 22

Figure 9 Combustion Chamber CAD model (Yousouf, 2020)..... 24

Figure 10 Cylinder-Head Air-Domain CAD model (Yousouf, 2020)..... 25

Figure 11 Port GOM inspection (Tibbott and Cooper, 2020) 25

Figure 12 Actual Vs simplified combustion chamber (Kim et al. 2019) 26

Figure 13 Engine Specifications (Yousouf, 2020) 28

Figure 14 Table of Equations and Values (Yousouf, 2020) 28

Figure 15 Experimental Mass flow graph (Yousouf, 2020)..... 34

Figure 16 RNG K-e turbulence model mass flow graphs (Yousouf, 2020) 36

Figure 17 RNG K-e turbulence model results (Yousouf, 2020) 36

Figure 18 Realisable K-e turbulence model mass flow graphs (Yousouf, 2020) 37

Figure 19 Realisable K-e turbulence model results (Yousouf, 2020) 37

Figure 20 Realisable K-e enhanced wall model graph (Yousouf, 2020)..... 38

Figure 21 CFD flow velocity verification (Yousouf, 2020) 39

Figure 22 Boundary conditions for boosted engine simulation (Yousouf,2020)..... 40

Figure 23 Base-line 2mm CFD post results (Yousouf, 2020) 41

Figure 24 Base-line 4mm CFD post results (Yousouf, 2020) 41

Figure 25 Base-line 6mm CFD post results (Yousouf, 2020) 42

Figure 26 Base-line 8mm CFD post results (Yousouf, 2020) 42

Figure 27 Port masking technique (Ye et al. 2018) 43

Figure 28 Intak-Port design parameters (Yousouf, 2020)..... 43

Figure 29 Base-model 2D sketch (Yousouf, 2020)..... 44

Figure 30 Optimised-model 2D sketch (Yousouf, 2020)..... 44

Figure 31 Base-line vs Optimised-model overlaid (Yousouf, 2020)..... 44

Figure 32 Base-line Vs optimised-design 3D sketch (Yousouf, 2020)..... 45

Figure 33 Optimised-port 2mm CFD post results (Yousouf, 2020)..... 46

Figure 34 Optimised-port 4mm CFD post results (Yousouf, 2020)..... 46

Figure 35 Optimised-port 6mm CFD post results (Yousouf, 2020)..... 47

Figure 36 Optimised-port 8mm CFD post results (Yousouf, 2020)..... 47

Figure 37 Base vs Optimised port, flow velocity at 6mm (Yousouf, 2020)..... 48

Figure 38 Base vs Optimised port, flow velocity at 2mm (Yousouf, 2020)..... 48

Figure 39 Base vs Optimised port, flow velocity at 8mm (Yousouf, 2020)..... 49

Figure 40 Base vs Optimised port, flow at 8mm top-view (Yousouf, 2020) 49

Figure 41 Base vs Optimised port, Turbulence at 8mm (Yousouf, 2020) 50

Figure 42 Base vs Optimised port, turbulence at 4mm (Yousouf, 2020) 50

Figure 43 Base vs Optimised port, mass flow results (Yousouf, 2020) 51

Figure 44 Base vs Optimised port, mass flow over VL graph (Yousouf, 2020) 51

Figure 45 Base vs Optimised Turbulent Kinetic Energy Graph (Yousouf, 2020) 52

Figure 46 Base vs Optimised port, Turbulent intensity Graph (Yousouf, 2020)..... 52

Figure 47 tumble across different engine RPM (Cameretti et al. 2017)..... 53

ABSTRACT

The stringent emissions legislation, and the future requirements to develop a high-end ICE technology, are the aim of every automotive company around the world (Adomeit *et al.* 2005). Gasoline direct injection systems in particular, has been widely utilised in the automotive industry today, due to their higher fuel economy and limited greenhouse gas emissions, according to Premnath and Khalek (2019). Kim *et al.* (2019) cited that, turbulence is one of the most paramount characteristics of the GDI engines, as it substantially influences burn rates, combustion stability, and heat transfer rates. Turbulence occurs during a large-scale flow motion generated by the intake port, and tumble-motion is the most dominant flow inside the combustion of a gasoline engine (Ikeya *et al.* 2015).

Through the Implementation of CFD simulation tool and CAE design, an optimised intake-port was developed, based on a base-port of a 1.6 GDI engine. The study results revealed that, the high-tumble optimised port generated higher in-cylinder turbulence flow-structure. Despite the minimal decrease of 2.3% in mass flow-rate due to the aggressively designed port-profile, the turbulent intensity was enhanced by 7.72%, resulting to a substantial increase in turbulent kinetic energy (17.8%).

ACKNOWLEDGEMENTS

الله الحمد، الى صالاته تم به نعمته الذي العالمين رب الله والشكر والحمد الله به سم
العمل هذا اكملت علاه في جل منه وتسهيل به توفيقه الذي

“In the Name of Allah, the Most Beneficent, the Most Merciful.
All the praises and thanks be to Allah, the Lord of All Worlds”

Special thanks to my ‘Big Family’ I am so grateful for your support and sacrifices.
Thank you Harag, Kheira, Nadir, Amine, Amaria, Zoulikha, and my life partner,
Emily.

Thank you Andrew Gibson! Thank you for sharing your valuable knowledge and
expertise, and for your consistent support and guidance.

Thank you Tim, for your advice on CFD procedures.

Thank you Andrew and Dave for your tremendous physical work on 3D scanning and
the Manufacturing process.

ABBREVIATIONS

CAE: Computer-aided engineering
CAD: Computer-aided design
CAM: Computer-aided manufacturing
CFD: Computational fluid dynamics
RE: Reverse engineering
IC: Internal combustion
ICE: Internal combustion engine
SI: Spark ignition
DI: Direct Injection
GDI: Gasoline direct injection
CIDi: Compression ignition direct injection
PFI: Port fuel injection
CC: Combustion chamber
VVT: Variable valve timing
VVT-i: Variable intake and exhaust valve timing
BSFC: Brake specific fuel consumption
PM: Particulate matter
NOx: Nitrogen oxides
TI: Turbulent intensity
TKE: Turbulent kinetic energy
TR: Tumble ratio
VL: Valve lift
HEV: Hybrid electric vehicle
WI: Water injection
EGR: Exhaust gas recirculation
c-EGR: Cooled exhaust gas recirculation

Chapter 1: Introduction

1.1. Background

Future guidelines to increase ICE thermal efficiency, limit exhaust emissions, and reduce fuel consumption, are motivating the development of highly advanced combustion concepts (Berntsson *et al.* 2011; Ye *et al.* 2018). Gasoline direct injection system coupled with turbocharging technology and downsizing, have substantially contributed to the development of IC engine, on meeting the future emission requirements, as mentioned by Iyer and Yi (2009). Wentsch (2018) cited that the automotive industry is striving to secure the internal combustion engine market-place, especially within the hybrid vehicle application in the long-term future.

Cylinder-head design, and intake port development is an important parameter since it impacts the volumetric efficiency and in-cylinder charge motion, as highlighted by Selvaraj (2011). The combustion process development is firmly related to the port geometry, both emissions and combustion performance are connected to turbulent intensity, and the quality of mixture formation, according to Adomeit *et al.* (2005) and Sakurai *et al.* (2017). An optimised intake-port layout aids on the generation of the first tumble peak, prompting an in-cylinder turbulence which consequently influences the combustion process (Singh *et al.* 2019).

CAE methods, including CAD and CFD have become a fundamental tool in today's engineering world. Due to the optimisation of IT technology and computing power, the development of CFD codes as discussed by Vaidya (2010) has received significant interest since 1990. In particular, some CFD codes for engine flow simulation have further developed through academic groups, as well as commercial ones.

In the present study, a GDI intake port was analysed throughout the application of CAE tools and CFD simulation, to assess the tumble motion and in-cylinder flow structure. Solidworks package was implemented for the design and modelling, and the simulation process was conducted on ANSYS workbench. Extensive validation work was conducted using different turbulence models simulation, experimental flow-bench testing, and analytical verification. Further optimisation work was carried out to alter the tumble charge, by means of geometrical port modification, where the main design parameters comprised port profile angle, chimney height, and port masking.

In line to what have been said, the project stimulated multiple aims which were paramount to the success of this research.

1.2. Motivation and Aims

- ❖ The main aim of this work was to critically analyse and assess the influence of intake-port geometry, on tumble generation, and in-cylinder turbulence intensity. The complex turbulent flow occurring during the intake stroke, has motivated the implementation of CFD engine simulation tool, which was an imperative technology and has been utilised extensively during this project.
- ❖ The second aim of this study included a comprehensive investigation, regarding turbulence modelling within a 3-D CFD domain, to discover the ideal turbulence model, which accurately predicts the fluid flow structure inside the cylinder. Thus, a solid flow-bench data was required, along with analytical verification, in order to set the benchmarks for the CFD results.

Chapter 2: Literature Review

2.1. IC Engine Efficiency, Emissions and Related Pollution

Energy and environmental concern such as the depletion of fossil fuel resources, and air pollution as mentioned by Mito *et al.* (2012, p. 45) have become a critical and even more problematic matter in modern world. Multiple researches has pointed to the automotive industry as a serious contributor to environmental damage, especially air or ambient impurity and human health infection. Furthermore, the automotive industry has a higher demand of oil, which massively translated to a deficiency to energy resources (Mito *et al.* 2012, p45-58). Similarly, Engeljehringer (2004), have also reported that motor vehicles are a contributor to air pollution. Specifically, the automotive sector shares a significant amount of CO₂ emissions, which is considered one of the greenhouse gases, leading to a possible impact on the world climate change. Engeljehringer discusses that, CO₂ emission is a result of the complete combustion process inside an internal combustion engine operating on fossil fuel, and could not be devaluated by any exhaust after treatment system.

Continuously, a recent research concerning air pollution and health, has declared that combustion of fossil fuels and biomass is the major source of air pollution as Hext *et al* mentioned. Highlighting airborne particulate matter as the greatest threat to human health, reveals a general toxicology laboratory. The untreated emissions of combustion consist of a compelling concentration of ultrafine, fine, and large particles, which are sourced from different fossil fuel engines, mentioning the mobility sector, including petroleum powered cars, buses, and trucks. Those last once are considered the primary source of air deterioration in cities and public areas. Whereas other type of mobilities, such as ships and aircraft, are the main source of air pollution near ports and airports zones (AOSO *et al*, 2019; HEI review panel, 2002; Hext *et al*, 1999).

Pollution is the greatest environmental antecedent of disease and early death in recent years. Late statistics results by Landrigan *et al.* (2018, p. 462) estimated 9 million premature deaths in 2015, caused by illnesses related to environmental pollution. The increasing application of petroleum powered cars, trucks, and buses are recognised as a key drivers of ambient air pollution says the lancet commission of pollution and health. Emissions associated to vehicles including black carbon, impacts human health but also regional temperature and extreme weather. The arctic and glaciated

regions are sensitive to melt as consequence of accumulated black carbon which heats the surface in places such as the Himalayas (AOSO *et al*, 2019; Landrigan, 2018).

In line to what have been discussed above and as a response from the automotive industry parties, multiple regulations regarding vehicles emissions have been set in place. And as stated in the latest Bosch automotive handbook, California announced the first emissions control regulations regarding petrol engines in the mid-1960s. those regulation have been then implemented widely in all industrialised countries, and developed further to become progressively tighter each year (Bosch, 2014, p. 532).

Future concerns to decrease exhaust emissions and fuel consumption are stimulating the development of highly advanced combustion concepts, and as Bengtsson *et al*. (2011, p. 2298) cited, diesel engines are highly developed to perform better in term of efficiency, but with the disadvantage of generating higher amount of NO_x and particles. In the other hand, petrol engines have shown a promising future regarding meeting the stringent emission legislation, but with lower efficiency in comparison to their counterparts. As an example and for the request to meet the strict fuel economy and emissions standards, Dongfeng Motor Corporation has upgraded a 1.8 litre four-cylinder turbocharged engine, based on a previous 2.0~2.3 litre engines to be mounted on SUV and mass production vehicles (Ye et al, 2018, p.1).

Europe, America, and Japan, as the leaders in the automotive technology development, have optimised engines and exhaust after-treatment equipment for extremely low emission level as declared by Kurt (2004). As a result, vehicle's level of emission today is very small compared to what they did emit before the implementation of after-treatment technologies. Almost every reduction of emissions limits has necessitated a corresponding development in design or technology of the engine or their exhaust after-treatment system, or even their fuel supply and lubricants (Kurt, 2004). From a design side for example, the development of a further advanced intake ports as cited by Karaya and colleagues (2019, p.1), is a key factor to the amount of air induced into the cylinder during the intake stroke, which thereby positively control the volumetric efficiency of the engine.

Automotive engineers are striving to design optimised engines with an enhanced brake-specific fuel consumption (BSFC), and as Tikar *et al*. (2019, p.1) discussed, the

in-cylinder flow structure of an IC engine controls the combustion process, and directly alters the emission formation and fuel consumption level. Moreover, the combustion efficiency of a direct injection engines is massively dependent upon the mixture of fuel and air. Thus, generating a combustible mixture will level up the overall engine efficiency and performance (Tikar, Maikhede and Nandgaonkar, 2019; Verma, Hill and Xu, 2018).

2.2. IC Engine Development and GDI Technology

IC engines has continued to be a dominant social and economic force for decades as highlighted by Qi *et al.* (2010). A combination of high-power output, efficient BSFC and extremely low emissions, are the dominant aims for every automotive company. Consequently, in order to achieve these aims, HEV technology is now the leading innovation within the market space. Thus, IC engines are paramount to reduce the overall impact of both ICE powered vehicles and hybrid-electric cars (Qi *et al.*, 2010).

Bandel *et al.* (2006) considered the existing technology involving engine downsizing, and/or down-speeding (longer gear ratio) as a novel achievement to reduce fuel consumption, this innovative measure, made it possible to shift engine operation mode toward higher efficiency area (i.e. at higher load). Furthermore, the combination of that technology with turbocharging resulted to a very interesting vehicle performance. The turbocharging technology related to both petrol and diesel engines is gaining an increasing attention for development each year, this technology has been introduced to passenger vehicles since mid-1970s (Arnold *et al.*, 2005; Bosch, 2014).

Data from Petitjean *et al.* (2004) revealed that for an equivalent power figure, turbocharging allows gasoline engine downsizing by 30%, increase torque and acceleration performance, while improving fuel economy by about 10%. This investigation by Petitjean and his colleague, was performed on multiple hundred sedan vehicles over a ten-year period, to compare turbocharged versus naturally aspirated engines. Likewise, it was cited by Kirwan *et al.* (2010) that turbo-diesel powertrain offers a very low fuel consumption and are effective in term of low-end torque.

Water Injection (WI) is another technology concerning ICE performance and emissions, which has recently gained a substantial researcher interest. Tornatore *et al.* (2017) discussed WI technology within downsized turbocharged SI engines, where the main objective of the study was to demonstrate that WI might be a reliable solution

to improve fuel efficiency, through advanced spark timing, and leaner mixture ratios that will benefit knock tendency. Alternatively, other strategies have been developed to address the propensity of knocking at high loads, for example Cooled EGR (c-EGR) as evident by many researchers, is a successful approach which mitigates knock at high loads, while reducing NO_x emissions (Johnson and Joshi, 2018; Qi *et al.*, 2010). Johnson and Joshi also cited that, c-EGR along with cylinder deactivation techniques, can improve fuel efficiency through reduced pumping work.

Fischer *et al.* (2017) investigation revealed that, engines with c-EGR systems reduce fuel consumption by more than 10%, simultaneously, particulate numbers are declined by over 75%. Engine knocking is confirmed to be eliminated and exhaust temperature was significantly reduced, as a result, substantial increase of engine compression ratio was achieved to further improve fuel economy across the entire engine map.

In the other hand, the development of Gasoline Direct Injection system (GDI) have contributed extensively to the optimisation of IC engines. As confirmed by Zhao, Lai, and Harrington (1999) the thermodynamic capabilities of petrol engines that are designed to inject fuel directly into the cylinder, has significantly enhanced fuel economy, transient response, and reduced cold-start hydrocarbon emissions. This technology as described by the above authors, has led to a considerable number of research and development projects, that aims to understand and optimise GDI combustion system. Fuel injection processes, spray atomization, charge cooling, mixture preparation and in-cylinder air movement are all being thoughtfully researched saying Zhao and colleague. Furthermore, Kirwan *et al.* (2010) confirmed that GDI fuel system provides the capability of utilising an increased scavenging without sacrificing hydrocarbon emissions, due to the separated control of fuel and air. It also provides the benefit of charge cooling that reduces knock, which is necessary for turbocharging applications.

The application of Direct Injection (DI) in petrol engine, certainly support the combination of previously discussed technologies such as turbocharging and engine downsizing, benefiting from the flexibility of DI system, make it possible to overcome the deficiencies of these technologies says Wirth *et al.* (2000). GDI engines offer several considerable advantages in comparison to Port Fuel Injection (PFI) engines, due to its ability of eliminating fuel wall film inside intake port as well as reducing the

throttling/pumping losses, while producing higher volumetric and thermal efficiency, better BSFC, and lower emission level (Smith and Sick, 2006). Bandel *et al.* (2006) discussed that turbocharged GDI engines with VVT technology make it possible to implement synergy effects, which aid on the achievement of a high fuel-efficient engine concept suitable for low emission.

GDI engines operates in both stratified charge mode and homogenous charge mode says Kume *et al.* (1996) in a study where the description of these different charges were explained as followed: A stratified charge where late injection is occurred, can be implemented at lower engine speeds/loads as it will enable the engine to burn a very lean mixture with high air/fuel ratios, however, more NO_x is emitted due to the lean burn. Thus, EGR is very effective during this operation mode to reduce NO_x level. Homogenous charge which requires early injection is utilised for higher load and higher speeds, the fuel is sprayed early at the intake stroke to allow more time for the air/fuel mixture formation, at this mode the engine operates with a near stoichiometric mixture, therefore, NO_x level is reduced without the use of EGR system (El-Adawy and Munir, 2017; Kume *et al.*, 1996).

Mixture formation in GDI engines plays an important role which alter the combustion characteristic, performance, and emission level. Karaya, Addepalli and Mallikarjuna (2019) suggested that in order to get better mixture formation, various aspects must be considered such as, intake port design and fuel spray dynamics.

2.3. IN-Cylinder Charge Motion

Cylinder-head design for an SI engine is important as it impacts performance, fuel economy and emissions. Specifically, designing an effective intake port is being widely investigated, since it determines the volumetric efficiency and in-cylinder charge motion which effects the thermodynamic properties of IC engine. More mass flowing into the engine can improve volumetric efficiency, whereas increasing intake air/charge motion and turbulence generation, will enhance combustion rate in order to develop a higher thermal efficiency and a better emission control (Selvaraj *et al.* 2011; Singh *et al.*, 2019).

Figure 1 and 2, shows the different in-cylinder charge movement across the three X – Y – Z axis, as described by several engineers in this field. The blue arrow represents the tumble flow-charge movement direction as it rotates around the Z axis, the green arrow represents the swirl movement as the air flow around the Y axis, which identify the centreline of the cylinder. Whereas the cross-tumble is a movement similar to the tumble charge, but it flows around the X axis, which is parallel to the port design, therefore the flow around this Axis is not considered (Kim *et al.* 2019; Schogl *et al.*, 2011).

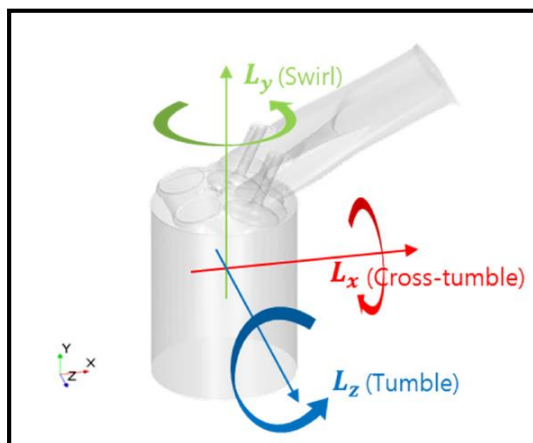


Figure 2 In-Cylinder Flow-Motion (Kim *et al.* 2019)

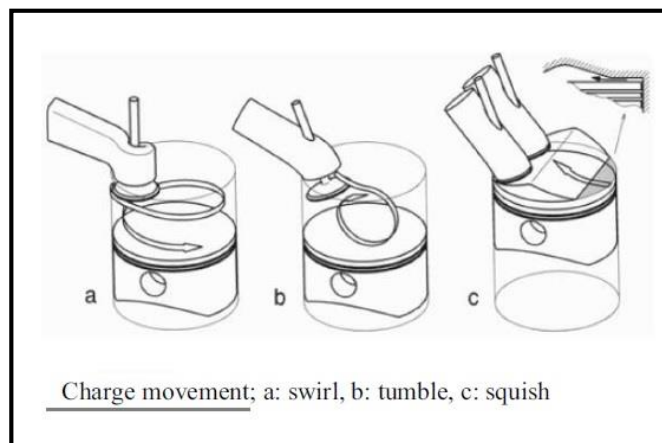


Figure 1 Charge Movement through port (Schogl *et al.* 2011)

Schogl *et al.* (2011) mentioned that controlling the charge motion into different direction inside the cylinder, directly depend on the port design layout or port geometry. Figure 3 shows the different port geometries with relation to the charge movement, where the authors describes the different charge modes as followed: Port layout 'a' represent the filling port which is efficient on delivering a higher mass flow, due to the smooth and fluent curvature of the design, steep angle and round profile. The disadvantage of this design is the lack of in-cylinder turbulence intensity,

as the flow is not intentionally guided or directed toward a certain path. Port layout 'b' represent the tumble-oriented port design. In opposing to the filling port, a tumble port introduces the required tumble flow structure to generate maximum turbulence. The narrow port angle, straight and guiding surface, and the square shape profile, is purposely designed to direct the fluid mass flow. Port layout 'c' represent the swirl movement, this design is not studied critically in this discussion, because it is mainly used on the compression ignition diesel engines, whereas this paper investigates specifically the gasoline direct injection system.

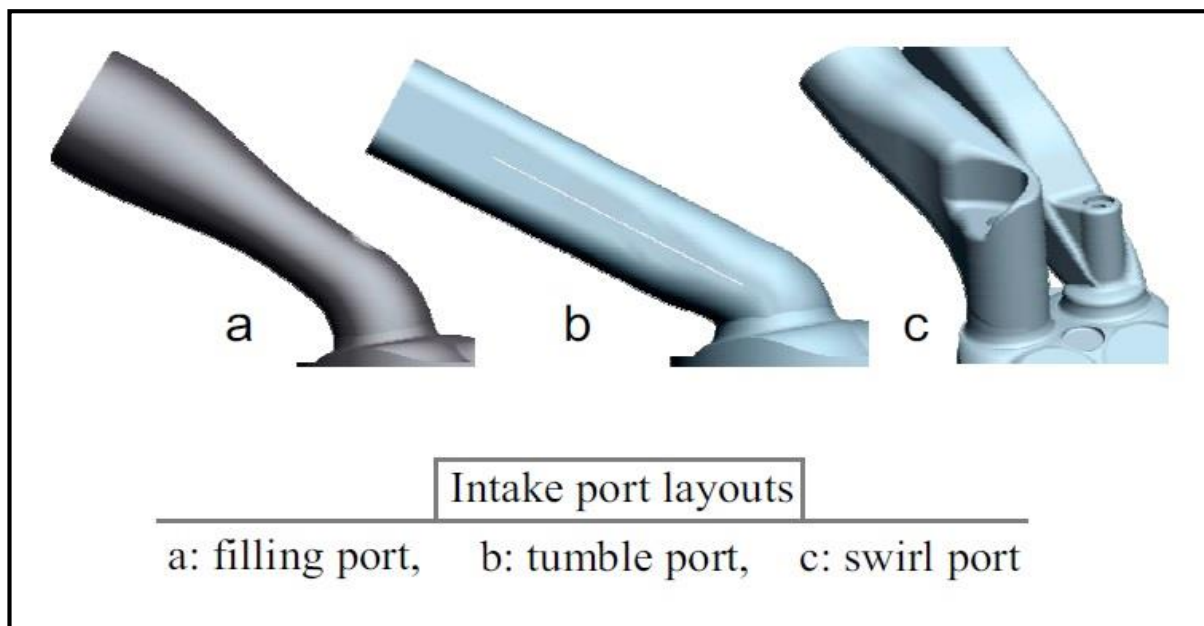


Figure 3 Intake Port Layouts and Different flow motions (Schogl *et al.* 2011)

In SI engines, turbulence generation is a key parameter as it directly affects the burn rates efficiency, heat transfer, and combustion stability as evident by Kim, *et al.* (2019). The authors continue addressing the criticality of this parameter, by confirming that turbulence is a determining factor on flame propagation in GDI engines, therefore, it alters the overall engine performance. Primarily, a higher turbulence level enables turbulent flame speed to increase, which leads to an improved thermal efficiency and mitigates knocking phenomena. Moreover, it has been discussed in multiple research papers regarding GDI technology that, increasing turbulence intensity can highly benefits the EGR tolerance (Berntsson *et al.*, 2011; Ogink and Babajimopoulos, 2016).

