



University of Wales Trinity Saint David

School of Engineering

*The Effects on Engine Emissions and Performance through the Changes of
Cylinder Head Port Design- An Experimental Investigation Using CAD
Software.*

Christopher Robbins

P146163

B.Sc. Automotive Engineering

23/04/18

Research Methods & Project M3X9486

Word Count: 16,500

Abstract

The process of understanding volumetric efficiency within an internal combustion engines' cylinder head port. Finding alternate means including turbocharging, supercharging, valve overlap/timing and different port modification. Understanding, all the variables within an engine operating cycle that effects volumetric efficiency e.g. bore, stroke, valve/s, temperature and port dimensions.

Finding that equipment used e.g. flow bench, rubber silicone mould- has its own disadvantages and that results collected will not be to the highest standard. The fact testing has been conducted once per test will give room for error.

Through the literature review vital information, for improving volumetric efficiency, was found in relation to inlet port modification. Also, the research spread that turbocharging and supercharging as well as raising the RPM range is best for improving volumetric efficiency overall but cost and reliability comes into play against port modification and testing to improve driveability, emissions and performance.

The modifications made through port design (1st modification) increased CFM readings while maintaining a good air velocity but the 2nd modification did not really increase the CFM readings throughout valve lifts compared to the 1st modification but velocity was affected in a negative way. Through research and testing it was to be the 1st modified port that flowed the best without affecting inlet air velocity in a negative manner.

Acknowledgements

To show a great appreciation to every one of the lecturers and staff at the University of Wales Trinity Saint David. Without the resources and knowledge shown by the staff, none of the work thus far would have been possible. A close thank you to Mr Andrew Gibson, Mr Andrew Harrison, Mr Tim Tudor and to Gareth and Dave with their help in the workshop. An additional thank you to the close friends and family who have been with me throughout.

List of Figures and Tables

FIGURE 1: A LIST OF FACTS ABOUT THE 1.0-LITRE ECOBOOST ENGINE (MEDIA.FORD.COM, 2018).....	11
FIGURE 2: HOW DIFFERENT PORT DESIGNS INCREASE EITHER SWIRL, TUMBLE OR SQUISH EFFECTS (SCH\ARGL, O., EDTMAYER, H., SCHMIDT, S., LEIBER, S).	12
FIGURE 3: SINGLE POINT/INDIRECT, MULTIPOINT/INDIRECT AND DIRECT INJECTION APPLICATIONS (SENDYKA AND NOG, 2018).....	13
FIGURE 4: CHANGE IN THE AMOUNT OF AIR AND FUEL PARTICLES FOR THE RATIO OF VOLUMETRIC EFFICIENCY (ENGINELABS, 2017).....	14
FIGURE 5: TURBOCHARGER SHOWING FLOW OF EXHAUST GASES AND INLET AIR (BRIGHTHUB ENGINEERING, 2017).	15
FIGURE 6: THREE TYPES OF SUPERCHARGERS- ROOTS, TWIN SCREW AND CENTRIFUGAL. (MECHANICAL BOOSTER, 2017).....	16
FIGURE 7: VALVE DURATION AND OVERLAP IN DEGREES. (ANON, 2017).	17
FIGURE 8: VERTICALLY PLACED INLET PORTS WITH A SPHERICAL PISTON CAVITY (KUWAHARA, K., WATANABE, T., TANADA, H., UEDA, K-2000).	18
FIGURE 9: TABLE OF PORT LOCATION (VERTICAL VS HORIZONTAL) WITH RELATION TO CLOCKWISE AND/OR ANTI- CLOCKWISE FLOW. (VASHISHTHA, A., RATHINAM, B., DELAHAYE, L., RAVET- 2012).	19
FIGURE 10: SPRAY OF FUEL ATOMS WITHIN THE AIR FLOW OF A PORT (PRATTE, 2017).	19
FIGURE 11: TWO TYPE OF EQUIPMENT USED TO MEASURE SURFACE ROUGHNESS (OLYMPUS-IMS.COM, 2018).....	24
FIGURE 12: ORIGINAL INLET PORT FLUID VOLUME.....	25
FIGURE 13: THE CYLINDER BORE USED IN THE SIMULATION TESTING. THE DIMENSIONS ARE ALL IN RELATION TO THE ACTUAL BORE OF THE 1600CC FORD ZETEC ENGINE.....	26
FIGURE 14: INLET MANIFOLD AND CFD TESTING WITHIN SOLIDWORKS (HAWKRIDGESYS.COM, 2018).	27
FIGURE 15: PROCESS FROM THE SCAN DATA OF A 3D SCANNER (LASERDESIGN.COM, 2018).....	28
FIGURE 16: SHOWING THE POINT CLOUD DATA CONVERTED INTO A MESH THAT CAN BE USED TO CREATE A SOLID- BODY.	32
FIGURE 17: THE INLET PORT AS SHOWN AS A SOLID-BODY WITHIN SOLIDWORKS AFTER THE MESH HAS BEEN LOFTED.	33

FIGURE 18: SHOWING THE PROCESS OF USING THE MOULD AS A CAVITY IN A READY-MADE CYLINDER HEAD TO CREATE THE PORT.....	34
FIGURE 19: DRAWING VIEW OF THE CYLINDER HEAD WITH VALVES AND BORE. A PICTURE OF THE VALVE CREATED IN SOLIDWORKS.....	34
FIGURE 20: THESE PICTURES SHOW THE FLUID VOLUME OF THE INLET TRACK AND INTO THE CYLINDERS BORE. THE INLET VALVES CAN BE SEEN TO BE OPEN ALLOWING THE TRANSFER OF FLUID INTO THE BORE.	35
FIGURE 21: FLOW AREA OF INLET PORT AND CYLINDER BORE WITH VALVES OPEN AT 2MM OF LIFT (SHOWING COMPUTATIONAL DOMAIN).....	36
FIGURE 22: SIDE AND ANGLED VIEW OF THE STANDARD PORT AND MODIFIED PORT WITH ITS FLUID VOLUME SEEN.	36
FIGURE 23: PORT SPLIT DIAMETER AND INNER DIAMETER OF THE VALVE SEAT FOR THE FIRST MODIFIED PORT. ...	37
FIGURE 24: PORT SPLIT DIAMETER AND INNER DIAMETER OF THE VALVE SEAT FOR THE SECOND MODIFIED PORT.	37
FIGURE 25: LIVE FLOW BENCH DATA AND POTENTIAL FLOW DATA (CFM/SQIN) AGAINST VALVE LIFTS.....	40
FIGURE 26: THREE DESIGNS OF THE INLET PORT- STANDARD PORT, STRAIGHT PORT (NO CHANGE IN SIZE) AND STRAIGHT PORT WITH AN ADDITIONAL 2MM IN DIAMETER AT THE PORT SPLIT.	41
FIGURE 27: DATA PLOTTED OF ORIGINAL SIMULATED PORT AND POTENTIAL CFM AGAINST VALVE LIFTS.	42
FIGURE 28: GRAPH SHOWING CFM READINGS FOR THE FIRST MODIFIED PORT IN THE SIMULATION SOFTWARE AND POTENTIAL AGAINST VALVE LIFTS.	43
FIGURE 29: GRAPH SHOWING CFM READINGS FOR THE SECOND MODIFIED PORT IN THE SIMULATION SOFTWARE AND POTENTIAL AGAINST VALVE LIFTS.	44
FIGURE 30: GRAPH SHOWING CFM RESULTS OF THE STANDARD PORT, 1ST MODIFIED PORT, 2ND MODIFIED PORT AND POTENTIAL AGAINST VALVE LIFTS.	45
FIGURE 31: SHOWING THE PRESSURE DROP BETWEEN THE START OF THE PORT THROUGH TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- ORIGINAL PORT.	46
FIGURE 32: SHOWING THE VELOCITY BETWEEN THE START OF THE PORT TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- ORIGINAL PORT.	46

FIGURE 33: SHOWING THE VELOCITY BETWEEN THE START OF THE PORT TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- 1 ST MODIFIED PORT.	47
FIGURE 34: SHOWING THE PRESSURE DROP BETWEEN THE START OF THE PORT THROUGH TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- 1 ST MODIFIED PORT.	47
FIGURE 35: SHOWING THE PRESSURE DROP BETWEEN THE START OF THE PORT THROUGH TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- 2 ND MODIFIED PORT.	48
FIGURE 36: SHOWING THE VELOCITY BETWEEN THE START OF THE PORT TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- 2 ND MODIFIED PORT.	48
FIGURE 37: THE ORIGINAL SIMULATED PORT WITH A 2MM INCREASE IN VALVE DIAMETER (32MM) AND POTENTIAL CFM AGAINST VALVE LIFTS.	51
FIGURE 38: SHOWING THE PRESSURE DROP BETWEEN THE START OF THE PORT THROUGH TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- ORIGINAL PORT WITH BIGGER VALVES (32MM).....	52
FIGURE 39: SHOWING THE VELOCITY BETWEEN THE START OF THE PORT TO THE BOTTOM OF THE CYLINDER. ENVIRONMENTAL PRESSURE IS STATED AT THE ENTRANCE AND 25 INCHES H ₂ O LESS IS STATED AT THE BOTTOM OF THE CYLINDER BORE- ORIGINAL PORT WITH BIGGER VALVES (32MM).....	52
FIGURE 40: CALCULATION FOR INLET RUNNER LENGTH (EXX.SE, 2017).	54
FIGURE 41: THEORETICAL HORSEPOWER CALCULATED BY USING BOTH LIVE AND SIMULATED FLOW READINGS..	55
FIGURE 42: CALCULATED VOLUMETRIC EFFICIENCY FOR THE LIVE AND SIMULATION TEST DATA AT DIFFERENT VALVE LIFTS.	56
FIGURE 43: CALCULATED VOLUMETRIC EFFICIENCY USING SIM FLOW READINGS AND POTENTIAL AT DIFFERENT VALVE LIFTS.	56
FIGURE 44: CALCULATED THEORETICAL CFM AND POTENTIAL CFM PLOTTED AGAINST VALVE LIFT.	58
FIGURE 45: TWO GRAPHS DEPICTING ALL TESTED CFM VALUES AND CALCULATED WITH THE POTENTIAL AGAINST VALVE LIFT.....	59

TABLE 1: EURO EMISSION STANDARD AND DATES.	23
TABLE 2: MATERIAL SPECIFICATION OF THE FORD ZETEC ALUMINIUM CYLINDER HEAD.	25
TABLE 3: MATERIAL SPECIFICATION OF THE FORD ZETEC INLET VALVE.	25
TABLE 4: MATERIAL SPECIFICATION FOR FORD ZETEC CYLINDER BLOCK BORE.	26
TABLE 5: INITIAL GANTT CHART FROM THE BEGINNING OF THE FIRST SEMESTER. SHOWING INITIAL AIMS OF THE PROJECT.....	29
TABLE 6: VALVE DIMENSIONS AND EFFECTIVE VALVE AREA OF THE INLET VALVE.	38
TABLE 7: VALVE LIFT CALCULATED FOR THE PERCENTAGE OF VALVE DIAMETER.	38
TABLE 8: THE FLOW-BENCH RANGES WITH THE CONVERSION NUMBERS.	39
TABLE 9: DATA COLLECT FROM THE FLOW-BENCH TESTING AT EACH VALVE LIFT. SHOWING THE TEST CFM AND THE CFM WITH THE USE OF THE EFFECTIVE VALVE AREA.	39
TABLE 10: LIVE AND POTENTIAL CFM READINGS.	40
TABLE 11: CFM READINGS FOR ORIGINAL SIMULATED PORT AND POTENTIAL AGAINST VALVE LIFTS.	42
TABLE 12: CFM READINGS FOR FIRST MODIFIED SIMULATED PORT AND POTENTIAL AGAINST VALVE LIFTS.	43
TABLE 13: CFM READINGS FOR FIRST MODIFIED SIMULATED PORT AND POTENTIAL AGAINST VALVE LIFTS.	44
TABLE 14: TABLE SHOWING CFM RESULTS OF THE STANDARD PORT, 1ST MODIFIED PORT, 2ND MODIFIED PORT AND POTENTIAL AGAINST VALVE LIFTS.	45
TABLE 15: CFM READINGS FOR EACH VALVE LIFT FOR THE SIMULATED PORT WITH AN INCREASE IN VALVE SIZE AND POTENTIAL CFM.	51
TABLE 16: THEORETICAL HORSEPOWER CALCULATED AT EACH VALVE LIFT FROM THE LIVE AND SIMULATED FLOW TEST SESSIONS.	55
TABLE 17: CALCULATED VOLUMETRIC EFFICIENCY FROM USING LIVE FLOW READINGS AND POTENTIAL AT DIFFERENT VALVE LIFTS.	56
TABLE 18: VALVE AND SEAT DIMENSIONS FOR CALCULATING THEORETICAL CFM.	57
TABLE 19: CALCULATED CURTAIN AREA FOR EACH VALVE LIFTS.	58
TABLE 20: CALCULATED THEORETICAL CFM FOR EACH TESTED VALVE LIFT.	58
TABLE 21: ALL CALCULATED, LIVE AND SIMULATED DATA FOR FLOW RATE AT THE SAME VALVE LIFTS.	59

Table of Contents

Abstract.....	1
Acknowledgements	2
List of Figures and Tables	3
Table of Contents.....	7
1. Introduction	10
1.1 The importance to industry.	11
1.2 The aims of the project.....	11
2. Literature review.....	12
2.1 Background	12
2.2 Fundamentals of an Inlet Port	13
2.3 Volumetric Efficiency	14
2.4 Methods to increase volumetric efficiency within a cylinder head	15
2.4.1 Turbocharging.....	15
2.4.2 Supercharging	16
2.4.3 Valve timing/overlap	17
2.4.4 Port design and modification	18
2.5 Draw backs to each method of improving volumetric efficiency.....	20
2.5.1 Turbocharging.....	20
2.5.2 Supercharging	20
2.5.3 Valve timing/overlap	21
2.5.4 Port design and modification	21
2.6 The important of volumetric efficiency within the automotive industry in relation to emissions, health issues and legislation.	22
2.6.1 Emissions	22
2.6.2 Health Issues	22
2.6.3 Legislation	23
2.7 Material Research- Cylinder Head Components.....	24
2.7.1 Surface Roughness.....	24
2.7.2 Cylinder Head- Intake Port	25
2.7.3 Inlet Valves	25

2.7.4	Cylinder Block- Bore	26
2.8	Software	27
2.8.1	Computational Fluid Dynamics- Solidworks Flow Simulation	27
2.9	Hardware	28
2.9.1	Flow Bench	28
2.9.2	3D Scanner.....	28
3.	Methodology.....	29
3.1	Initial test plan- Gantt chart.....	29
3.2	Process of inlet port to CAD inlet port.....	30
3.2.1	Process of making a mold of an inlet port	30
3.2.2	Collection of initial flow bench data of the standard inlet port.....	31
3.2.3	Scanning the inlet mold to acquire x, y, z point cloud data	32
3.2.4	Converting the point cloud data into a mesh in Solidworks	32
3.2.5	Processing the mesh into a solid working body within Solidworks.....	33
3.2.6	Creating cylinder head with inlet port, valves and valve seats	34
3.3	Process of collecting initial simulated data	35
3.3.1	The use of Xpress Flow in Solidworks	35
3.3.2	The change in use from Xpress Flow to Solidworks Flow Simulation.....	35
3.3.3	Process of using Solidworks Flow Simulation	36
3.4	Differences between the standard and two modified ports.....	37
3.4.1	Modified Port 1- Diameter at port split and at valve seat	37
3.4.2	Modified Port 2- Diameter at port slit and at valve seat	37
4.	Discussion.....	38
4.1	Live data.....	38
4.1.1	Valve calculations.....	38
4.1.2	Live flow bench data.....	39
4.2	Simulation data	41
4.2.1	Original Simulated Port CFM against Potential CFM	42
4.2.2	1 st Modified Simulated Port CFM against Potential CFM.....	43
4.2.3	2 nd Modified Simulated Port CFM against Potential CFM	44
4.2.4	Comparison between the standard port, 1 st modified port, 2 nd modified port and potential CFM ..	45

4.2.5	Original Simulated Port Pressure and Velocity trace	46
4.2.6	1 st modified Simulated Port Pressure and Velocity trace	47
4.2.7	2 nd modified Simulated Port Pressure and Velocity trace	48
4.2.8	Analysis and Evaluation of Velocity and Pressure traces for the standard, 1 st modified and 2 nd modified inlet ports.....	49
4.2.9	CFM, Velocity and Pressure trace for a 2mm increase in valve diameter.	51
4.2.10	Analysis and Evaluation of Velocity and Pressure traces for the standard port with the increase in valve diameter (30mm-32mm).	52
4.3	Calculated data	54
4.3.1	Pulse Tuning- Calculating optimum intake runner length for a specific RPM	54
4.3.2	Theoretical Horsepower.....	55
4.3.3	Calculated Volumetric Efficiency.....	56
4.3.4	Calculated CFM.....	57
4.4	Live, Simulated and Calculated CFM comparison.....	59
5.	Conclusion	61
5.1	Live data.....	61
5.2	Simulation data	62
5.3	Calculated data	63
5.4	Recommendations for improvement	64
5.4.1	Cylinder head.....	64
5.4.2	Engine and additional components	65
5.4.3	Testing	65
5.5	Overall conclusion	66
6.	References	67
7.	Appendices	70

1. Introduction

This major project report considers the design and efficiency of inlet ports in automotive internal combustion engines (ICE). The effects that port design has on volumetric efficiency, pressure changes and velocity will be examined and evaluated to see the limiting factors of overall performance of the internal combustion engine.

