2016-04-07

Critical Flow Analysis and Understanding of Automotive Exhaust Ports

Kevin Boutsen
University of Miami, kevbou9790@gmail.com

UNIVERSITY OF MIAMI

CRITICAL FLOW ANALYSIS AND UNDERSTANDING OF
AUTOMOTIVE EXHAUST PORTS

By

Kevin Boutsen

A DISSERTATION

Submitted to the Faculty
of the University of Miami
in partial fulfillment of the requirements for
the degree of Doctor of Philosophy

Coral Gables, Florida

May 2016

UNIVERSITY OF MIAMI

A dissertation submitted in partial fulfillment of
the requirements for the degree of
Doctor of Philosophy

CRITICAL FLOW ANALYSIS AND UNDERSTANDING OF
AUTOMOTIVE EXHAUST PORTS

Kevin Boutsen

Approved:

Michael Swain, Ph.D. Associate Professor of Mechanical and Aerospace Engineering	Landon Grace, Ph.D. Assistant Professor of Mechanical and Aerospace Engineering
Ryan Karkkainen, Ph.D. Assistant Professor of Mechanical and Aerospace Engineering	Guillermo Prado, Ph.D. Dean of the Graduate School
Matthew Swain, Ph.D. President Analytical Technologies Incorporated Miami, Florida	

BOUTSEN, KEVIN (Ph.D., Mechanical Engineering)
Critcal Flow Analysis and Understanding of Automotive Exhaust Ports (May 2016)

Abstract of a dissertation at the University of Miami.

Dissertation supervised by Professor Michael Swain.
No. of pages in text. (110)

The exhaust process in an internal combustion engine directly affects the ability to produce power. Currently, experimental testing is used to test and perfect the exhaust process using a steady state low pressure flow bench. The flow through the exhaust port is initially critical which is not captured by the conditions of testing set on a flow bench. The goal of this study was to understand critical “blowdown” flow in exhaust ports to ultimately extract more performance out of an engine. A sonic flow bench was used in order to study the critical flow phase of an exhaust stroke.

Blowdown testing using a sonic flow bench alongside CFD testing and low pressure flow bench testing was performed on three cylinder heads and a sonic nozzle. The three methods combined allowed to come to conclusions on how the blowdown phase works and how it can be optimized.

The overall results revealed that the blowdown event can be split into two phases in which the nature of the flow is characterized differently. At high pressures, immediately following the opening of the valve, a development phase occurs where the mass flow rate starts from zero and eventually reaches a theoretical mass flow rate. This zone is very responsive to changes in geometry in the flow region affecting the overall mass flow rate

during blowdown by more than 10%. This phase is characterized by the flow accelerating through the valve opening thus meaning it is highly transient. The second phase of the flow is called the fully developed region where the flow reaches a quasi-steady state matching theoretical values for mass flow rate.

The high pressure development phase is most important for port design. By using a low pressure flow bench this is not captured as this is the result of a transient effect. Blowdown in an engine occurs extremely fast in time. This means that the blowdown flow never reaches a quasi-steady state and is always in the transient development phase. Valve and seat design has a large influence on performance. The results from testing valves and seat at different angles and with different design ideas allowed to conclude that a shape that resembles a venturi works best at low pressures however the opposite is needed for good blowdown. Abrupt transitions in shape promote the transient phase to last longer and go above the theoretical maximum. Using the right combination of blowdown performance and low pressure performance is therefore the goal to increase exhaust performance as a whole. This study shows the tools and designs needed to increase blowdown performance and in turn increase the overall performance of an exhaust stroke.

ACKNOWLEDGMENTS

I would like to thank my advisor Dr. Michael Swain for all his support during the research and the invaluable help and ideas provided throughout.

I would like to thank Dr. Matthew Swain for all his help during the project and for all the data collection and analysis.

I would like to thank my committee members for all their suggestions and continuously challenging me throughout the project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

Chapter 1 Introduction

Motivation and Goals

The exhaust process in an internal combustion engine directly affects the ability to produce power. The efficiency of that phase is very important for engineers as it can provide significant improvements in performance. Currently, experimental testing is used to test the exhaust process using a steady state low pressure flow apparatus called a flow bench. Low pressure testing doesn't mimic the flow conditions in the engine as the pressures are much greater at the beginning of the process and the flow is transient in the engine. The flow through the exhaust port is initially critical due to the very high pressure differential between the cylinder and the downstream pressure (atmospheric). It then transitions into sub-critical flow once the pressures have lowered.

Critical flow through a valve implies that any downstream change in pressure will not affect the mass flow rate through the system. It is currently assumed in engines that critical flow occurs through the entire valve curtain during blowdown with no downstream effects on flow rates. Exhaust valves open under high cylinder pressures causing gases to quickly exit through the valve. This causes a rapid drop in pressure. This is referred to as blowdown. A study published in 2013 by Jeremy Decker and the Internal Combustion Laboratory at University of Miami shows that this is not the case [1]. Downstream changes have an effect on the critical air flow. The end of that study is the starting point for this study. The dissertation showed the unexpected phenomenon experimentally using a purpose built sonic flow bench for analysis. It did not, however, explain and provide an understanding of why the changes occurred.

The goal of this project is to understand critical “blowdown” flow in exhaust ports to ultimately extract more performance out of an engine. The understanding and explanation of the flow is important to be able to design the most efficient exhaust shape.

Experimental testing was done to understand flow in an iterative manner by modifying existing exhaust ports. This allowed the recording of what changes affect the flow rate during an exhaust event. The use of Computational Fluid Dynamics (CFD) to visualize the flow and compare it to experimental data is a challenge and was used to provide an additional level of understanding.

The use of sonic nozzles to capture the ideal behavior of a gas through a small orifice has been proven to work very successfully. These nozzles are used as flow meters for high pressure differentials. The exhaust port should behave in the same way. The testing of a sonic nozzle in addition to exhaust ports allowed the verification of the experimental process. It also gave an opportunity to understand how the downstream modifications could alter flow through a small orifice and what different phases occur in the blowdown process.

Why study critical “blowdown” flow?

Critical “blowdown” exhaust flow has to be investigated because the theory for designing engines is being challenged by the findings of this research and the research published in 2013 [1]. Experimental results statistically show a change in flow rate proving the downstream shape can affect the flow. The improvements in performance on the whole engine due to blowdown performance are hard to predict due to the fact that most engine simulation software such as Ricardo Wave currently assume choked sonic flow for blowdown. The engine however can be tested on dynamometer and by only changing the exhaust port shape one can record the power change.

Weld Tech, a C.N.C. head porting company for high performance engines, has reported increases of performance of half a percent in power output when “blowdown” performance was increased. Interestingly, Weld Tech stated “bettering the flow on a steady airflow, low pressure flow bench resulted in lower engine performance” which shows that low pressure exhaust testing is insufficient. This has been confirmed in the research published in 2013 by Jeremy Decker [1].

It is important to investigate this phenomenon in more detail to fully understand blowdown and pinpoint the mechanism which allows the flow to be altered downstream. This study is designed to investigate this matter using different methodologies to advance the overall knowledge on the process.

Chapter 2 Theory, Design and Testing of Current Exhaust Ports

Theory

The exhaust process occurs when the exhaust valve opens while the piston is still descending and continues as the piston is moving up within the cylinder pushing the burnt air-fuel mixture out of the system. Figure 2-1 is a diagram showing the path the exhaust gas takes out of the cylinder when the piston is moving up [2]. The pressure differential between the cylinder and the exhaust manifold is very high hence this is the initial and primary mechanism of exhausting the burnt gases. The pressure at the exit of the exhaust system is atmospheric whereas when the exhaust valve opens the pressure inside the cylinder can have a value from 4 to 16 atmospheres.