Numerous studies have already investigated intake port flow characteristics, by both experimental bench testing and CFD simulations. In 2000, The Mitsubishi GDI engine adopted a pair of upright intake ports to activate rotating in-cylinder flow, reverse

tumble, and control air/fuel mixture. The optimization of intake port design for the Mitsubishi GDI engine was the key to achieve a high intensity of the reverse tumble while maintaining the charge coefficient high. Also, the refinement of port design enhances the flow in the intake sides of the intake valves, which maximized the reverse tumble motion inside the cylinder (Kuwahara *et al.* 2000). In 2009, Ford Motor Company utilised 3D CFD techniques to investigate the upfront optimisation of the in-cylinder flow of the 3.5 litre V6 EcoBoost engine says Iyer and Yi (2009).

Sadakane *et al.* 2005 demonstrated that on the develop of a V-6 stoichiometric GDI engine, Toyota highlighted the optimization of the intake port as one of the special technologies that contributed to the achievement of that successful high-performance engine. The authors confirmed the design of an intake-port that obtains both of a high flow coefficient and high tumble ratio, was a key aspect to improve power performance, where dual VVT-i (variable intake and exhaust valve timing) systems were employed to decrease pumping losses and increase the expansion ratio, in order to improve fuel consumption.

Berntsson *et al.* (2011) conducted further research in attempt to enhance fuel efficiency and meet the emission reduction requirements, the study confirmed that GDI system combined with turbocharging and downsizing is a cost-effective technology to meet the above objectives. In order to investigate the turbulence induced by the in-cylinder charge motion, the generated intake flow-field was varied through numerous port concepts, aiming to evaluate if further increase in tumble will potentially lead to higher efficiency when compared with baseline model. The study combined two series of testing, one aiming to evaluate tumble in low to medium load and engine speed by mainly focusing on efficiency, and the other was performed at high load region to examine any negative consequences of increased tumble. Similar research was completed by Qi *et al.* (2010).

In a recent study completed by Saw, Addepalli and Mallikarjuna (2019), four-stroke GDI engine was analysed by mean of CFD simulation, in order to generalise the effect of intake port configuration and inlet valve angle, and related effect on the in-cylinder charge motion, mixture formation, and combustion parameters. Kim *et al.* (2018), utilised CFD tools to predict the influence of intake port geometry on the tumble

generation and turbulence characteristics, by implementing Zero-Dimensional internal combustion engine model. Sutar *et al.* (2017), investigated tumble effect on SI-DI engine performance, utilising ANSYS-ICE as a simulation software.

Chapter 3: Methodology

High tumble intake port design was found to stimulate the incoming intake charge and consequently, develop high tumble motion within the combustion chamber (Singh *et al.* 2019). The authors further demonstrated throughout their recent study that the enhancement of turbulence intensity with intake port design, generated an increase in the Turbulence Kinetic Energy (TKE) during the compression phase near the spark plug. Thus, burning rate of charge was significantly increased due to TKE enhancement, which resulted in an improved EGR tolerance, reduced cyclic variability, and increased thermal efficiency by shifting to higher compression ratio operation mode.

Bialy *et al.* (2014) investigated the intake-port design of a diesel engine with compressed natural gas direct injection, utilising simulation tools and adopting a data collection and analysis, which can be explained as followed: **A.** three-dimensional scanning of the real object using ZScanner 700; **B.** development of geometrical model in CAD using CATIA V5; **C.** intake-flow computational analysis in the AVL_Fire software. Similar methodology was adopted by Masi, Artico, and Gobbato (2015), in the study of measurement of the intake and in-cylinder flow field to investigate the reliability of CFD steady-state simulations for actual engines. In this research however, the authors performed the CFD analysis by using release 9 of CD-Adapco Star CCM+ finite volume code. Furthermore, DeLima *et al.* (2017) completed a project on combustion analysis in IC engine adapted with pre-chamber, utilising three-dimensional model obtained from the same Reverse Engineering (RE) method discussed in the previous research papers. The numerical simulations were performed via the commercial software Converge-CFD.

The flow investigation of the GDI intake-port in this study, adopted a similar methodology which will be discussed in detail in this chapter. The design and modelling were completed utilising Solidworks-2019 version, and ANSYS-IC Engine software was used for the CFD simulation. Moreover, this research incorporated

additional software to aid on the design, RE process, and validation (i.e. Geomagic-studio-12, Gom-inspect).

In order to ensure a well organised procedure, and successful project strategy, the engineering work was approached using a structured development planning shown in figure 4.

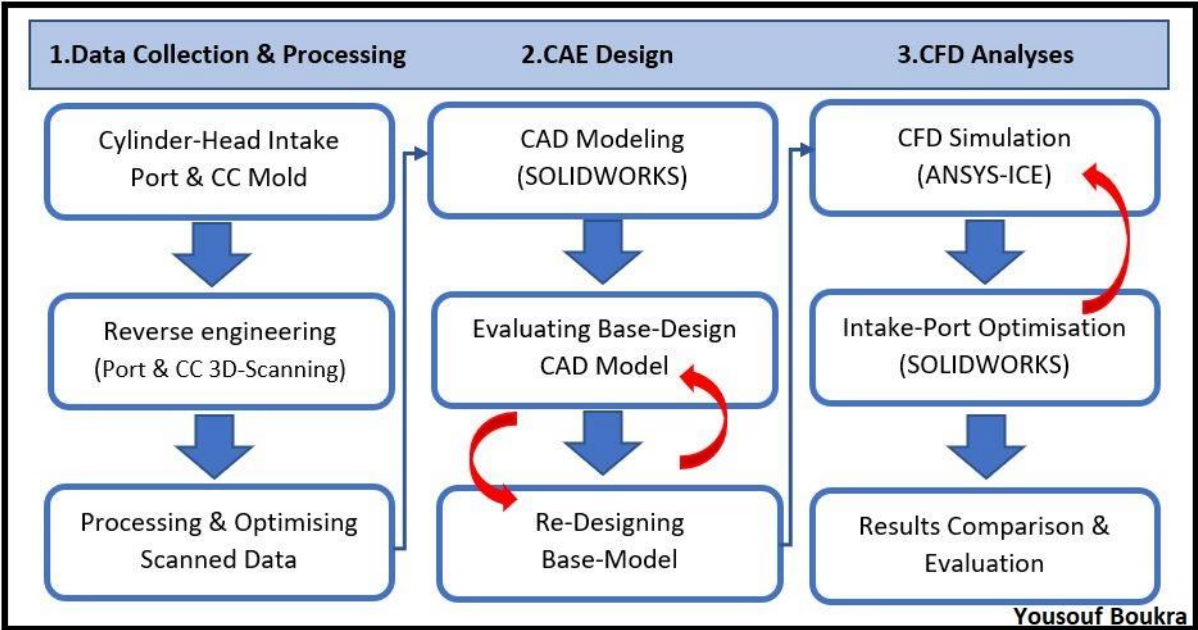


Figure 4 Project development plan (Yousouf, 2020)

3.1. Data Collection and Processing

Obtaining a geometrical solid shape of both the intake-port and the combustion chamber (CC), was the first step in the *Data Collection & Processing* phase toward implementing the reverse engineering techniques, in order to produce a computational model of the base-design. At this first step of the primary phase, a silicon model was created to represents the negative domain (air-path) of both the intake-port and the combustion chamber (see Appendix 1 & 2).

Post obtaining the silicon models, NEXTENGINE 3D-Scanner was utilised to transform the intake port and CC shapes into digital entities (Appendix 3).

Throughout the 3D scanning process, a cloud points digital data was obtained and used as the base reference for modelling of the solid CAD geometry. Although the used scanner is limited in its capacity to capture some of the critical points of the silicon parts, the data was adequate after passing on multiple design optimisation to rectify the missing points from the files. To accomplish this, the cloud points files were

imported into Geomagic-studio-12 software, which was chosen due to the advanced and highly accurate rectifying tools embedded on its platform.

After generating primary surface files from the cloud points, the missing data was exposed and multiple optimisation, and repair tools (e.g., hole filling, bridging, and curvature extension) were implemented.

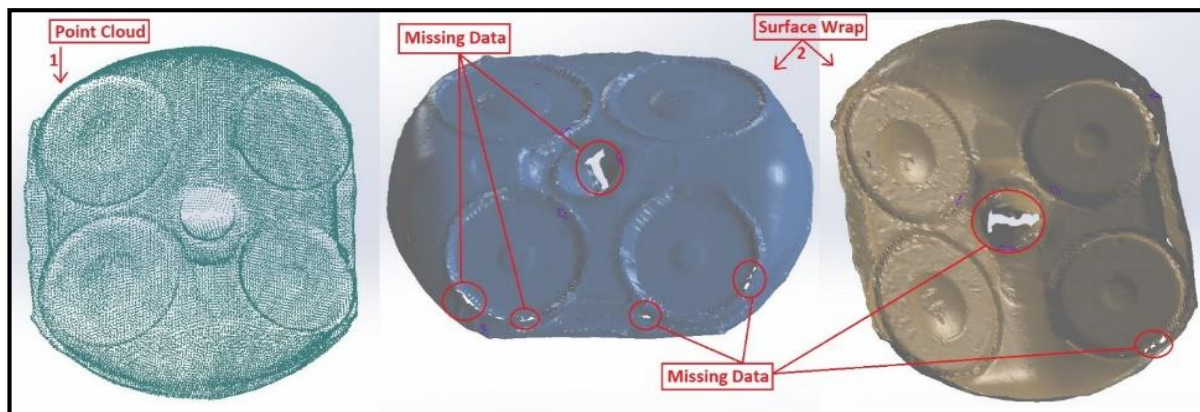


Figure 6 Highlight of missing data from scanned files (Yousouf, 2020)

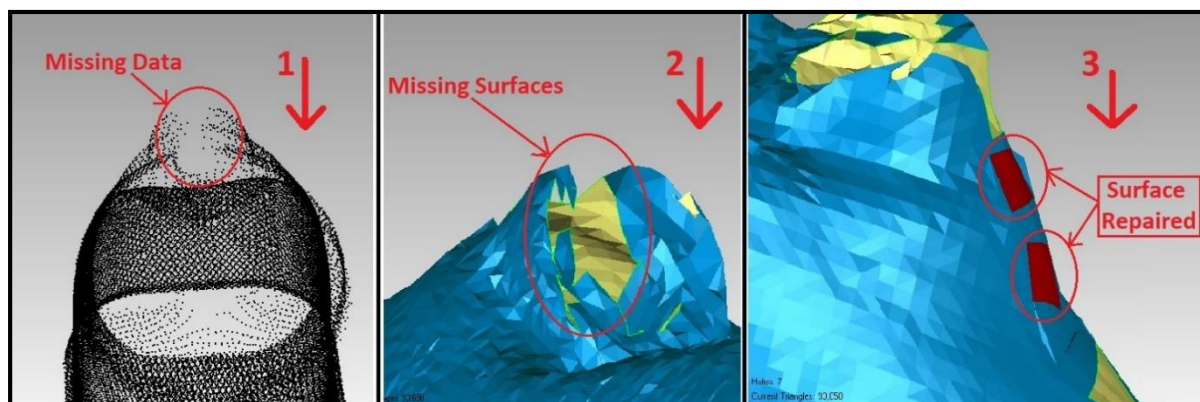


Figure 5 Surface File Repair (Yousouf, 2020)

This enabled the models to be fixed and generated clean surface files, as presented in figure 7 and 8. Due to the complexity of the procedure followed, it is not feasible to include the complete method, but figure 5 and 6, illustrate some of the key techniques which were implemented to repair the surface files.

Kumar *et al.* (2013) indicates that the methodology of using reverse engineering (RE) techniques, to digitalise real life data for further CFD/CAE investigation and development, has been widely recognised as an important technique in the product design cycle. RE enable the creation of surface models by three-dimensional scanning techniques, which will make it possible to redesign and manufacture variety of parts and tools in a short development period says the authors. Thus, the application of

reverse engineering significantly decreases the manufacturing cost, and considered to be time efficient for product realisation process.

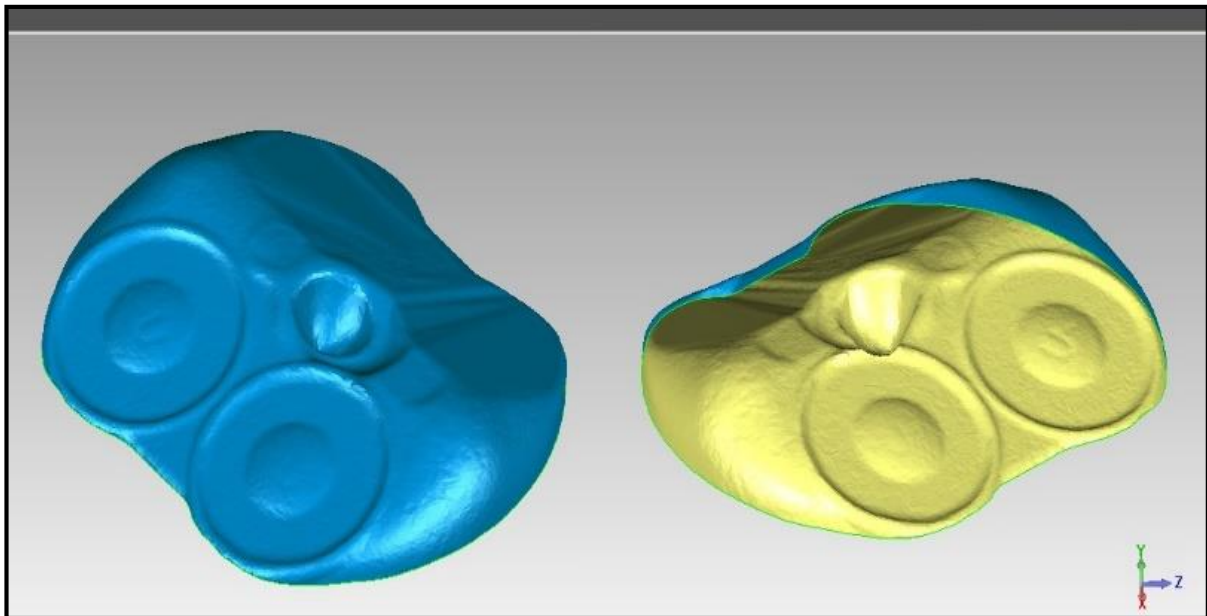


Figure 7 Combustion Chamber Clean Surface File (Yousouf, 2020)

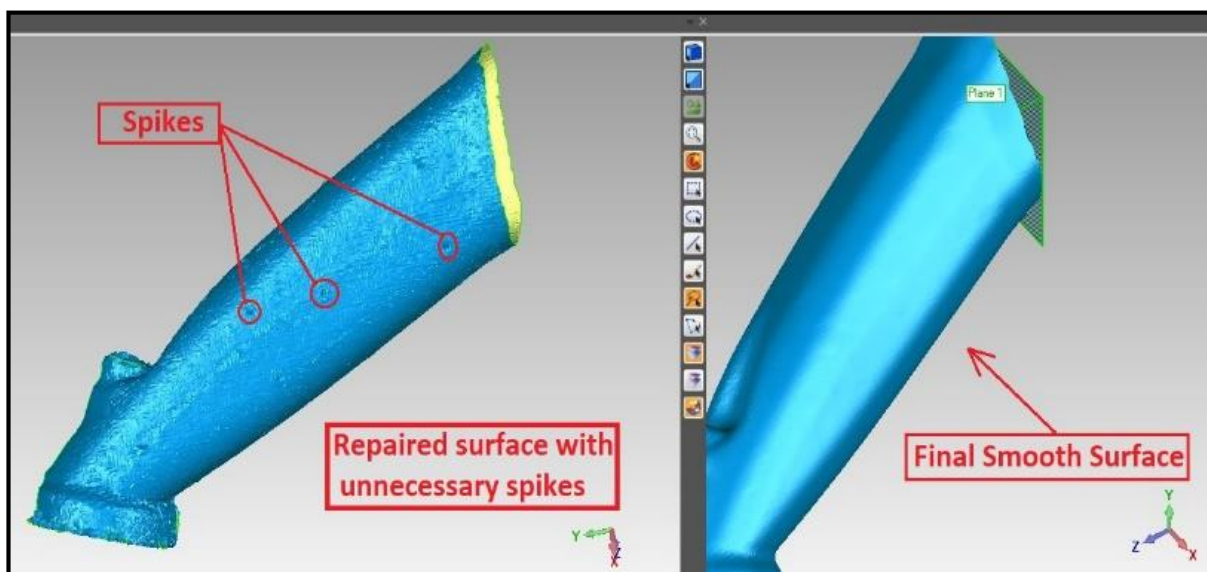


Figure 8 Intake port Clean Surface File (Yousouf, 2020)

3.2. CAE Design and Validation

Second phase in the project methodology plan, involved the CAD modelling and product validation process. At this stage, Solidworks was utilised with the aid of Geomagic-for-Solidworks Add-in. The integration of Geomagic within Solidworks platform made it easier to generate solid CAD geometry, and it provided high design accuracy, due to the surface utilised being generated from Geomagic software as explained earlier.

To design a solid shape based on the surface files, two different methods were performed. Regarding intake-port design, solid-modelling method was adopted, whereas combustion chamber CAD model was obtained through surface-modelling.

3.2.1. Solid-Modelling Method:

The process of creating a solid model based on plan sections as a reference, can be explained in the steps below:

- a) Creating a cross sectional plan in the intake-port surface file.
- b) Manually sketching the lines using projection view as a reference.
- c) Repeating step 'b' for each plan.
- d) Lofting the cumulated sketches to create a solid CAD geometry.

Although this method was successful in creating the solid model (see Appendix 4), it is very limited in terms of accuracy and cannot be reliable with small size geometries, where critical curvature and shapes are determining factors and must be effectively modelled. Therefore, Geomagic-for-Solidworks was found to be very accurate in terms of geometrical quality, and time efficient compared to the conventional method.

Utilising Geomagic within Solidworks platform (see Appendix 5), provides many advantages which can be listed as followed:

- Apart from creating numerous plans in a cross-sectional configuration, similar to the conventional method (step 'a'), it was not necessary to manually perform the sketching steps, as the feature automatically creates a sketch between the intersection of the plan and the surface shape. Thus, step 'b' and 'c' which are time consuming especially as the number of plans increases, was eliminated.

- By passing the interaction of manual sketching, and relying upon the program to automatically project the intersection sketch based on the numerical data, the drawing was highly accurate with zero deviation. Therefore, numerous plans were created to capture all the critical curvature from the surface model, which could not have been sketched in the conventional method.

3.2.2. Surface-Modelling Method:

In order to create a solid body from the combustion chamber (CC) surface file, the surface-Modelling technique was implemented as followed:

- a) All the individual CC surfaces were merged into one single surface.
- b) Additional surface was extruded at the bottom to create a closed geometry file.
- c) The bottom surface was knitted to the CC surface to produce a solid body.

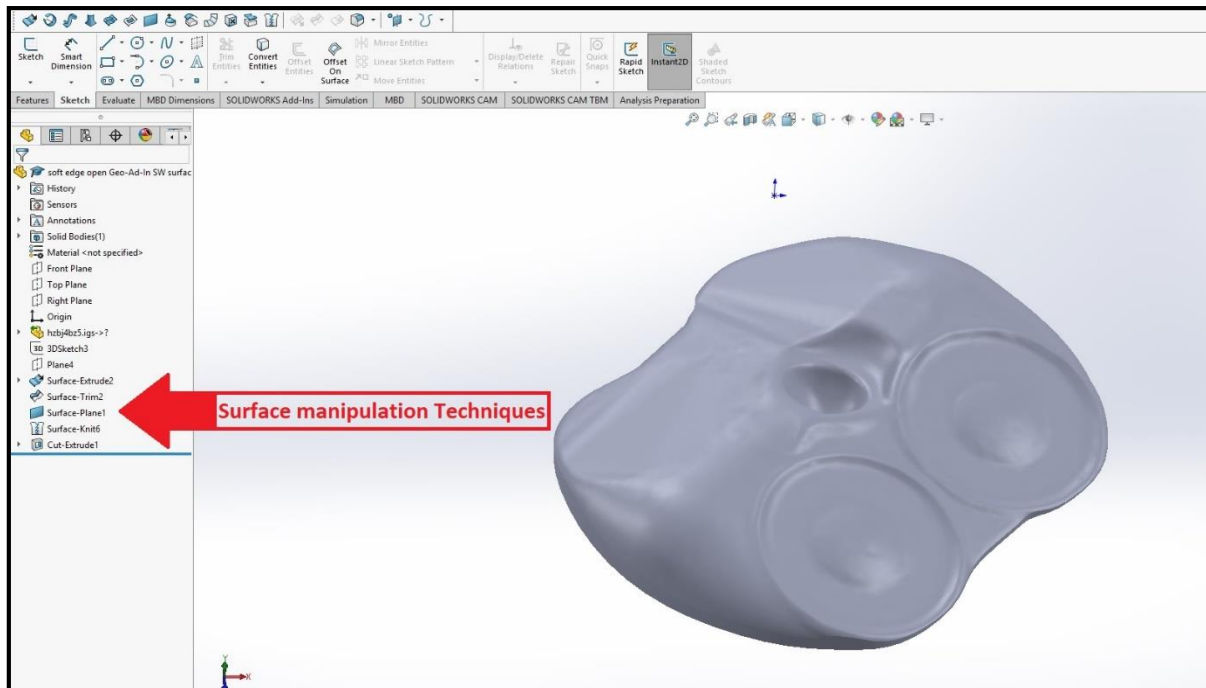


Figure 9 Combustion Chamber CAD model (Yousouf, 2020)

3.2.3. CAD models Validation:

The acquired CAD models (figure 10) were then evaluated against the original scan files using GOM-Inspect software, by overlaying the scanned data with the CAD models. the software automatically highlight the deviation points and calculate the accuracy of the solid geometry.

As evident in figure 11, it is clear the higher accuracy of the achieved design as the port for example here, perfectly overlays with the scanned data. This inspection was conducted by Andrew Tibbott and David Cooper, in addition to the combustion chamber evaluation in the university workshop.

Furthermore, an experimental validation was carried through volume measurement.