Using Solidworks (CAD Software), a standard port is tested, evaluated and redesigned to increase the volumetric efficiency throughout incremented valve lifts. Taking in mind that the inlet window is a very short period with most valves only opening to a maximum of 12% of an engine bore diameter the changes will have to benefit the engine at these specific increments.

As the chosen cylinder head is from the automotive realm that has been designed to meet emission regulation as well as supplying high power outputs at lower revolutions per minute (RPM) to achieve better driveability, the inlet port will only be modified slightly regarding removing any casting marks and to achieve a smoother port with a lower degree of roughness. An area of research will be through the changes of port design from the perspective of performance, emissions and driveability.

To achieve initial live data a flow bench is used to find out how well the 4-valve head flowed through-out different valve lifts (1.5mm-12mm) in 1.5mm increments. This gives a starting point when comparing the simulation data and calculated data. Similar readings through the live, simulation and calculations give a great credibility when proving and explaining upgrades to the inlet port design.

A mold of the inlet port is taken so that it can be replicated in Solidworks for simulation testing and improvements made to a credible standard. Allowing modifications to be made within the software cuts down time, money and resources and gives an insight on where enhancements can be made.

1.1 The importance to industry.

As Euro emission regulations are becoming stricter manufactures must find alternative ways to reduce engine emissions. One alternative is to reduce engine size but a reduction in engine size means a reduction in performance and this is seen a backwards step to the public when considering purchasing a new car.

This brings the want and need of upgrades to achieve a higher performing engine with a lower capacity and lower emissions. Upgrades such as turbo/supercharging, longer strokes and direct injection can achieve a higher performance from a smaller capacity engine and this has been successfully undertaken in industry e.g. Ford's Award Winning 1.0-Litre EcoBoost engine. The EcoBoost combines a longer stroke for a reduction in friction, a turbocharger for added performance (volumetric efficiency) and direct injection for a precise and minimum amount of fuel usage possible to achieve its great emission levels and miles per gallon MPG (Media.ford.com, 2018).

Ford 1.0-litre EcoBoost petrol engine named 2017 International Engine of the Year “Best Engine Under 1.0-litre” for the sixth year in a row.
Fuel-efficient, compact and powerful engine has won 10 International Engine of Year awards since launch in 2012; powers one in five Ford vehicles sold in Europe
Ford to offer 1.0-litre EcoBoost with innovative cylinder deactivation technology for further reduced CO2 emissions and improved fuel-efficiency from early 2018.

Figure 1: A list of facts about the 1.0-Litre EcoBoost Engine (Media.ford.com, 2018).

1.2 The aims of the project.

- To increase the volumetric efficiency of a cylinder head (Ford Zetec 1.6, 16 valve) by modification of inlet port design.
- To not effect engine emissions, driveability and performance in a negative way when the inlet port is modified, through mathematical calculations, and live and simulated data.

2. Literature review

This section will look through alternative methods to increase volumetric efficiency in an ICE (Internal Combustion Engine) with emissions and performance being the main factors. A list of the fundamental positive and negative effects these alternative methods have upon the overall application will be shown.

An in-depth investigation of the way port design can affect the air entering the combustion chamber regarding velocity at alternative valve lifts and the way velocity and turbulence do not come hand in hand. The size, shape, path and material used for automotive inlet ports will show how detrimental the design process is when achieving an effective flowing port through valve lifts and maintaining a high volumetric efficiency. The process and equipment used will be looked at and examined to show how inlet ports can be analysed and modified.

2.1 Background

The design of cylinder head ports has changed dramatically over the years of the internal combustion engine. The changes include materials used such as cast iron and aluminium, and port shapes/design e.g. round, square style. Different port designs effect the way the air travels through it to the combustion chamber as seen in the diagram (Smith and Wenner, 1977).

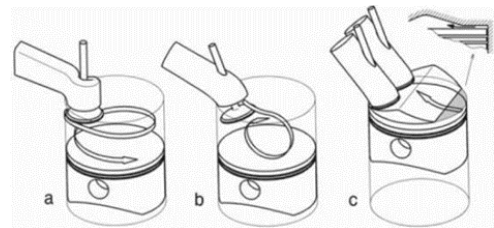


Figure 2: How different port designs increase either swirl, tumble or squish effects (Sch\argl, O., Edtmayer, H., Schmidt, S., Leiber, S).

An inlet ports size has a direct correlation to the size of an engine. This is to maintain broad flow across the RPM range and to avoid choking (ports too small for engine size) and poor driveability/ low torque (ports too big for engine size). Additionally, port design can affect the way the air and fuel mix and the velocity that it travels through the port to the cylinder and the total volumetric efficiency (Vizard, D, 2012).

2.2 Fundamentals of an Inlet Port

An inlet ports task, in automotive terms, is to act as a passage to guide atmospheric air to behind the inlet valve where once opened the air can enter via a difference in pressure (Atmospheric pressure and a negative pressure creates a vacuum to draw the air into the combustion chamber). The port on an automotive engine is usually situated between inlet manifold and the cylinder bore of the block.

More prominence has been placed on port design by the recurrence of the direct injection petrol engines and inlet port design is as well influenced by external elements, such as packaging, valve/valve train design, volume and application (Mohammadebrahim, A., Hannani, S.K. and Shafii, B., 2014).

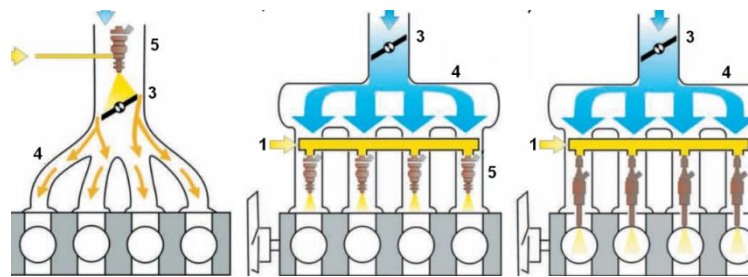


Figure 3: Single point/indirect, multipoint/indirect and direct injection applications (Sendyka and Nog, 2018).

On indirect and carburation type applications the fuel also must travel through the inlet port and different parameters are needed for these type of port applications. In the past, with carburation units, inlet ports needed to be relatively rough to create a turbulent air flow which led to better emulsifying of the air and fuel to achieve as close to a stoichiometric ratio as possible. The same principle applies to indirect fuel injection engines dependant on where in the inlet system the injector is positioned i.e. injector position by throttle butterfly or behind the inlet valve (Vizard, D, 2012).

An injector that is placed right behind the intake valve is still indirect but the fuel doesn't travel the length of the port to arrive at its destination. This gives more freedom to port design and to create a smoother surface for high air velocity and less risk of carbon build up.

2.3 Volumetric Efficiency

The term volumetric efficiency refers to the ratio of the volume of air that is theoretically capable of being consumed into the cylinders combustion chamber (potential) against the actual air volume that makes it into the cylinders combustion chamber, with all influences reflected. (Smith, L., Fickenscher, T., and Osborne, R- 1999). Restrictions such as valve size, port size, carburettor and throttle body size etc. affect the ratio of the actual volume of air entering the cylinder (Baechtel.J, 2011).

$$VE \% = (\text{Measured CFM} \div \text{Potential CFM}) \times 100$$

(Baechtel.J, 2011)

Most automotive production engines achieve a volumetric efficiency is around 70 to 80%, but on higher performance vehicles 80% to the mid 90's has become well achievable. Highly tuned race engines, for example drag racing engines, regularly surpass 100% volumetric efficiency because of the use of heavily modified intake systems, exhaust scavenging and cam timing (e.g. 130% efficient = air is 30% more dense than that of the air entering the system), (Burgess and Gollan, 2010).

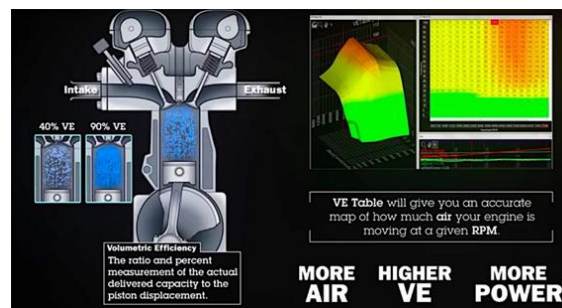


Figure 4: Change in the amount of air and fuel particles for the ratio of volumetric efficiency (EngineLabs, 2017).

2.4 Methods to increase volumetric efficiency within a cylinder head

2.4.1 Turbocharging

Turbocharging has been around since the early 20th century, when Alfred Buchi a Swiss engineer was the first to use exhaust gases to power an engine turbine and this brought a steady introduction of turbocharging into the automotive industry (Morotta. J, 2010).

A turbocharger uses the spent combustion gases (exhaust gases) to power a turbine on one section of the turbocharger unit. The turbine is connected to the compressor wheel via a shaft, sealed at each end, which draws the ambient air in and compresses it (Davis and Perkins-Davis, 2001). As the air is compressed the temperature increases and so the density of the air decreases which is not desired for an engine so the compressed air is directed through an intercooler which acts as a radiator (Stone, 1989).

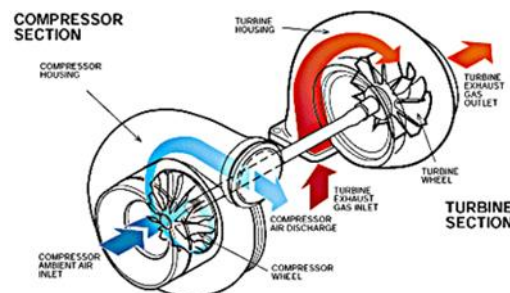


Figure 5: Turbocharger showing flow of exhaust gases and inlet air (Brighthub Engineering, 2017).

The air exiting the turbocharger doesn't reach back to ambient temperature but does decrease significantly, thus increasing the density of the inlet air and reducing the temperature. This air entering through the inlet port and into the combustion chamber increases the volumetric flow-rate/efficiency. (Charge air after cooling, 2017).

2.4.2 Supercharging

The use of a supercharger requires a direct connection to the engine to power it usually in the form of a belt which takes power from the engine to achieve more power. A supercharger increases the velocity of air travelling to the combustion chamber but as no need for exhaust gases the inlet air is much cooler and thus the use of an intercooler is not needed (Servati, H. and DeLosh, R).

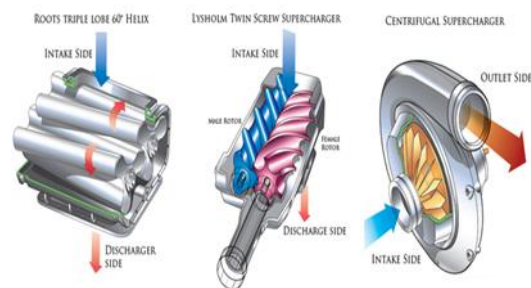


Figure 6: Three types of superchargers- roots, twin screw and centrifugal. (Mechanical Booster, 2017).

There are three main types of superchargers from the roots triple lobe helix, twin screw and the centrifugal type. The roots type like the twin screw rotates so that the air is forced into smaller spaces where it is then compressed and exited at a higher velocity. A centrifugal type looks like a turbocharger on the outside and contains a compressor wheel also. It is the most efficient out of the 3 because of the size and load taken on the engine (Dietsche, K. (2011).

2.4.3 Valve timing/overlap

The function of valve overlap is to assist the exiting process of the exhaust gases by opening the inlet valve early when the exhaust valve is still open. By the upwards movement of the piston and the valve overlap acts as a vacuum which draws more air into the combustion chamber which in turn drives the spent gases out into the exhaust manifold (Baechtel. J, 2011). This process can also help raise the volumetric efficiency by pulling in more air into the combustion chamber. As seen in the depiction, the inlet valve opens at 46° BTDC and the exhaust valve closes at 43° ATDC giving a total valve overlap of 89° . An engine with high lift cams and long cam durations will exaggerate this degree of overlap allowing for more time for the inlet air to enter the combustion chamber and exhaust gases to exit (Vizard. D, 2012).

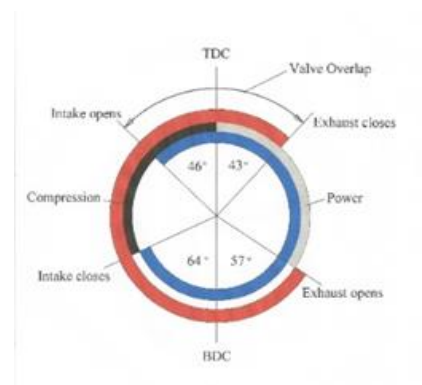


Figure 7: Valve duration and overlap in degrees. (Anon, 2017).

2.4.4 Port design and modification

An inlet ports design is essential for achieving the best possible flow for the inlet air but another side is to create a tumbling effect for air and fuel mixture. Although, on direct injection engines the air and fuel don't mix until the inlet valve has started to open which means a cylinder heads port on a direct injection engine can be made very smooth where all the casting marks and imperfections are removed (Articles.sae.org- 2017).

On an indirect or carburation fuel type system, the air and fuel both travel through the inlet port so turbulence is needed for better atomization of the air and fuel as a smooth port would create puddling of the fuel thus increases emissions- unburnt fuel/hydrocarbons and an increase in CO. The use of sequential injection and the placement of an injector closer to the valve eliminates puddling and therefore the rise in emissions (Malkhede, D. and Khalane, H- 2015).

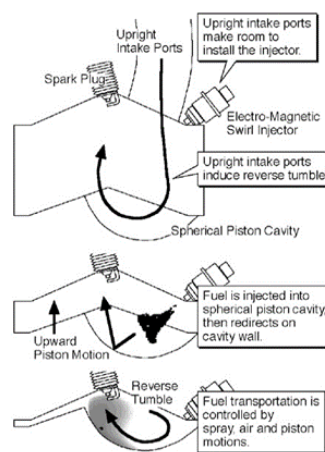


Figure 8: Vertically placed inlet ports with a spherical piston cavity (Kuwahara, K., Watanabe, T., Tanada, H., Ueda, K-2000).

A technical paper by Mitsubishi shows an actual change in port location instead of design, where the inlet port is orientated vertically instead of horizontally apposed as on most internal combustion engines.

This allows a straighter port to the combustion chamber which in-turn can be used to direct the air to a specific design of piston crown e.g. spherical piston cavity.

The air entering does not need a rough port surface to create turbulence for the air and fuel to mix as the injector is placed internally this creates a better flowing port but in-turn the whole design allows for a tumbling effect within the combustion chamber. This allows a better atomization of the fuel within the air and a better coverage of the combustion chamber allowing for a more efficient burn and flame front (Kuwahara, K., Watanabe, T., Tanada, H., Ueda, K-2000).

A study by Renault for intake port design/location found that intake entry, in-line with the cylinder axis, had better performance in relation to the total mass flow rate and permeability compared to horizontally opposed ports. As a percentage, the flow was found to be approximately 7 to 9 % and 7.7 to 9.6 % higher than inlet entry, of a port design perpendicular to cylinder axis for different pressure drop conditions (Vashishtha, A., Rathinam, B., Delahaye, L., Ravet- 2012).

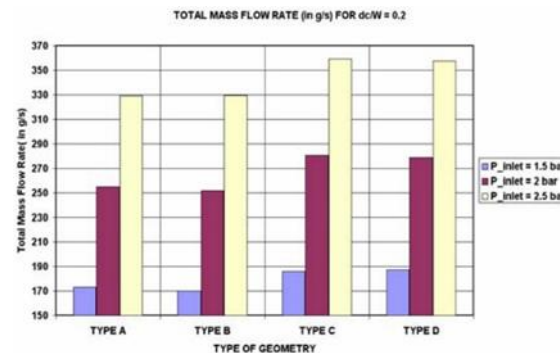


Figure 9: Table of port location (vertical vs horizontal) with relation to clockwise and/or anti-clockwise flow. (Vashishtha, A., Rathinam, B., Delahaye, L., Ravet- 2012).

Figure 9 shows that horizontally apposed ports (Type A & B) have a lower mass flow rate than that of vertically positioned ports (Type C & D). The direct of flow i.e. clockwise or anti-clockwise makes no significant difference to the flow. Testing was completed on a 2-stroke, 2-cylinder engine for cost and manufacturing time/cost.

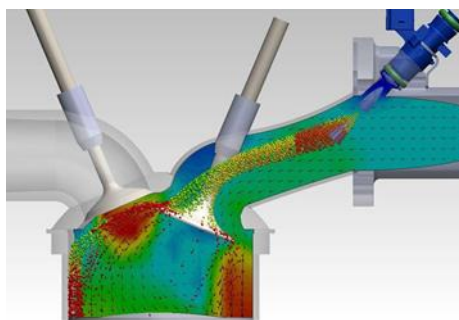


Figure 10: Spray of fuel atoms within the air flow of a port (Pratte, 2017).

2.5 Draw backs to each method of improving volumetric efficiency

2.5.1 Turbocharging

When it comes to draw backs of turbocharging there are a few. As turbochargers use exhaust gases to power them, this creates unwanted heat. With inlet air temperature, you want it to be as cold/dense as possible for a better combustion (Power boost technology- 2007).

This disadvantage means more components are needed to try and reduce the turbochargers outlet air temperature including pipe work and an intercooler. These additional components and possible internal upgrades (to make sure engine withstands the increase boost) makes the cost of this upgrade difficult for some Charge air after cooling- 2017).