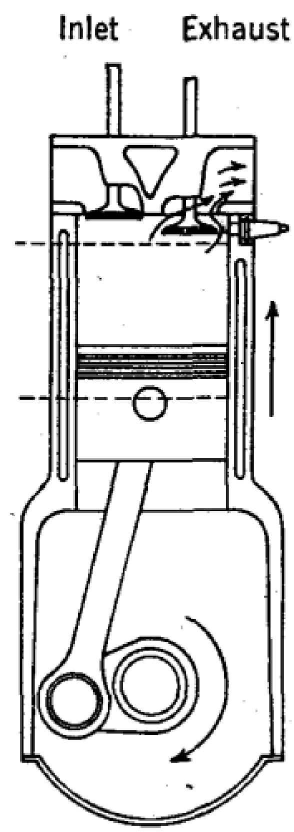

Figure 2-1 - Diagram showing the exhaust process in a 4 stroke internal combustion engine [2]

Exhaust flow occurs in two different phases due to the high pressure differential; there is a critical phase and a sub-critical phase. Critical flow occurs when the pressure differential between two volumes is high enough. It occurs in cases when there is a restriction in area such as a valve in a pipe. The flow conditions upstream of the restriction are subsonic and due to conservation of mass, the fluid flow is required to increase its velocity through the smaller area. In this case, the velocity required to conserve mass reaches sonic speeds causing the mass flow rate to be restricted by the upstream pressure. The downstream pressure in turn has no effect on the mass flow. Mass flow rate is fixed by the upstream pressure when the pressure differential is high enough. The velocity through the valve is fixed independent of upstream pressure. The mass flow rate can however be changed by changing the pressure of the fluid upstream due to a change in density.

For choked flow to occur for an ideal gas the following condition must be met:

$$\frac{p_e}{p_c} \leq \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}} \text{............ (1)}$$

Where

$$\gamma = \frac{c_p}{c_v} \text{............ (2)}$$

In the case of the engine, with a valve restriction as shown in Figure 2-1, p_e and p_c are exhaust manifold and cylinder pressure. The condition for which the flow is critical is determined by the fluid property which is the specific heat ratio γ. Specific heat ratio is primarily a function of temperature and it decreases with an increase in temperature [3].

The specific heat ratio for exhaust gases is: $\gamma = 1.28$

For air at 70°F: $\gamma = 1.40$

Therefore critical flow occurs at pressure ratio 0.549 for exhaust gases and 0.528 for air [4]. This means that if the cylinder pressure is at least 1.82 times as high as exhaust (atmospheric) pressure the flow will be critical. It will be dictated only by the velocity of the fluid through the valve curtain area and the upstream pressure.

To summarize, according to ideal gas theory, when the critical pressure ratio is reached, further reductions in the downstream pressure have no effect on upstream flow [5]. The mass flow is at its maximum and constant. The key word in this statement is "ideal" gas which obeys the ideal gas laws making the derivation and use of the equation 1 possible. The ideal gas is composed of many randomly moving point particles that do not interact except when they collide elastically. The equation above also includes assumptions of one dimensional flow of an isentropic, adiabatic gas but give insight into the flow's behavior [4]. Real gases behave differently and it could be the reason this flow phenomenon is being challenged in this research. Nevertheless, it is accepted by engine designers that the shape of the port has no effect on the blowdown phase. This is challenged by the research results in Jeremy Decker's work [1].

Several studies have documented changes in flow rate beyond the critical pressure on certain geometries. In case of a convergent divergent nozzle, if it is shaped well the maximum flow ratio from experimental and theoretical calculations agree very closely [6].

This type of nozzle, also called a sonic nozzle is used as a flowmeter in industry due to the fact that the geometry agrees with the theory for restricted sonic flow. In the case of this type of nozzle there are actually effects that are not taken into account in theory. According to Massey, in practice, the position of the critical velocity occurs slightly downstream of the minimum area due to viscous effects [5]. The fact that the flow is transient in the problem definition may also add to the complexity of the flow being fully formed. When the valve opens the flow does not immediately settle to a quasi-steady state since the valve in an engine is moving and the pressure upstream is changing at a rapid rate. Square edged orifices have also been known to behave differently than theory suggests. When the flow is critical, the minimum flow area can move to a position downstream from the nozzle [6]. This minimum area is called a vena contracta and is visualized in Figure 2-2 for a square edged orifice flow [7].

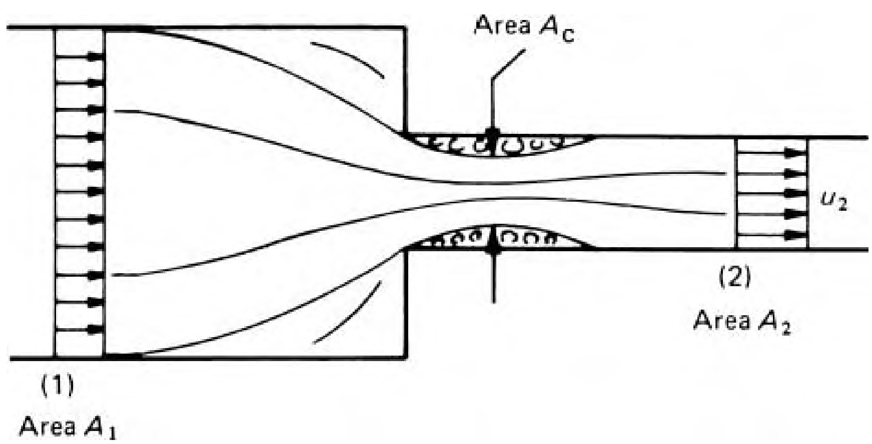

Figure 2-2 - Visualization of the vena contracta [7]

The diameter of the vena contracta can change with flow rate and could affect the flow especially on a more complex geometry such as an engine valve. The theory also could be challenged by the fact that the localized pressures in blowdown flow might change the pressure ratio close the valve curtain area. Most studies on sonic flows through area reductions are done at steady state and the transient effects are not investigated. In the case of an engine this is very important and could play an important role in explaining why the flow rate in an exhaust port can vary with downstream changes at pressure ratios above 1.82.

Design and Testing of Current Exhaust Ports

It is important to have an understanding of how exhaust ports are currently being designed and why this technique is deemed inappropriate and incomplete for a full understanding of exhaust flow. To improve this technique it is important to recognize what needs to be improved and if the improvement is worth the required effort.

Currently the design and testing of exhaust port flow is done at steady state at a pressure differential of 28 inches of water (1.012 psi) on a flow bench. The flow is recorded at different valve lifts and these numbers are used to predict how the flow will behave in a transient manner in the engine. This pressure differential is extremely low compared to blowdown exhaust pressures. The flow is therefore turbulent but much slower. Ports are designed to flow more air through trial and error and experience. The goal is to get the most air to exit in a given time out of the exhaust port on the flow bench which should in turn translate to increased engine performance. This has been shown to be the case in intake valve flow where the pressure upstream and downstream of the valve are closer in value. The flow is in majority subsonic and this technique can show relatively good correlation between increased air flow and power increase on the engine. For exhaust flow, testing gives insight into what is happening in sub-critical region however does not take into account the critical flow region. The setup for a flow bench is illustrated in Figure 2-3.