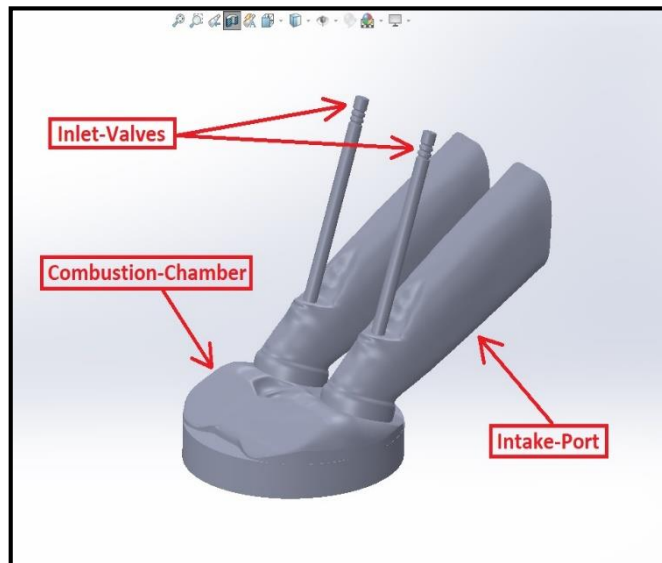


Figure 10 Cylinder-Head Air-Domain CAD model (Yousouf, 2020)

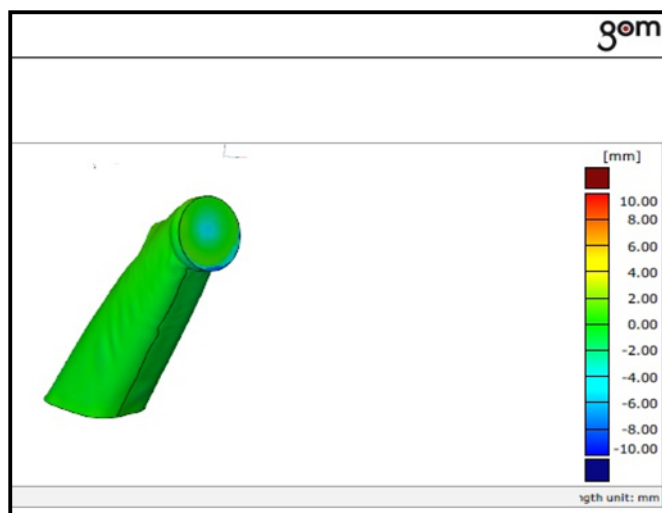


Figure 11 Port GOM inspection (Tibbott and Cooper, 2020)

Both intake-port and combustion chamber volume were measured, following the method described by Hunter (2017) and DeLima *et al.* (2017), where a controlled amount of liquid was injected in the CC and intake port to obtain the internal size (see Appendix 6 & 7). The recorded measurements were eventually compared to the CAD volumes in Solidworks, and the results indicated 99% accuracy in terms of port design, and 93% for the combustion chamber.

In relation to the CC, a 93% model validation is considered highly accurate for the current investigation. The influence of combustion chamber on turbulence flow is fairly low, which has been confirmed by the comparison conducted by Kim *et al.* (2019).

The study investigated the difference between an actual pentroof geometry CC and a simplified pentroof CC (See Appendix A), throughout the assessment of inertia which is the quality that determines the rotational characteristic in the 3-D space. As mentioned by the authors, for a normal pancake-shaped CC geometry, the moment of inertia 'I' is identified as followed:

$$I_z = \iiint_{vol} \rho(x^2 + y^2) dV$$

$$= m \left(\frac{B^2}{16} + \frac{H^2}{12} \right)$$

Where B and H represents cylinder bore and height, respectively. After The simplified pentroof geometry is applied, the volume integral obtained is:

$$I_{z, pent} = 8 \int_0^R \int_0^{\frac{H}{2}} \int_0^{\sqrt{R^2 - x^2}} \rho(x^2 + y^2) dz dy dx - 4 \int_0^R \int_{\frac{H}{2} + ax}^{\frac{H}{2}} \int_0^{\sqrt{R^2 - x^2}} \rho(x^2 + y^2) dz dy dx$$

$$= \rho \left[\frac{(64a^3 + 192a)R^5 + (45\pi a^2)HR^4 + 120aH^2R^3 + 30\pi H^3R^2}{360} \right]$$

Noted that the volume between the two CC geometries is no more equal at this stage. After the theoretical calculation completed by Kim and colleague (2019). The moment of inertia for each geometric assumption was plotted in figure 12, along with the results obtained from the CFD simulation of both CC geometries.

It is evident that, even with a significantly simplified CC model, the Inertia is not substantially influenced. Moreover, the current investigation is conducted specifically at the intake-stroke, where the CC geometry has no influence on the Inertia as observed in the graphs here between -360° and -180°, both theoretical calculation and CFD results show no deviation on Inertia at this specific crank position.

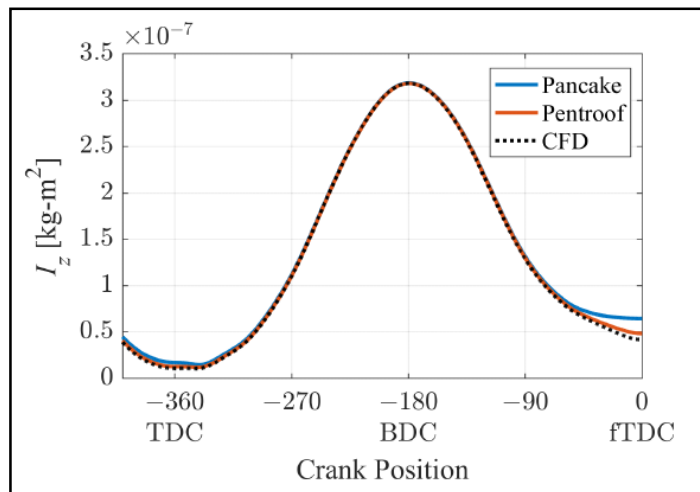


Figure 12 Actual Vs simplified combustion chamber (Kim et al. 2019)

3.3. CFD Simulation and Flow Validation

As Slater (2008) confirmed, there is a professional argument on exact methods for verification and validation of CFD simulations. Although CFD is maturing, it still an emerging technology, it is a complex innovation comprising strongly coupled non-linear partial differential equations. Those equations are used to computationally model theoretical and experimental models, in a distinct domain which involves of complex geometry shape. Furthermore, a detailed assessment of errors and unpredictability must be concerned with the three roots of CFD (theory, experiment, and computation). However, the development of CFD code has received a large motivation since last three decades, due to the availability of computing power and optimisation of CFD codes for engine flow simulation. Thus, the implementation of CFD is rapidly expanding with the available resources (Slater, 2008; Vaidya, 2010).

In line with what have been cited above, every engine optimisation related to the internal flow characterisation, must incorporate some sort of experimental and/or theoretical validation. Masi, Artico, and Gobbato (2015) performed an experimental characterisation of the local flow field parameters in an A-type industrial fan test rig, by mean of pressure tappings drilled along the surface of cylinder-head. Where stainless-steel capillaries, were inserted for pressure measurement as shown in Appendix-8. This method enabled both the mass flow, and the tumble coefficient to be measured and compared to the CFD results for verification. Likewise, Zotti, Masi, and Antonello (2012) utilised the same test rig and instrumentation set-up.

Sutar *et al.* (2017) utilised AVL's method based on vane anemometer to experimentally measure the tumble flow and validate the CFD results (see Appendix 9). Whereas other researchers used the application of Particle Image Velocimetry (PIV), to measure the internal flow parameter. PIV system consists of double pulse laser of 200 mJ/pulse capacity, charge coupled device (CCD) camera, and an advanced flow-bench rig (Bevan and Ghandhi, 2004; El-Adawy and Munir, 2017; Krishna *et al.*, 2013).

The verification of CFD simulation results require complex experimental instruments and hardware. This posed a challenge for the completion of the current study, as unfortunately, the available tools were inadequate. However, due to the extensive research confirming the validity of CFD tools, and the experimental flow-bench

completed in the automotive/motorsport laboratory, the CFD results were eventually been able to be successfully verified.

3.3.1. Flow-bench Experiment:

The test rig used for the experiment was the SF-110 simple flow-bench, where the only flow parameter that can be measured was the mass flow rate based on the pressure drop. The experiment set-up of the cylinder-head flowing procedure can be seen in Appendix-10 and engine specifications are listed in figure 13. In addition, camshaft profile was measured to find the Maximum valve lift opening, which was measured to be 8mm.

GDI Engine	Ford EcoBoost 2013 Model
Configuration	4 Cylinder
Displacement	1.6 : 1596 cc
Cylinder Bore	79.0 mm
Piston Stroke	81.4 mm

Figure 13 Engine Specifications (Yousouf, 2020)

In order to validate the CFD results, the model used to solve the simulation needed to be verified against the experimental data. Mass flow rate was the key parameter for comparison, however, the CFD set-up had to be adapted to the experimental set-up first, as the flow-bench utilised is unable to run at higher pressure.

That was achieved by setting boundary conditions matching that of the experiment, where outlet pressure was calculated to be 3726.4 pascal, and operation temperature 300 Kelvin. Figure 14 shows the equations used to calculate the mentioned parameters based on flow-bench values. Experimental results and validation are discussed further down in chapter four.

Equations Used	
Ideal Gas Law: $P \cdot V = n \cdot R_u \cdot T \Rightarrow [P / (R_u \cdot T)] = [n / v]$	
Air Density (ρ) = Pressure / ($R_u \cdot T$)	
Mass Flow = Volumetric Flow * Air Density	
Known value & calculated results	
Density of water	997
Pressure drop Height (inch)	15
inch to mm	25.4
mm to meter	0.001
Gravity	9.81
R_u	287
Temperature in Kelvin	300
Outlet Pressure (Pascal)	3726.4
CFM to M ³ /s conversion rate	0.0005

Figure 14 Table of Equations and Values (Yousouf, 2020)

3.3.2. CFD Set-up Procedures:

Ansys-ICE is the software used to perform the CFD simulation. This system was introduced to Ansys workbench in 2012, and provided enormous benefits and solutions which were unachievable with the use of fluent platform manually. The new IC Engine analysis system combine the best of both CFD solvers and chemistry tools, this includes Ansys-Forte, Ansys-Fluent, Ansys-CFX and Ansys-ChemKin-Pro all in one platform. Thus, the system delivers the most comprehensive solutions available for IC engine flow and combustion simulation (Ansys Inc. 2012).

Some of the system's advantages can be summarised as followed:

- ❖ Fast, accurate, transient, and steady-state flow solvers.
- ❖ Reduce the setup time from many hours to under an hour for a complete case, due to engine-specific Ansys-Forte interface and streamlined Ansys-ICE.
- ❖ Fully automatic and dynamic solution adaptive meshing, that refines and coarsens based on solution variables for both high accuracy and speed.
- ❖ Seamless combination of fluid and thermal solvers, that enables thermal structural durability.
- ❖ Advanced solver physics setups (e.g., spray injection, combustion simulation, energy model, turbulence model, etc.).
- ❖ Automatic geometry preparation: move and deactivates valves, manipulation of piston-bowl and cylinder length, creation of different inlet/outlet plenum.
- ❖ Automatic generation of tumble/swirl planes at the given position.

(Ansys Inc. 2012; 2020).

Ansys-ICE incorporates four different internal combustion engine simulators, Cold Flow, Port Flow, DI-CI Combustion, and GDI Combustion Simulation. Each simulator varies in term of the parameters or problems which can be solved, that depends on the input data and the allowable solving time per case. For-example, Cold Flow simulators solving time is around forty hours per setup case (Ansys Inc. 2015). Cold Flow simulator solve the air flow throughout the complete intake and compression stroke, with variables such as connecting-rod length, piston bowl geometry and intake port parameters.

However, Port Flow simulator solves the air flow with relation to intake port parameters only, such as valve displacement, port angle and port design characteristics. The solver runs the calculations under steady-state condition for multiple valve lifts, thus, it takes significantly less time (5 to 8 hours per setup case) compared to Cold Flow simulator which takes over 40 hours. Port Flow simulator was selected for this application, as only the intake port design parameters are investigated, where the effect of intake port design on in-cylinder turbulence can be adequately analysed. Limiting other variables such as piston-bowl to interfere.

The procedures followed to setup a case within Ansys-ICE are demonstrated below:

- ❖ Step 1: IC Engine analysis system was created in the workbench, and the properties were selected, where Port Flow simulation type was chosen.
- ❖ Step 2: includes the import of external geometry developed in CAD earlier, within the DesignModeler.
- ❖ Step 3: involved the data input process within the Input-Manager tab. And the selection of post processing planes.
- ❖ Step 4: After all the inputs were established (e.g., Valve, intake ports, cylinder-length, etc.), geometry decomposition was performed.
- ❖ Step 5: at this stage, a parameter set was inserted to allocates 4 different valve lifts to be solved (i.e. 2mm, 4mm, 6mm, and 8mm).
- ❖ Step 6: after checking the geometry for verification, an adaptive Hybrid-mesh was generated.
- ❖ Step 7: at this step, the solver settings were selected, where boundary conditions, Monitoring, and post processing settings were initiated.
- ❖ Step 8: Fluent launcher was opened, and all the necessary physics variables were interpolated (e.g., turbulence model, fluid properties, etc.).
- ❖ Step 9: after all the settings were established, solution was performed automatically for all the valve lift design-parameters (this process last between 5 to 8 hours, depends on the turbulence model selected).
- ❖ Step 10: this is the final step, where the solution was obtained, and result investigation started within CFD-post.

3.3.3. Turbulence Modelling:

Fluid flow around building, airplanes, cars, and inside engines, are considered turbulent. Also the flow and combustion inside an IC engine and turbines are highly turbulent, and can be described by numbers of characteristics features including: diffusivity, irregularity, high Reynolds number, three dimensionality and dissipation (Davidson, 2018; Sjerić, Kozarac and Bogensperger, 2012).

A thorough understanding of the in-cylinder turbulent flow is important for a successful modelling of IC engine combustion. When the flow is turbulent, it is better to decompose the instantaneous variables (such as velocity and pressure, eq.1) into mean value and fluctuating value, according to Davidson (2018). Those are:

$$\begin{aligned} U_i &= \bar{U}_i + u_i & \dots\dots\dots (eq.1) \\ P &= \bar{P} + p. \end{aligned}$$

The Continuity equation and the Navier-Stokes equation are:

$$\frac{\partial \rho}{\partial t} + (\rho U_i)_{,i} = 0 \quad \dots\dots\dots (eq.2)$$

$$\frac{\partial \rho U_i}{\partial t} + (\rho U_i U_j)_{,j} = -P_{,i} + \left[\mu \left(U_{i,j} + U_{j,i} - \frac{2}{3} \delta_{ij} U_{k,k} \right) \right]_{,j} \quad \dots\dots\dots (eq.3)$$

After derivation and elimination of the dilatation term in the right-hand side of eq.3, equation below is obtained:

$$\frac{\partial \rho U_i}{\partial t} + (\rho U_i U_j)_{,j} = -P_{,i} + [\mu (U_{i,j} + U_{j,i})]_{,j}. \quad \dots\dots\dots (eq.4)$$

After inserting eq.1 into the continuity equation eq.2 and the Navier-Stokes eq.4, and after time averaging, the time averaged continuity equation and Navier-Stokes equation are obtained as followed:

$$\frac{\partial \rho}{\partial t} + (\rho \bar{U}_i)_{,i} = 0 \quad \dots\dots\dots (eq.5)$$

$$\frac{\partial \rho \bar{U}_i}{\partial t} + (\rho \bar{U}_i \bar{U}_j)_{,j} = -\bar{P}_{,i} + [\mu (\bar{U}_{i,j} + \bar{U}_{j,i}) - \rho \overline{u_i u_j}]_{,j}. \quad \dots\dots\dots (eq.6)$$

(Davidson, 2018).

As expressed by Versteeg and Malalasekera (2007), The $\overline{u_i u_j}$ (Reynold stress tensor) appeared in the right hand-side of eq.6, represents correlations between fluctuation velocities (additional unknown stress term due to turbulence). Thus, a model is needed for $\overline{u_i u_j}$ to close eq.6 system, which are cited by Davidson as followed: Algebraic models which is also known by 0-D models, One-equation models, Two-equations models, and Reynolds stress models.

Only the two-equation turbulence models will be discussed in this investigation, as it is widely used in CFD application today, and specifically to solve ICE internal flow problem as confirmed by many researchers (Barbouchi and Bessrour, 2009; Han and Reitz, 2007).

Two-equation models are the most common type of turbulence models used for engineering problems, and they are classified within the eddy-viscosity models class, according to Davidson (2018). Two transport equation are derived which represents transport of two scalars, in this case those are referred to as K - ϵ . Differential equations of the turbulent kinetic energy K and its dissipation rate ϵ , are derived from the RANS (Reynold Averaged Navier-Stokes) equations. By using as assumption which links the Reynolds stress tensor to the velocity gradients and an eddy viscosity, the Reynold stress tensor is then computed (Sjeric, Kozarac and Ormuz, 2016).

With $P_\epsilon = c_{\epsilon 1}(\epsilon/k)P_k$ transport equation for the dissipation is:

$$(\rho \bar{U}_j \epsilon)_{,j} = \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \epsilon_{,j} \right]_{,j} + \frac{\epsilon}{k} (c_{\epsilon 1} P_k - c_{\epsilon 2} \rho \epsilon) \quad \dots\dots\dots(\text{eq.7})$$

And for boundary-layer flow, the above equation (eq.7) will be:

$$\frac{\partial \rho \bar{U} \epsilon}{\partial x} + \frac{\partial \rho \bar{V} \epsilon}{\partial y} = \frac{\partial}{\partial y} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial y} \right] + c_{\epsilon 1} \frac{\epsilon}{k} \mu_t \left(\frac{\partial \bar{U}}{\partial y} \right)^2 - \rho c_{\epsilon 2} \frac{\epsilon^2}{k} \quad \dots\dots\dots(\text{eq.8})$$

$$\text{Diss.Term} = -c_{\epsilon 2} \rho \frac{\epsilon^2}{k}.$$

(Davidson, 2018).

3.3.4. Theoretical Verification:

The theoretical verification involved the calculation of air velocity, by utilising the pressure data obtained from the experimental test, and Bernoulli's equation:

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g z_1 = P_2 + \frac{1}{2}\rho v_2^2 + \rho g z_2$$

- P is the fluid pressure
- ρ is the fluid density
- v is the fluid velocity
- g is the acceleration due to gravity
- z is the elevation of the fluid above a fixed reference point

(Arakeri, 2000).

$$V^2 = (2P \div \rho) \rightarrow V = \sqrt{(2 \times 3726.4) \div 1.29} \rightarrow V = 75.92 \text{ m/s}$$

The velocity figure calculated above is not an absolute figure, because many parameters must be recognized as variants and may inhibit the final value. Thus, the comparison completed further down (see Chapter 4) between this value and the obtained CFD flow-velocity, cannot be considered a validation. However, the comparison attempts to verify if the velocity of the flow within the CFD simulation, is within a reasonable value.

Chapter 4: Results Discussion

4.1. CFD Model Validation

In order to validate CFD mass-flow results, the experimental flow-rate must be converted to kg/s first, as the results were recorded in cfm unit. With reference to figure 14, the conversion was calculated as followed:

- ❖ Converting CFM values to M³/s (volumetric-flow).
- ❖ Calculating average-pressure based on outlet-pressure and atm-pressure.
- ❖ Calculating air-density: Air-density = average-pressure / (Ru x T).
- ❖ Obtaining Mass-Flow (Kg/s) = volumetric-flow (M³/s) x air-density.

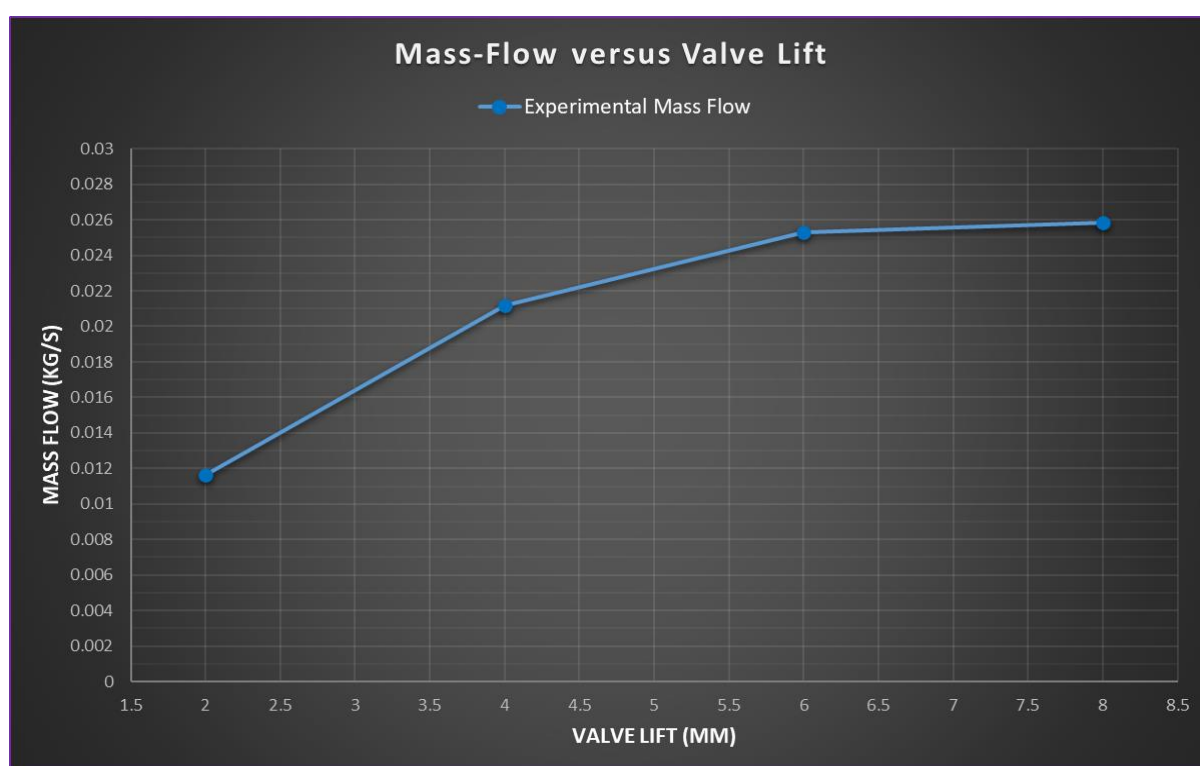


Figure 15 Experimental Mass flow graph (Yousouf, 2020)

The line graph in figure 15, represent the experimental mass-flow in kg/s versus valve lift opening. Measurement were recorded in an increment of 2mm.

During this stage, the CFD simulation was initiated in attempt to discover an accurate turbulence model, that can match the mass-flow values measured in the flow-bench.

Through a thorough investigation into previous researchers attempts, and efforts taken in order to simulate internal flow for an ICE application, the following outcomes were found: Vaidya, (2010) utilised standard k- ϵ turbulence model to simulate in-cylinder

flow, for the study of mixture formation process in GDI engines. Sjerić, Kozarac, and Bogensperger (2012) cited that k - ϵ turbulence model is the most commonly used for combustion modelling. The in-cylinder flow analysis study conducted by Krishna *et al.* (2013) revealed that, TKE and TR is over predicted by standard k - ϵ model compared to RNG k - ϵ model. Thus, the RNG-model was found to be suitable for predicting flow characteristics inside IC engines. Bozza *et al.* (2018) confirmed that RNG version of k - ϵ model with all advanced wall treatment functions, is the most suitable for the application of IC engines. Bozza and colleagues described the RNG version as a considerably accurate model compared to the standard version, due to the RNG dissipation equation being a modelled form of a term arising from the ReNormalization Group, which takes into account the influence of the mean flow distortion on the turbulence formation. Similarly, Karaya *et al.* (2019); Radicchi *et al.* (2017); Saw *et al.* (2019), all implemented RNG k - ϵ turbulence model to investigate the flow parameters of an IC engine.

Based on the literature discussed above, k - ϵ turbulence model was chosen to be implemented in the current investigation. Taking the literature as a reference, the standard version of the model was eliminated, due to the fact that this version of k - ϵ over predicts the TKE and TR (Krishna *et al.* 2013). However, Both RNG k - ϵ and Realisable k - ϵ models were applied, and the results are discussed further down.

Furthermore, wall treatment function was considered in this investigation, as there was a lack of information regarding the influence of this parameter on turbulence prediction. Thus, the three version (Standard, Non-equilibrium, and Enhanced wall functions) were tested individually for both RNG and Realisable models.

With the three wall functions, a total of six different turbulence models were solved to predict the mass-flow, and the turbulent through the base-port design. In order to stabilise the flow parameters, and make sure the results are convergent, each of the six models was simulated for three complete cycles unless otherwise stated.

Prior running the CFD simulation, the boundary conditions were set to match that of the experimental setup, See Appendix-11.