2.5.2 Supercharging

The same as said for turbocharging can be related to supercharging in the sense of cost. Although superchargers don't necessarily need intercoolers (compressing air and a rise in temperature is directly related) this doesn't make them a cheaper alternative. The main disadvantage for superchargers are that they are run directly off of the engine so to power them they take vital engine power to supply additional power back. Superchargers are also usually left for higher output engines where the disadvantage of taking power isn't felt as much as smaller capacity engines (Tingwall, E- 2017).

2.5.3 Valve timing/overlap

An upgrade for change in valve timing with an increase in valve overlap can be a much cheaper option than the previous upgrades stated. The reason for this can be placed down to only the cost of the camshafts and no additional components (unless double springs are needed).

Although camshafts allow more air to enter as they may hold the valves open longer this doesn't necessarily mean an increase in volumetric efficiency as much as turbocharging and supercharging. Longer valve over-lap with lengthen the vacuum time between the exhaust and inlet cycle to increase the flow but not as efficient as boosting (Vizard. D, 2012).

2.5.4 Port design and modification

The main disadvantage to this process is that that there is no strict right or wrong way to porting. Taking in consideration to engine size and existing port size just a little smoothing out of the port can increase volumetric efficiency but could also hinder engine torque (Chen et al., 1995).

Porting is a very precise task and significant engine power can be lost if the ports shape, size and direction are changed. Between every slight change within the port it must be tested on a flow bench to see how these changes have affected flow and this task cannot be left till the end when too much modification or material is removed (Baechtel. J, 2011).

This type of upgrade is good in the sense of cost, but in time and effort a different upgrade could be chosen depending on cost.

2.6 The important of volumetric efficiency within the automotive industry in relation to emissions, health issues and legislation.

2.6.1 Emissions

As volumetric efficiency is key to a more efficient combustion which reduces emissions it should be a main part of the manufacturing process of auto-manufactures to upgrade and modify ports. Many manufactures have decreased engine size whilst increasing performance to match a bigger capacity engine. This is usually achieved by adding longer stroke bores to reduce friction, turbocharging/supercharging (increase air flow), a more efficient fuel delivery (direct injection) and modified ports for better intake flow and efficiency.

A more efficient combustion reduces the emissions that an internal combustion engine produces e.g. carbon monoxide (CO), unburned hydrocarbons (HC), nitrogen oxides (NO_x), carbon dioxide (CO₂) and particulate matter (PM) (Rizvi, 2017).

Port design can help prevent a ‘puddling’ effect of the fuel, which increases hydrocarbons. Achieving a better ratio between the air reaching the combustion chamber and the potential, will make for a more efficient combustion with less emissions being created.

2.6.2 Health Issues

A big reason to reduce emissions with internal combustion engines is because of the negative effect it poses to humans and animals. A scientific research found a relation between exhaust emission/waiting in traffic against a higher risk of developing Dementia (Alzheimer's Research UK, 2017).

As internal combustion engines create emissions this effects the quality of air which increases the health risk to the planet and its population (Union of Concerned Scientists, 2017).

2.6.3 Legislation

The biggest legislation to impact the automotive industry was the introduction of the Euro Emission Regulations which was introduced in 1992 with Euro 1 and is currently a significantly way through Euro 6. These made it compulsory for vehicles to go through emissions test during MOT testing. This process was aimed at reducing emissions and to make manufactures produce more environmentally friendly cars to meet these regulations (See Table 1).

Euro Emission Standard	Start (Year)	Ended (Year)
Euro 1	1992	1995
Euro 2	1995	1999
Euro 3	1999	2005
Euro 4	2005	2008
Euro 5	2008	2014
Euro 6	2014	+

Table 1: Euro Emission Standard and dates.

As new Euro standards were introduced the limit for specific emission gases e.g. Carbon Monoxide, Hydrocarbons, Nitrogen Oxide and Particulate matter- were significantly reduced so manufactures would have to manufacture engines that would meet these standards so that they could sell these vehicles.

‘Vehicle emissions are presently tested above the New European Drive Cycle (NEDC) chassis dynamometer process. The whole vehicle is tested and all emissions are expressed in g/km’ (In-Service performance 6/VI Vehicles, 2015).

2.7 Material Research- Cylinder Head Components

Material research is very important as different materials react differently under diverse conditions. As an example: Aluminium is a softer more malleable material compared to Cast Iron and therefore will act differently especially under temperature changes. In relation to testing the flow characteristics of an inlet port it is important to know the type of metals used, their properties and most importantly the roughness of the finished product as this will affect the velocity and the turbulence of the air through the port (Thomas, 1999).

2.7.1 Surface Roughness

Surface roughness indicates the state of a machined surface. Roughness can be measured two ways- contact and non-contact type (Groover, 2011).

The contact way uses a stylus which can go over the material in a linear or areal method to obtain the data. As the stylus is very delicate only machined surfaces can be measured and not cast surfaces like cylinder heads (Kalpakjian and Schmid, 2001).

The non-contact method can be used on slightly more coarse materials as these systems never harm the sample and can even measure soft or viscous materials as

a light is used instead of a stylus. This method would have a better chance at measuring a cast surface but as an inlet port is a cylindrical area the machine would most definitely not be able to collect data (Olympus-ims.com, 2018).

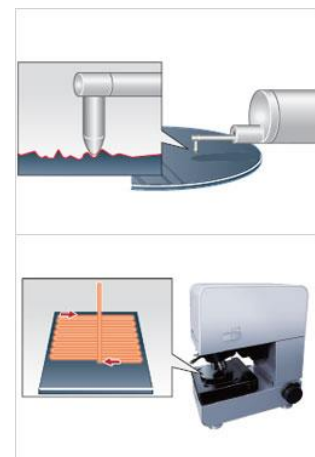


Figure 11: Two type of equipment used to measure surface roughness (Olympus-ims.com, 2018).

2.7.2 Cylinder Head- Intake Port

It is vital to specify the roughness of an intake port to get reliable and correct results when it comes the simulation testing process. The reason for this is under live conditions to original port has not been ported and/or polished so the intake track is rough and course where the cylinder head has been casted (Thomas, 1999).

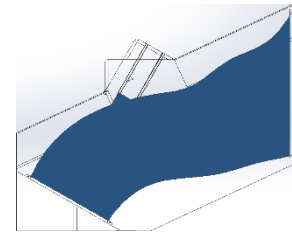


Figure 12: Original Inlet port fluid volume.

A roughness reading needs to be taken so that it can be processed in the Solidworks simulation package. When the port and cylinder head is replicated in Solidworks with the correct material specification it is still a smooth metal and doesn't take into consideration the material roughness.

The reason this is so paramount is because a highly smoothed/polished port creates less turbulence for the air flow which in turn doesn't mix the air and fuel as efficiently but can improve the velocity of the air and the volumetric efficiency. A polished port may not produce more power at low rpms because of the lack of air velocity.

Cast Aluminium Cylinder Head	201.0 T43 Insulated Mold Casting (SS)
-------------------------------------	--

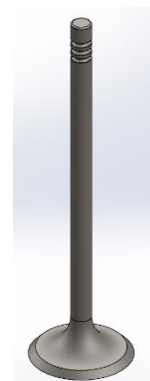
Table 2: Material specification of the Ford Zetec Aluminium cylinder head.

2.7.3 Inlet Valves

The inlet valves are another key component that restricts air flow so again it is important to specify the correct material of the valves to obtain valuable and reliable data.

High Carbon Steel Inlet Valve	AISI4340, steel Normalised
--------------------------------------	-----------------------------------

Table 3: Material specification of the Ford Zetec inlet valve.



2.7.4 Cylinder Block- Bore

The cylinder bore is less detrimental to the overall simulation results and the reliability achieved. Even though the block is cast iron and very rough the bore is honed and machined to smooth but crosshatched finish. The bore itself doesn't affect the air travelling through the port via its material compound but bore and stroke size does affect air velocity as a bigger bore allows for bigger valves and therefore a bigger port.

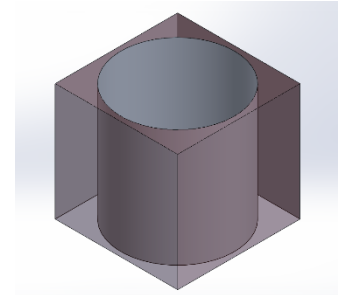


Figure 13: The cylinder bore used in the simulation testing. The dimensions are all in relation to the actual bore of the 1600cc Ford Zetec engine.

Cast Iron Cylinder Block (Bore)	Gray Cast Iron
---------------------------------	----------------

Table 4: Material Specification for Ford Zetec cylinder block bore.

2.8 Software

2.8.1 Computational Fluid Dynamics- Solidworks Flow Simulation

A software commonly used to create real life objects into a programme to be improved and adapted e.g. change in thickness, reposition of bolt holes and change of material. Also used to design completely new items from scratch to connect to another component. Components can be made from CAD files on a CNC machine for real life testing and/or use.

The Solidworks flow simulation package allows for the same testing to be made as from a flow bench but with much more variety in setting features. The flow simulation enables the component to be quickly and effectively simulated regarding fluid flow, heat transfer and fluid forces and in turn components can be compared allowing for design optimisation and greater performance (Solidsolutions.co.uk, 2018).

CFD is used to calculate the mass-flow rates, pressure drops, heat transfer rates, and fluid dynamic forces such as lift, drag and pitching moments for all sorts of components. These can be analysed on an objects external or internal features to see the objects efficiency in different areas (Concept, Product, Design, Engineering, Manufacture, Analysis, & Testing - PES - Performance Engineered Solutions, 2018).

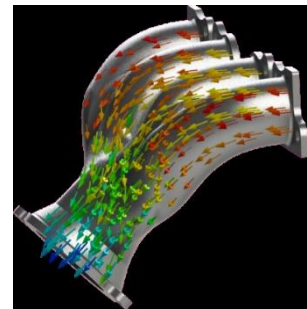


Figure 14: Inlet manifold and CFD testing within Solidworks (Hawkridgesys.com, 2018).

2.9 Hardware

2.9.1 Flow Bench

A flow bench is used in the automotive industry to test the flow capabilities of specific air flow passages but is most generally used in testing cylinder head ports and/or intakes. Typically, a flow bench draws air through to simulate the intake cycle of an engine when the piston is travelling between Top Dead Centre (TDC) and Bottom Dead Centre (BDC). This is achieved by creating a pressure difference in the ‘dummy’ cylinder bore and the external air. The pressure inside the bore is lower than that of the environmental pressure outside of the unit which in turn creates the flow of air through the port into the bore. The opposite can be said of the exhaust side as when this is tested the flow bench drives the air out of the machine and through the exhaust port.

2.9.2 3D Scanner

A 3D scanner is a great tool when the need to analyse, modify or test a live component within a software package is needed. By testing in computer software’s, the cost and time possibilities can be severely reduced due to no material wastage and if an incorrect change is made most of the time it can just be removed.

A 3D laser scanning is a non-contact, non-damaging technology that digitally captures the form of objects using a laser emitted light. Through this process, they create point clouds of data from the external surface of an object. This allows for this data to be placed into a software like Solidworks to be reformed, modified and/or tested (Laserdesign.com, 2018).

As seen in the figure to the right is the different routes to go for each outcome for each file type of the scan data. For the instance of this report an x, y, z file was used that was adapted by a tool called Scan-3D in Solidworks which created a mesh so that a solid body could be made and tested (Wright, 2018).

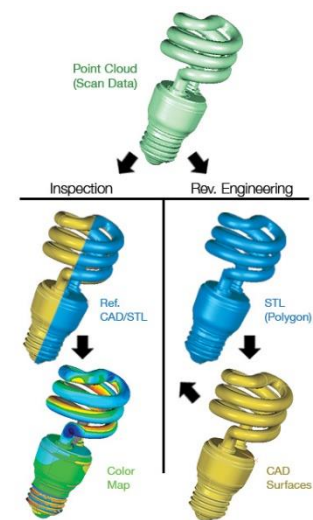


Figure 15: Process from the scan data of a 3D scanner (Laserdesign.com, 2018).

3. Methodology

This section will follow the process of the test plan for the main report and will look at each step starting from initial live flow bench testing, through to the final simulation flow testing of the standard and modified ports. A detailed account of how the initial data was collected, the mold taken of the port, the mold being scanned, the port being replicated in Solidworks, initial simulation testing and the testing of the modifications made to the port within Solidworks.

3.1 Initial test plan- Gantt chart

The initial term plan can be seen in the form of a Gantt chart for term 1 (October- December). This was created to make sure tasks were started and completed in time and that no tasks were over-running and preventing other tasks from being started. A Gantt chart is a great way to track the progress of a project with multiple tasks (PM Project Management, 2017).

	Start	Duration	Finish	Behind Plan	Seminar Paper Deadline	Seminar Presentation
Activity	Actual Start	Actual Duration	Weeks			
			1 2 3 4 5 6 7 8			
			Oct 9th 16th 23th 30th Nov 16th 13th 20th 27th			
Initial Research	1	2				
Mould of cylinder head	3	1				
Mould scanned to computer	4	1				
Flow Bench Test	4	1				
Further Research	3	4				
Methodology Review	3	4				
Research Review	5	2				
Creating solidworks model from scan	4	N/A				
Seminar Paper Complete	5	6				
Power Point Complete	5	6				

Table 5: Initial Gantt chart from the beginning of the first semester. Showing initial aims of the project.

3.2 Process of inlet port to CAD inlet port

Before any testing is started it is essential to familiarise yourself on how to use the flow bench safely and in an effective manner. It is a must to wear ear defenders whilst the machine is in use for prolonged periods of time to avoid any hearing damage. The HASAWA (Health and Safety at Work Act- 1974) was seen to whilst all live testing was conducted.

3.2.1 Process of making a mold of an inlet port

As a simulated test of the cylinder heads port flow is needed to be compared against actual live data a mold is needed of the port in question. A silicone substance is used with a catalyst substance of the ratio 1:1 and once it has been mixed the solution can be poured into the desired port.

The process usually takes 24hrs but 2 days is preferred as the mold must be completely hardened so the process of scanning can be achieved properly.

Once the mold is set correctly the use of a lubricant e.g. WD40- is needed to allow the mold to be removed with limited damage. A must is to remove the two inlet valves so the mold isn't broken on removal. A scribe or a screwdriver can be used initially to free the mold but after a force is applied to the valve throat end this will allow the mold to be removed undamaged.

The mold then needs to be cleaned up to remove any additional debris and flaking of the mold so an accurate scan can be taken of it to achieve the best model replica within Solidworks and Ansys software.

3.2.2 Collection of initial flow bench data of the standard inlet port

The standard inlet port chosen on the Ford Zetec 16v head needed to be flowed to collect the initial live base data so that a comparison could be made later between the simulated and live data to show the accuracy/ reliability of the data gathered.

Before testing began, an inlet valve needed to be measured for the diameter of the head and stem in metric (mm) and imperial (inches). This was so the effective valve area could be calculated and then how much valve lift for each test increment. The effective valve area is calculated using the formula- $0.785(Dh^2 - Ds^2)$. This was calculated in inches and the 'Ds' represented the stem diameter and 'Dh' represented the head diameter. Then by multiplying the valve lift against the valve diameter in 'mm' the lift could be calculated for the inlet valve.

Before testing an adapter needs to be fitted and bolted to the flow bench to represent the actual size of the engines' cylinder bore. Also, between the flow bench and adapter and the adapter and the cylinder head it needs to be sealed with sealant or blue-tac to prevent air leaks which would then give inaccurate results. For the cylinder head to the adapter the sealant goes around the outside of the combustion chamber and the cylinder head is held down with multiple G-clamps to secure the head level with the adapter.

Then, a plate which has a bolt per valve (two) is mounted to the topside of the cylinder head- this is so the valves can be opened by a turn of each bolt. Also, two DTI gauges are used to measure how much the valves are opened via each turn of the bolts. The DTI gauges are pre-loaded upon the top of the both bolt heads. The valves are opened by the set amount calculated from the valve dimensions.

With all the valves closed and the Flow-bench on range 1 the machine can be turned on (must be on inlet range to create air flow through inlet track and past valves and not on exhaust). The machine needs to be taking up to 25" (60cm) of H₂O for testing purposes and to check initially that the sealant has sealed the head and adapter and that there are no leaks.

Finally, by using the measurements calculated from the valve dimensions the valves were opened using the bolts and DTI gauges in increments for each test part. This would then give a reading on the flow scale as a percentage and if the percentage increased too high or near the end of the scale the test needed to be stopped so that a higher range could be selected to prevent the corks from being blown out and a specialist called in to recalibrate the machine. Once the reading was recorded the machine was turned off to prevent over-heating.

3.2.3 Scanning the inlet mold to acquire x, y, z point cloud data

To be accurately scanned the mold needed to be slightly whitened to avoid the transparency of the silicone disturbing the scan data collected. Also reference marker dots were placed spaced out over the mold in red marker to allow the scanner to pick up these specific points during the process. The whole process can take several hours depending on the size and shape of the object being scanned.

This data can then be opened in Solidworks but is not recognised as a solid body or mesh but is a 3D image with no data/reference points.

3.2.4 Converting the point cloud data into a mesh in Solidworks

As the point cloud data had an excessive amount of points referenced the file could not be converted into a stl. file then opened as a solid body as this would crash the system and take up a large memory file size.