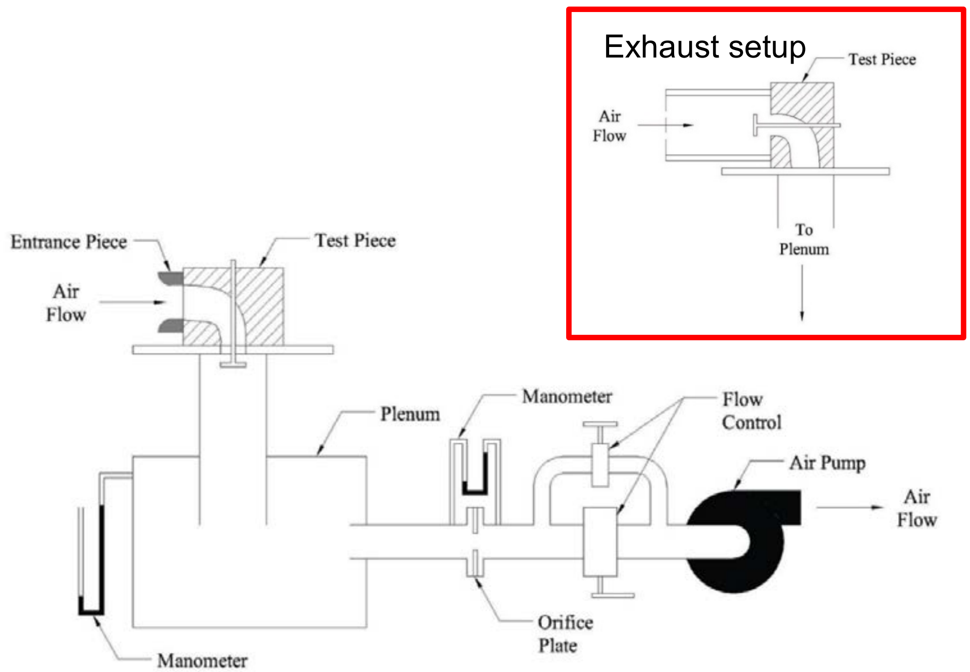

Figure 2-3 - Intake and Exhaust Flow Bench setup

The air pump shown in black pulls the air through an orifice plate of known flow characteristics and the pressure differential between the upstream and downstream is set to 28 inches of water. The pressure in the plenum can then be measured and this can be converted to the volume flow rate across the valve curtain. In the exhaust case the test piece is inverted so that the flow goes through the valve in the correct direction.

According to Weld Tech, a current high performance supplier of cylinder heads *“bettering the flow on a steady airflow, low pressure flow bench resulted in lower engine performance”*. This was done on a NASCAR cylinder head at Weld Tech and the results were duplicated in the University of Miami Internal Combustion Engines lab for confirmation. The results are shown below and are part of the work done in the previous study by Jeremy Decker [1].

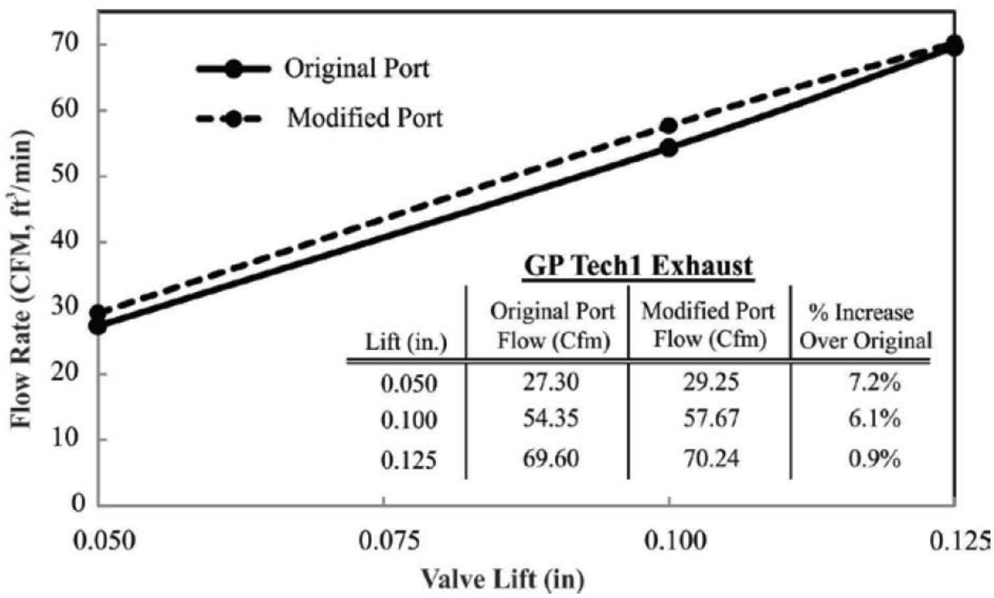

Figure 2-4 - Flow Bench results for GP Tech1 NASCAR Exhaust [1]

The modified exhaust port is more efficient on the flow bench however does not give as much power in the engine. The answer to this had to lie in the critical flow changes that cannot be tested on a regular flow bench.

Change in Testing Methods

A change in testing methods was necessary to capture what happens at higher pressure differentials as well as seeing if the flow would change in critical conditions contrary to what the belief is. The study by Jeremy Decker [1] created a new testing method for exhaust flow to capture the critical "blowdown" phase of flow.

Some criteria needed to be met to capture the exhaust flow in its entirety better:

- Test of cylinder pressures two orders of magnitude above current flow bench are needed.

- Testing had to be done in a transient way with pressures much closer to exhaust pressures.

- The measuring device needs to capture pressure changes at a high frequency to observe possible changes in transient flow.

The setup used for the new testing was created during the study by Jeremy Decker and was used throughout this research. The results revealed that blowdown performance cannot be measured on a conventional flow bench and is the basis of the future work and goals set for this research. Understanding the reason why and how the flow changes can be found using this experimental setup. This is because it allows to capture performance changes that match with the subsequent test on the engine as a whole which gave more performance. The new setup is illustrated in Figure 2-5.

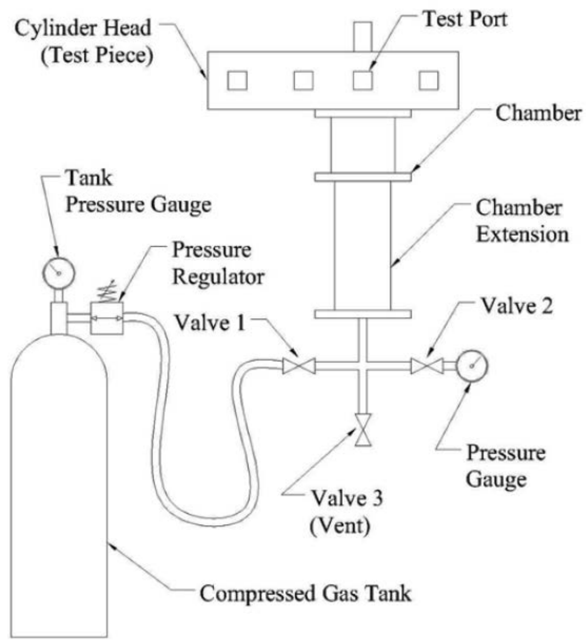

Figure 2-5 - Blowdown Flow Bench Design

The setup of this new flow bench utilizes a tank at high pressure to supply gas into the chamber using Valve 1. Once the chamber is filled to a certain pressure, it can be released by opening the exhaust valve to a set lift. A pressure gauge attached to the pressurized chamber reads the pressure change once the release has been triggered. This method therefore allows to simulate blowdown at one lift position. In the engine the blowdown occurs as the valve is moving however this was deemed to be too complex of a solution to implement. In addition to this, blowdown occurs in the beginning of the valve opening and does not occur once the valve curtain area is too high.

The pressure sensor used for the critical flow testing is an Omegadyne PX209-300G10V. The pressure range is 0-300 psi with an output voltage of 0-10V. The sensor is connected to a Fluke 192B ScopeMeter oscilloscope which records the pressure drop during blowdown. The pressure range is within the testing range that is of interest to this project. The blowdown will be started with a chamber at a gage pressure of 120 psi.

The head initially tested on the regular flow bench was retested on the blowdown flow bench to see what the results yield and if they correlate to the performance readings from the engine.