4.1.1. RNG K- ϵ Turbulence Model:

Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experimental Mass Flow	K- ϵ RNG standard (max inflation layers = 12)	
2	0.011657429	0.010259	93.60%
4	0.021206235	0.020586	98.50%
6	0.025315833	0.025956	98.70%
8	0.025854468	0.028077	95.90%
			Overall 96.69 %
Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experimental Mass Flow	K- ϵ RNG non-equilibrium (max inflation layers = 12)	
2	0.011657429	0.01006	92.60%
4	0.021206235	0.02045	98.20%
6	0.025315833	0.02557	99.50%
8	0.025854468	0.02756	96.80%
			Overall 96.78 %
Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experimental Mass Flow	K- ϵ RNG enhanced wall t (max inflation layers = 12)	
2	0.011657429	0.010559	95.05%
4	0.021206235	0.021491	99.30%
6	0.025315833	0.02665	97.40%
8	0.025854468	0.02884	94.50%
			Overall 96.59%

Figure 17 RNG K- ϵ turbulence model results (Yousouf, 2020)

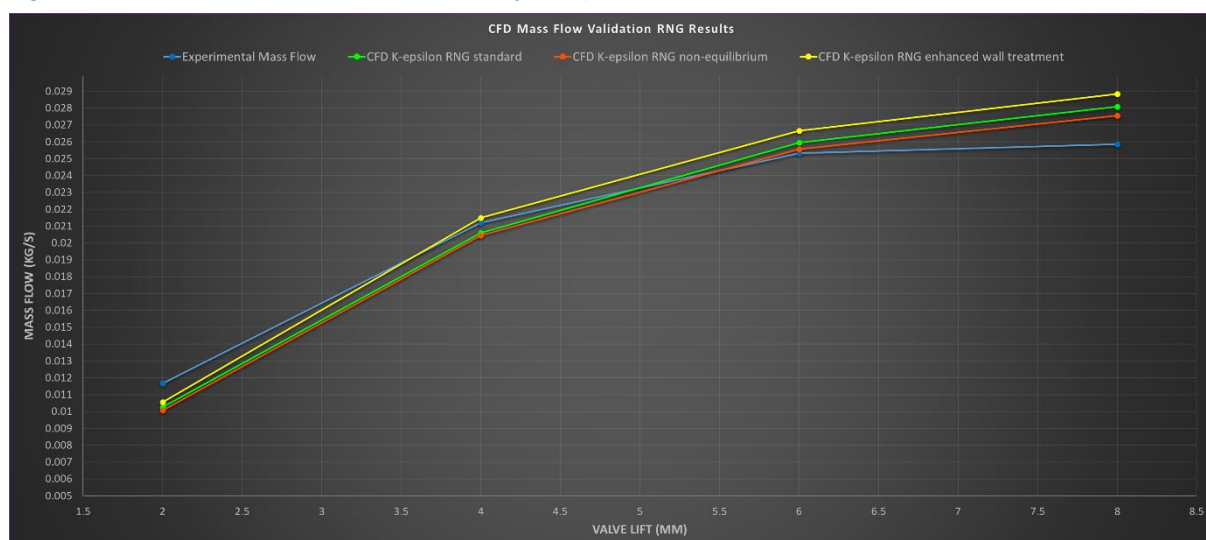


Figure 16 RNG K- ϵ turbulence model mass flow graphs (Yousouf, 2020)

As demonstrated in the figures above, the RNG k- ϵ turbulence models revealed an accuracy of 96%, for all three wall-function variations. Thus, the results are in correlation with the experimental (blue-line) figure of mass-flow, confirming that the CAD model is carefully designed and precisely replicates the physical cylinder-head.

Despite the significantly accurate results of this model, the deviations of CFD mass-flow values, which are less at minimum VL and more at maximum VL compared to flow-bench values, raised a concern in term of the validity of the model and the predictability of mass-flow rate. Specifically, the inclined slopes of the graphs at

maximum VL, indicating the port is yet to stagnate (choke) in term of intake-air mass-flow.

4.1.2. Realisable K- ϵ Turbulence Model:

Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experemental Mass Flow	K-e Realisable standard (max layer = 12)	
2	0.011657429	0.01028	93.70%
4	0.021206235	0.020667	98.70%
6	0.025315833	0.02587	98.90%
8	0.025854468	0.028377	95.30%
			Overall 96.67 %
Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experemental Mass Flow	K-e Realisable non-equilibrium (max layer = 12)	
2	0.011657429	0.0099937	92.30%
4	0.021206235	0.020366	97.97%
6	0.025315833	0.02544	99.75%
8	0.025854468	0.027303	97.30%
			Overall 96.83 %
Valve Lift (mm)	1 Port / Mass Flow (Kg/s)	CFD Mass Flow (Kg/s)	Accuracy Percentage
	Experemental Mass Flow	K-e Realisable enhanced wall t (max layer = 12)	
2	0.011657429	0.0116	99.75%
4	0.021206235	0.02143	99.47%
6	0.025315833	0.025961	98.74%
8	0.025854468	0.027872	96.24%
			Overall 98.55 %

Figure 19 Realisable K-e turbulence model results (Yousouf, 2020)

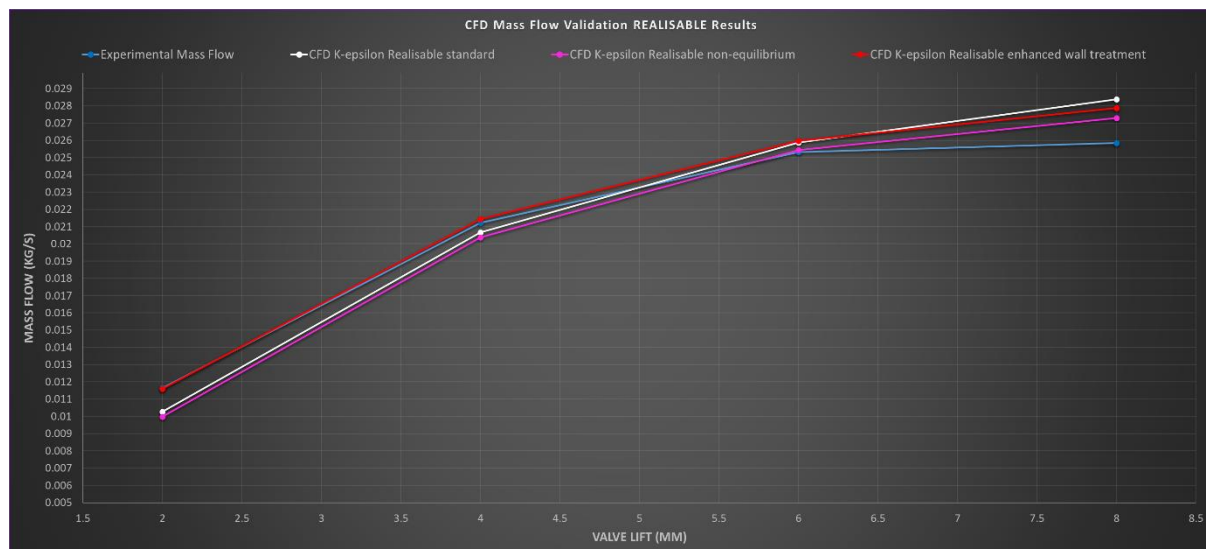


Figure 18 Realisable K-e turbulence model mass flow graphs (Yousouf, 2020)

Realisable K- ϵ turbulence model with standard and non-equilibrium wall functions, revealed similar results to the previously analysed RNG models. The accuracy of the realisable model with the two mentioned wall functions was also 96%, and the slope at minimum and maximum VL remained sharp.

However, Realisable k- ϵ with enhanced-wall function (red-Line) produced an exceeding mass-flow accuracy of 98.55%. As evident in figure 20, there was almost no deviation at 2mm and 4mm VL underscoring over 99.4%. A merging divergence

appeared at 6mm VL of less than 1.5%. The only diversion can be seen at maximum VL, although the slop was less inclined, pinpoint toward a near stagnation mark. That last model was simulated for an additional 2 complete cycles to ensure the stability of the results. This was considered adequate, as by analysing other researchers' work, their graphs revealed a substantial similarity to the obtained results (see Appendix 12 & 13). Validation results completed by Schogl *et al.* (2011), and Selvaraj *et al.* (2011) shows an identical deviation at maximum VL on their presented graphs, compared to the graph below.

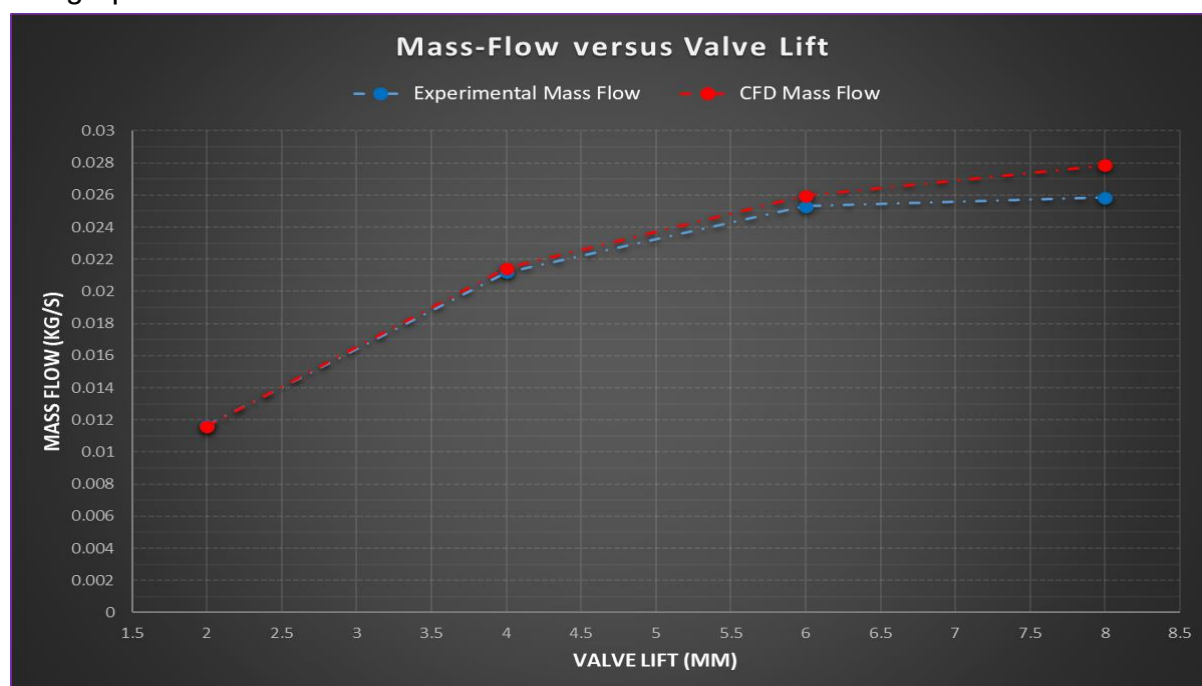


Figure 20 Realisable K- ϵ enhanced wall treatment turbulence model graph (Yousouf, 2020)

The Realisable k- ϵ turbulence model with enhance-wall function, is the most accurate version amongst all the six models solved (see figure 20). That was also confirmed by Sugash *et al.* (2019), during the advanced investigation of various turbulence models, for modelling airflow in a four-stroke single cylinder engine. The study evaluated four turbulence models: Standard-Chen K- ϵ , Standard-Re-no K- ϵ , RNG K- ϵ , and Realisable K- ϵ . Results highlighted that Realisable K- ϵ is significantly enhanced model and can be highly trusted, as the model largely correlates to the experimental data (Sugash *et al.* 2019). Same turbulence model was implemented by Singh *et al.* (2019).

4.1.3. Flow-Velocity Verification:

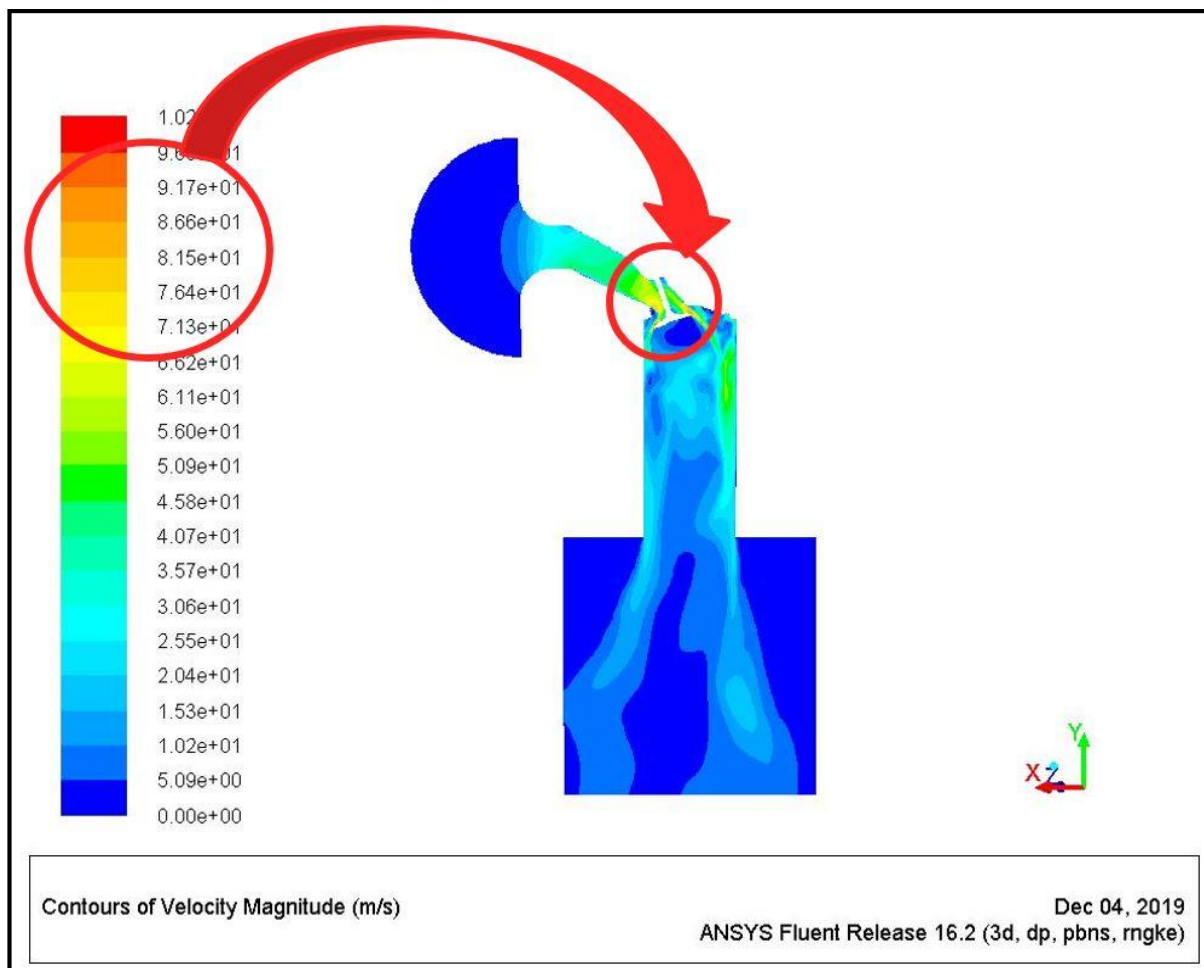


Figure 21 CFD flow velocity verification (Yousouf, 2020)

It is well observed throughout the velocity magnitude colour contours, that the flow simulation of the intake-port range between 65 to 85 m/s (around valve curtains). Considering the theoretically calculated velocity ($V = 75.92$ m/s), the simulated flow-velocity can be confirmed to be well correlating to the theoretical values, which further ensured the validity of both the designed CAD model, and the turbulence model.

4.2. Base-Model Analysis

Post validating the designed model, and selecting the convenient turbulence model to solve in-cylinder flow of the GDI engine. The CFD boundary conditions were adjusted to meet the engine operating conditions, which were obtained from a Ford 1.6 Ecoboost GDI engine pressure data. This data was sourced from Ford-company, where the pressure measurements were based on a dynamometer test. Note that the pressure used to simulate the engine flow, was the specific boost pressure at 2000rpm. And precisely, the pressure occurring during the intake-stroke phase.

The tables below, summarise the meshing results and the setup parameters utilised. Note that engine cylinder temperature parameters were not considered for this simulation, due to the flow parameters being tested only during the intake stroke where there is not a substantial difference on the charge temperature. Singh *et al.* (2019) completed a CFD simulation to study the intake-port geometry influence on flow characteristic, where boundary conditions also excluded the operating temperature and was set at 300 kelvins. Similarly, Kim *et al.* (2018) set the simulation Temperature at 298K (see Appendix 14 & 15).

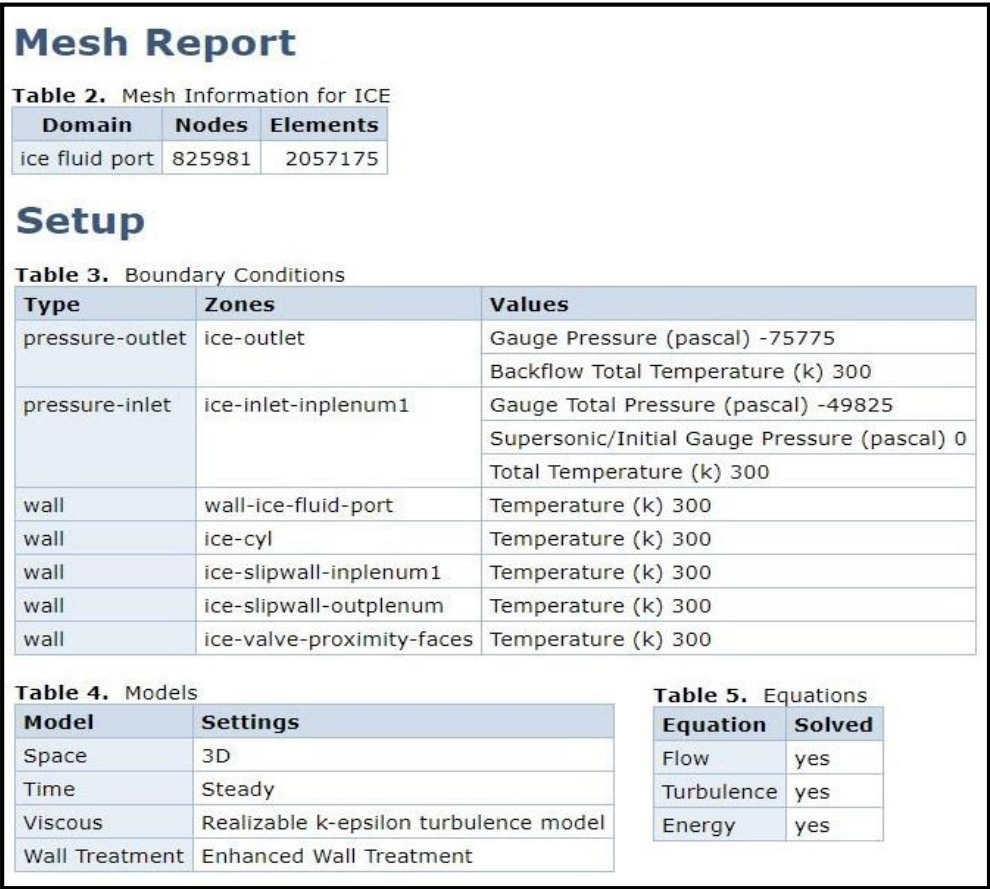


Figure 22 Boundary conditions and setup parameters for boosted engine simulation (Yousouf,2020)

The operating condition were chosen to be selected at 2000 rpm, as with reference to the literature review related to pollution, the main problem which is currently investigated is the emissions within urban area and cities environment. Thus, a low engine rpm is optimal for this study. After performing 5 simulation runs on the base-line design, with 2000 iteration each. The results were exported for post processing, and the outcomes are discussed below:

2mm valve lift:

Figure 23 here, exemplify the flow velocity contours throughout the port, around the valve curtains, and inside the cylinder. By dividing the flow around the intake-valve into two section (front 'F.S' and back 'B.S'), tumble '3' and cross-tumble '2' can be investigated. It is evident at point '1' that there is no sharp flow separation at the back of the port, leading to smooth charge to flow from the back of the valve and create a cross-tumble charge '2' inside the cylinder.

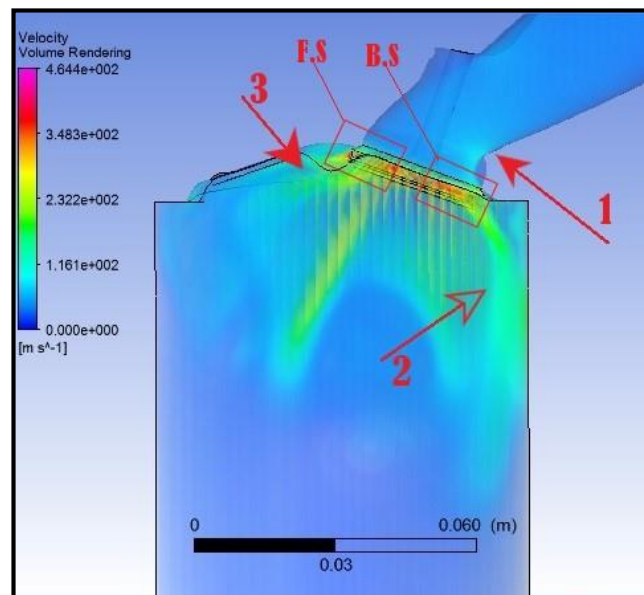


Figure 23 Base-line 2mm CFD post results (Yousouf, 2020)

4mm valve lift:

At 4mm VL the flow velocity started developing at the back of the valve '1', due to the high descending port-angle, resulting to cross-tumble charge '2' to overtake the tumble charge '3'. The high cross-tumble charge reduced the strength of tumble-flow inside the cylinder, even though the flow velocity was higher at the front side of the valve curtain.

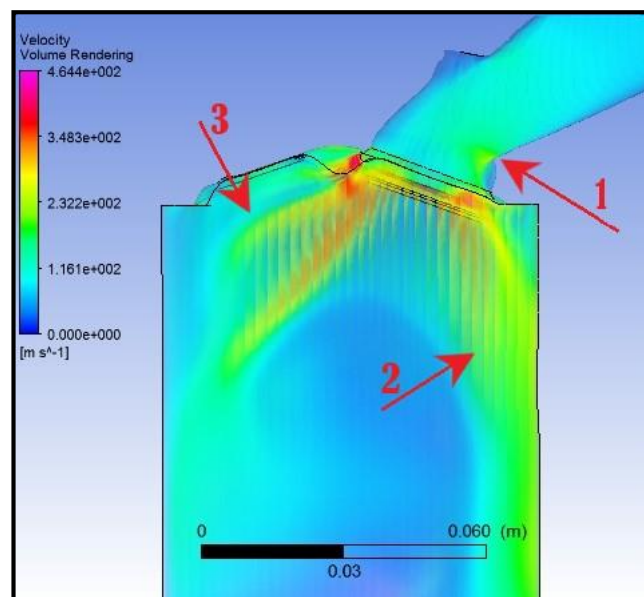


Figure 24 Base-line 4mm CFD post results (Yousouf, 2020)

6mm valve lift:

Despite that the tumble '4' increased at 6mm VL, due to the enhanced flow velocity at front side of the port '2'. The cross-tumble flow '3' equivalently developed, as a consequence of the previously generated back flow at the port neck which prompt a high flow velocity at the back of the valve curtain '1'.