In Solidworks there is an add-in called Scan-3D which allows big point cloud data to be converted into a mesh made up of

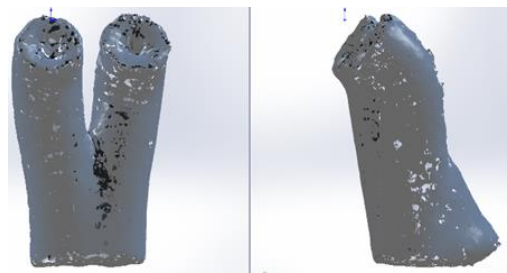


Figure 16: Showing the point cloud data converted into a mesh that can be used to create a solid-body.

faces which takes less time and memory to adapt. This allowed Solidworks to recognise the data as a mesh and allowed points, edges and faces to be selected.

3.2.5 Processing the mesh into a solid working body within Solidworks

The data now being a mesh gave the opportunity to make the mesh into a solid body so it could be used effectively in Solidworks.

To achieve a solid body several planes needed to be inserted through and all the way along the mesh so that the outline of the mesh could be sketched and lofted between the planes thus making a solid body.

As there were a limited amount of mesh edges that revolved around the mesh this gave its limitation to how many planes could be used without intersecting sketches and failed lofts. The planes were placed parallel along the mesh at 10mm apart until the end of the mesh (valve seat area) where the planes were placed at an angle to achieve the correct shape of the port.

On each plane the tool 'convert entities' was used to connect the edges that revolved around the mesh at that plane. This was undertaken at every plane so that a spline tool sketch could be created around the mesh to give an accurate copy of its shape.

A loft tool was used to create the mesh shape as a solid body between each spine sketch as accurately as possible. This was until the port divided into two where it has two inlet valves. Then because of this in total three loft sections had to be used- one for each valve section (two lofts) and one for the main body of the port. As for where the main body and two sections meet the spline tool was just used multiple times to allow the lofts to connect making one solid body. This achieved a smooth constant face without protruding edges and multiples faces along the port surface.

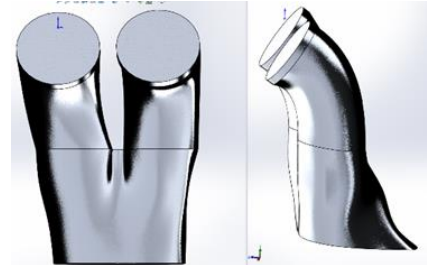


Figure 17: The inlet port as shown as a solid-body within Solidworks after the mesh has been lofted.

3.2.6 Creating cylinder head with inlet port, valves and valve seats

As the important part of testing the flow is the actual port design itself the outer part of the cylinder could just be replicated using a square sketch extrude for the size of the inlet port to be cut from.

To achieve the port shape cut in the cylinder head a Solidworks assembly needed to be created. From this the inlet port body can be mated to the cylinder head in such a way that it is level and the inlet area is at the correct angle for a pent-roof type combustion chamber.

When all the inlet port is submerged into the cylinder head it then can be cut out to leave the correct port shape for testing. The tool used is called 'cavity' which allows the selection of the object that is wanted to be cut from and the object that is going to be cut. After saving, the cylinder head can be opened on its own as a part and it will have the inlet port cavity in it ready for simulated testing.

Seen to the right is the solid port and the cylinder head that how hollowed using the port.

Also, the cylinder head, bore and valves can be seen to be mated to the right and image below. The valve was created from measurements taken in the initial live flow bench testing. The correct length, width of the stem, width of the face and angle of the seat were created from a sketch then a 'revolve' tool used.

The cylinder bore was made from the dimensions of the engine with a bore of 79mm and a stroke of 81.9mm. By using specific mates in the assembly, the port could then be tested in the Solidworks Flow Simulation software.

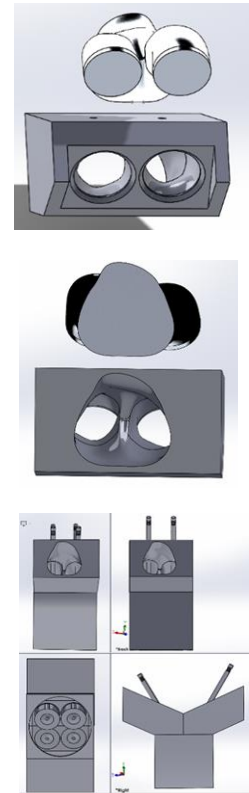


Figure 18: Showing the process of using the mould as a cavity in a ready-made cylinder head to create the port.

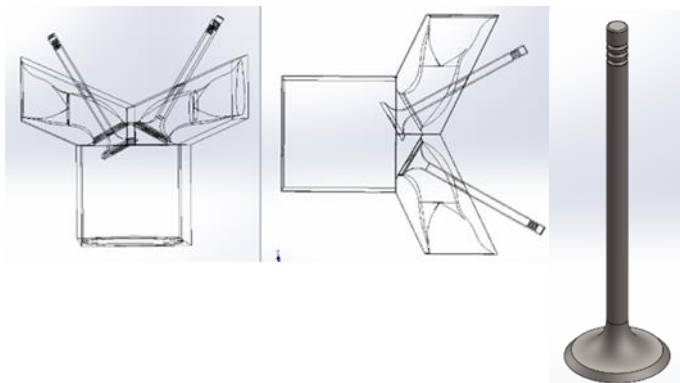


Figure 19: Drawing view of the cylinder head with valves and bore. A picture of the valve created in Solidworks.

3.3 Process of collecting initial simulated data

3.3.1 The use of Xpress Flow in Solidworks

To avoid learning a new software to collect simulated flow data it was decided to use Xpress Flow with Solidworks instead of Ansys to reduce testing time and to reduce the chance of user error.

Before testing can be undertaken a translucent plate is needed to be placed over the inlet and outlet side of the inlet port so when testing takes place their faces can be chosen to show the flow area.

The problem with Xpress Flow that was found out after the tutorials had been completed was that only one inlet and one outlet can be chosen which leads to a problem with more than one inlet valve as the port separates in two thus leaving one inlet side and two outlets. As this was the case it was decided that Solidworks Flow Simulation software had to be used instead to achieve accurate results. The software is good for achieving flow images for explanations of flow systems but for actual reliable, credible data it is not enough.

3.3.2 The change in use from Xpress Flow to Solidworks Flow Simulation

The first obvious benefit of Solidworks flow simulation over Xpress flow was the capabilities of more than one inlet and outlet section. Other benefits included specification of valve lift, cylinder area, combustion chamber type and more accurate reliable results. Also, the air velocity, pressure and CFM can all be recorded and viewed for all tested valve lifts. Below, images are shown from the software where lids are placed to specify the fluid volume of the port and cylinder.

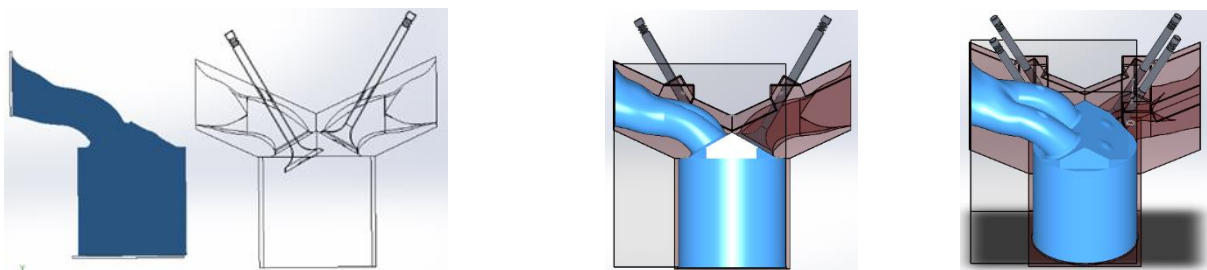


Figure 20: These pictures show the fluid volume of the inlet track and into the cylinders bore. The inlet valves can be seen to be open allowing the transfer of fluid into the bore.

3.3.3 Process of using Solidworks Flow Simulation

The same initial process was undertaken as with using Xpress Flow regarding sealing the flow area with lids to specify the area to create the flow or pressure. The benefit with this software is that boundary conditions and goals can be set before testing is started to achieve accurate results and results that are wanted.

As for boundary conditions, an environmental pressure can be stated at the inlet and for the outlet a set pressure of 25 Inches of H₂O (95098 Pascal's) to replicate the flow set-up on the flow bench in the live testing (creating a pressure drop and air flow). This pressure drop replicates the process in which the flow bench works so that data can be gathered and compared for live and simulated testing.

Before testing is carried out goals need to be specified so the data wanted is recorded and can be viewed after. The goals can range from the minimum, average and maximum values for pressure, velocity etc during the test.

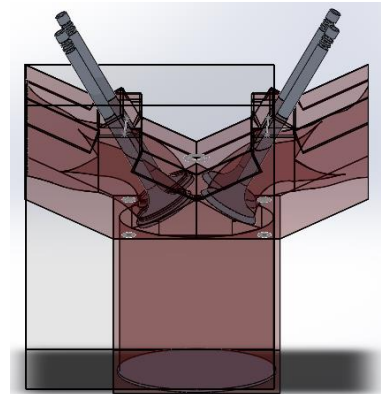


Figure 21: Flow area of inlet port and cylinder bore with valves open at 2mm of lift (showing computational domain).

In this report Volume flow rate goals were set at Lid1 and Lid2 which were placed at the start of the port and bottom of the cylinder bore. The readings give an average, max and min value which are recorded at each valve lift. As this is in CFM and not CFM/sqin like the flow bench data the readings need to be divided by the effective valve area figure that was calculated for live testing (2.1 Inches= effective valve area for two valves).

The images below show the fluid volumes for the standard port and the 1st modified one. This shows the significant difference in port design but not actual size to decrease port resistance and increase air flow into the cylinder. A distance mate at the flat part of the bottom of each inlet valve and the combustion chamber, which are parallel throughout valve lift, is used and changed to set each valve lift correctly to obtain reliable results to be compared.

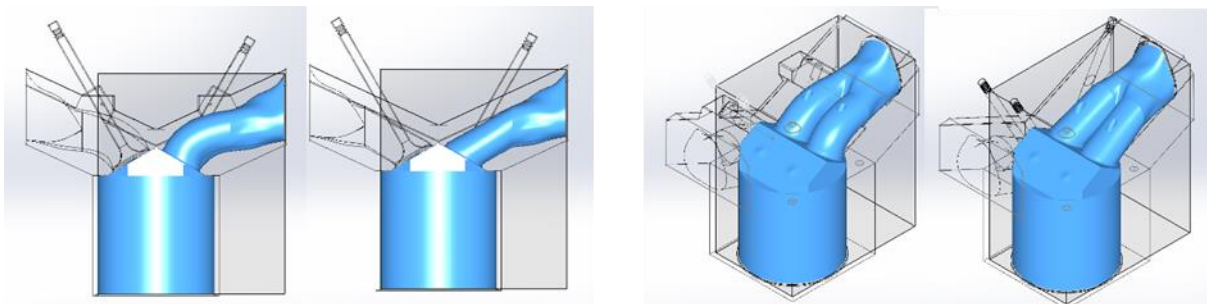


Figure 22: Side and angled view of the standard port and modified port with its fluid volume seen.

3.4 Differences between the standard and two modified ports

3.4.1 Modified Port 1- Diameter at port split and at valve seat

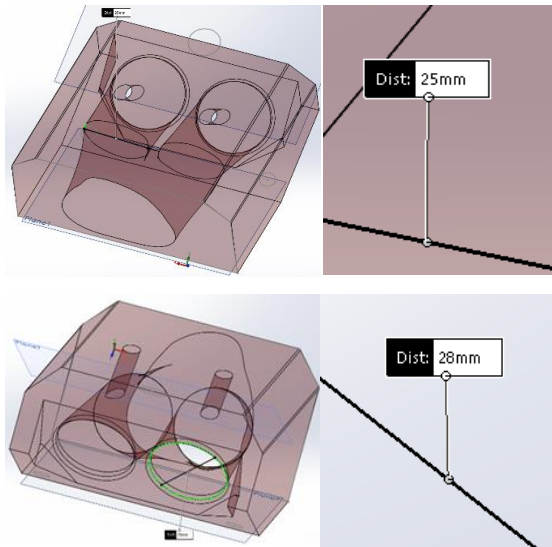


Figure 23: Port split diameter and inner diameter of the valve seat for the first modified port.

The first modified port doesn't incorporate a change in port diameter at the split or at the seat but the design has been changed to obtain a straight as possible port at the port split to the valve seat.

This modification was performed to try and see if the straight track of intake port increased the flow rate throughout the valve lifts (1.5mm-12mm in 1.5mm increments).

3.4.2 Modified Port 2- Diameter at port slit and at valve seat

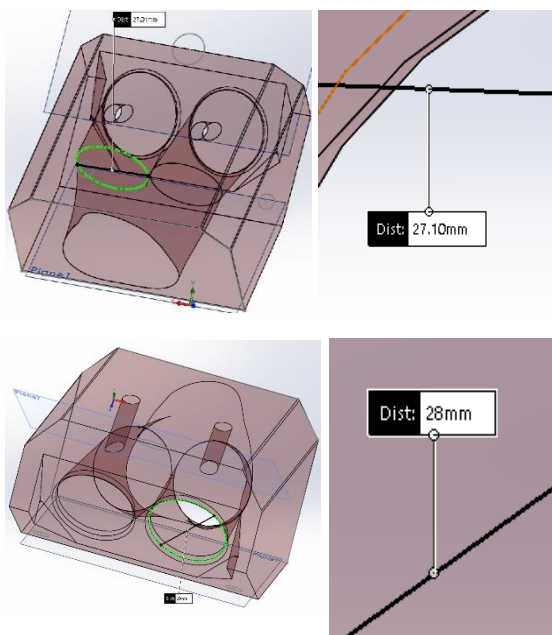


Figure 24: Port split diameter and inner diameter of the valve seat for the second modified port.

The second modification adds an additional 2mm (27mm) in diameter at the port split and retains a valve seat diameter of 28mm. This achieves an almost constant port diameter down each port split to the valve seat with only a different of 1mm from the split to the seat.

The design was incorporated from the first modification with only a change in diameter. The closer diameter at the seat and split did not give a sufficient increase in air flow compared to the first upgrade.

4. Discussion

This section will display the results collected through live and simulation testing of the inlet port and show calculated data to solidify the reliability of the results collected. Each piece of data collected will be thoroughly investigated for irregularities and the reliability and effectiveness questioned. The results will be heavily evaluated and analysed so that a comparison can be made between each set of results.

4.1 Live data

4.1.1 Valve calculations

Before the cylinder head port can be tested a few calculations must be made from the valve dimensions to find the correct amount of valve lift increments via the valve L X D (Lift x Diameter).

	Valve Head Diameter (mm)	Valve Stem Diameter (mm)	DH Valve Head Diameter (in)	DS Valve Stem Diameter (in)	Effective Valve Area (in ²): (0.785(DH ² -DS ²))x2
Inlet Valve	30	6	1.1811	0.24	2.1

Table 6: Valve dimensions and effective valve area of the inlet valve.

The dimensions were measured using a digital micrometer and the effective valve area number of 0.785 was used for the curtain area of the valve.

Valve Lift x Diameter (LXD)	0.05	0.1	0.15	0.2	0.25	0.3	0.35	0.4
Inlet Valve Lift (mm)	1.5	3	4.5	6	7.5	9	10.5	12

Table 7: Valve lift calculated for the percentage of valve diameter.

By using the lift percentage in the first row multiplied by the valve head diameter (30mm diameter) the valve lifts can be calculated. Each tested valve lift is at every 1.5mm increment starting at 1.5mm of lift and ending at 12mm of valve lift. Generally, valves only open to around 12% that of the cylinder bore diameter i.e. (79mm/100) x12= 9.48mm maximum valve lift. This is as the air flow velocity does not continue to increase past this point and is restricted by the actual port itself. Another calculation shows that the valve is restrictive until it reaches a quarter of its diameter of lift (D/4) i.e. 30/4 = 7.5mm of lift. There is almost 2mm of lift between the first and second calculations where the port becomes more restrictive than the valve.

4.1.2 Live flow bench data

The flow range figures are used to calculate the actual CFM from the flow scale to achieve the correct data. As the valve lift increases and so does the velocity of air and in most cases the range must be changed to obtain the correct flow-scale. Therefore, the range numbers are needed to achieve the correct CFM data for each valve lift.

Range	1	2	3	4	5	6
Intake	34.98	70.98	147.8	293	440.5	592.5

Table 8: The flow-bench ranges with the conversion numbers.

Test No.	1	2	3	4	5	6	7	8
1. Pressure	25	25	25	25	25	25	25	25
2. Valve Lift	1.5	3	4.5	6	7.5	9	10.5	12
3. Flow Range	2	3	3	4	4	4	4	4
4. Flowscale (%)	55	52.5	74.2	45	49	51	52	53
5. Flowscale/100	0.55	0.525	0.742	0.45	0.49	0.51	0.52	0.53
6. Raw Test Flow	39.04	77.58	109.6	131.8	143.6	149.4	152.4	155.3
7. Leakage	0	0	0	0	0	0	0	0
8. Test CFM	39.04	77.58	109.6	131.8	143.6	149.4	152.4	155.3
9. Test CFM/Sqin	18.59	36.94	52.2	62.78	68.36	71.15	72.54	73.94
10. Potential Flow	25.3	47.3	65.7	80.2	91.1	98.2	101.5	101.1
11. Potential Flow (%)	0.73	0.78	0.79	0.78	0.75	0.72	0.71	0.73
12. Max Flow	0.742							
13. Average Flow	0.54							

Table 9: Data collect from the flow-bench testing at each valve lift. Showing the test CFM and the CFM with the use of the effective valve area.