Figure 2-6 shows that the original chamber works better at extracting the gases from the exhaust due to downstream changes from the valve. This is contrary to the results from low pressure testing. This goes against the theory discussed earlier where both ports should perform the same way regardless of shape. It does however correlate with engine performance observed. The reason why a change occurs is unknown and finding the reason is one of the goals of this study. Understanding the flow required further experimental testing to determine what areas of the port affect the critical flow. The use of CFD allowed visualization of what is happening. Further testing on a sonic nozzle also helped gain an understanding of this change in flow. It also confirmed that the testing procedure used and apparatus are a valid form of testing transient sonic flow.

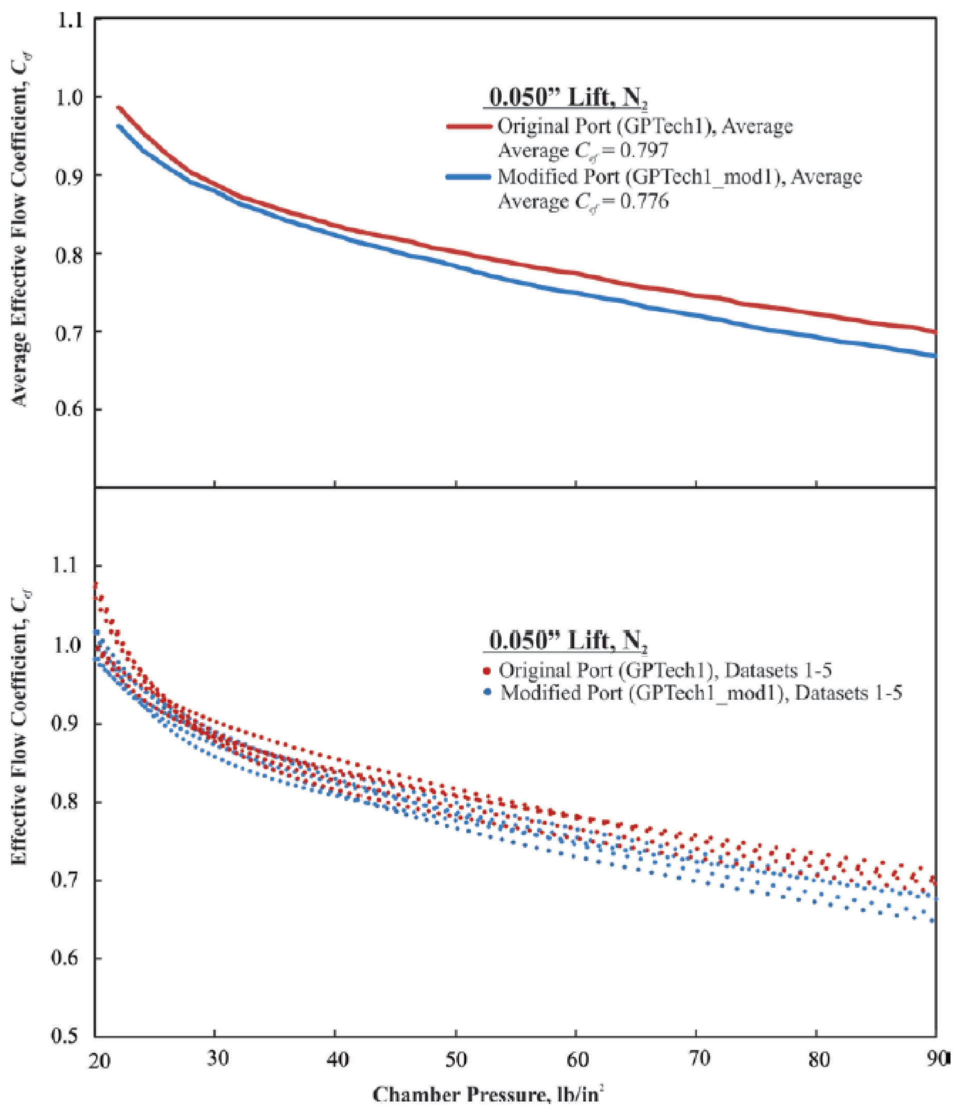

Figure 2-6 - Effective flow coefficient vs chamber pressure results for the original and modified GP Tech1 cylinder head on blowdown flow bench

Chapter 3 Blowdown Testing of Cylinder Heads

Methodology and Planning

Understanding exhaust flow in a comprehensive manner is the goal of this study. It allows for better exhaust design and improved methods of optimization. Testing of different styles of ports is necessary to understand the effects of different downstream shapes on flow. This was done using a combination of experimental and CFD testing in the optic to work together to provide an understanding of the flow and a possibility to provide a better design methodology for design. Capturing the behaviors observed in the previous study [1] using an established methodology of testing and analysis on different styles of ports will provide a new understanding.

Two different types of ports are experimentally tested to understand how changes affect each shape respectively. Both cylinder heads tested have the exact same valve size and valve seat shape. One type of exhaust port is die-cast and one is created with a sand-core. The die-cast cylinder head part number is RF-E9SE-6090-D7A. The sand-core cylinder head part number is RF-YF2E-6090-A22A. The sand-core cylinder head has an advantage in manufacturing because the port can be shaped in a more complex manner with smoother curvature. This technique is usually used for high performance applications. Conversely, a die-cast manufacturing technique restricts what you can achieve and sharp angles in the shape of the exhaust are usually the result and used on lower performance vehicles.

In an effort to see if there are any flow changes and understand them, different downstream configurations will be tested on each port. Since the two ports have the same valve and seat angle when open it is also possible to compare them to each other. The possibilities for the modification of downstream flow are numerous including:

modifications downstream of the port, inside the port, at the valve and seat as well as in the combustion chamber. To show a change in the flow due to downstream effects a statistical method is used to establish if the flow has changed between two different shapes. The number of samples of air flow needed for each test and how to compare two pressure curves statistically was studied for the data to be accurate and precise. This is discussed in Appendix A and Appendix B.

Experimental Testing

Two previously untested exhaust ports were chosen for testing using the blowdown flow bench. A die-cast (low performance) and a sand-core (high performance) exhaust were chosen for the experimental analysis. The cylinder heads are both from 3.8L V6 engines from Ford. These two port types were tested on the blowdown flow bench with different downstream configurations. Statistical analysis was done to determine if downstream changes have an effect on the flow or not. All testing is completed at 0.1 inch lift for the cylinder head.

Test of Downstream Extension Tubes

The first step in the testing was to create a simple way to test the pressure drop in both ports with a change to their downstream shape without changing the valve curtain area. This was done by using a downstream divergent tube extension added to the port exiting into environment. This is pictured in Figure 3-1. It changes the flow shape 2.5 inches from the exhaust valve and should therefore not change the flow through the valve according to theory. Tests with and without the extension were carried out to see if they had an effect on mass flow. The modifications made to the GP Tech1 head in the previous study were much closer to the valve curtain [1]. In this case the modification is 2.5 inches downstream which is less likely to affect the flow but is the first step in the investigation.