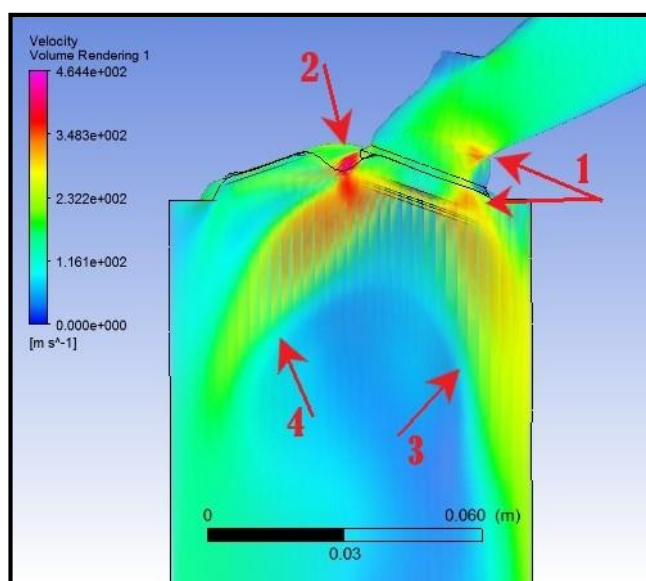


Figure 25 Base-line 6mm CFD post results (Yousouf, 2020)

8mm valve lift:

At maximum VL, it is evident that flow velocity is identical at both back and front side of the port. Resulting to nearly symmetrical in-cylinder flow between tumble '4' and cross-tumble '3'. And as Singh *et al.* (2019) cited, cross-tumble flow directly influences, and weaken the tumble strength flowing from the front of the intake port. This is clearly confirmed at the current study as in the CFD figures discussed here.

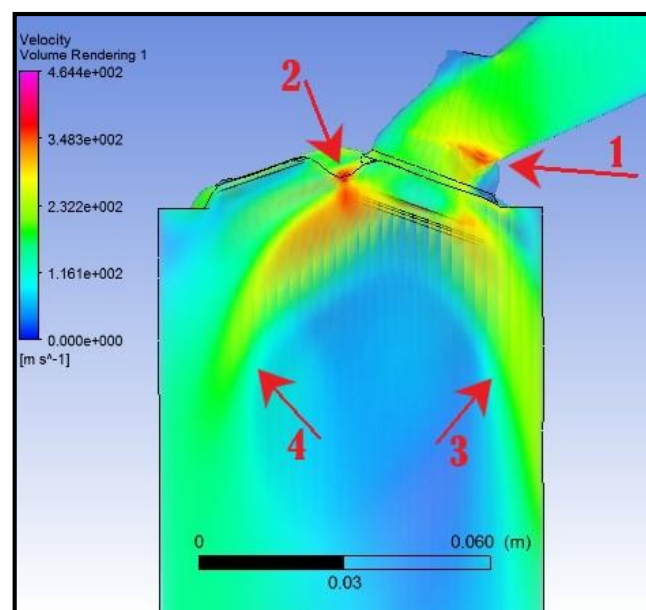


Figure 26 Base-line 8mm CFD post results (Yousouf, 2020)

4.3. Optimised-Design Analysis

4.3.1. Design Parameters:

Designing an intake-port geometry that promote high tumble charge, depends on multiple parameters. The approach which was taken to optimise the base-line model analysed above, comprises of 3 imperative design variables.

- *Port profile angle*: as considered by Falfari, Brusiani, and Bianchi (2012), as well as Singh *et al.* (2019), Port profile angle is very important to direct the flow charge toward the front-side of the intake-valve, in order to enhance the tumble motion.
- *Port chimney height*: port chimney height as demonstrated in figure 28, is a key parameter to control cross-tumble flow at the back of the valve. And as previously analysed in the CFD figure, there was a high velocity flow at port-neck due to the height of the chimney which created an empty domain for the air to flow through. Schogl *et al.* (2011) also included chimney height, in the design parameters of tumble-port geometry.
- *Port masking*: port masking triggers the flow separation at the port back section and guide the air-charge toward the frontal area of the port. Thus, increases the overall tumble motion. Ye *et al.* (2018) significantly increased tumble flow at low valve lift, by performing a masking optimisation technique on an intake port of a turbocharged gasoline engine (see figure 27).

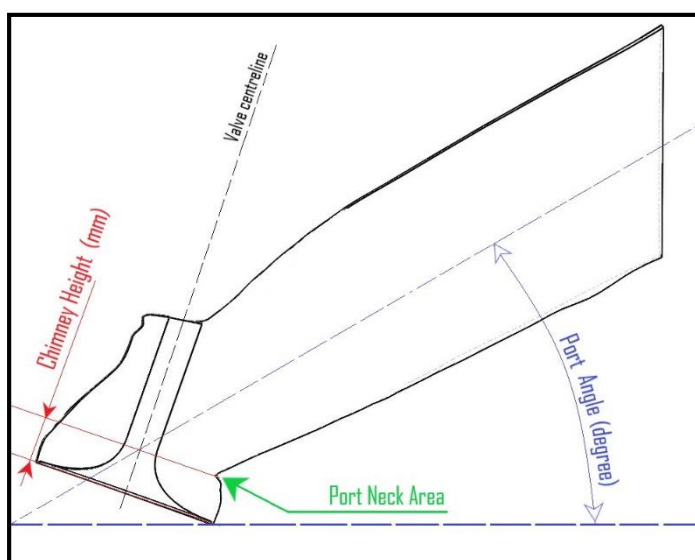


Figure 28 Intak-Port design parameters (Yousouf, 2020)

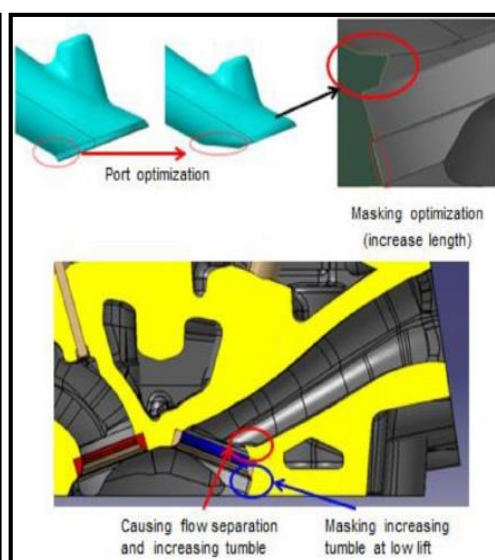


Figure 27 Port masking technique (Ye *et al.* 2018)

4.3.2. Design Concept:

With reference to the CFD analysis of the base-port, and the parameters discussed above. Two optimised-port concepts were developed, after numbers of drafts and primary sketches. However, this paper will only discuss the final optimised-design. Appendix 16 demonstrates both concepts.

As per figure 31, it is clear that valve angle and valve seat diameter were not changed. This was very important in order to achieve a feasible design, which can be realistically implemented in the stock cylinder-head, as the valve seat diameter was already stretched to the maximum in the base-line model. Valve-angle was not modified for a cost-efficient measure, in order to prevent the

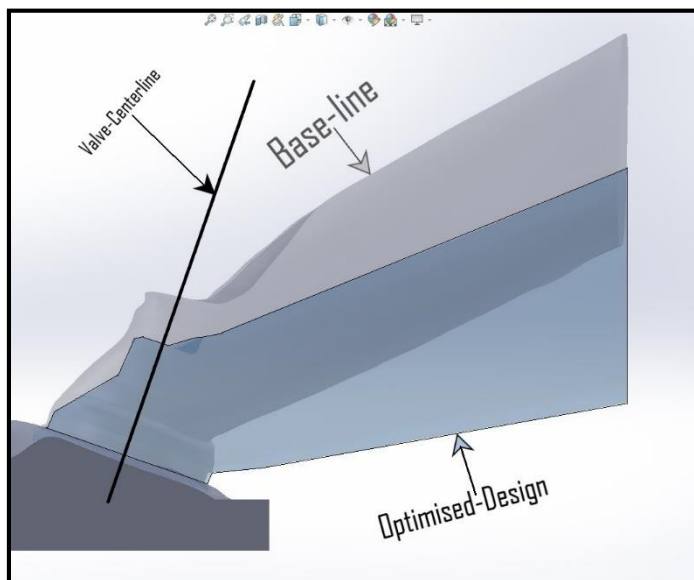


Figure 31 Base-line vs Optimised-model overlaid (Yousouf, 2020)

complete head modification during the manufacturing process.

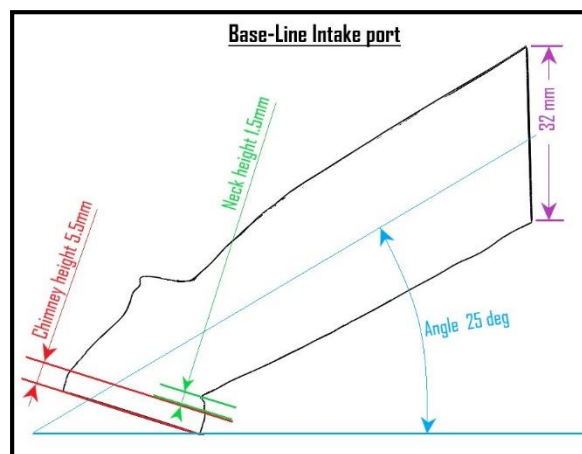


Figure 29 Base-model 2D sketch (Yousouf, 2020)

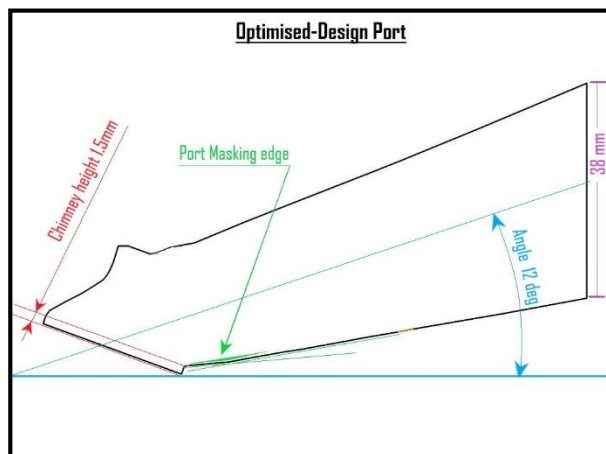


Figure 30 Optimised-model 2D sketch (Yousouf, 2020)

As illustrated in the 2D sketches here, the major changes which defines the tumble port design from the baseline are the port angle, chimney height, and port masking.

Port angle have been reduced by 13 degrees compared to the base-line, this change was very aggressive and influenced the mass-flow rate drastically. Therefore, to overcome that problem, port-manifold entrance was enlarged from 32mm to 38mm, which successfully prompt an extra air-flow inside the cylinder. Chimney height was significantly reduced from 5.5mm to 1.5mm, this design prevented the air-flow from changing directions as it comes closer to the valve-seat. Port masking was applied for two reasons; one to eliminate the port-neck geometry which was drawing the flow-charge from the back section, and two to create flow separation for a higher tumble charge, as the technique was previously successful according to Ye *et al.* (2018).

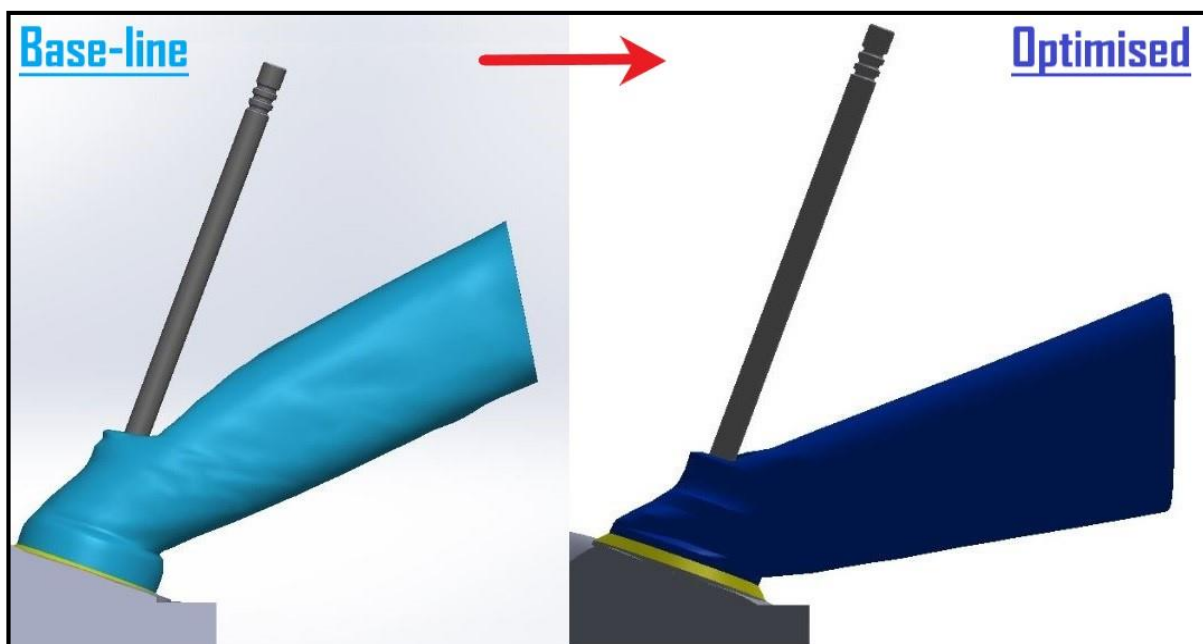


Figure 32 Base-line Vs optimised-design 3D sketch (Yousouf, 2020)

4.3.3. Optimised-Design CFD Results:

Similar to the base-line model, the optimised tumble-port was computationally tested for 5 simulation runs, with 2000 iteration each (see Appendix 17). The final results were exported for post processing, and are discussed as followed:

2mm valve lift:

Although the masking technique did not stop the air to flow from the back section '1' at low VL, it did significantly enhance the front section flow-velocity compared to the base-line. Thus, a substantial tumble charge started building as pointed in '2'.

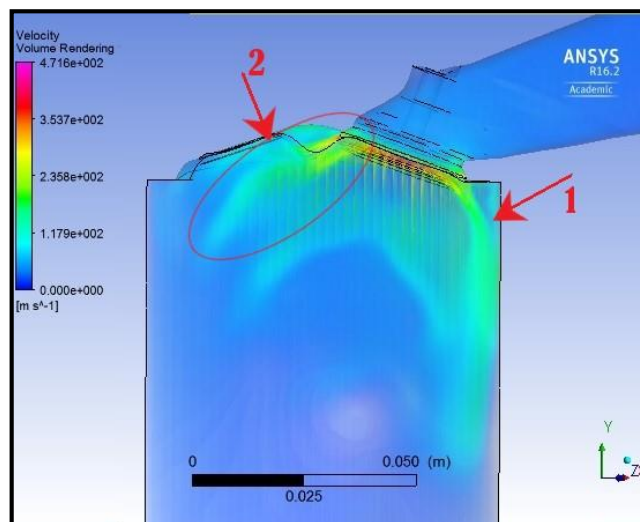


Figure 33 Optimised-port 2mm CFD post results (Yousouf, 2020)

4mm valve lift:

The over developed cross-tumble charge seen in the base-line at 4 mm VL, has been significantly controlled throughout the optimised design. The tumble-charge marginally exceeded the air-flow at the back section, which in turn minimised the cross-tumble strength.

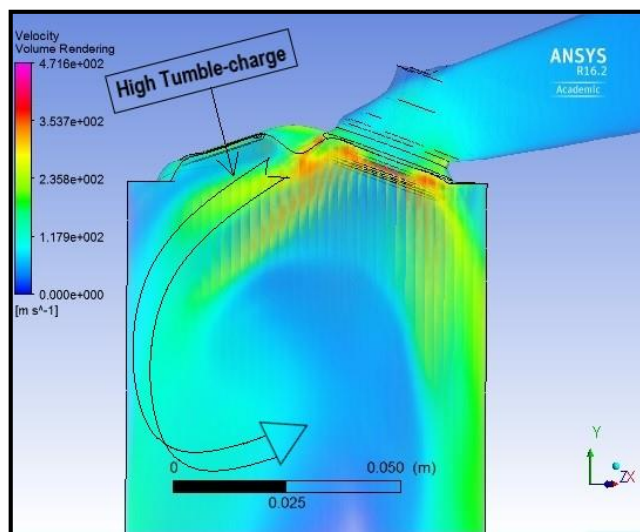


Figure 34 Optimised-port 4mm CFD post results (Yousouf, 2020)

6mm valve lift:

At 6mm VL, the asymmetry of the flow was significant, as the tumble charge '1' positively took over the cross tumble. the tumble-oriented port design noticeably increased the front section flow velocity and restrained the velocity at port back-section.

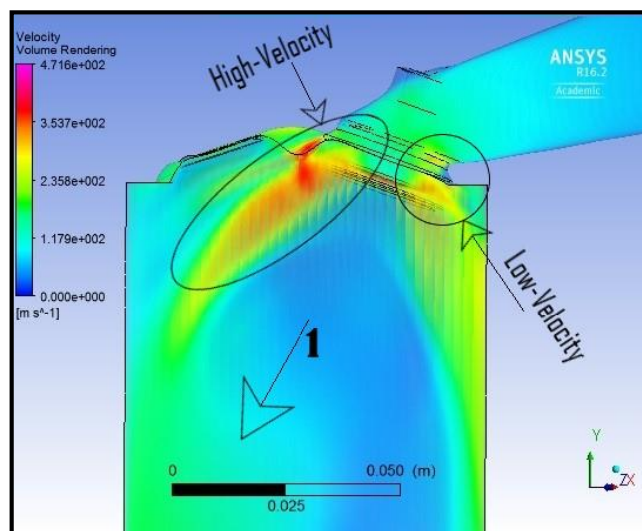


Figure 35 Optimised-port 6mm CFD post results (Yousouf, 2020)

8mm valve lift:

At maximum VL, the flow velocity at port back section diminished, and the cross-tumble charge further reduced. Opposingly, the frontal section flow velocity increased, pushing a larger amount of air through the front side of the port. Consequently, tumble charge dominated the in-cylinder flow structure.

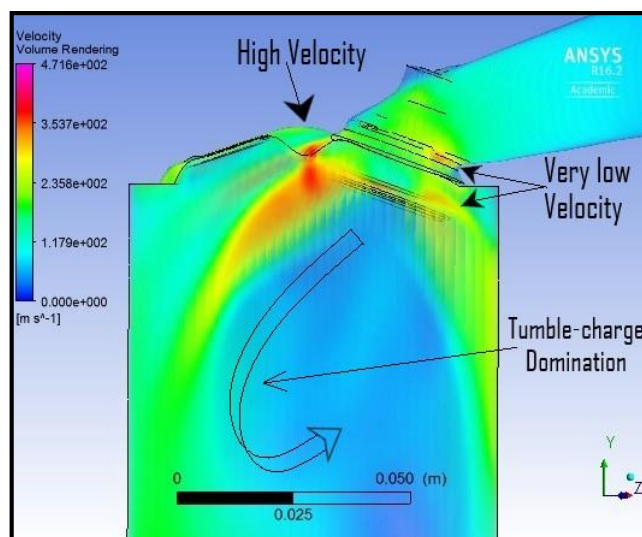


Figure 36 Optimised-port 8mm CFD post results (Yousouf, 2020)

4.4. Designs Comparison

4.4.1. Intake-port Flow-velocity:

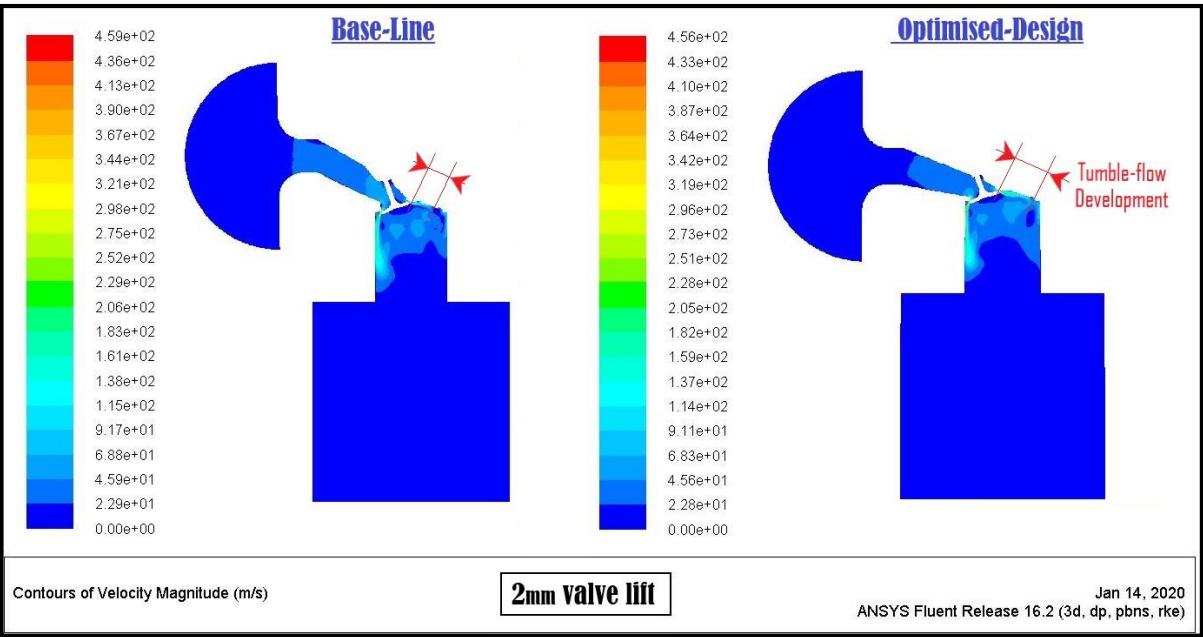


Figure 38 Base vs Optimised port, flow velocity comparison at 2mm (Yousouf, 2020)