In row number 8 & 9 the flow increase in relation to the valve lift as the area between the valve and valve seat is increasing. The difference in pressure between the entrance of the port (environmental) and the flow bench internals (25 Inches H₂O less than environmental) creates a higher pressure at the entrance than in the cylinder bore thus creating a vacuum from the pressure difference when the valve begins to open.

As stated previously the air velocity will only keep increasing with valve lift to a specific extent (7.5mm- 9.48mm of valve lift) and although the table data doesn't reflect on this greatly, the data increase does start to flatten around these valve lifts. See flow graph below for better display of results.

The initial flow bench data from the live testing gave good comparable results compared to the potential flow readings found in the flow bench user manual.

As explained previously, the increase in CFM should start to flatten out at around 7.5mm- 9.48mm of valve lift as this is where the port is more restrictive in design than the valve itself. Most engines are limited to a specific valve lift because of camshaft timing and the area of which the piston and valve meets.

Valve Lift (mm)	Live CFM/sqin	Potential CFM/ sqin
1.5	18.59	25.3
3	36.94	47.3
4.5	52.2	65.7
6	62.78	80.2
7.5	68.36	91.1
9	71.15	98.2
10.5	72.54	101.5
12	73.94	101.1

Table 10: Live and Potential CFM readings.

Initially, the live and potential flow readings, for a pent-roof combustion chamber, are quite similar as the valves are only slightly open and very restrictive but as the valve lifts increase the difference can be seen to increase until at 9mm of lift where both readings flatten out because of what was explained prior.

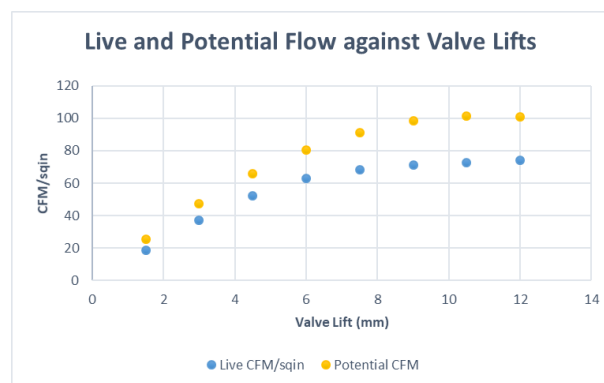


Figure 25: Live flow bench data and potential flow data (CFM/sqin) against valve lifts.

The difference in flow between 6mm of lift and 12mm of lift is only 23.44 CFM but between 1.5mm of lift and 6mm is 92.8 CFM which is almost 4 times in difference. This shows that the Zetec head flows 4 times quicker at low valve lift compared to high valve lift on the inlet side.

4.2 Simulation data

In section 4.4 a comparison will be made using the live, simulated, calculated and potential CFM. The comparison will also be made for the modified ports tested within the simulation in Solidworks

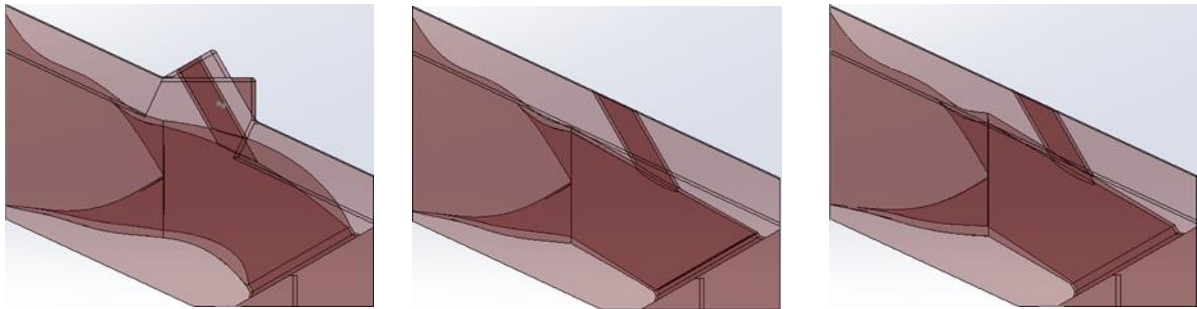


Figure 26: Three designs of the inlet port- Standard port, straight port (no change in size) and straight port with an additional 2mm in diameter at the port split.

Above are the three tested simulation inlet ports. The first from left is the standard port with the natural bend from the port split to the valve seat, the middle shows a more direct, straighter port from the port split to the valve seat with no increase in port diameter and the right port incorporates the first modification but with an increase in diameter by 2mm at the port split to create the same diameter at the smaller diameter of the valve seat.

Modification 1- This modification was undertaken to see if port design affected CFM, Velocity and Pressure without a change in port diameter. The change was made on the assumption that the air entering would be taken a shorter, less resistant path that would slow the intake air less than the standard port.

Modification 2- The modification was decided on when the CFM readings and Velocity readings of the first modification came back positive with an increase in all. It was decided to see how a change in diameter in the intake track would affected CFM, Velocity and Pressure. *See Methodology for an in-depth explanation of the changes.*

4.2.1 Original Simulated Port CFM against Potential CFM

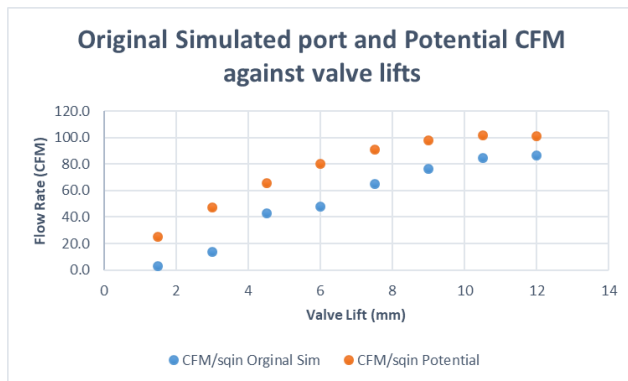


Figure 27: Data plotted of original simulated port and potential CFM against valve lifts.

	CFM/sqin	
	Original Sim	Potential
1.5	2.6	25.3
3	13.4	47.3
4.5	42.8	65.7
6	48.0	80.2
7.5	65.2	91.1
9	76.4	98.2
10.5	84.8	101.5
12	86.3	101.1

Table 11: CFM readings for original simulated port and potential against valve lifts.

In the graph above it can be seen like in section 4.1.2 the standard port CFM with the potential. In this instance the readings are for the simulated data which differ slightly to the live readings. The CFM appears to hold its increase in flow and begins to flatten around 10.5mm of valve lift, which gives it a higher volumetric efficiency but only at very high valve lifts. See 4.3.2 for volumetric efficiency comparison.

As there is still potential between the live and simulated data there needs to be modifications made through research to increase the CFM to around the potential for an increase in volumetric efficiency. Both data readings start to flatten out and this is due to the pressure difference in the port and bore where the pressure is near enough the same because of the high valve lift creating less velocity but more CFM.

It is important to take into consideration that making a port bigger and bigger will increase CFM readings but velocity is very much affected at low and high valve lifts. Increasing port size too drastically can lose the engines low RPM torque meaning that higher RPM's are needed to achieve maximum power and torque.

4.2.2 1st Modified Simulated Port CFM against Potential CFM

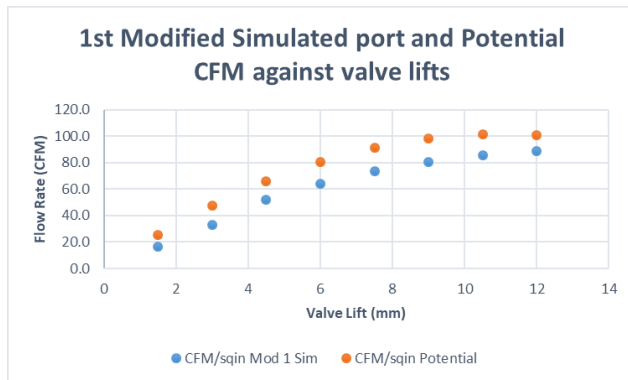


Figure 28: Graph showing CFM readings for the first modified port in the simulation software and potential against valve lifts.

	CFM/sqin	
	Mod 1 Sim	Potential
1.5	16.3	25.3
3	33.2	47.3
4.5	51.8	65.7
6	64.3	80.2
7.5	73.3	91.1
9	80.2	98.2
10.5	85.6	101.5
12	88.6	101.1

Table 12: CFM readings for first modified simulated port and potential against valve lifts.

These data plots show a closer relation between the first modified port and the potential for this type of cylinder head. An increase in CFM is positive for an increase in volumetric efficiency but the velocity of the air entering is just as important to maintain power and torque figures.

It is clear to see that a straighter port increases the CFM output throughout valve lifts to be closer to the potential CFM but it is unknown what affects this modification has upon velocity and pressure in the port and cylinder.

The pressure and velocity plots will be compared further into the discussion to show how port design and size can affect these variables.

4.2.3 2nd Modified Simulated Port CFM against Potential CFM

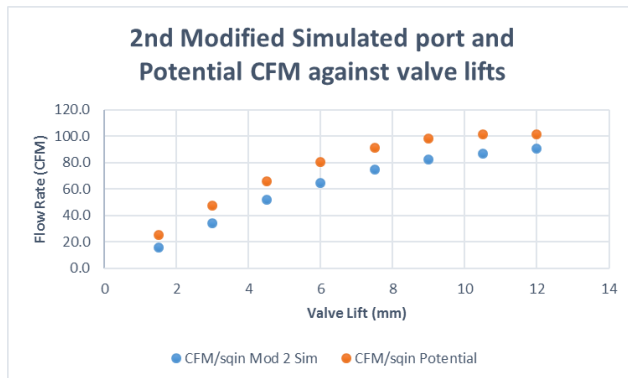


Figure 29: Graph showing CFM readings for the second modified port in the simulation software and potential against valve lifts.

	CFM/sqin	
	Mod 2 Sim	Potential
1.5	15.8	25.3
3	34.2	47.3
4.5	51.7	65.7
6	64.7	80.2
7.5	74.7	91.1
9	82.3	98.2
10.5	86.6	101.5
12	90.8	101.1

Table 13: CFM readings for first modified simulated port and potential against valve lifts.

In the 2nd modified port the flow readings are again an improvement on the standard simulation port but comparisons between the two modifications will be analysed in the next section. It can clearly be seen that these readings follow a closer relation to the potential and seem to dip around the same valve lifts with only the simulated port getting close to the potential at the highest measured lift of 12mm.

The reason for the simulated data to take longer to flatten out at higher lifts can be placed down to the limitations of the software, mesh reliability and surface roughness specification (Chen et al., 1995).

4.2.4 Comparison between the standard port, 1st modified port, 2nd modified port and potential CFM

The graph shows the CFM readings for all the simulated ports and the potential for the specific cylinder head type tested (pent-roof).

The standard, 1st modified and 2nd modified ports all are below the potential flow readings for this type of cylinder head but the two modified heads have an improved CFM reading across the valve lifts compared to the standard port.

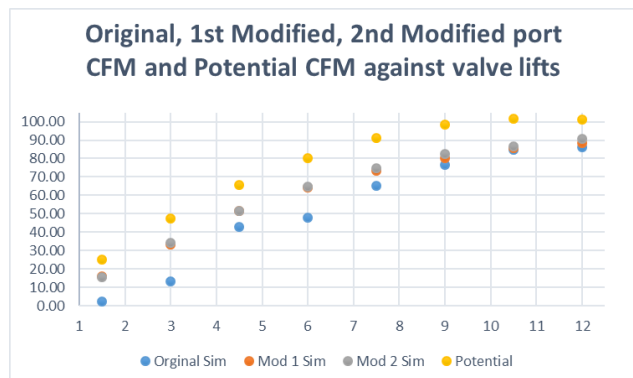


Figure 30: Graph showing CFM results of the standard port, 1st modified port, 2nd modified port and potential against valve lifts.

As for the two modified ports there isn't a significant difference in flow readings to justify the 2nd modification (2mm increase in diameter at port split) as there is only a difference around 9mm of valve lift and above but even this is only slight. At 6mm of valve lift the 1st modified port achieves a flow reading of 64.27CFM/sqin and the 2nd modified port achieves a figure of 64.68CFM/sqin. Its only at an excessive, unrealistic valve lift of 12mm that they differ slightly with the 1st modified port achieving 88.59CFM/sqin and the 2nd modified port achieving 90.81CFM/sqin.

By using the calculation that the valve is the restrictive part until 4/D (quarter of its diameter of valve lift), (30/4= 7.5mm of valve lift). At 7.5mm of lift and above this is where the flow readings start to change as the port becomes the restrictive part and the slightly bigger port on the 2nd modified port has an increase flow rate.

	CFM			
	Original Sim	Mod 1 Sim	Mod 2 Sim	Potential
1.5	2.62	16.35	15.84	25.3
3	13.42	33.20	34.20	47.3
4.5	42.78	51.76	51.66	65.7
6	48.03	64.27	64.68	80.2
7.5	65.18	73.25	74.67	91.1
9	76.38	80.21	82.33	98.2
10.5	84.76	85.56	86.57	101.5
12	86.27	88.59	90.81	101.1

Table 14: Table showing CFM results of the standard port, 1st modified port, 2nd modified port and potential against valve lifts.

As valve lift is dependent on combustion chamber size and valve timing the benefits of the 2nd modified port is inconsequential as it is highly unlikely that the valve will reach this amount of lift and thus the increase in flow readings between the 1st and 2nd modified port is not important. A decision on how well the modifications perform can't be justified until the velocity of the air and pressure entering through the port is seen and analysed.

4.2.5 Original Simulated Port Pressure and Velocity trace

Velocity trace and pressure trace for 1.5mm-12mm of valve lift in order of appearance.

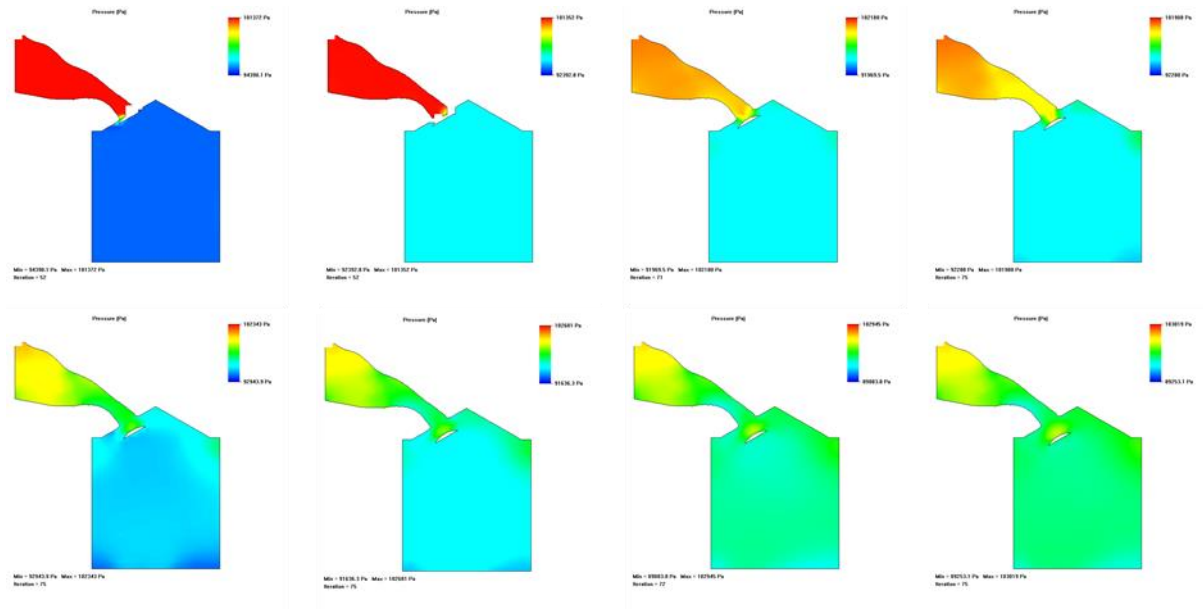


Figure 31: Showing the pressure drop between the start of the port through to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- Original Port.

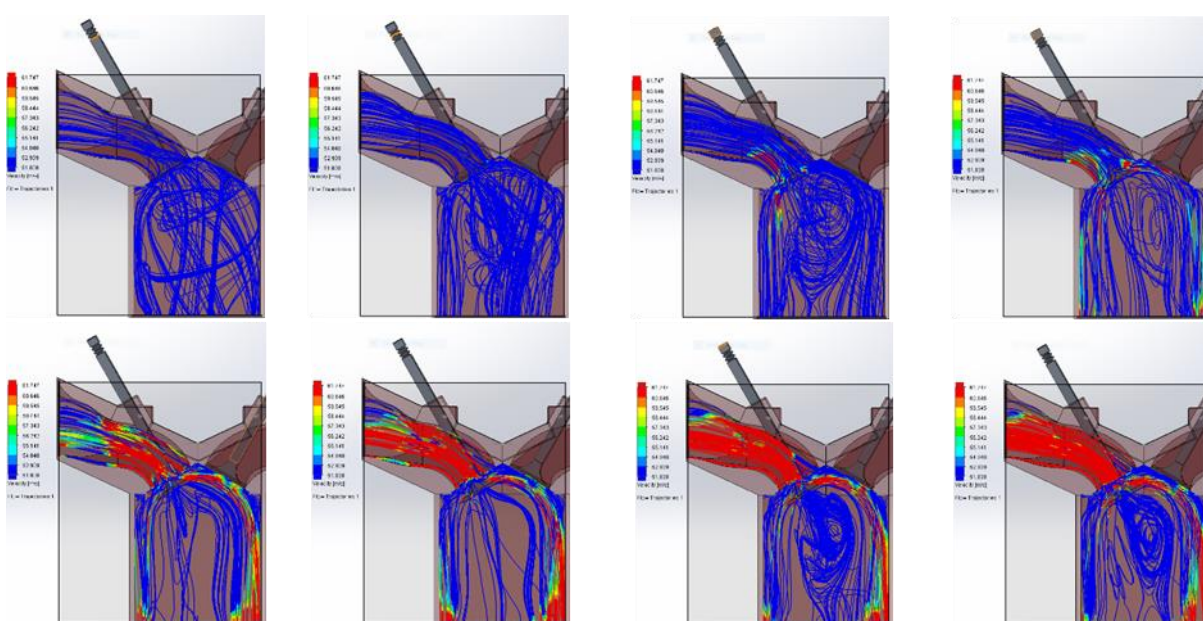


Figure 32: Showing the velocity between the start of the port to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- Original Port.