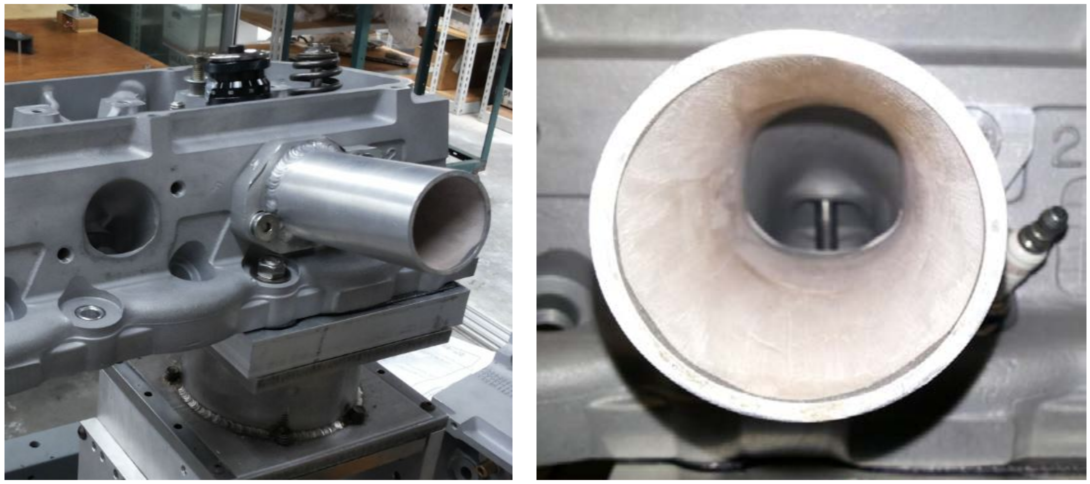

Figure 3-1 - Downstream tube configuration as extension to the port

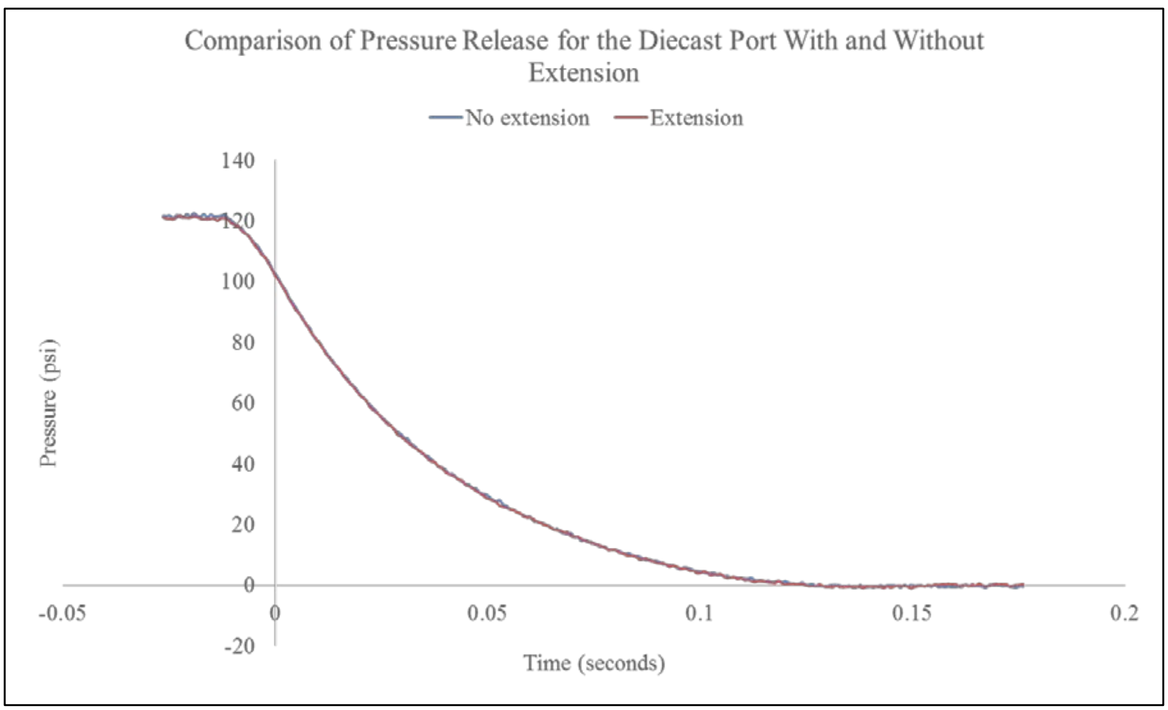

Figure 3-2 - Pressure release through the die-cast port with and without the tube extension

Figure 3-2 shows the pressure release for the die-cast port which is the low performance port with a very abrupt turn close to the valve (short side radius). The pressure in the graph is the gage pressure. The graph presents the pressure drop with and without the tube extension. The pressure drop in the chamber seems to be matched all along the curve. It indicates that the downstream change has no effect on the flow rate out of the cylinder. This is confirmed statistically by using the Nash-Sutcliffe method to detect a change in flow. This method is explained and detailed in Appendix B.

A statistical analysis using Analysis of Variance (ANOVA) was also performed to know how many samples needed to be taken of each configuration. In order to get accurate results for the pressure release, the minimum number of samples necessary per configuration is 7. This is discussed in Appendix A. The raw data gathered in the runs is aligned at 100 psi for different runs. This is because the starting pressure between configurations is not always the same. The crossing point between the data samples is at 100 psi because it is deemed that the valve is fully open at that pressure. From that point if there is a difference in curve path then there is a difference between the two configurations.

Converting Pressure Data to Mass Flow Rate

In order to understand the meaning of the pressure drop it is useful to plot mass flow rate versus pressure. This allows to visualize the performance of the modifications made to the downstream shape. The mass flow rate from the port can also be compared to a theoretical flow rate for a nozzle with the same flow area. This can show how far the results for the port are from the ideal case and how much performance could be gained assuming the flow cannot surpass the theoretical flow. The mass flow rate graph allows to compare data from two different tests but also benchmark it against the theoretical flow rate. The experimental pressure data is converted to mass flow rate using the following process.

For each time step n, the temperatures of the gases within the chamber are estimated assuming adiabatic, isentropic expansion:

$$T_{n+1} = T_n \left(\frac{P_{n+1}}{P_n}\right)^{\frac{\gamma-1}{\gamma}} \text{............ (3)}$$

Where T_{n+1} and T_n are the temperatures, P_{n+1} and P_n are the pressures at time step $n + 1$ and n of the expansion and γ is the specific heat ratio of the gas. The mass of the gas in the chamber can be calculated at each time step using the ideal gas law:

$$m = \frac{P_c V_c}{R T_g} \text{............ (4)}$$

Where P_c is the chamber pressure (absolute), V_c is the chamber volume, R is the specific gas constant and T_g is the estimated gas temperature. The mass flow rate between each time step is them calculated using:

$$\dot{m} = \frac{m_2 - m_1}{t_2 - t_1} \text{............ (5)}$$

The mass flow rate conversion above using raw pressure data is very sensitive to any data collection error due to the fact that the mass flow rate is a derivative of mass. The pressure data has to be fitted with a smoothed line in order to get a realistic mass flow rate curve. The curve fitting method chosen is an “acsplines” fit using the graphing program GNUPlot. This type of fit approximates the data using a natural smoothing spline. This is done by constructing a piecewise curve from segments of cubic polynomials. These cubic polynomials are weighted by the data points. The magnitude of the weights determines the number of segments used to make the curve. The fit is smoother with a lower weight but further away from each data point. A higher weight allows the curve to be closer to the data points but provide less smoothing. A balance for the weighting needs to be found and is different for every data set.

Figure 3-3 displays the averaged pressure data for the test on the die-cast port and the different curve fits with different weights. It is noticeable from the plot that the data set shown in Figure 3-2 has been cut off for the curve fitting. This is done to keep only flow data when the valve is fully open. Curve fitting the whole curve would be problematic due to the fact that some of the curve includes data when the valve is still closed.