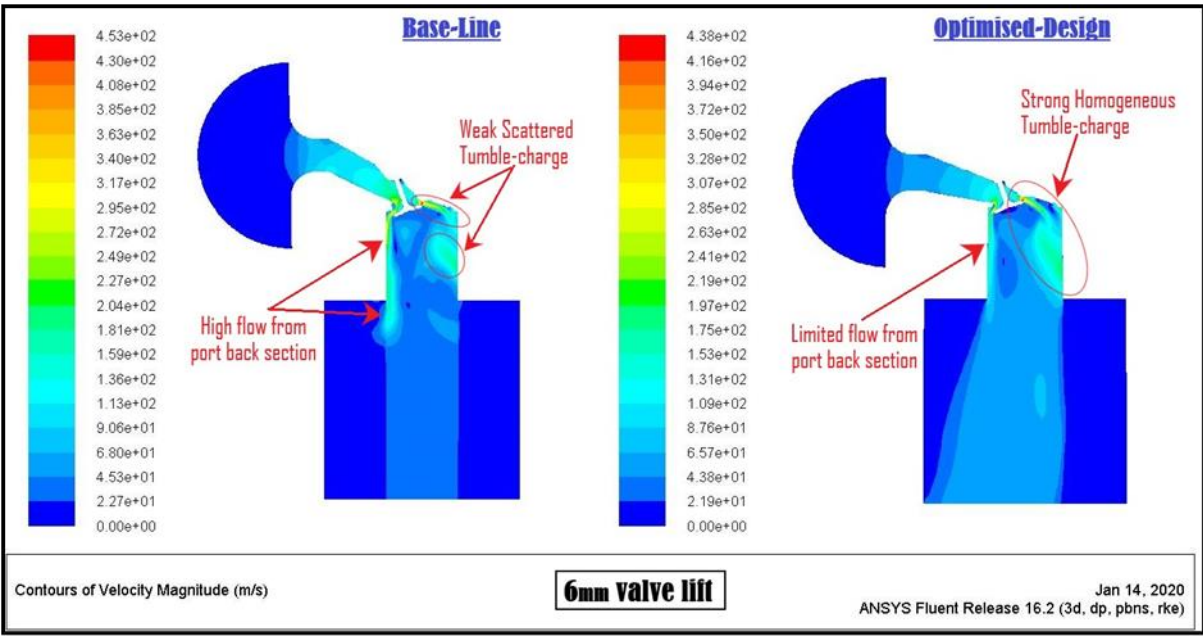
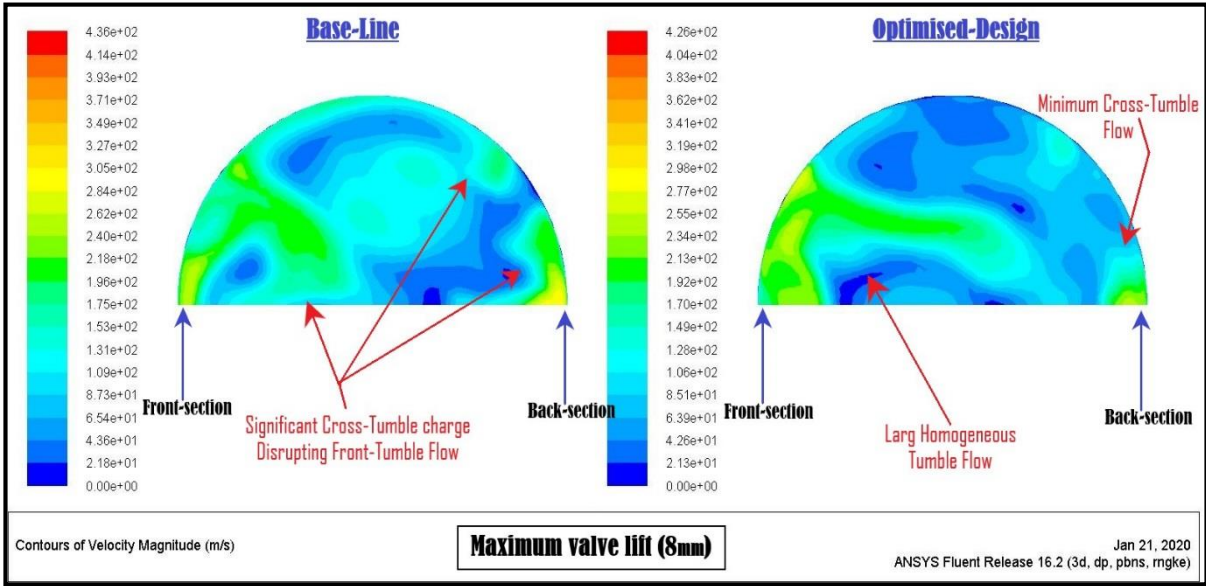
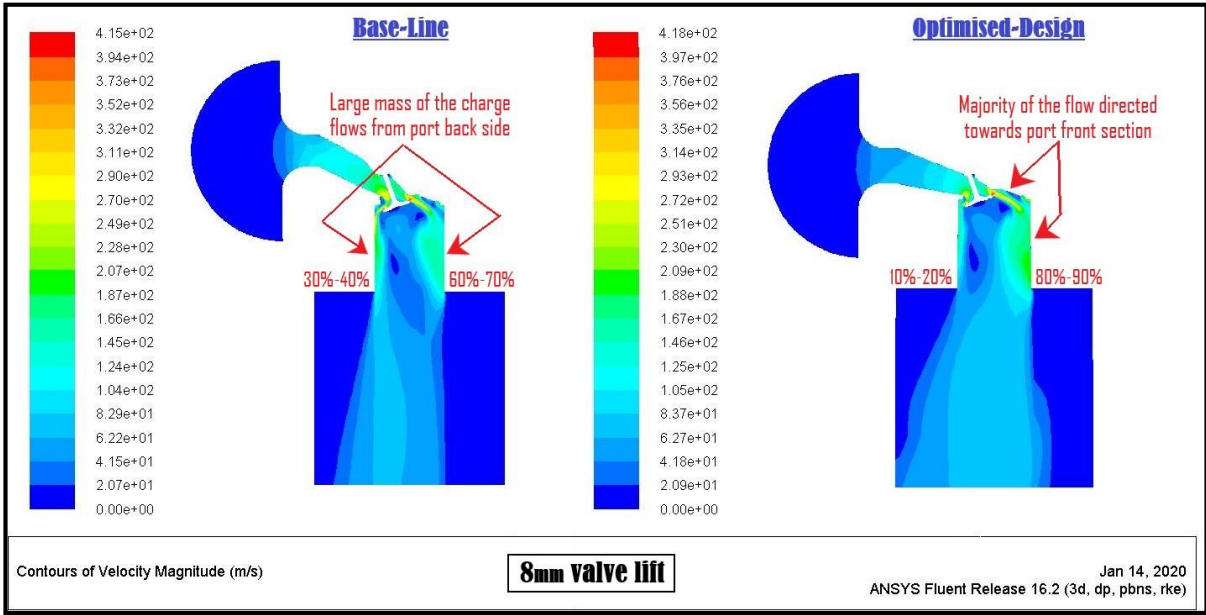


Figure 37 Base vs Optimised port, flow velocity comparison at 6mm (Yousouf, 2020)



4.4.2. In-Cylinder Turbulence:

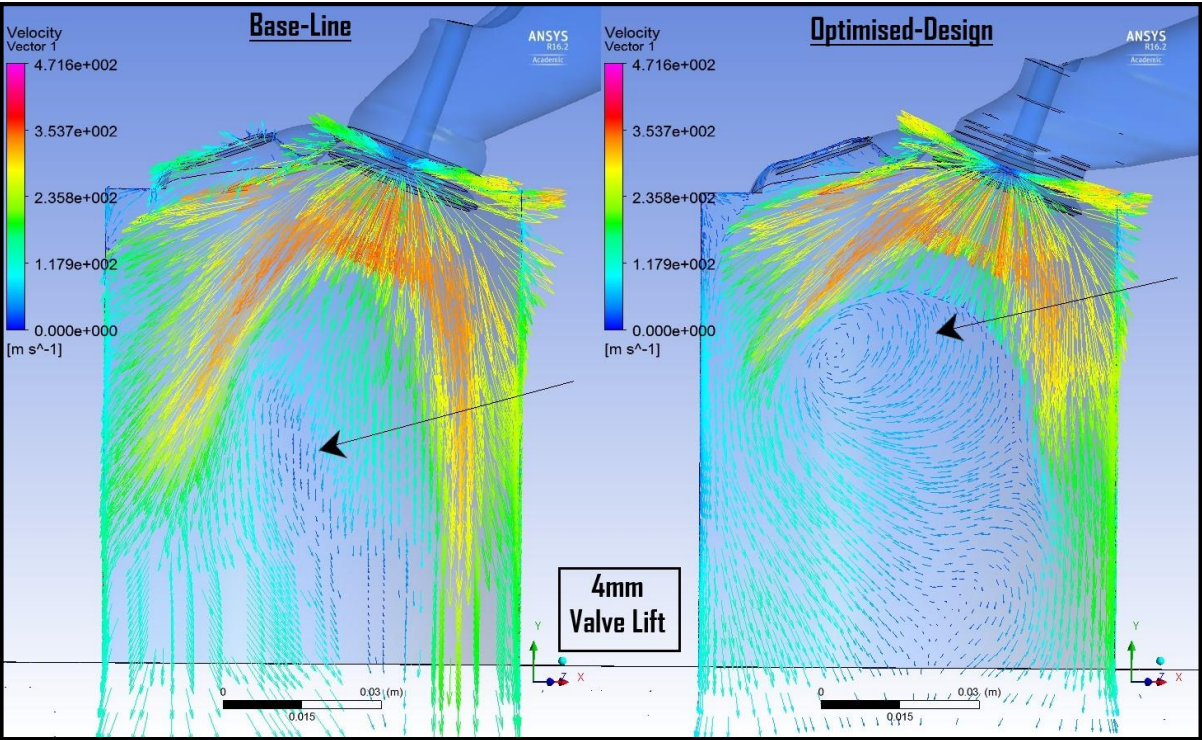


Figure 42 Base vs Optimised port, IN-cylinder turbulence comparison at 4mm (Yousouf, 2020)

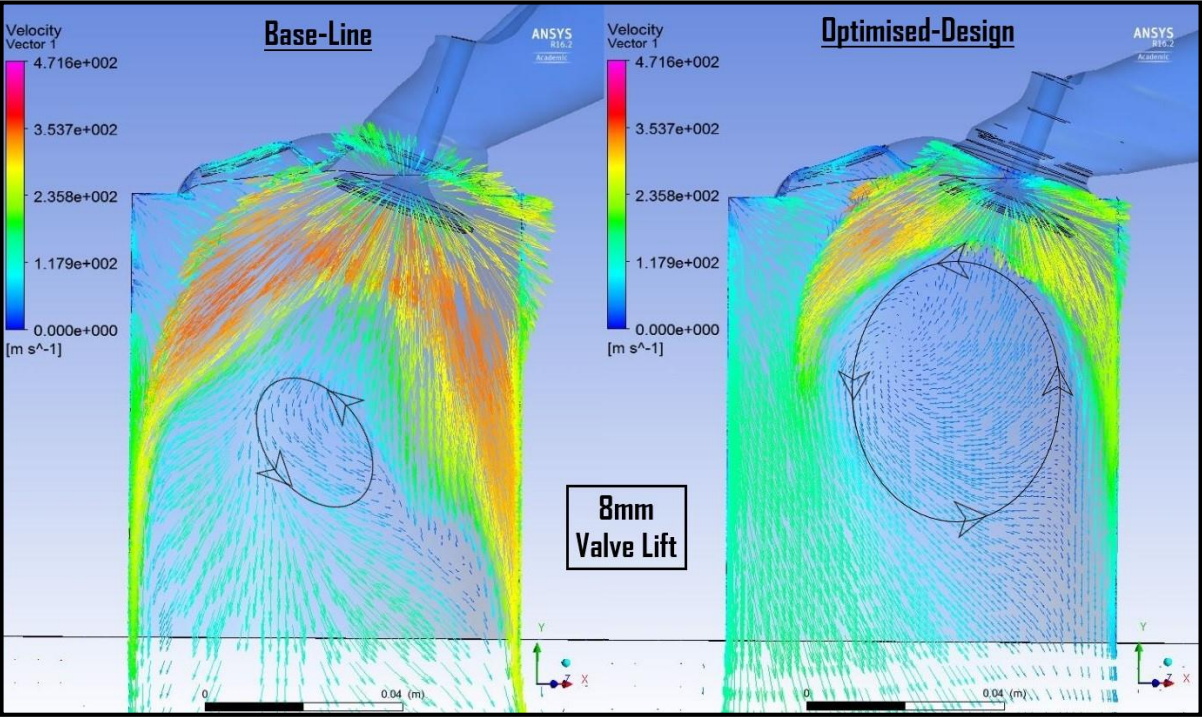


Figure 41 Base vs Optimised port, In-cylinder Turbulence comparison at 8mm (Yousouf, 2020)

With reference to the literature discussed, and the CFD figures presented above, it is evident that the optimised high tumble port, significantly increase the in-cylinder turbulence structure.

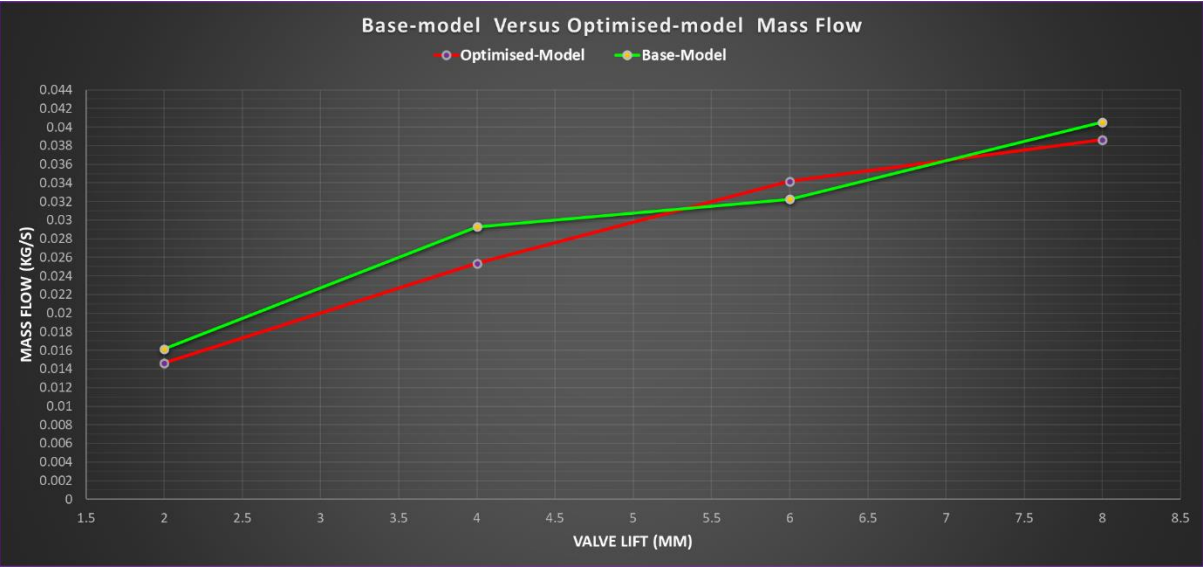


Figure 44 Base vs Optimised port, mass flow over VL graph (Yousouf, 2020)

ICE/CFD Mass Flow (Kg/S)	ICE/CFD Mass Flow (Kg/S)	Mass Flow Rate	Percentage
Base-model Port	Optimised-model Port		
0.016146	0.014635		4.9% Decrease
0.029273	0.025359		7.2% Decrease
0.032232	0.034162		3.0% Increase
0.040524	0.038637		2.4% Decrease
		Overall mass flow Reduction = 2.3%	

Figure 43 Base vs Optimised port, mass flow results (Yousouf, 2020)

Mass flow figures above, indicates that the introduced port design slightly influenced the amount of air induced inside the cylinder. This was expected as previous researchers critically discussed the trend between high-tumble port and flow coefficient. Ye *et al.* (2018) discussed the trend between high-tumble and medium-tumble port configurations, and the results revealed a noticeable reduction in flow coefficient regarding high-tumble port compared to its counterpart (see Appendix 18). The optimised-port, in the other hand, shows a steady and constant flow development compared to the bas-line, which nearly stalled between 4mm and 6mm VL before it increases again at maximum VL. Iyer and Yi (2009) confirmed through a dynamometer test comparison study, between high and medium tumble ports that, a high tumble port mitigates knock and improves combustion stability due to the consistent flow structure. Thus, a 2.3% reduction in mass-flow is minimal compared to the benefits gained from the high turbulent intensity and TKE which are discussed further down.

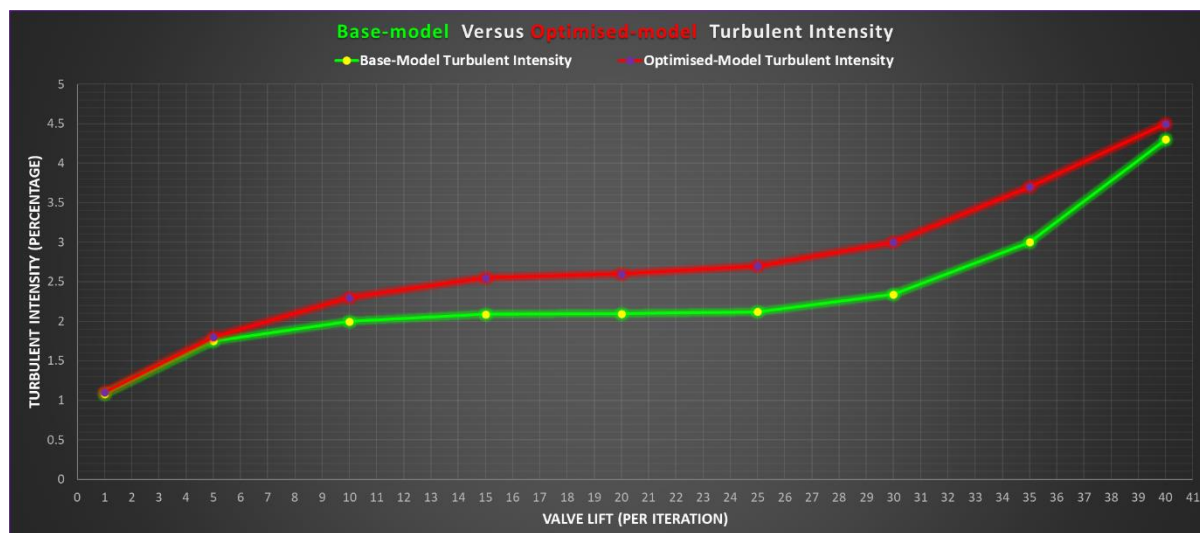


Figure 46 Base vs Optimised port, Turbulent intensity Graph (Yousouf, 2020)

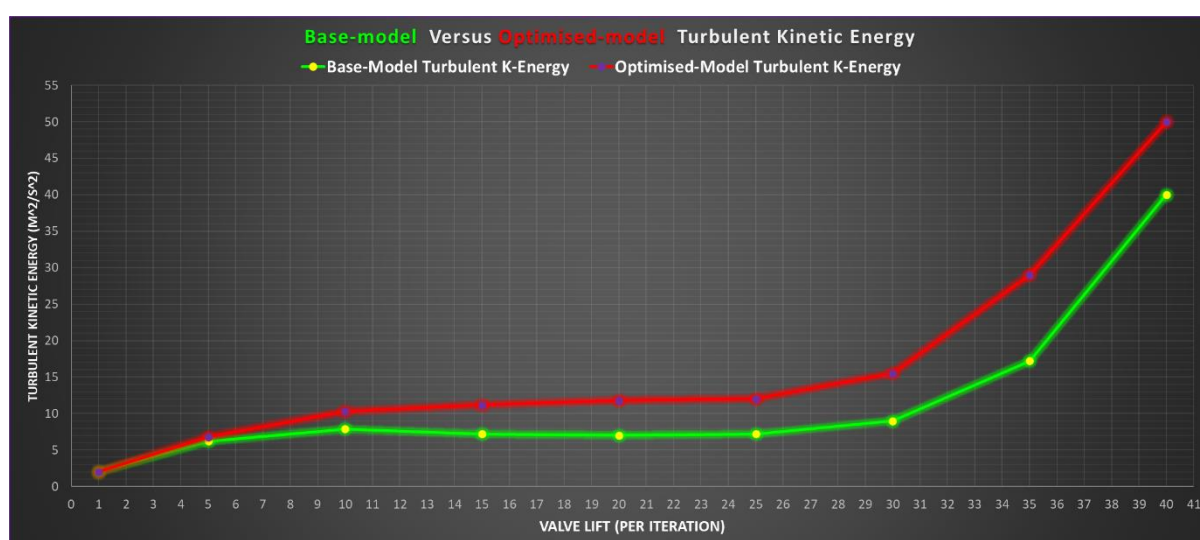


Figure 45 Base vs Optimised Turbulent Kinetic Energy Graph (Yousouf, 2020)

Despite the minimal decrease in the mass flow rate, the optimised-port significantly enhanced turbulent intensity by 7.72%, which resulted to 17.8% increase in turbulent kinetic energy as demonstrated in the above graphs. The important of that significant achievement can be justified as followed:

In 2009 Ford Motor Company, optimised the in-cylinder flow of the 3.5 litre V6 Ecoboost engine, by developing a high tumble intake-port through the application of 3D CFD tools. The optimised tumble-port was than tested in a monitored facilities using an engine dynamometer and the results revealed that: the higher turbulent intensity developed during the intake stroke was maintained during the compression stroke, which translated into a faster burn rates; Air-fuel mixture improved significantly

due to the high-tumble port configuration; the high in-cylinder turbulent intensity improved knock resistance as per dynamometer test measurement; tumble-port is able to achieve a higher engine-load and needs less fuel enrichment, which in turn translated into fuel consumption benefits; the coherent high tumble motion of the optimised port eliminated flow fluctuation sensitivity, which led to an improved combustion stability (Iyer and Yi, 2009).

As mentioned earlier, the flow structure investigation was conducted during low engine speed (i.e. 2000rpm). However, in order to approximately understand the influence of turbulence increase on medium and high engine-rpm, multiple research papers have been studied. Cameretti *et al.* (2017), investigated the influence of different port geometries on tumble and in-cylinder turbulent, across a wide range of engine rpm (i.e. 2000, 4000, and 6000rpm). The results can be observed in the figure below.

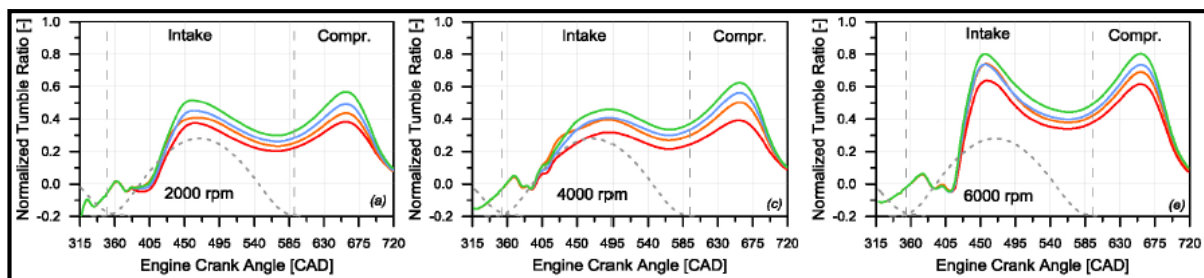


Figure 47 tumble across different engine RPM (Cameretti *et al.* 2017)

As cited by Cameretti and colleagues, the trend on tumble ratio development between the 4 different port geometries, is quite similar even if a certain shift occurs at increasing engine speeds. Also, the tumble ration increases as the engine speed goes higher, leading to an increased turbulent intensity inside the cylinder.

The Influence of engine rpm on tumble and turbulent has been substantially investigated by various researchers. It is an important parameter in the development of a successful engine, that can operate efficiently across a wide engine-map. Consequently, it is hoped that future research will build upon this project by continuing to investigate the effect of engine speed, on the previously discussed flow characteristics.

CONCLUSION

In the current research, a 1.6 Ford Ecoboost GDI intake-port has been investigated, throughout the application of reverse engineering tools, CAE design, and CFD simulation, aided by experimental validation and theoretical work. The study aimed to analyse the influence of port geometry on the tumble-motion and in-cylinder flow structure. After an extensive computational work, and design concepts, a tumble-oriented port was developed where the following results were reported, as the concluding factors which also would have been addressed during the final viva:

- ❖ Amongst 6 different turbulence settings, Realisable K- ϵ turbulence model with enhanced-wall treatment function, was found to be the most accurate model on mass-flow and in-cylinder turbulent prediction.
- ❖ The optimised-port generated higher and coherent tumble motion in comparison to the base-line design.
- ❖ The high tumble charge generated by the optimised port, marginally influenced mass flow rate, with a reduction of 2.3%.
- ❖ The new optimised intake-port enhanced the turbulent intensity by 7.72%, which resulted in a significant increase in TKE (17.8%).
- ❖ An extensive literature review proved that, the increase of turbulent intensity directly influences burn rates, enhances air-fuel mixture and combustion stability, reduces knock sensitivity, and improves BSFC.

Taken together, it is concluded that a high tumble port can significantly enhance engine performance, including thermal efficiency, fuel economy, and emissions level.

A successful manufacturing of the optimised intake-port was achieved, through CNC machining at UWTSD manufacturing laboratory. The manufactured prototype (see Appendix 19) was intended to be used for validation procedures, to experimentally verify the minimum reduction on mass flow rate. Unfortunately, due to unforeseen circumstances of the Covid-19 pandemic, the mentioned work was unable to be completed. However, the extensive research discussed above, correlates to the findings and results of the current project.