4.2.6 1st modified Simulated Port Pressure and Velocity trace

Velocity trace and pressure trace for 1.5mm-12mm of valve lift in order of appearance.

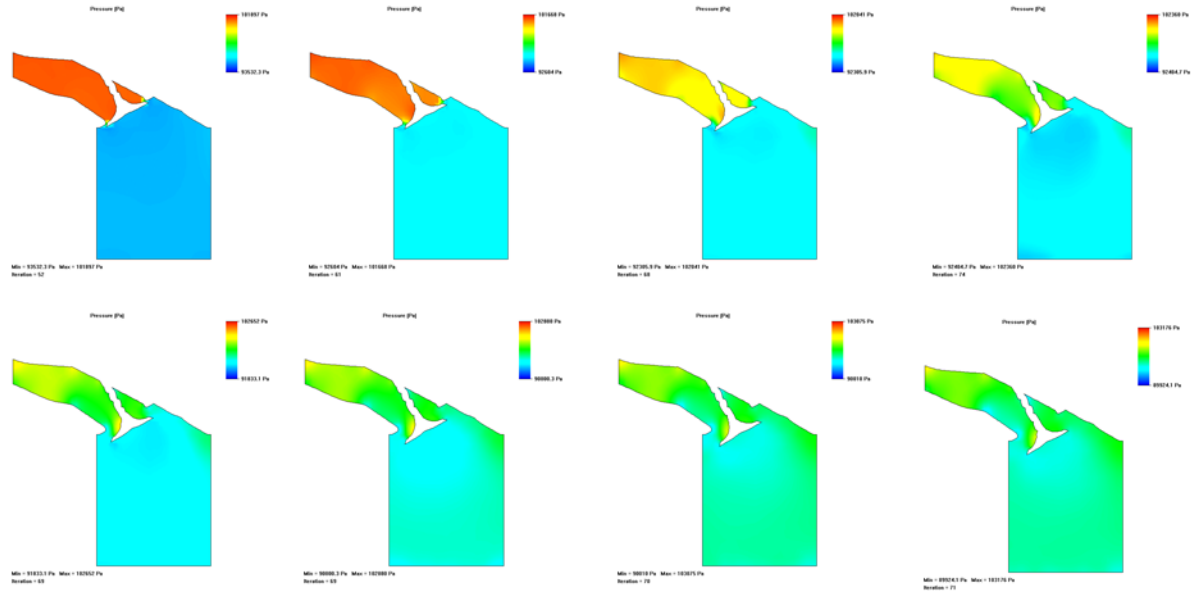


Figure 34: Showing the pressure drop between the start of the port through to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- 1st Modified Port.

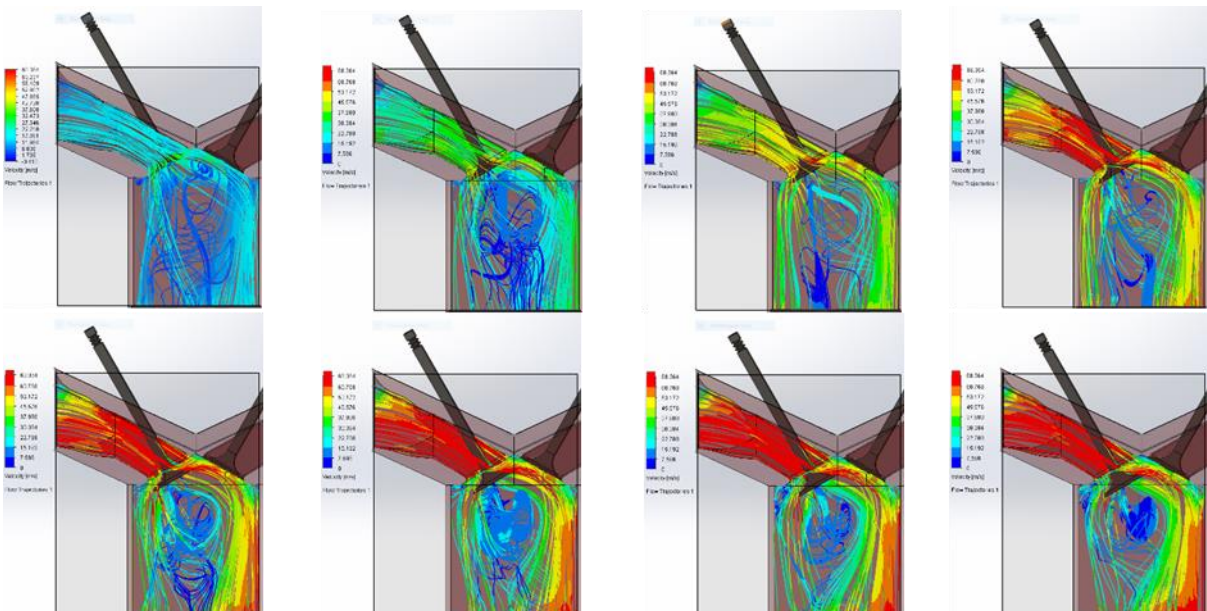


Figure 33: Showing the velocity between the start of the port to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- 1st Modified Port.

4.2.7 ^{2nd} modified Simulated Port Pressure and Velocity trace

Velocity trace and pressure trace for 1.5mm-12mm of valve lift in order of appearance.

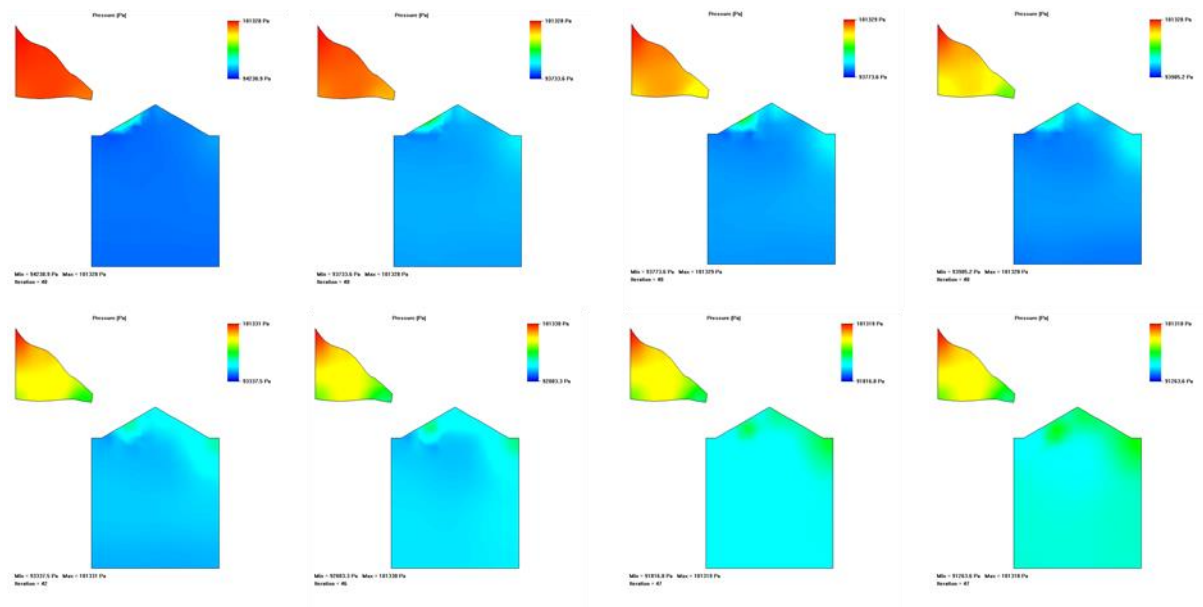


Figure 35: Showing the pressure drop between the start of the port through to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- ^{2nd} Modified Port.

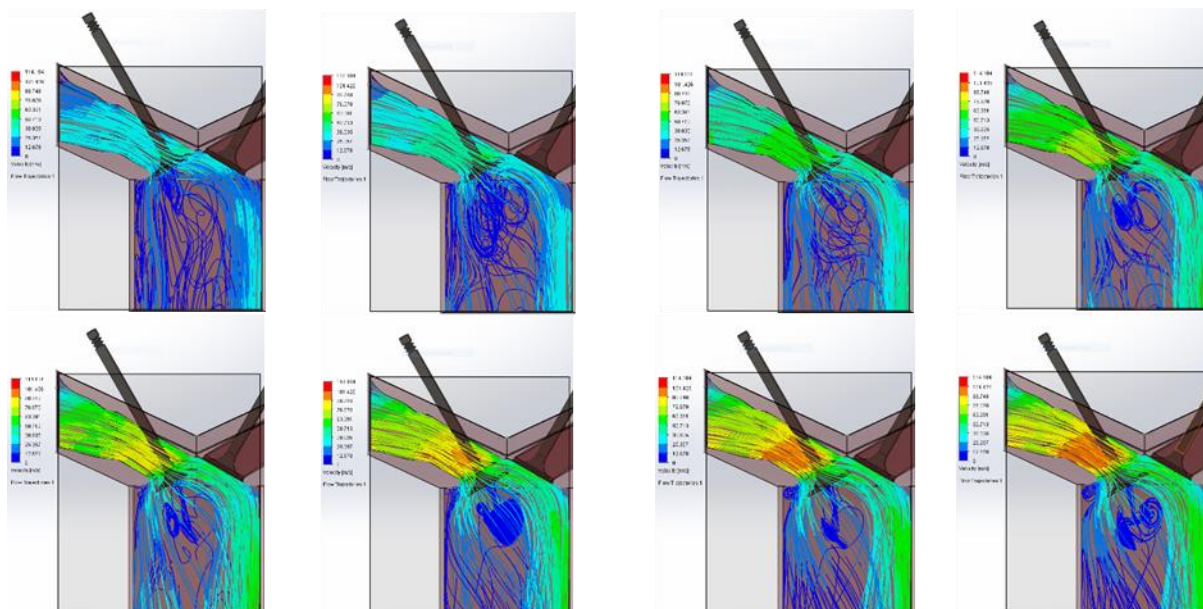


Figure 36: Showing the velocity between the start of the port to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- ^{2nd} Modified Port.

4.2.8 Analysis and Evaluation of Velocity and Pressure traces for the standard, 1st modified and 2nd modified inlet ports.

In section 4.2.5- 4.2.7 the data traces for velocity and pressure throughout all tested valve lifts can be seen for the standard, 1st modified and 2nd modified ports. This data was captured to backup or disprove the changes made throughout the different designed ports tested. An increase in CFM is great but if the velocity is significantly reduced the engine that the cylinder head is placed upon may not make the desired and expected power and torque figures.

At 1.5mm of valve lift the pressure trace for all can be seen to show high pressure in the port and a low pressure in the cylinder bore. This is because of the stated environmental pressure at the start of the intake track and the 25 Inches of H₂O of pressure less stated at the bottom of the cylinder bore. The small valve lift really shows the pressure difference which creates the increase in velocity through the port itself.

The 1st modification shows a quicker drop in high pressure in the intake track and a quicker rise in pressure in the cylinder bore, during valve lifts, showing the difference in efficiency that the 1st port design has over the original. Even though the difference in pressure averages out the quickest in the 1st modified port than the other two the velocity keeps increasing as seen with the red colour even at high valve lifts.

The 2nd modification had a similar CFM reading to the 1st modification but a slower pressure change throughout the valves lift which is like the first port. The velocity traces do not show an as high velocity reading even though the CFM is same to the 1st modification. This is a prime example of a well flowing head that may not make good torque and power outputs when linked to an engine dyno.

The standard port has a start in high velocity (red) at around 4.5mm but is only a little amount compared to the low velocity (blue) around this valve lift. The 1st modified port shows high velocity readings (red) at 4.5mm of valve lift but much more of an average velocity readings (green) than the standard port. The first modified port has an increase in CFM and velocity compared to the standard port but the 2nd modified port doesn't show an increase in velocity to high velocity (red) but has a more consistent mid-range velocity (green).

Regarding the standard, 1st modified and 2nd modified port the 1st modified port shows the best in simulation data with higher CFM readings without a negative effect to the ports velocity. The 1st modified port would be the best for the automotive cylinder head with an increase in volumetric efficiency through CFM throughout the valve lifts which should increase torque and power. The constant increase in velocity would show that torque wouldn't be affected in a negative way at low and high RPM levels.

This shows that for this port an increase in port diameter of 2mm is not needed for an increase at low valve lifts and only showed some increase and unrealistic valve lifts and even then, the increase was insignificant. The improved inlet track with a less restrictive path does increase the CFM and velocity but again an increase in diameter from the standard port is not needed from the results shown.

An improvement in CFM may be shown if the size of the valves is increased slightly as they are the restrictive part at low valve lifts. *See next section for a brief look at an increase in valve size by 2mm (30mm-32mm).*

4.2.9 CFM, Velocity and Pressure trace for a 2mm increase in valve diameter.

A brief consideration into the effects of valve size in relation to CFM/sqin show that initially at 1.5mm of valve lift the standard port with bigger valves increase CFM/sqin to over the potential giving more than 100% volumetric efficiency. This changes from 3mm-12mm of valve lift but still the CFM/sqin readings are very close to the potential and improved the flow rates greatly.

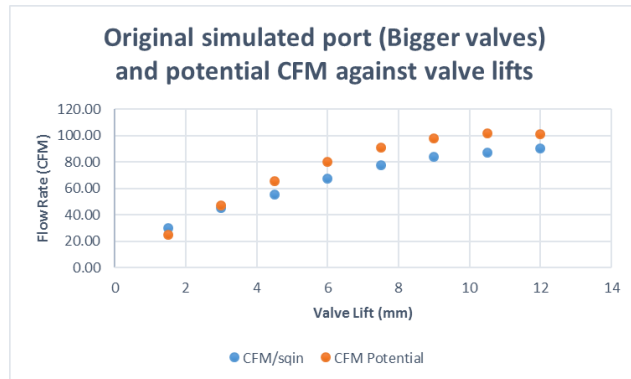


Figure 37: The original simulated port with a 2mm increase in valve diameter (32mm) and potential CFM against valve lifts.

Even though the CFM/sqin has increased the velocity needs to be known as this is detrimental to engine torque and power output. Another adaption would be the increase the valve size of the 1st modified port to see if the potential CFM can be reached throughout the valve lifts. *See section 4.4 to see a comparative between all the modifications and the next section for the velocity and pressure traces.*

	CFM/sqin	
	Original Sim (bigger valves)	CFM Potential
1.5	29.77	25.30
3	45.51	47.30
4.5	55.60	65.70
6	67.20	80.20
7.5	77.39	91.10
9	84.15	98.20
10.5	86.87	101.50
12	90.41	101.10

Table 15: CFM readings for each valve lift for the simulated port with an increase in valve size and potential CFM.

4.2.10 Analysis and Evaluation of Velocity and Pressure traces for the standard port with the increase in valve diameter (30mm-32mm).

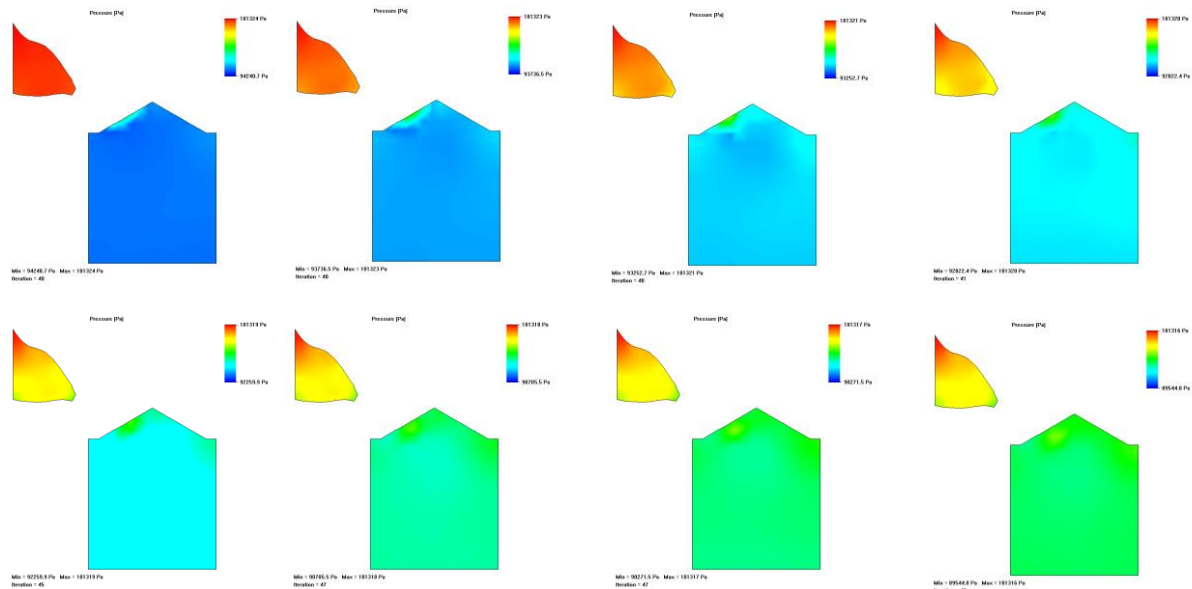


Figure 38: Showing the pressure drop between the start of the port through to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- Original port with bigger valves (32mm).