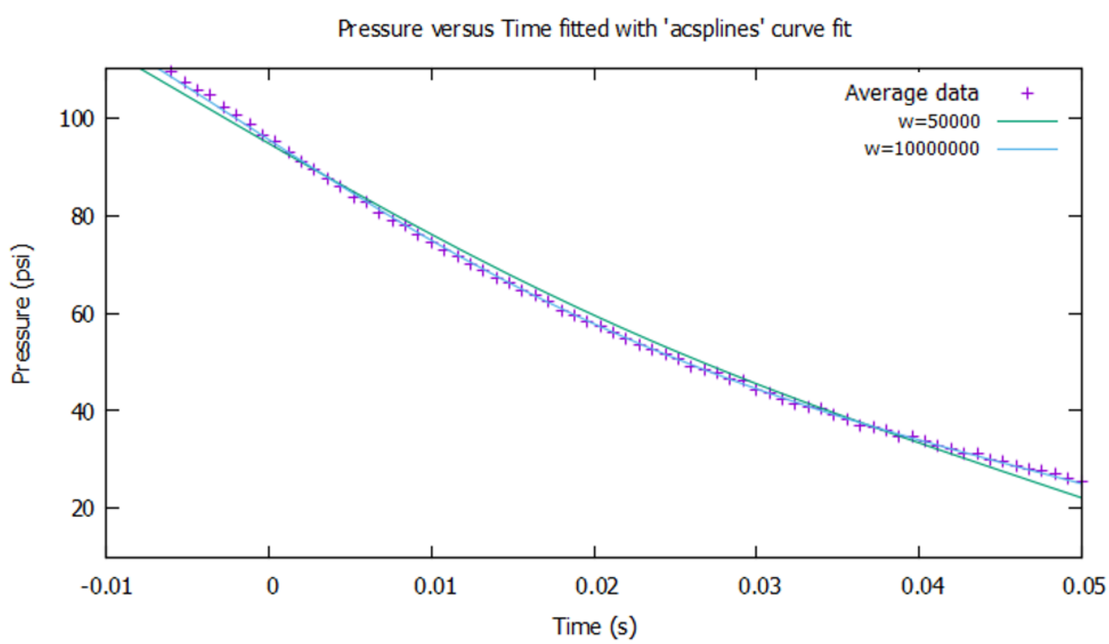

Figure 3-3 - Pressure data curved fitted using “acsplines” in GNUPlot

Two different weighting factors are shown in Figure 3-3. The weighting factor of 50000 does not follow the data closely enough therefore a higher factor is necessary. A weight of 10 million is chosen for this specific test as it follows the curvature of the data smoothly. Once the data is averaged and curve fit, the mass flow rate can be found for the range in which the valve is fully open. The range of data that will be converted to mass flow rate is from 100 psi (gage) to 30 psi (gage). The pressure is given as gage pressure for the raw data however for the mass flow rate graphs and all further analysis, the results will be given in terms of absolute pressure. In absolute terms, the mass flow rate will be plotted from 115 psi to 45 psi.

The experimental flow rate can be compared to a flow rate for a venturi nozzle of the same area. It is important to understand how the flow across the valve compares to the ideal case when making modifications in order to see what is occurring and how flow develops in a transient experimental setup.

The mass flow rate from equation 5 at different pressures can be compared to the theoretical mass flow rate which is calculated using the following equation and derived from ideal gas laws:

$$\dot{m}_{theoretical} = A_f \frac{P_c}{\sqrt{RT_c}} \sqrt{\gamma} \left(\frac{2}{\gamma+1}\right)^{\left(\frac{\gamma+1}{2(\gamma-1)}\right)} \text{............. (6)}$$

Where A_f is the flow area (valve curtain area) and T_c is the gas temperature in the chamber. It is important to note that the pressure P_c in this equation is the absolute pressure.

Using the technique and equations above it is now possible to display the results in terms of mass flow rate for the die-cast port with and without a downstream tube and it all compared to theoretical flow rate. Figure 3-5 and Figure 3-6 present the results.

In order to compare multiple experiments with slightly different conditions all versus theoretical a non-dimensional coefficient of mass flow rate is introduced. This allows to compare experiments with different initial and geometric conditions.

$$C_{MFR} = \frac{\dot{m}}{\dot{m}_{theoretical}} \ldots\ldots\ldots\ldots (7)$$

The coefficient of mass flow rate allows to compare any experiment to a theoretical value of 1 which will help visualize how different shapes compare. This is presented in Figure 3-7.

A drawing of the valve and valve seat open at 0.1 inch lift is included in Figure 3-4 for the die-cast cylinder head. The drawing shows where the minimum flow area is within the system and will give insight into what is happening if different seat and valve geometries are tested. The drawing shows that the surface where the minimum area occurs at 45º. There are three cuts in the valve seat at 30º, 45º and 60º which provides a smooth transition into the minimum area. The zone inside the red lines is where the minimum area occurs. This drawing can be compared to drawings of further testing done to understand what kind of shape could better or worsen the blowdown flow.

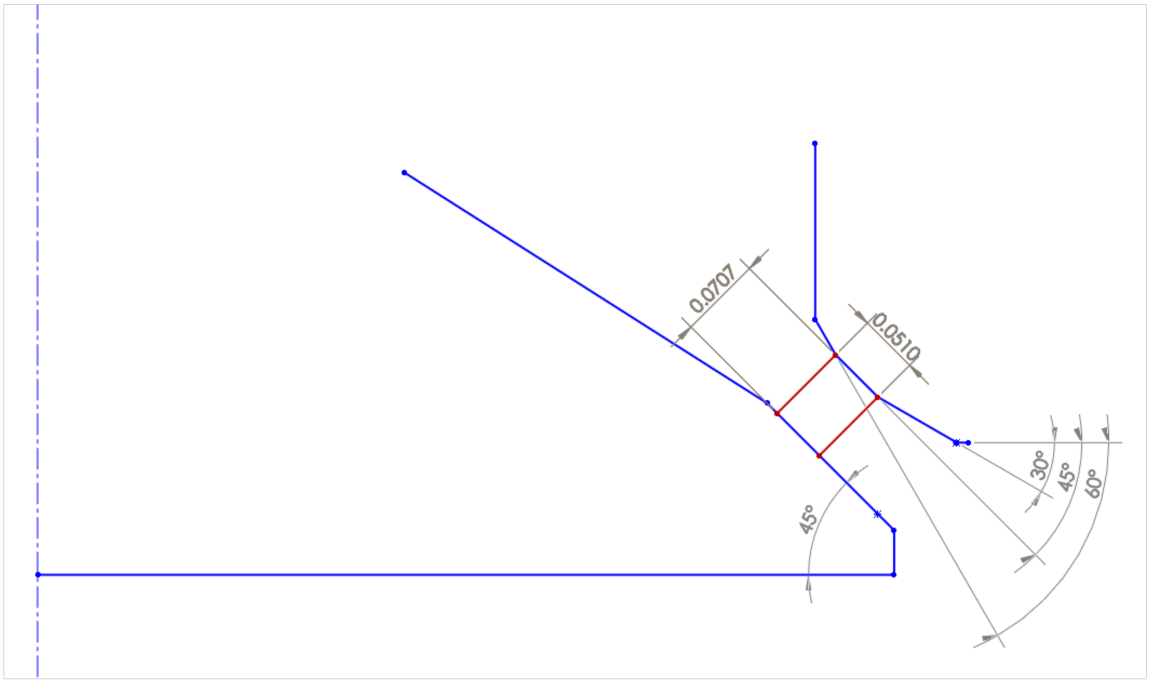

Figure 3-4 - Drawing of the die-cast port valve curtain area, including the valve outline (bottom), seat outline (top) and the location of the minimum area (red)