REFERENCES

- Academy of Science of South Africa, et al. (2019) 'Air Pollution and Health – A Science-Policy Initiative' *Annals of Global Health*, 85(140). DOI: 10.5334/aogh.2656
- Adomeit, A., Hopp, M., Schmidt, A., Lang, O., and Schmitt, P. (2005) 'CAE-Based port development and flow design for SI engines', *SAE International Journal*. DOI: 10.4271/2005-01-0243
- Adomeit, A., Jakob, M., Pischinger, S., Brunn, A., and Ewald, J. (2011) 'Effect of intake port design on the flow field stability of a gasoline DI engine', *SAE International Journal*. DOI: 10.4271/2011-01-1284
- ANSYS Inc (2020) Comprehensive IC Engine Flow and Combustion Development. Available at: <https://bit.ly/2yINfoD> (Accessed: 15th November 2019)
- ANSYS Inc. (2012) 'Automotive Simulation', *World Congress*, 30th October. Available at: <https://bit.ly/2JO4RSs> (Accessed: 9th January 2020).
- Arnold, S., Balis, C., Jeckel, D., Larcher, S., Uhl, P., and Shahed S. (2005) 'Advances in turbocharging technology and its impact on meeting proposed California GHG emission regulations', *SAE International Journal*. DOI: 10.4271/2005-01-1852
- Bandel, W., Fraidl, G., Kapus, P., Sikinger, H., and Cowland, C. (2006) 'The Turbocharged GDI Engine: Boosted Synergies for high fuel economy plus ultra-low emission', *SAE International Journal*. DOI: 10.4271/2006-01-1266
- Barbouchi, Z., and Bessrour, J. (2009) 'Turbulence study in the internal combustion engine', *Journal of Engineering and Technology Research*, 1(9), pp. 194-202. Available Online At: <http://www.academicjournals.org/jetr>
- Berntsson, A., Josefsson, G., Ekdahl, R., Ogink, R., and Grandin, B. (2011) 'The effect of tumble flow on efficiency for a direct injection turbocharged downsized gasoline engine', *SAE International Journal of Engines*, 4(2), pp. 2298-2311. DOI: 10.4271/2011-24-0054
- Bevan, k., and Ghandhi, J. (2004) 'PIV measurements of in-cylinder flow in a four-stroke utility engine and correlation with steady flow results', *SAE International Journal*. DOI: 10.4271/2004-32-0005

Bialy, M., Wendeker, M., Magryta, P., and Czyz, Z. (2014) 'CFD model of the mixture formation process of the CNG direct injection engine', *SAE International Journal*. DOI: 10.4271/2014-01-2575

Bosch, R. (2014) *Automotive Handbook*. 9th Ed. Sussex: Robert Bosch GmbH.

Bozza, F., Bellis, V., Berni, F., D'adamo, A., and Maresca, L. (2018) 'Refinement of 0D turbulence model to predict tumble and turbulent intensity in SI engines. Part1: 3D analyses', *SAE International Journal*. DOI: 10.4271/2018-01-0850

Bozza, F., Toedossio, L., Bellis, V., Fontanesi, S., and Lorio, A. (2018) 'Refinement of 0D turbulence model to predict tumble and turbulent intensity in SI engines. Part2: model concepts, validation and discussion', *SAE International Journal*. DOI: 10.4271/2018-01-0856

Cameretti, M., Bellis, V., Romagnuolo, L., Iorio, A., and Maresca, L. (2017) '3D CFD analyses of intake duct geometry impact on tumble motion and turbulence production in SI engines', *SAE International Journal*. DOI: 10.4271/2017-01-2199

Davidson, L. (2018) 'An introduction on turbulence models', *Chalmers University of Technology*.

De Lima, B., Thiriet, A., Teixeira, A., and Valle, R. (2017) 'Three-dimensional model obtained from reverse engineering for analysis of combustion in internal combustion engine adapted with pre-chamber', *SAE International Journal*. DOI: 10.4271/2017-36-0346

El-Adawy, M., Heikal, M., Aziz, A., Siddiqui, M., and Munir, S. (2017) 'Characterization of the inlet port flow under steady-state conditions using PIV and POD', *Energies Journal*. DOI: 10.3390/en10121950

Falfari, S., Brusiani, F., and Bianchi, G. (2012) 'Assessment of the influence of intake duct geometrical parameters on the tumble motion generation in a small gasoline engine', *SAE International Journal*. DOI: 10.4271/2012-32-0095

Fischer, M., Kreutziger, P., Sun, Y., and Kotrba, A. (2017) 'Clean EGR for gasoline engines – innovative approach to efficiency improvement and emissions reduction simultaneously', *SAE International Journal*. DOI: 10.4271/2017-01-0683

Fogla, N., Bybee, M., Mirzaeian, M., Millo, F., and Wahiduzzaman, S. (2017) 'Development of a K-k- ϵ phenomenological model to predict in-cylinder turbulence', *SAE International Journal*. DOI: 10.4271/2017-01-0542

Han, Z., and Reitz, R. (2007) 'Turbulence modelling of internal combustion engines using RNG k- ϵ models', *CST Combustion Science and Technology*, 106(6), pp. 267-295. DOI: 10.1080/00102209508907782

HEI Review Panel. (2013) *Understanding the health effects of ambient ultrafine particles*. Health Effects Institute, Boston, MA.

Hunter, J. (2017) 'The Little NB2 Miata that could – Part1 the BP-VE / BP-Z3', 5523 *motorsports*, 20 January. Available at: <https://bit.ly/2yIpGfI> (Accessed: 19th November 2019).

Ikeya, K., Takazawa, M., Yamada, T., Park, S., and Tagishi, R. (2015) 'Thermal efficiency enhancement of a gasoline engine', *SAE International Journal*. DOI: 10.4271/2015-01-1263

Iyer, C., and Yi, J. (2009) '3D CFD upfront optimisation of the in-cylinder flow of the 3.5L V6 Ecoboost engine', *SAE International Journal*. DOI: 10.4271/2009-01-1492

Johnson, T., and Joshi, A. (2018) 'Review of vehicle engine efficiency and emissions', *SAE International Journal*. DOI: 10.4271/2018-01-0329

Karaya, Y., Addepalli, S., and Mallikarjuna, J. (2019) 'Comparison of conventional intake port and swirl intake port on mixture formation in a GDI engine – a CFD analysis', *SAE International Journal*. DOI: 10.4271/2019-01-0010

Kim, Y., Kim, M., Kim, J., Song, H., Park, Y., Han, D. (2018) 'Predicting the influences of intake port geometry on the tumble generation and turbulence characteristics by zero-dimensional spark ignition engine model', *SAE International Journal*. DOI: 10.4271/2018-01-1660

Kim, Y., Kim, M., Oh, S., Shin, W., Cho, S., and Song, H. (2019) 'A new physics-based modelling approach for a 0D turbulence model to reflect the intake port and chamber geometries and the corresponding flow structure in high-tumble spark-ignition engines', *Energies Journal*. DOI: 10.3390/en12101898

Kirwan, J., Shost, M., Roth, G., and Zizelman, J. (2010) '3-Cylinder turbocharge gasoline direct injection value solution for low CO₂ and NO_x emissions', *SAE International Journal*. DOI: 10.4271/2010-01-0590

Krishna, A., Mallikarjuna, J., Davinder, K., and Babu, Y. (2013) 'In-cylinder flow analysis in a two-stroke engine : a comparison of different turbulence models using CFD', *SAE International Journal*. DOI: 10.4271/2013-01-1085

Kumar, A., Jain, P., and Pathak, P. (2013) 'Reverse Engineering in Product Manufacturing: An Overview', *DAAAM International Scientific Book 2013*, pp. 665-678. DOI: 10.2507/daaam.scibook.2013.39

Kume, T., Iwamoto, Y., Lida, K., Murakami, M., Akishino, K., and Ando, H. (1996) 'Combustion control technologies for direct injection SI engine', *SAE International Journal*. DOI: 10.4271/960600

Kurt, E. (2004) 'Trends of future emission legislation and its measurement requirements', *SAE International Journal*. DOI: 10.4271/2004-01-3291

Kuwahara, K., Watanabe, T., Tanada, H., Ueda, K., and Ando, H. (2000) 'Intake-port design for Mitsubishi GDI engine to realize distinctive in-cylinder flow and high charge coefficient', *SAE International Journal*. DOI: 10.4271/2000-01-2801

Landrigan, F., Fuller, R., Acosta, N., Adeyi, O., Arnold, R., Basu, N... Zhong, M (2018) 'The lancet commission on pollution and health', *The Lancet Commissions*. DOI: 10.1016/S0140-6736(17)32345-0

Masi, M., Artico, L., and Gobbato, P. (2015) 'Measurements of the intake and in-cylinder flow field to investigate the reliability of CFD steady-state simulations for actual engine', *SAE International Journal*. DOI: 10.4271/2015-24-2404

Mito, Y., Tanzawa, K., Watanabe, M., and Eiyama, Y. (2012) 'Advanced combustion performance for high efficiency in new I3 1.2L supercharged gasoline engine by effective use of 3D engine simulation', *SAE International Journal*. DOI: 10.4271/2012-01-0422

Ogink, R., and Babajimopoulos, A. (2016) 'Investigating the limits of charge motion and combustion duration in a high-tumble spark-ignition direct-injection engine', *SAE International Journal*. DOI: 10.4271/2016-01-2245

Petitjean, D., Bernaradini, L., Middlemass, C., and Shahed, S. (2004) 'Advanced gasoline engine turbocharging technology for fuel economy improvement', *SAE International Journal*. DOI: 10.4271/2004-01-0988

Premnath, V., and Khalek, I. (2019) 'Particle emissions from gasoline direct injection engines during engine start-up (cranking)', *SAE International Journal*. DOI: 10.4271/2019-01-1182

Qi, Y., Liu, H., Midkiff, K., and Puzinauskas, P. (2010) 'A feasible CFD methodology for gasoline intake flow optimization in a HEV application – Part 2: Prediction and optimization', *SAE International Journal*. DOI: 10.4271/2010-01-2238

Radicchi, F., Gomes, P., Franieck, E., Lopes, G., Valle, R., Silva, B., and Reis, L. (2017) 'Study of the effects of flow box utilisation on charge motion and mixture formation in an internal combustion engine', *SAE International Journal*. DOI: 10.4271/2017-36-0335

Sadakane, S., Sugiyama, M., Kishi, H., Abe, S., Harada, J., and Sonoda, Y. (2005) 'Development of a new V-6 high performance stoichiometric gasoline direct injection engine', *SAE International Journal*. DOI: 10.4271/2005-01-1152

Saito, K., Sekiguchi, K., Imatake, N., Takeda, K., and Yaegashi, T. (1995) 'A new method to analyse fuel behaviour in a spark ignition engine', *SAE International Journal*. DOI: 10.4271/950044

Saito, K., Sekiguchi, K., Imatake, N., Takeda, K., and Yaegashi, T. (1995) 'A new method to analyse fuel behaviour in a spark ignition engine', *SAE International Journal*. DOI: 10.4271/950044

Saito, K., Sekiguchi, K., Imatake, N., Takeda, K., and Yaegashi, T. (1999) 'The health effects of PM_{2.5} including ultrafine particles', *Concawe*. 99(60). DOI: 10.4271/950044

Sakurai, Y., Nakao, Y., Hisano, A., Saitou, M., and Tanaka, K. (2017) 'Optimization of intake port for improvement of fuel consumption and torque', *SAE International Journal*. DOI: 10.4271/2017-32-0055

Saw, O., Addepalli, S., and Mallikarjuna, J. (2019) 'Effects of cylinder head geometry on mixture stratification, combustion and emissions in a GDI engine – a CFD Analysis', *SAE International Journal*. DOI: 10.4271/2019-01-0009

Schogl, O., Edtmayer, H., Schmidt, S., Leiber, S., and Schabetsberger, T. (2011) 'Design of a tumble-orientated intake port layout for a gasoline combustion process used in power sport application', *SAE International Journal*. DOI: 10.4271/2011-32-0589

Selvaraj, B., Sridhara, S., Indraprakash, G., Senthilkumar, A., and Pangaonkar, A. (2011) 'Effects of intake port geometry on the performance of an SI engine', *SAE International Journal*. DOI: 10.4271/2011-32-0506

Singh, A., Poonia, S., Jalan, A., Singh, J., and Kumar, N. (2019) 'Intake and Exhaust Ports Design for Tumble and Mass Flow Rate Improvements in Gasoline Engine', *SAE International Journal*. DOI: 10.4271/2019-01-0763

Sjeric, M., Kozarac, D., and Bogensperger, M. (2012) 'Implementation of a single zone k- ϵ turbulence model in a multi zone combustion model', *SAE International Journal*. DOI: 10.4271/2012-01-0130

Sjeric, M., Kozarac, D., and Ormuz, K. (2016) 'Cycle-simulation turbulence modelling of IC engines', *International Journal of Automotive Technology*, 17(1), pp. 51-61. DOI: 10.1007/s12239-016-0004-2

Slater, J. W. (2008) 'Overview of CFD Verification and Validation', NASA, 17 July. Available at: <https://www.grc.nasa.gov/WWW/wind/valid/tutorial/overview.html> (Accessed: 7th January 2020).

Smith, J., and Sick, V. (2006) 'A multi-variable high-speed imaging study of ignition instabilities in a spray-guided direct-injection spark-ignition engine', *SAE International Journal*. DOI: 10.4271/2006-01-1264

Sutar, P., Dsouza, A., Sonawane, S., Rairikar, S., Kavathekar, K., Thipse, S., Marathe, N., Bhure, S., and Mapkar, M. (2017) 'Study of in-cylinder tumble effect on spark ignition direct injection SIDI engine performance using gasoline, CNG, & E85 fuel and simulation technique', *SAE International Journal*. DOI: 10.4271/2017-26-0076

Tikar, S., Malkhede, D., and Nandgaonkar, M. (2019) 'Optimization of in-cylinder flow and swirl generation analysis for a naturally aspirated diesel genset engine for emission reduction through intake port design', *SAE International Journal*. DOI: 10.4271/2019-28-0024

Tornatore, C., Siano, D., Marchitto, L., Iacobacci, A., Valentino, G., and Bozza, F. (2017) 'Water Injection: a technology to improve performance and emissions of downsized turbocharged spark ignition engines', *SAE International Journal*. DOI: 10.4271/2017-24-0062

Vaidya, A. (2010) *CFD Simulation of Internal Combustion Engines: Study of Mixture Formation Process in GDI Engines*. Saarbrücken: LAP Lambert Academic Publishing.

Verma, I., Hill, C., and Xu, M. (2018) 'Multi-objective adjoint optimisation of flow-bench port geometry', *SAE International Journal*. DOI: 10.4271/2018-01-0772

Versteeg, H. K., and Malalasekera, W. (2007) *An introduction to computational fluid dynamics: the finite volume method*. 2nd Ed. Essex: Pearson Education Limited.

Wentsch, M. (2018) 'Analysis of injection processes in an innovative 3D-CFD tool for the simulation of internal combustion engines', *Springer Fachmedien Wiesbaden GmbH*. DOI: 10.1007/978-3-658-22167-6

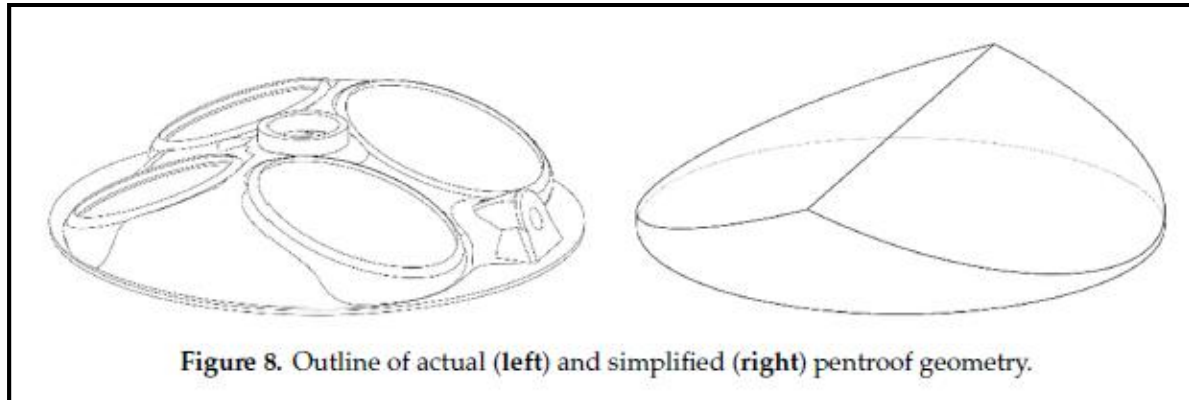
Wirth, M., Mayerhofer, U., Piock, W., and Fraidl, G. (2000) 'Turbocharging the DI Gasoline Engine', *SAE International Journal*. DOI: 10.4271/2000-01-0251

Ye, Y., Wang, J., Yao, K., Zhao, M., Dieterich, C., Souren, M., and Morcinkowski, B. (2018) 'Development of a new 1.8L down-speeding turbocharged gasoline engine with miller cycle', *SAE International Journal*. DOI: 10.4271/2018-01-1712

Zhao, F., Lai, M., and Harrington, D. (1999) 'Automotive spark-ignition direct-injection gasoline engines', *Progress in energy and combustion science*, 25(5), pp. 437-562.

APPENDIX

Appendix A



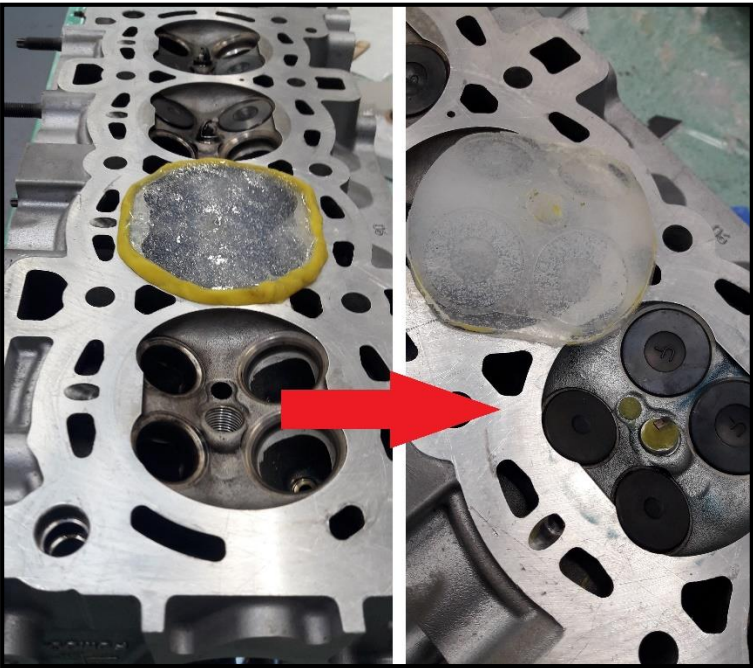
Kim et al. (2020)

Appendix 1



Yousouf (2020)

Appendix 2



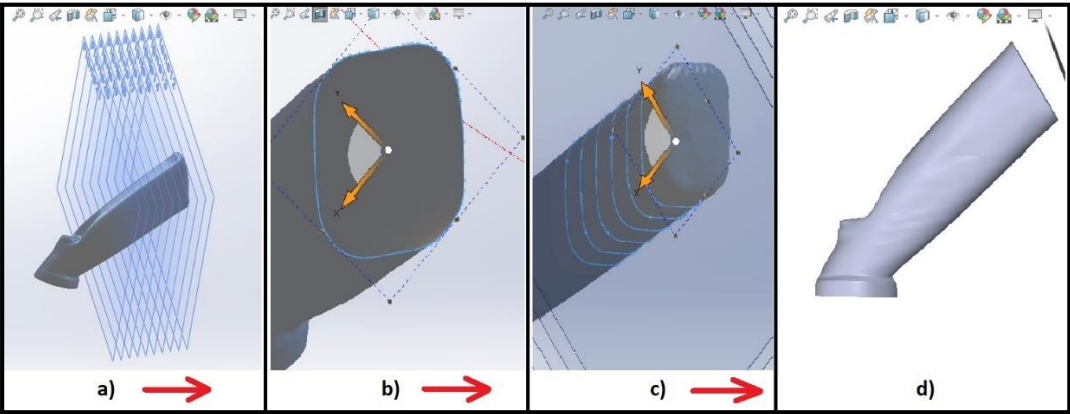
Yousouf (2020)

Appendix 3



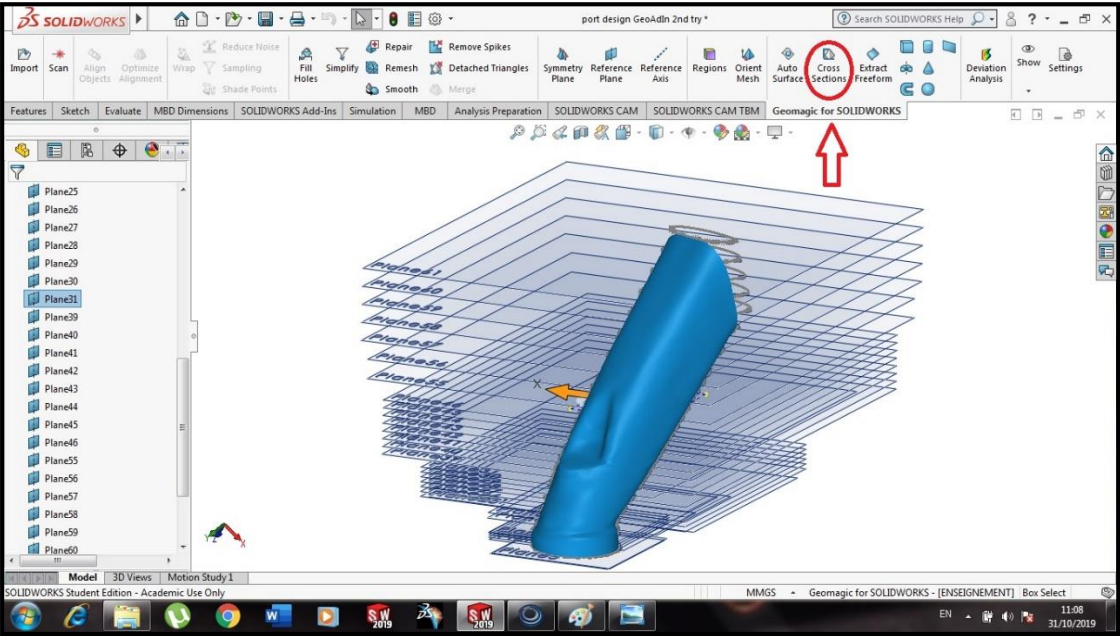
Yousouf (2020)

Appendix 4



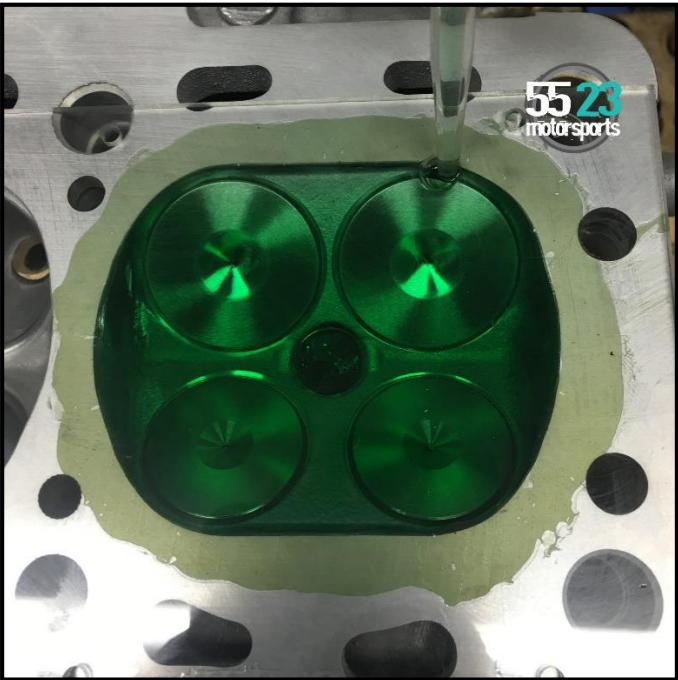
Yousouf (2020)

Appendix 5



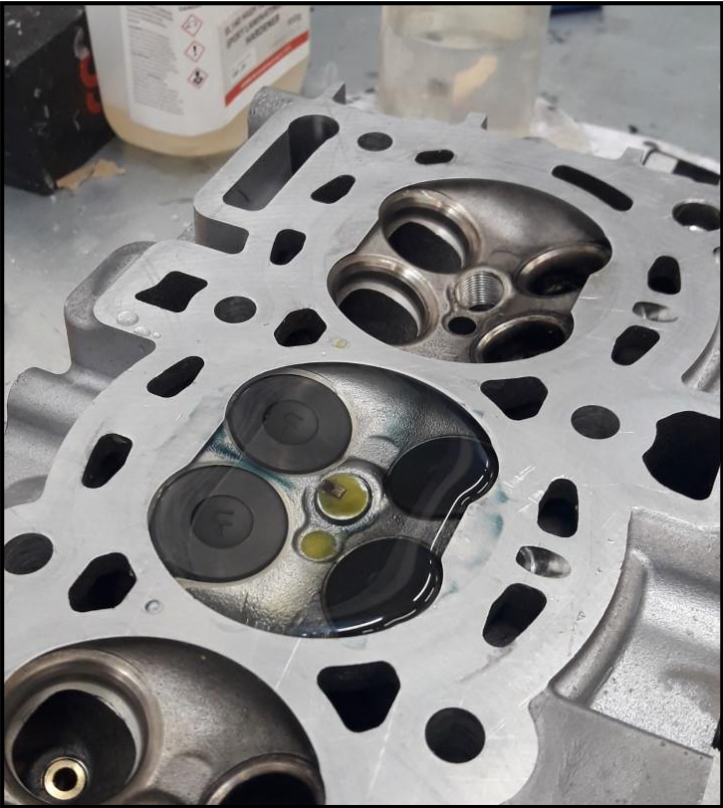
Yousouf (2020)