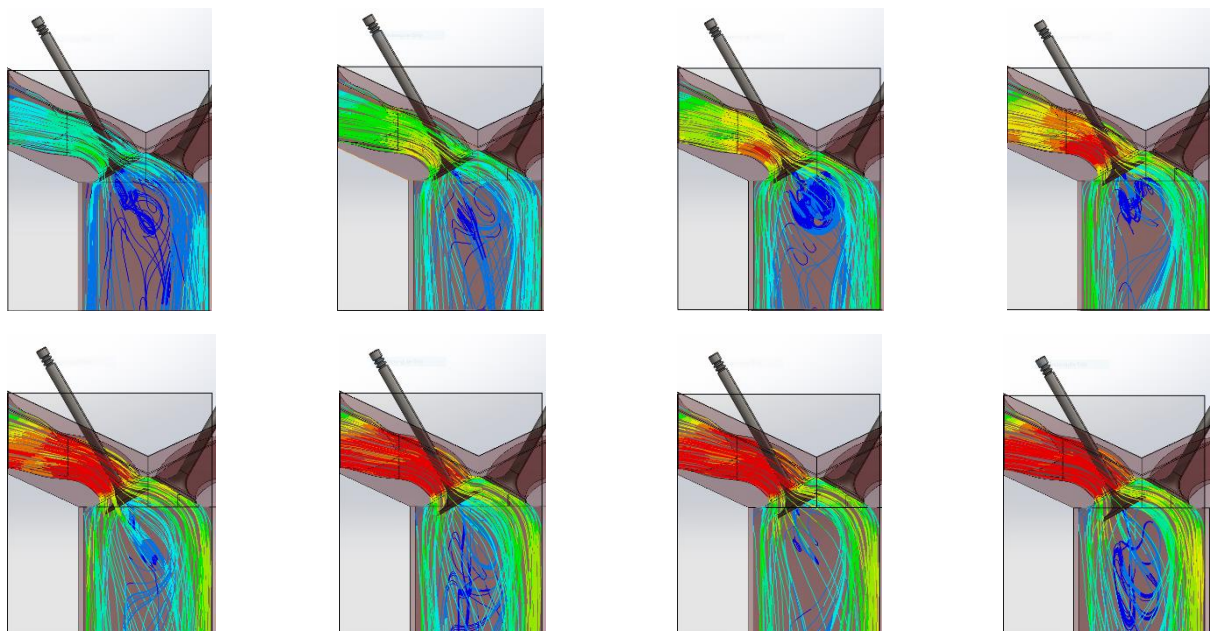


Figure 39: Showing the velocity between the start of the port to the bottom of the cylinder. Environmental pressure is stated at the entrance and 25 Inches H₂O less is stated at the bottom of the cylinder bore- Original port with bigger valves (32mm).

Theoretically, having bigger valves should increase CFM but like having a more aggressive camshaft the torque and power is more likely to be pushed higher up in the RPM range. As the valve is the restricted part below a quarter of its diameter ($4/D$) in lift the bigger valves should increase CFM at lower valve lifts but may affect the velocity of the air entering at higher valve lifts due to a more restrictive port.

Comparing the velocity and pressure traces from the bigger valve standard port to the original and 1st modified port it can be seen that CFM has increased significantly without a downfall in air velocity entering the through the port. The pressure change also is less drastic through valve lift which is the key to maintaining high air velocity.

An average in air pressure throughout the port and cylinder would decrease air flow but on a real life application there is the downwards movement (intake stroke) on the piston which pulls the air through the port. All the data has been tested through the performance of the cylinder head without any live testing on a complete engine. There are many more variables to consider when looking at a complete well performing engine compared to a well performing cylinder head.

A well flowing cylinder head may look impressive but dissapointing power and torque gains may be seen when equiped to a complete engine for dyno testing. As said before CFM isnt the end all of cylinder head performance and the velocity and pressure needs to be noted through valve lifts to achieve a well performing cylinder head (Chen et al., 1995).

4.3 Calculated data

4.3.1 Pulse Tuning- Calculating optimum intake runner length for a specific RPM

Calculation of cylinder volume	
Cylinder bore (mm)	79
Cylinder stroke (mm)	81.4
Number of cylinders	4
Cubic capacity (cm ³)	1595.9837299121102

Calculation of ideal runner diameter	
☒ Use values from previous calculation	
Cubic capacity (cm ³)	1595.9837299121102
Number of cylinders	4
Target RPM	1000
Runner diameter (cm)	1.505872293724785

Calculation of runner length	
☒ Use values from previous calculation	
Target RPM	1000
Runner diameter (cm)	1.505872293724785
Duration intake valve (degrees)	232
Runner length (cm)	366.11294042202934 or 183.05647021101467

Calculation of cylinder volume	
Cylinder bore (mm)	79
Cylinder stroke (mm)	81.4
Number of cylinders	4
Cubic capacity (cm ³)	1595.9837299121102

Calculation of ideal runner diameter	
☒ Use values from previous calculation	
Cubic capacity (cm ³)	1595.9837299121102
Number of cylinders	4
Target RPM	5000
Runner diameter (cm)	3.3672328142021492

Calculation of runner length	
☒ Use values from previous calculation	
Target RPM	5000
Runner diameter (cm)	3.3672328142021492
Duration intake valve (degrees)	232
Runner length (cm)	73.45254246106516 or 36.72627123053258

Figure 40: Calculation for inlet runner length (Exx.se, 2017).

This allows the pressure wave in the inlet to arrive at the valve at the correct time even though the runner length isn't the correct length. Additionally, the duration of which the valve is open effects the length of the inlet runner and larger cam durations would allow slightly smaller port lengths (Enginebasics.com, 2017).

At **1000rpm** the optimum intake length would be **3661mm/366.1cm/3.66m** this figure divided by 2 or 4 gives:

/4 = 0.915m/91.5cm/915mm. Intake runner would nearly be a meter.

/2 = 1.83m/183cm/1830mm. Intake runner would nearly be 2 meters.

At **5000rpm** the optimum intake length would be **735mm/73.5cm/0.735m** this figure divided by 2 or 4 gives:

/4 = 0.18m/18cm/180mm. Intake runner would nearly be 0.2 of a meter.

/2 = 0.36m/36cm/360mm. Intake runner would nearly be 0.4 of a meter.

4.3.2 Theoretical Horsepower

By using an equation that takes the number of cylinders, flow reading (at specific valve lifts) and efficiency for the SuperFlow SF600 at 25 Inches of H₂O the theoretical Horsepower can be calculated.

Maximum Horsepower achieved was 104.61HP for the simulation flow readings and 79.86HP for the live data flow readings.

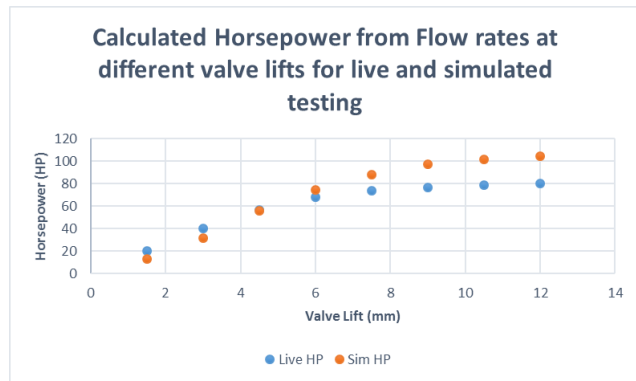


Figure 41: Theoretical Horsepower calculated by using both live and simulated flow readings.

$$\text{Horsepower} = (\text{Observed CFM} \times \text{Power Coefficient} \times \text{Number of Cylinders})$$

Baechtel, J. (2011).

As explained in the surface roughness in the Research section the roughness reading affects the flow of air in the inlet port and the turbulence. This can be a reason towards the higher flow readings and higher Horsepower readings for the simulated test compared to the live test.

Also, the process of molding the port, reliability of the scan data and the process of making the port in Solidworks gives room for error and lack of efficiency. The readings on both aren't too far apart and as mentioned above these conditions will affect the reliability and credibility of the results taken.

Valve Lift (mm)	Live HP	Sim HP
1.5	20.1	12.96
3	39.89	31.6
4.5	56.38	55.58
6	67.8	74.09
7.5	73.83	88.27
9	76.84	96.98
10.5	78.34	101.35
12	79.86	104.61

Table 16: Theoretical Horsepower calculated at each valve lift from the live and simulated flow test sessions.

4.3.3 Calculated Volumetric Efficiency

The volumetric efficiency can be calculated using the flow data (CFM) achieved on a flow bench or from CFD in Solidworks, divided against the potential CFM for that specific combustion chamber type and multiplied by 100. *See equation below.*

$$VE = \left(\frac{\text{Recorded CFM}}{\text{Potential CFM}} \right) \times 100$$

Baechtel, J. (2011).

Valve Lift (mm)	Live CFM	Potential CFM	VE (%)
1.5	18.59	25.3	73.4783
3	36.94	47.3	78.0973
4.5	52.2	65.7	79.4521
6	62.78	80.2	78.2793
7.5	68.36	91.1	75.0384
9	71.15	98.2	72.4542
10.5	72.54	101.5	71.468
12	73.94	101.1	73.1355

Table 17: Calculated Volumetric Efficiency from using live flow readings and potential at different valve lifts.

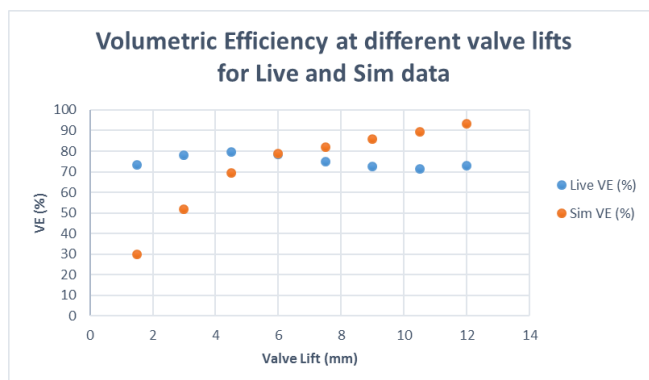


Figure 42: Calculated volumetric efficiency for the live and simulation test data at different valve lifts.

Valve Lift (mm)	Sim CFM	Potential CFM	VE (%)
1.5	7.567429	25.3	29.91078
3	24.46802	47.3	51.72943
4.5	45.65682	65.7	69.49288
6	63.0619	80.2	78.6308
7.5	74.6653	91.1	81.95971
9	84.2507	98.2	85.79502
10.5	90.80914	101.5	89.46714
12	94.34061	101.1	93.31415

Figure 43: Calculated Volumetric Efficiency using sim flow readings and potential at different valve lifts.

Comparing the live and simulated data for volumetric efficiency the simulation data achieves a higher volumetric efficiency but this is at 12mm of valve lift which would be non-existent in the automotive realm especially for the cylinder head tested. Compared at 6mm both achieve around 78% efficiency which is a more achievable/ realistic valve lift.

The live data achieves a more flat/ constant volumetric efficiency but the simulation data achieves higher readings (in green). This is down to surface finishes and limitations to the Solidworks flow software as the flow readings keep increasing through the valve lifts which wouldn't happen in a live test session as seen in the data.

4.3.4 Calculated CFM

A theoretical flow rate can be calculated using the dimensions of an inlet valve but first the curtain area must be calculated. For this calculation the flow efficiency will be assumed as 95% for the valve area. The valve seat inside diameter can easily be calculated using the seat angle (45°) to give a close reading for the actual throat diameter to calculate the theoretical flow rate at different valve lifts.

	Specifications
Valve Diameter (mm)	30
Seat Width (mm)	1.5
Valve Seat Angle (°)	45

Table 18: Valve and seat dimensions for calculating theoretical CFM.

$$\text{Valve seat id} = \text{valve dia.} - (\text{seat width} \times \cos.45)$$

$$28.94 = 30 - (1.5 \times \cos.45)$$

For the calculated 28.94mm at 1mm of lift the numbers must be in m².

$$\text{Curtain Area} = \pi \times \text{Diameter} \times \text{Lift}$$

$$0.0909 \times 10^{-3} = \left(\frac{28.94}{1000}\right) \times \pi \left(\frac{1}{1000}\right)$$

To calculate at alternative valve lifts the curtain area equation needs to be repeated for those valve lifts so the correct CFM can be theoretically calculated. *See following page for alternative valve lift CFM readings.*

Calculating CFM for 1mm of valve lift and assuming efficiency of 95%.

$$CFM = A \times C \times 213685.34$$

$$18.47 = (0.0909 \times 10^{-3}) \times (0.95) \times (213685.34)$$

The theoretical flow rate for 1mm of valve lift with an efficiency of 95% is 18.47CFM.

By changing the valve lift figure in the curtain area equation, the correct curtain area figure can be found for each valve lift and thus finding the theoretical CFM for each valve lift to be compared to the live and simulated data.

As expected as the valve lift increasing so does that of the curtain area, giving more room for air the exit the port and enter the cylinder. The bigger number for the curtain area will produce a higher CFM through calculation.

Valve Lift (m)	Curtain Area (m ²)
1.5	0.000136
3	0.00027
4.5	0.00041
6	0.00055
7.5	0.00068
9	0.00082
10.5	0.00095
12	0.0011

Table 19: Calculated Curtain area for each valve lifts.

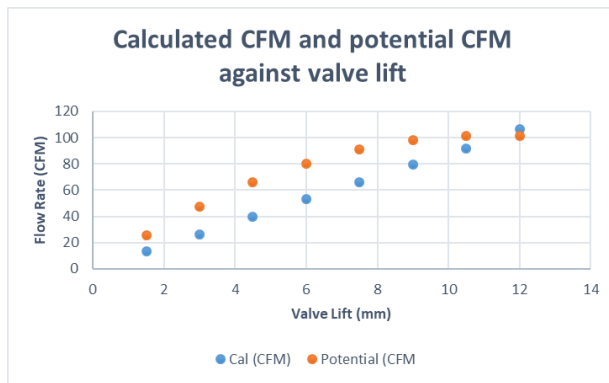


Figure 44: Calculated theoretical CFM and potential CFM plotted against valve lift.

	Calculated (CFM)	Potential (CFM)
1.5	13.14	25.3
3	26.1	47.3
4.5	39.63	65.7
6	53.16	80.2
7.5	65.73	91.1
9	79.26	98.2
10.5	91.83	101.5
12	106.3	101.1

Table 20: Calculated theoretical CFM for each tested valve lift.

The equations used to calculate the theoretical CFM use the valve dimensions and seat dimensions but do not consider the port minimum diameter and port length and therefore without added parameters the data is given very linearly with no drop off in CFM as what would be seen in the live and simulated test sessions.

As plotted with the potential CFM, the readings only are similar at low and high-end valve lift. As there is no consideration for pressure differences the calculated data will continue to increase and that's the reason the two data sets meet at high valve lift. Realistically, the calculated would have a similar curve to the potential but with less CFM at each valve lift as when the live data is compared.

There is no consideration into air temperature, pressure and surface finish which will all effect final CFM readings. See next section for a comparison between live, simulated and calculated data.

4.4 Live, Simulated and Calculated CFM comparison

This section compiles all the data gathered throughout the report on flow rate against specific valve lifts in the correct testing specifications.

Valve Lift (mm)	Flow Rate CFM/sqin						
	Calculation		Live	Simulation			
	Potential	Original	Original	Original	Mod1	Mod2	Bigger Valves
1.5	25.3	13.1	18.6	2.6	16.3	15.8	29.8
3	47.3	26.1	36.9	13.4	33.2	34.2	45.5
4.5	65.7	39.6	52.2	42.8	51.8	51.7	55.6
6	80.2	53.2	62.8	48.0	64.3	64.7	67.2
7.5	91.1	65.7	68.4	65.2	73.3	74.7	77.4
9	98.2	79.3	71.2	76.4	80.2	82.3	84.1
10.5	101.5	91.8	72.5	84.8	85.6	86.6	86.9
12	101.1	106.3	73.9	86.3	88.6	90.8	90.4

Table 21: All calculated, Live and Simulated data for flow rate at the same valve lifts.