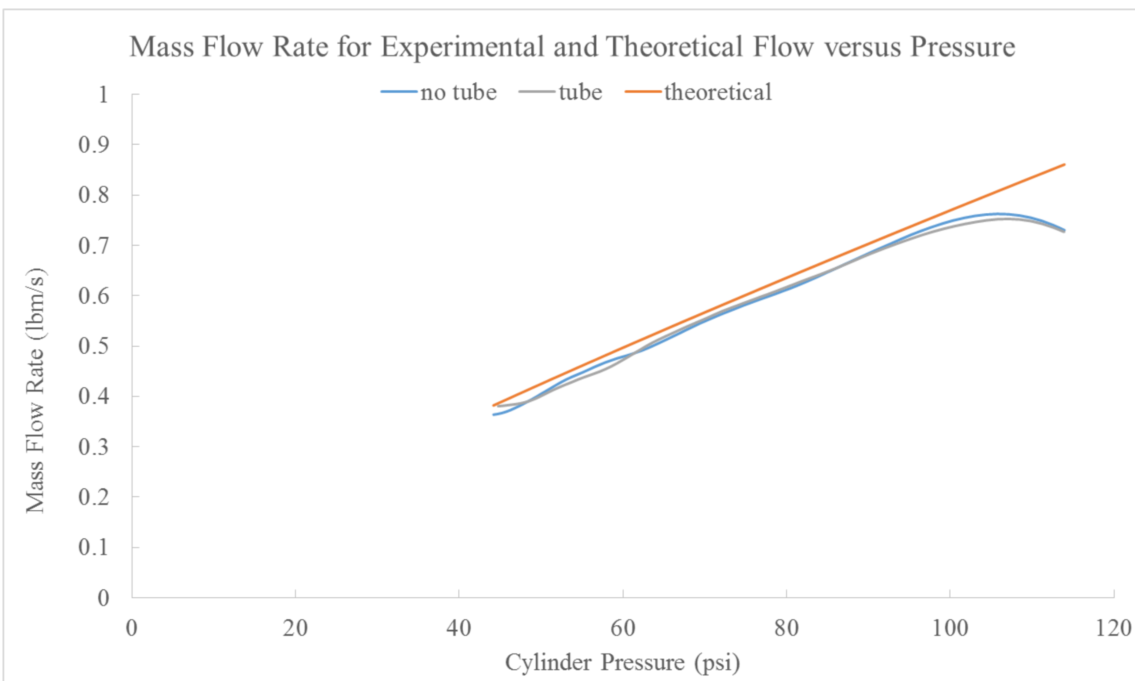

Figure 3-5 - Mass flow rate versus pressure for the die-cast port with and without extension tube

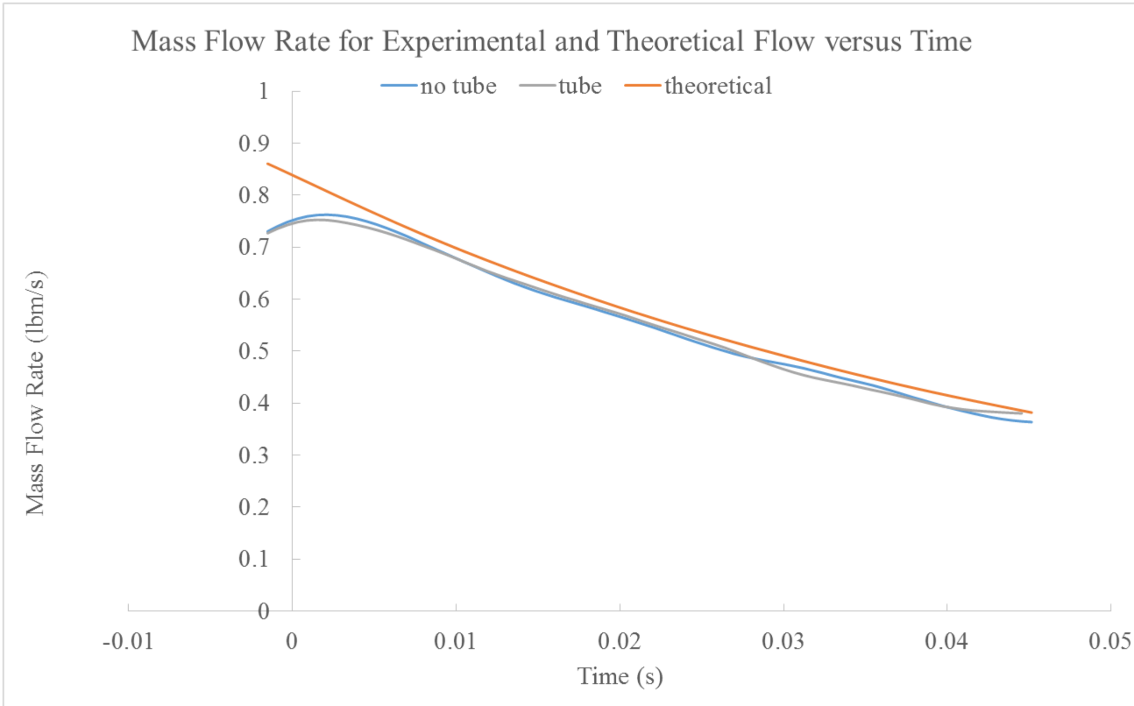

Figure 3-6 - Mass flow rate versus time for the die-cast port with and without extension tube

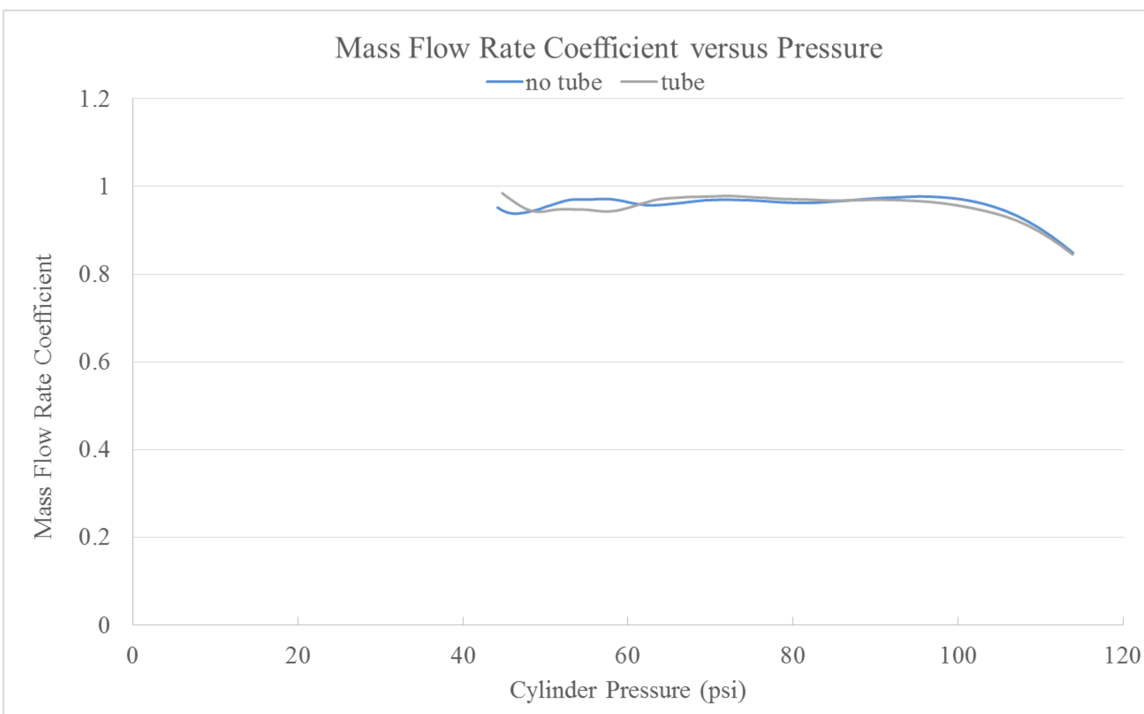

Figure 3-7 - Mass flow rate coefficient versus pressure for the die-cast port with and without extension tube

Figure 3-5 confirms what is seen in the pressure trace, the mass flow rate versus pressure with and without the extension tube are closely matched. The plot of mass flow rate versus time is presented as well to show the mass flow rate progression as the pressurized cylinder empties. From 45 to 100 psi the mass flow rate seems to follow the slope of the theoretical flow rate but is lower throughout the range. In that range the flow is 3.8% lower than theoretical. Above 100 psi the mass flow rate is much lower than theoretical and seems to be developing up to its maximum. Figure 3-6 shows the phenomenon against time, clearly the flow has not yet reached its stable configuration where the transition from subsonic to sonic is made at the minimum area. This is explained by the fact that the flow is still developing in the early portion of the blowdown. The flow seems to not have stabilized from the initial opening at 135 psi down to 100 psi. This is not visible from the pressure trace hence why the mass flow traces are important for analysis. The description of the above results are confirmed by Figure 3-7, the mass flow rate coefficient flattens under 100 psi due to the stabilization of the flow. It is also clear that the coefficient is below the theoretical value of 1.