Appendix 6



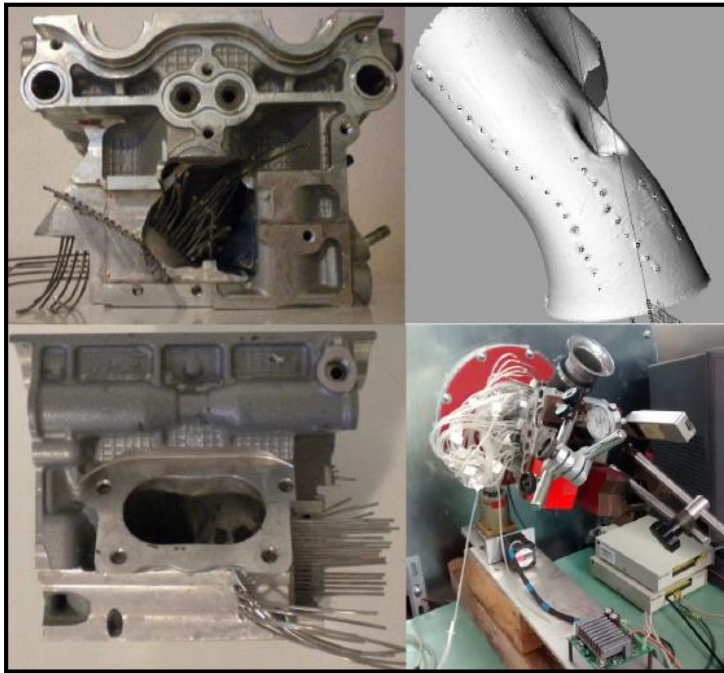
Hunter (2017)

Appendix 7



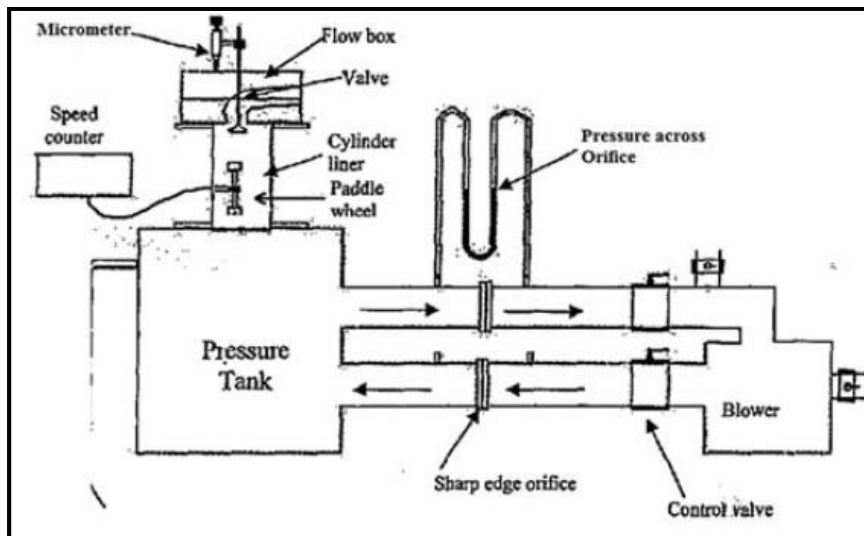
Yousouf (2020)

Appendix 8



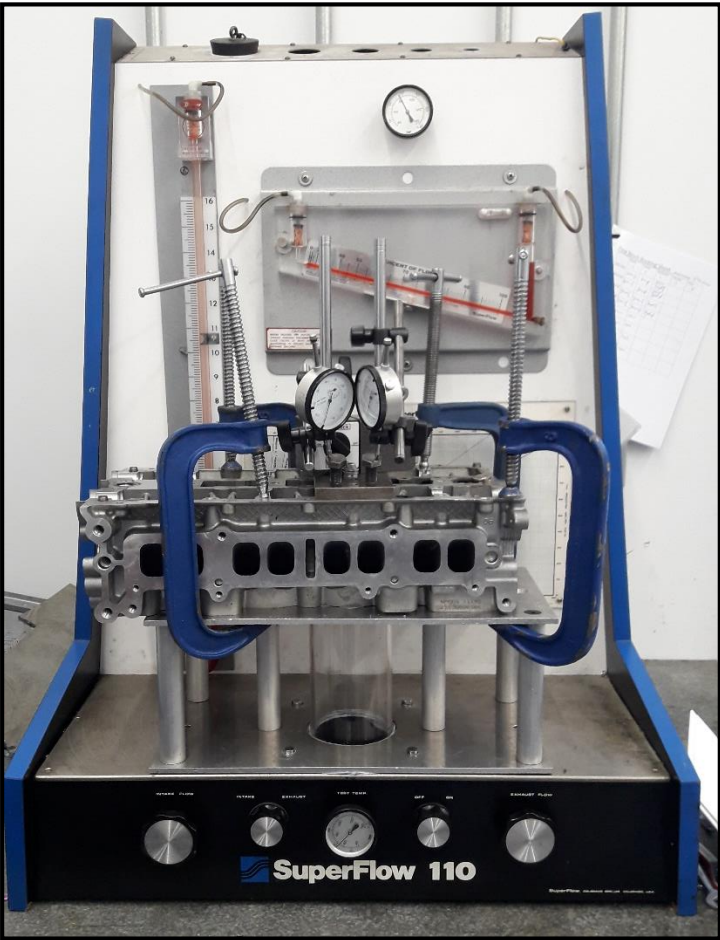
Masi, Artico, and Gobbato (2015)

Appendix 9



Sutar et al. (2017)

Appendix 10



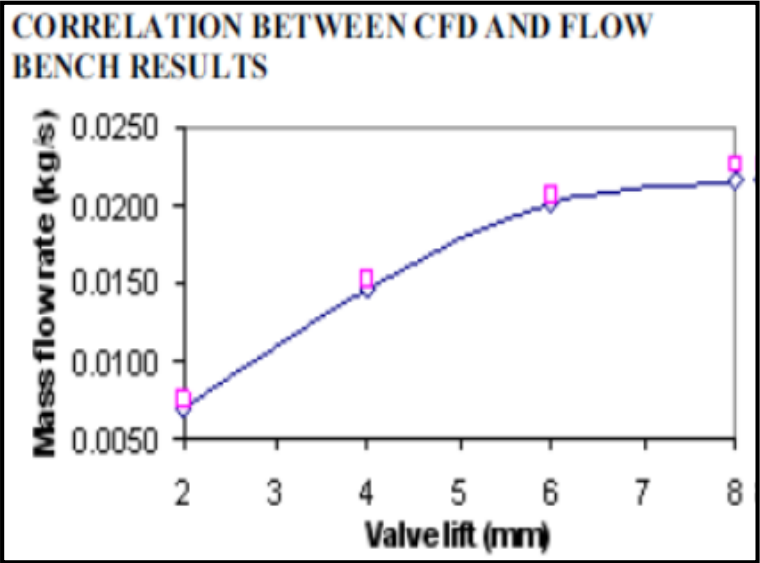
Yousouf (2020)

Appendix 11

Table 3. Boundary Conditions		
Type	Zones	Values
pressure-outlet	ice-outlet	Gauge Pressure (pascal) -3726.4
		Backflow Total Temperature (k) 303.4
pressure-inlet	ice-inlet-inplenum1	Gauge Total Pressure (pascal) 0
		Supersonic/Initial Gauge Pressure (pascal) 0
		Total Temperature (k) 300
wall	wall-ice-fluid-port	Temperature (k) 300
wall	ice-cyl	Temperature (k) 300
wall	ice-slipwall-inplenum1	Temperature (k) 300
wall	ice-slipwall-outplenum	Temperature (k) 300
wall	ice-valve-proximity-faces	Temperature (k) 300

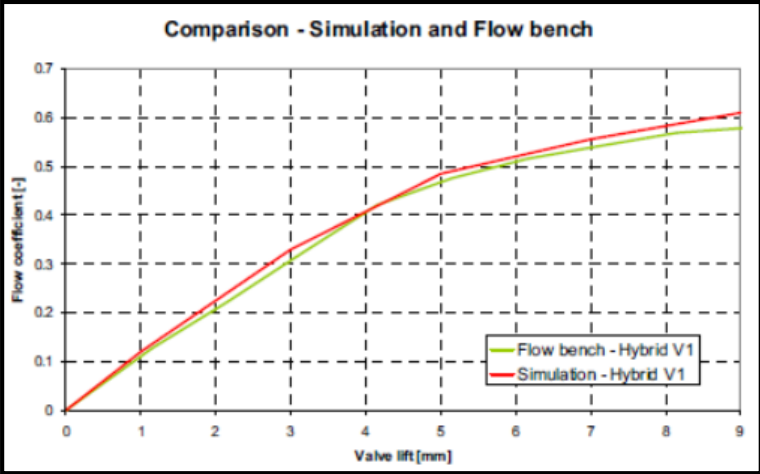
Yousouf (2020)

Appendix 12



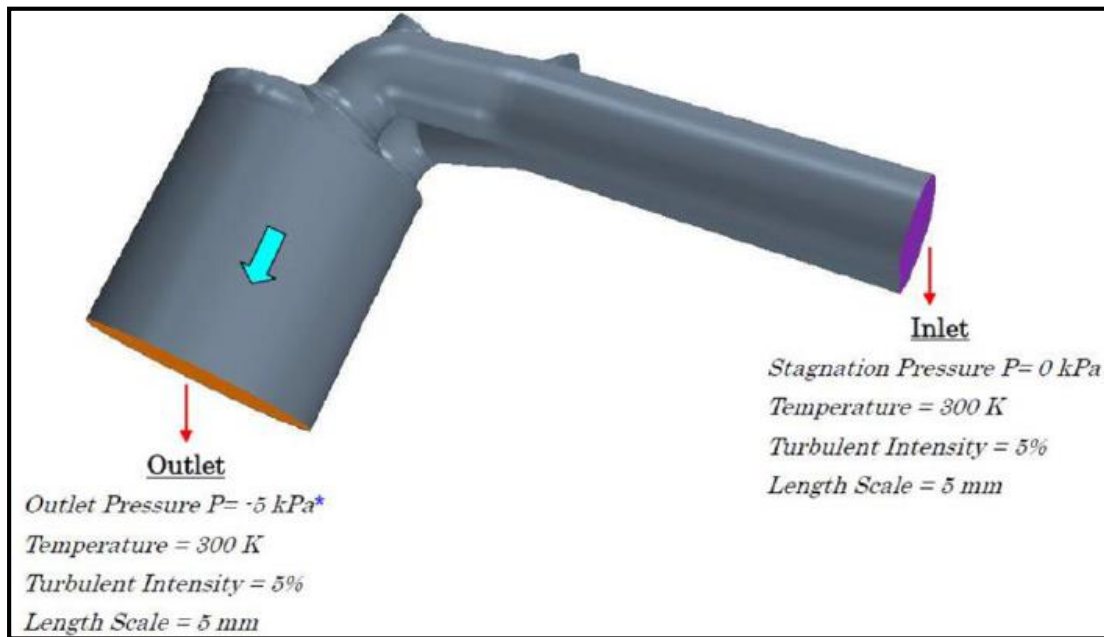
Schogl et al. (2011)

Appendix 13



Selvaraj et al. (2011)

Appendix 14



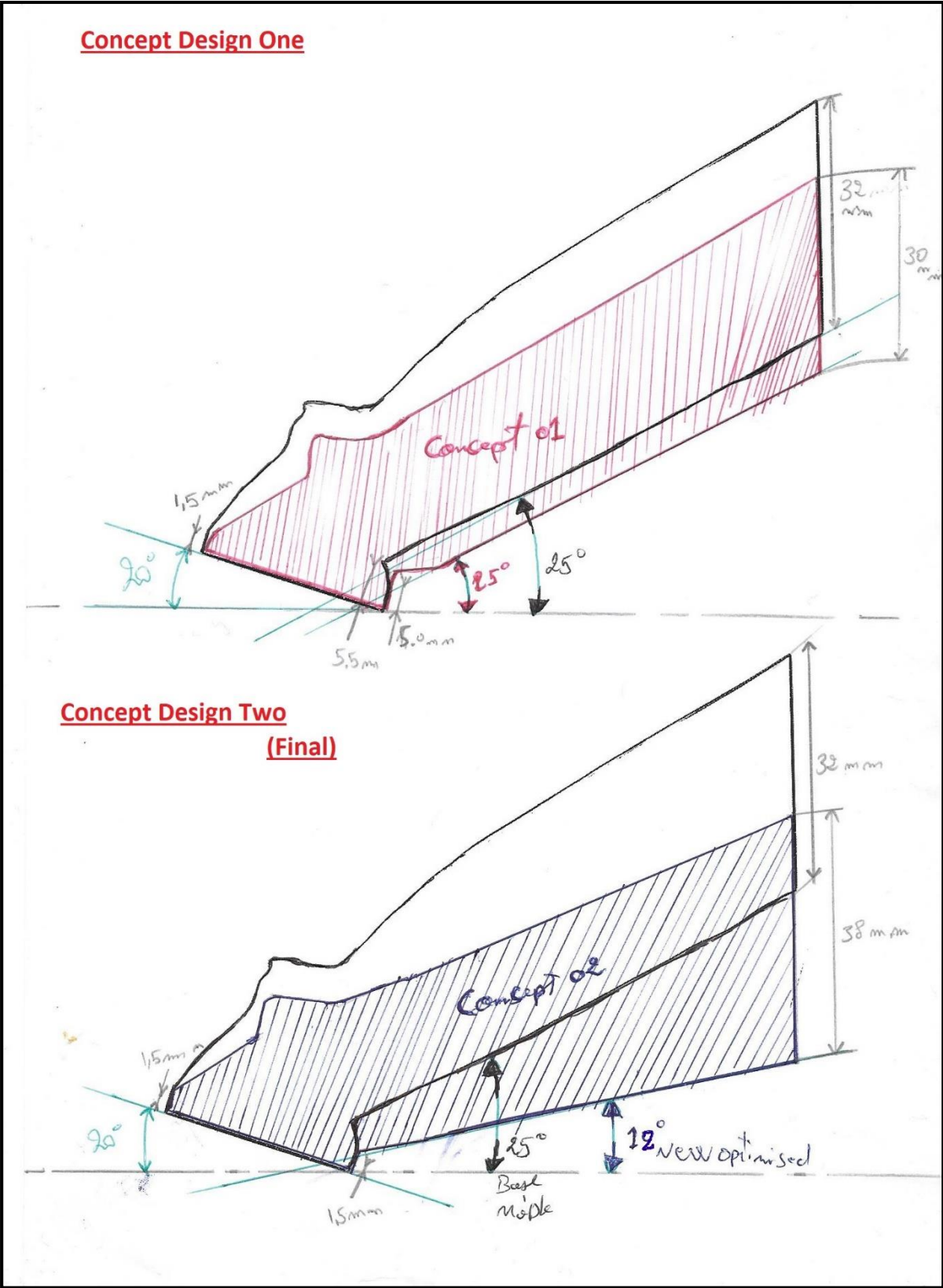
Singh et al. (2019)

Appendix 15

Bore	75.6 mm
Stroke	89 mm
Compression ratio	10.5:1
Displacement	399.5 cc
Intake valve diameter	30.5 mm
Exhaust valve diameter	25 mm
EVO	160 °afTDC
EVC	391 °afTDC
IVO	343 °afTDC
IVC	666 °afTDC
$T_{im}/T_{wall}/T_{em}$	298 K
P_{im}/P_{em}	70 kPa/100 kPa
Engine speed	1600 rev/min
Port design	A

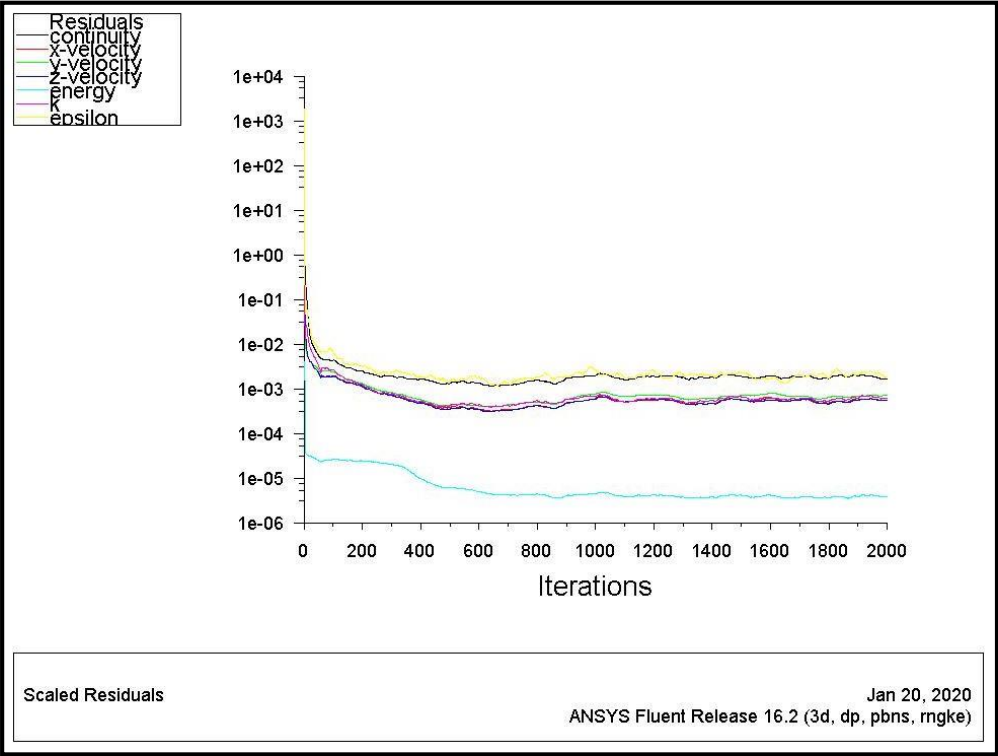
Kim et al. (2018)

Appendix 16



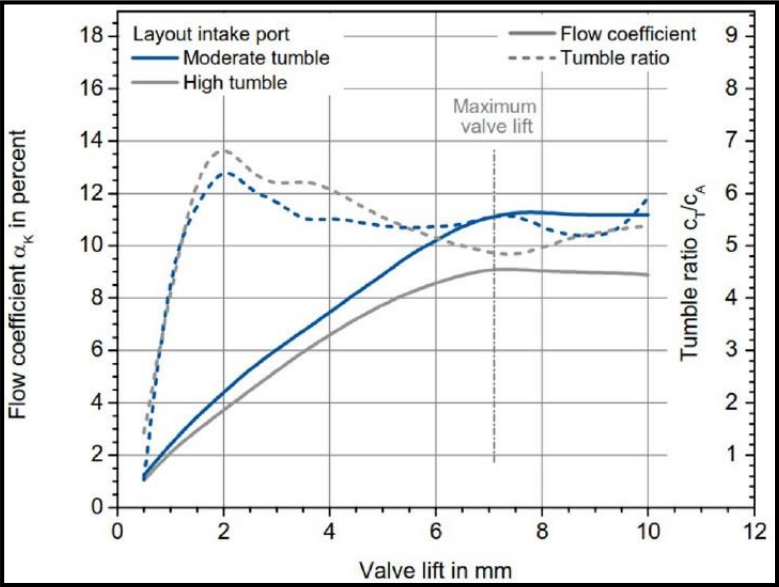
Optimisation Concepts Design (Yousouf, 2020)

Appendix 17



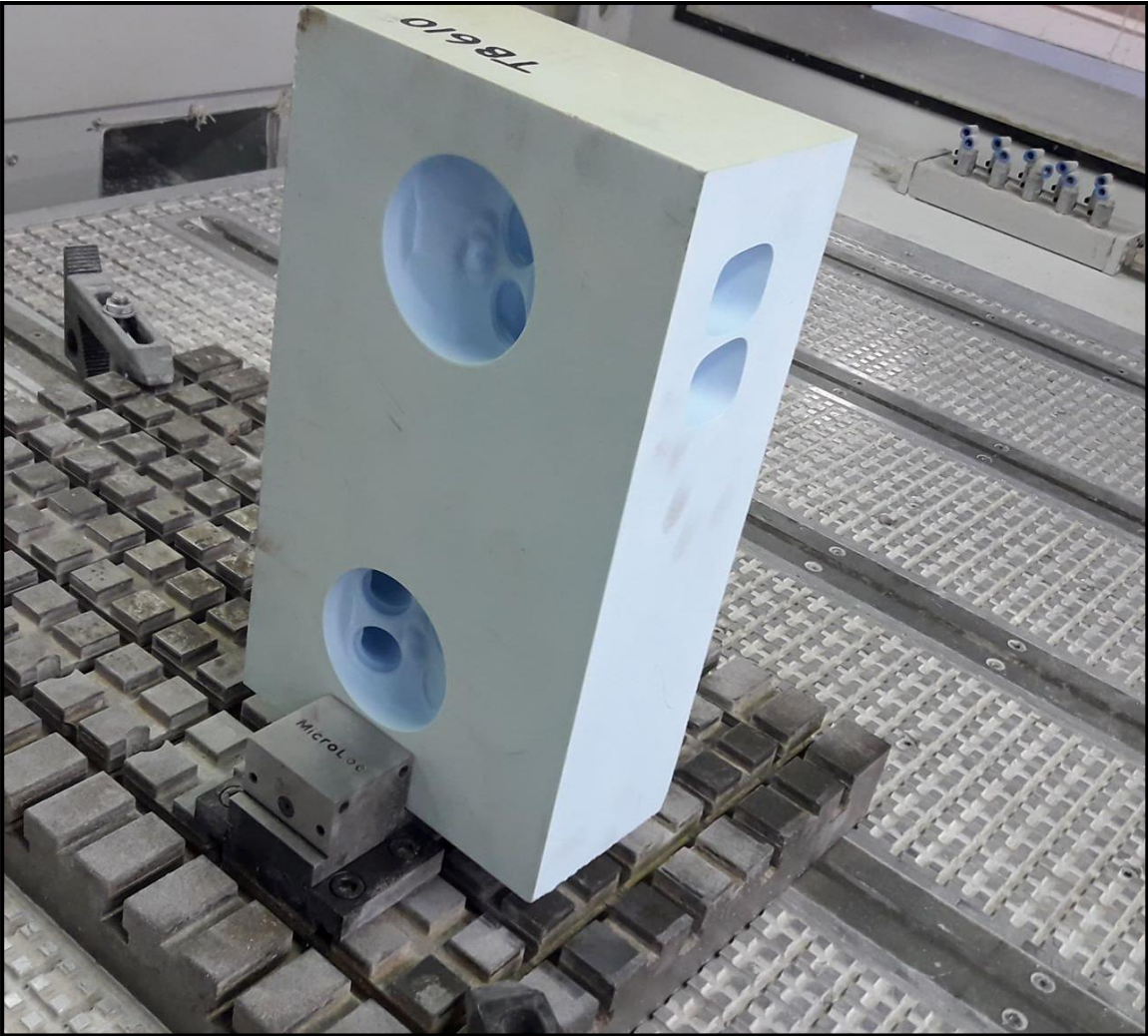
Yousouf (2020)

Appendix 18



Ye et al. (2018)

Appendix 19



Manufactured Cylinder-Head Prototype (Yousouf, 2020).