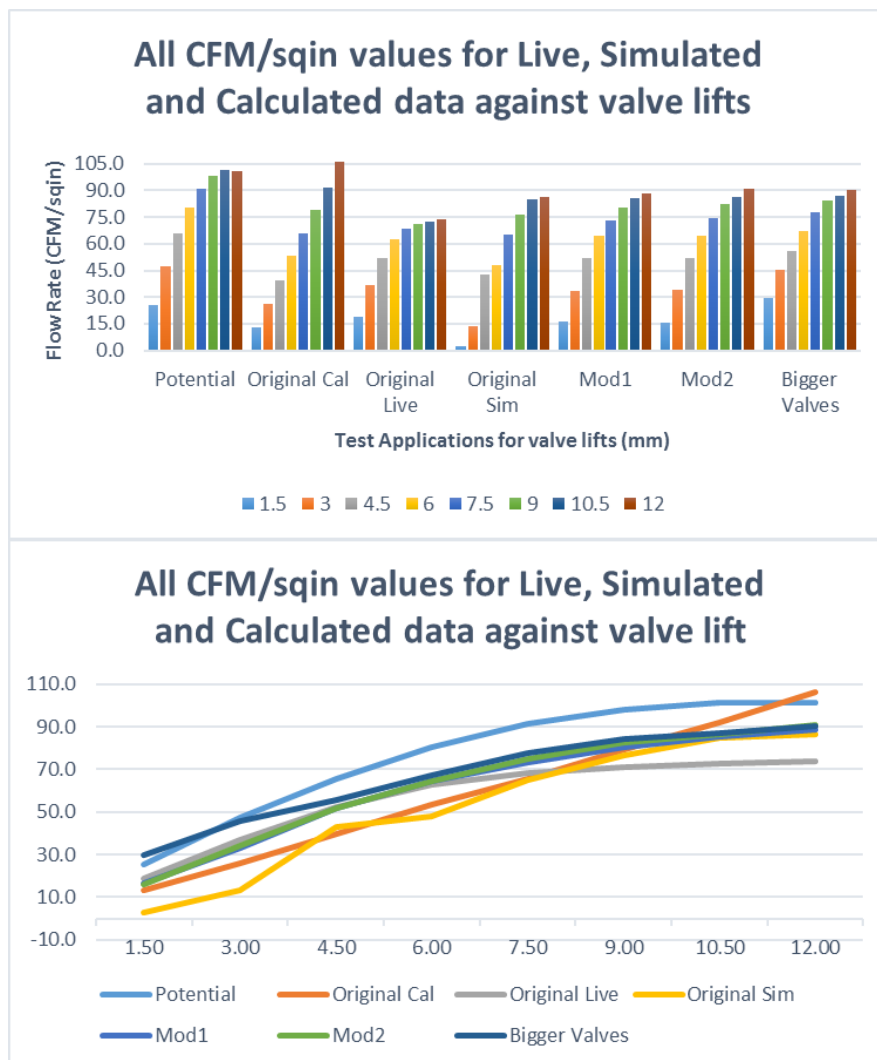


Figure 45: Two graphs depicting all tested CFM values and calculated with the potential against valve lift.

Looking at the graphs above the original port that was tested live has the lowest CFM values above 7.5mm of valve lift showing that the port restricts the air flow above this valve lift as at a quarter of a valve diameter of lift the port becomes more restrictive than the valve ($4/D$).

The original live and the two modified ports have very similar readings at lower valve lifts but at 7.5mm of lift the two modified ports have an increase CFM value compared to the original port. Again, this shows where the port becomes more restrictive than the valve and its amount of lift.

Unfortunately, the original simulation port does not give great data readings for lower valve lifts but between 4.5mm and 9mm of valve lift the data is very similar to the calculated data. The low data readings could be placed down to poor mesh set-up before testing and/or errors with the model and simulation process.

As the calculated data is based upon engine measurements and not efficiencies and actual testing the data does not flatten out at high valve lifts and keeps a linear upwards pattern where it should flatten out like all the tested data sets. Even though, the calculated gives a reliable data reading for up to max valve lift, often meeting the same figures as the tested ports, it can't be taken as very accurate compared to the actual live and simulation data which captures velocity, pressure and CFM.

The only time the potential CFM is met by any of the data traces is when the bigger valve original port is at low valve lifts between 1.5mm and 3mm. This shows that the increase in valve diameter increases CFM and by looking back at section 4.2.10 the velocity of the air entering the cylinder through the port has stayed relatively high compared to the other simulation sessions.

For outright CFM output at all valve lifts without affecting intake air velocity the best modification would be the 1st modification in relation to port modification but the standard port with bigger valves has a great CFM rate at low valve lifts and average at mid and high-range without affecting the velocity of the entering air.

To complete solidify the result a possible design of incorporating the 1st modification but with the slightly bigger valves to see how CFM and velocity act during the valve lifts.

5. Conclusion

To conclude, through live and simulation testing and calculated results the efficiency and performance has been found for the original cylinder head in live and simulation testing and modifications made throughout simulation testing to see what areas of port design can the CFM and velocity be improved throughout valve lifts.

5.1 Live data

The live data gave a real insight into the flow capabilities of the standard cylinder head port for a 1.6, 16 valve Ford Zetec and data in the flow manual gave the potential flow figures for the type of cylinder head tested (Pent-roof).

Testing took place twice and an average of the data was found even though testing figures were very close throughout valve lifts. The data taken was a good base point in relation to any modifications made in the simulation software but air velocity is very important to for a well flowing head, such a parameter the flow bench could not give.

If the testing was undertaking three or more times with an average and the temperature of the air considered in the data then more accurate results might have been achieved. Furthermore, a better method to seal the dummy cylinder bore and cylinder head (Blue-tac) would have been better even though no leaks were detected before testing.

The live data was reliable in the sense that each time the cylinder head was tested very similar results were recorded and through research the data came back conclusive. No anomalies were found in the data and the CFM readings increased in a steady linear state and started to flatten out around the same valve lift as the potential readings (as seen in the flow bench manual).

5.2 Simulation data

Initial simulation readings showed a higher CFM value than that of the live tested port. After this, research gave a figure for the surface finish for a cast Aluminium cylinder head but unfortunately the limitations of the flow software in Solidworks gave no change in flow characteristics throughout any valve lifts.

Even though initial simulation data came back inconclusive the further data that came from tested, backed up from research, showed that the data that was being achieved was realistic according to calculated data for the specific engines cylinder head. See as all the CFM figures were all in the same range gives consistency even through the modification made within Solidworks.

With more testing time an additional simulation session would have incorporated the best CFM and velocity port (1st Modification) with the increase in valve size to see how this would affect CFM and velocity throughout the valve lifts.

Additionally, with more time, more advanced software could have been used with Ansys being a prime example. This should generate more accurate results being as more variable can be set before testing takes place in a more advance software. The downside is the complexity and the time it would take the set-up an accurate model, and to test to achieve useful results. With this Solidworks Flow Simulation was still a very useful tool and with even more testing time and additional knowledge even more reliable results could possibly be obtained.

5.3 Calculated data

The calculated data was achieved from equations from multiple engineering books/ journals. The equations for potential Horsepower, potential CFM and potential volumetric efficiency were all derived from- Camshafts and Camshaft Tuning for High Performance Engines by D. Hammill and Performance Automotive Engine Math (SA Design-Pro) by J. Baechtel.

These books and their engine mathematical equations helped understand the theoretical potential from the chosen engines cylinder head. Although these equations are only based upon engine dimensions and known measurements, and not efficiencies and variables like air temperature, they still give an accurate representation for what could and can be achieved. As seen in the graphs produced the only downfall of the equations are the fact they don't represent a drop in CFM at high valve lifts compared to potential, live and simulated data.

5.4 Recommendations for improvement

5.4.1 Cylinder head

Regarding the cylinder head tested improvements that could be made that would benefit a more efficient combustion process would include the move to direct-injection from indirect this is so the fuel wouldn't have to travel the length of the inlet port and be caught behind the inlet valves. Also, the injection can occur at the exact time as the fuel is being injected directly into the cylinder for combustion. Moving the injector out of the port gives more freedom on designability for manufactures of the port as they do not have to work around the injector which is not made to be aerodynamic.

The use of direct-injection would also allow the inlet port to be slightly port and very much polished to a satin finish without the concern of fuel puddling in the port and the increase of hydrocarbons because of the unburnt fuel.

The use of a higher lift camshaft or, if there is sufficient room, bigger valves could be used to theoretically allow more air to enter the combustion chamber but this is all substantial to engine size and power requirements in relation to RPM and emissions and application e.g. automotive or motorsport.

5.4.2 Engine and additional components

As explained previously a longer stroke bore would reduce friction and the losses that come from it, thus producing a more efficient engine. As seen in the research section there are many alternatives to achieving a high volumetric efficiency without the time and research that port design does and these include turbocharging and supercharging. For the instance of turbocharging by using spent exhaust gases that power the turbocharger and increases air density and speed a higher volumetric efficiency can be achieved with possibly less research and more ease than port design. But cost does not give this application the total favour and the increase chance of components that could go wrong.

Apart from turbocharging, higher-lift camshaft, bigger valves and port design the only other way to increase CFM and air flow is to increase to RPM as an engine is essentially a pump that pulls fresh air in and spent gases out. The faster an engine revs the more air can travel into the cylinder bore but this increases heat and decreases the life of moving components.

With insight knowledge of port flowing and the use of a flow bench the CFM and torque and power can be increased easy dependent upon the application with little in the need of money but for big power gains the easiest and most rewarding route would be charging the intake air.

5.4.3 Testing

The testing could have been improved with additional time allowing for more live testing, more simulation testing and an increase knowledge of the subject area. Additional knowledge would decrease the time for testing as the process would be known and the chance of mistakes less likely to happen. As the flow software was first used for this report a more in-depth knowledge needs to be known on the effects of setting up the model for testing.

Although, a lack of initial knowledge for the subject of simulation flow testing the result that were collated were right in the range with the live data and calculated data given some validity to the test results obtained. If the project was to be run again valuable time would be saved in the process of modelling the port from real life into a simulation ready state. This would give more time for the understanding for the simulation software and the possibility of more modification being made from research and test results

5.5 Overall conclusion

Generally, the report went well with very little errors to slow down testing and progress. As said previously if simulation modelling time was reduced even more data could have been gained from port design and a more in-depth conclusion drawn.

By completing live testing twice, the data was defined as reliable as both data sets were almost identical throughout the valve lifts with no leakage present.

To conclude, on the simulation modifications made to the inlet port, the first modification would be the best in terms of port design as it achieves a higher than standard CFM value throughout valve lifts but continues to supply high air velocity throughout the intake port. If more time was available the modifications made would be replicated on the actual port and flow readings obtained but it was found that the effectiveness of porting would not be able to be achieved to obtain similar figures from the simulation design.

Research found suggests that if port modification is completed correctly and checked throughout, significant power gains can be achieved with little cost. Furthermore, this route will increase the volumetric efficiency which is the main aim of this project without effecting the driveability and emissions of the engine.

In terms of references to back up the report it was found that, although there is a lot of data and knowledge of port modification and design, the subject area is dated in terms of when relative papers were published.

In conclusion, it was found that CFM and in-turn volumetric efficiency can be increased and improved by port modification and design. Further testing also displayed that an increase in inlet valve size can also increase CFM at specific valve lifts but with little room in increasing valve size a re-designed port and careful testing would achieve great flow results if the velocity of the air isn't negatively affected by the change in design.

6. References

- Media.ford.com. (2018). Ford 1.0-Litre EcoBoost Wins International Engine of the Year for the 6th Year Running | Ford of Europe | Ford Media Centre. [online] Available at: <https://media.ford.com/content/fordmedia/feu/en/news/2017/06/21/ford-1-0-litre-ecoboost-wins-international-engine-of-the-year-fo.html> [Accessed 5 Jan. 2018].
- MOHAMMADEBRAHIM, A., HANNANI, S.K. and SHAFII, B., 2014. Investigating the effect of intake port geometric parameters and blockage on flow coefficient and in-cylinder flow: Application to engine port design. *Scientia Iranica.Transaction B, Mechanical Engineering*, 21(2), pp. 438-448.
- Sendyka, B. and Nog, M. (2018). Combustion Process in the Spark-Ignition Engine with Dual-Injection System.
- Olympus-ims.com. (2018). Surface Roughness | Olympus IMS. [online] Available at: <https://www.olympus-ims.com/vi/knowledge/metrology/roughness/> [Accessed 10 Jan. 2018].
- Solidsolutions.co.uk. (2018). Computational Fluid Dynamics (Flow) SOLIDWORKS. [online] Available at: <https://www.solidsolutions.co.uk/solidworks/simulation/packages/flow-cfd.aspx> [Accessed 11 Jan. 2018].
- Iyer, C. and Yi, J., "3D CFD Upfront Optimization of In-Cylinder Flow of the 3.5L V6 EcoBoost Engine," SAE Technical Paper 2009-01-1492, 2009.
- Dong, J., Gesheng, L., Yue, Z., and Liang, Z., "Impact of Reformed Ethanol on the Volumetric Efficiency in I.C. Engines," *SAE Int. J. Fuels Lubr.* 3(2):38-46, 2010.
- Okanishi, N., Fukutani, I., and Watanabe, E., "Volumetric Efficiency Characteristics of a Crankcase-Supercharged 4-Stroke Cycle Engine with Rotary Disc Valves," SAE Technical Paper 900174, 1990.
- Wilson, T., Haste, M., Xu, H., Richardson, S. et al., "In-cylinder Flow with Negative Valve Overlapping - Characterised by PIV Measurement," SAE Technical Paper 2005-01-2131, 2005.
- Sandquist, H., Wallesten, J., Enwald, K., and Strömberg, S., "Influence of Valve Overlap Strategies on Residual Gas Fraction and Combustion in a Spark-Ignition Engine at Idle," SAE Technical Paper 972936, 1997.
- Stone, R. and Kwan, E., "Variable Valve Actuation Mechanisms and the Potential for their Application," SAE Technical Paper 890673, 1989.

- Zirngibl, S., Held, S., Prager, M., and Wachtmeister, G., "Experimental and Simulative Approaches for the Determination of Discharge Coefficients for Inlet and Exhaust Valves and Ports in Internal Combustion Engines," SAE Technical Paper 2017-01-5022, 2017.
- Ma, J., Stuecken, T., Schock, H., Zhu, G. et al., "Model Reference Adaptive Control of a Pneumatic Valve Actuator for Infinitely Variable Valve Timing and Lift," SAE Technical Paper 2007-01-1297, 2007.
- 570sx.blogspot.co.uk. (2017). Turbocharger Understanding Guide How a Turbo Works Boost Basics. [online] Available at: <http://570sx.blogspot.co.uk/2009/10/turbocharger-understanding-guide-how.html#!/2009/10/turbocharger-understanding-guide-how.htm>.
- Kirkpatrick, A. and Ferguson, C. (2000). Internal combustion engines: Applied thermosciences. 2nd ed. New York: John Wiley & Sons.
- Stoakes, G. (2012). Level 3 diploma principles of light vehicle technology. 1st ed. Oxford: Heinemann.
- Stoakes, G. (2011). Level 2 principles of light vehicle maintenance and repair. 1st ed. Oxford: Heinemann.
- Zammit, S. (1996). Motor vehicle engineering science for technicians. 1st ed. Harlow, England: Addison Wesley Longman Limited.
- Twigg, P. (1996). Science for motor vehicle engineers. 1st ed. Waltham, Mass.: Butterworth-Heinemann.
- Heisler, H. (2001). Advanced vehicle technology. 1st ed. Oxford [u.a.]: Butterworth-Heinemann.
- Chen, A., Lee, K., Yianneskis, M. and Ganti, G. (1995). Velocity characteristics of steady flow through a straight generic inlet port.
- Baechtel, J. (2011). Performance automotive engine math. North Branch, MN: CarTech.
- Vizard, D. (2012). David Vizard's how to port & flow test cylinder heads. North Branch, MN: CarTech.
- Bettes, H. (2014). Engine airflow hp1537. New York: HP Books.
- Baechtel, J. (2011). Performance automotive engine math. North Branch, MN: CarTech.
- Smith, P. and Morrison, J. (2009). Scientific design of exhaust & intake systems. Cambridge, Mass: Bentley.
- Iyer, C. and Yi, J., "3D CFD Upfront Optimization of the In-Cylinder Flow of the 3.5L V6 EcoBoost Engine," SAE Technical Paper 2009-01-1492, 2009.
- Okanishi, N., Fukutani, I., and Watanabe, E., "Volumetric Efficiency Characteristics of a Crankcase-Supercharged 4-Stroke Cycle Engine with Rotary Disc Valves," SAE Technical Paper 900174, 1990.
- Thomas, T. (1999). Rough surfaces. London: Imperial College Press.
- Kalpakjian, S. and Schmid, S. (2001). Manufacturing engineering and technology. Upper Saddle River, NJ: Prentice Hall.

- Groover, M. (2011). Fundamentals of modern manufacturing. John Wiley.
- Laserdesign.com. (2018). What Is 3D Scanning | Laser Design. [online] Available at: <https://www.laserdesign.com/what-is-3d-scanning> [Accessed 21 Feb. 2018].
- Wright, I. (2018). Quality Basics: How Does 3D Laser Scanning Work? [online] engineering.com. Available at: <https://www.engineering.com/> [Accessed 21 Mar. 2018].
- Concept, Product, Design, Engineering, Analysis, & Testing - PES - Performance Engineered Solutions. (2018). Computational Fluid Dynamics - Concept, Product, Design, Engineering, Manufacture, Analysis, & Testing - PES - Performance Engineered Solutions. [online] Available at: <https://www.pes-performance.com/analysis-and-simulation/computational-fluid-dynamics/> [Accessed 21 Mar. 2018].
- Hawkridgesys.com. (2018). SOLIDWORKS Flow Simulation for CFD Analysis | Hawk Ridge Systems. [online] Available at: <https://www.hawkridgesys.com/products/solidworks-flow-simulation> [Accessed 5 Apr. 2018].

7. Appendices