From this specific test the die-cast port behaves in a different way than the NASCAR GP Tech1 cylinder head because the flow does not change by altering the downstream shape. This might be because the modification is too far from the valve and if modifications were to be made inside the port like the GP Tech1 port changes would occur. There is a discrepancy between the ideal and experimental case. This means that further modifications could potentially increase the flow rate through the valve and somehow the flow is currently restrained by the design of the cylinder head. The development region (above 100 psi) might also affect the flow if it can be altered to develop faster and reach ideal flow rate values faster after opening. More results will follow in order to begin to understand why the flow past the valve does not behave ideally.

Figure 3-9 and Figure 3-10 present the results for the same testing done on the sand-core cylinder head which has a higher performance exhaust port with a larger radius of curvature. The port also has a smaller exit diameter than the die-cast port. The statistical analysis using the Nash-Sutcliffe method shows there is a significant difference in pressure by adding the downstream tube to the port (Appendix B). Looking at Figure 3-9, the mass flow rate seems to differ along the pressure range between both tests. From 115 to 90 psi, which is the initial developing phase of blowdown, the tube increases the mass flow rate through the valve curtain. From 90 psi to 45 psi, the flow rate then converges for both configurations and is slightly higher for test with no tube however the difference is negligible.

The biggest difference is made in the high pressure region thus showing that geometry changes can have an effect on the development of blowdown before the flow reaches a quasi-steady state agreeing with the theoretical curve. The flow takes longer to develop for the sand-core port than the die-cast port. The die-cast port had fully developed at 100 psi starting from 135 psi.

Figure 3-8 shows the shape of the valve curtain area for the sand-core port. Compared to the die-cast drawing presented in Figure 3-4, the differences are small. The width of the minimum distance location is slightly reduced for the sand-core cylinder head and its placement is more upstream towards the combustion chamber. The seat angle is the same and the minimum distance is the same. The seat has three cuts of 60º, 45º and 30º.

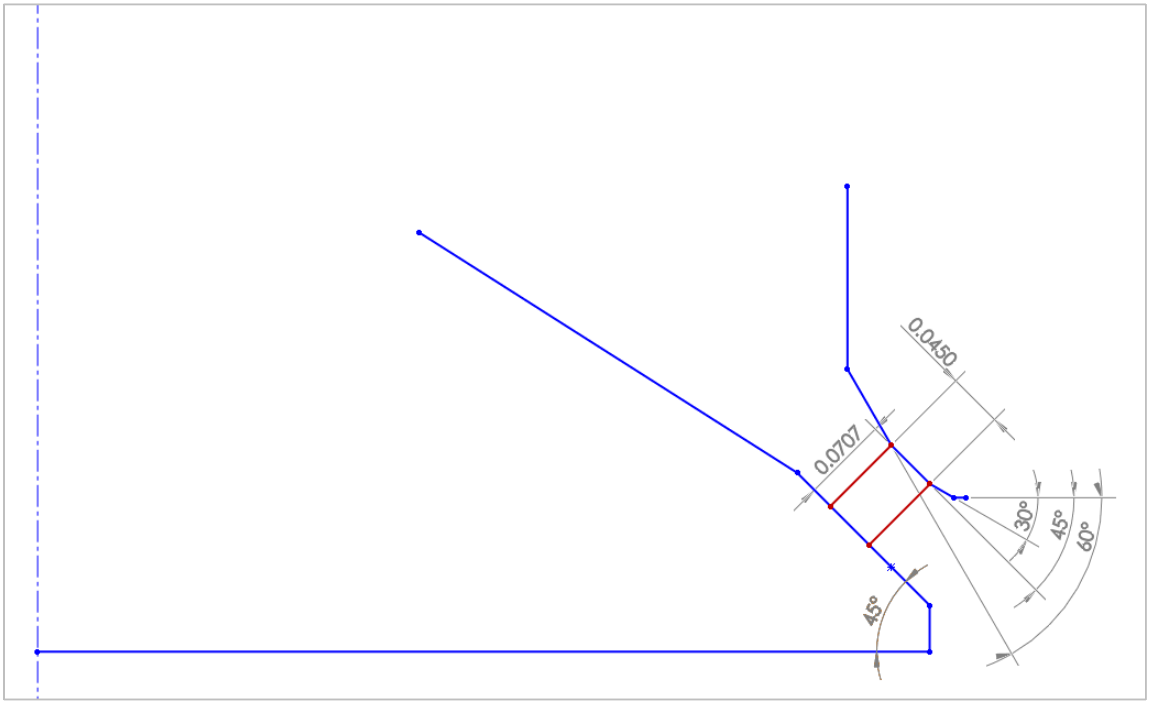

Figure 3-8 - Drawing of the sand-core port valve curtain area, including the valve outline (bottom), seat outline (top) and the location of the minimum area (red)

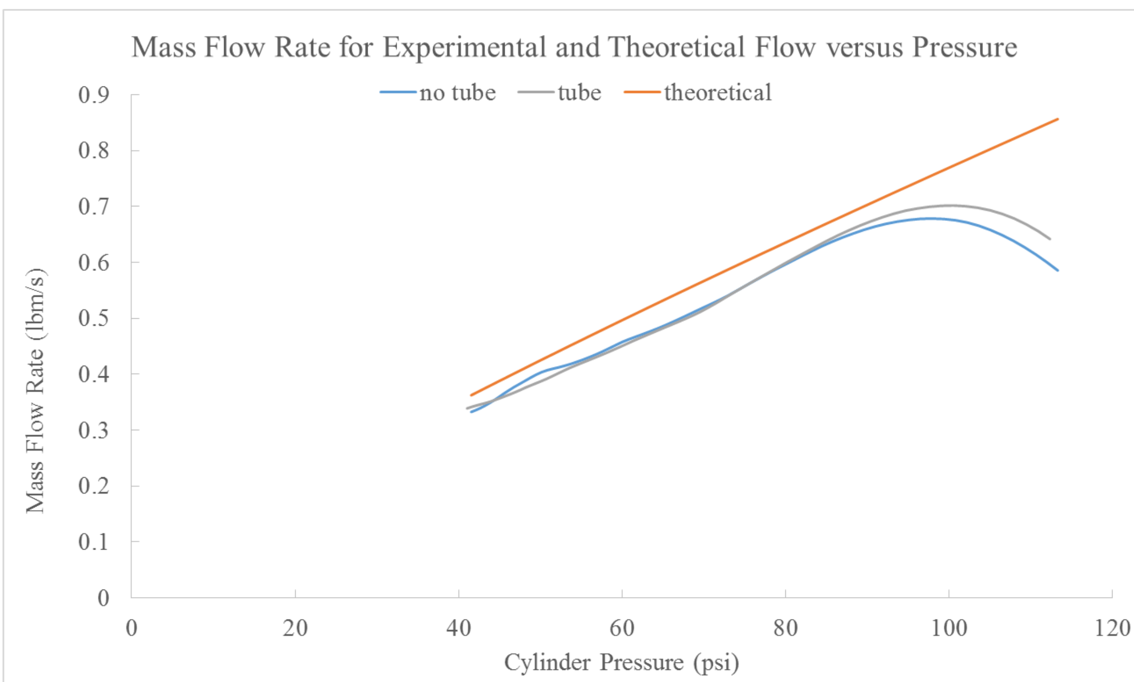

Figure 3-9 - Mass flow rate versus pressure for the sand-core port with and without extension tube

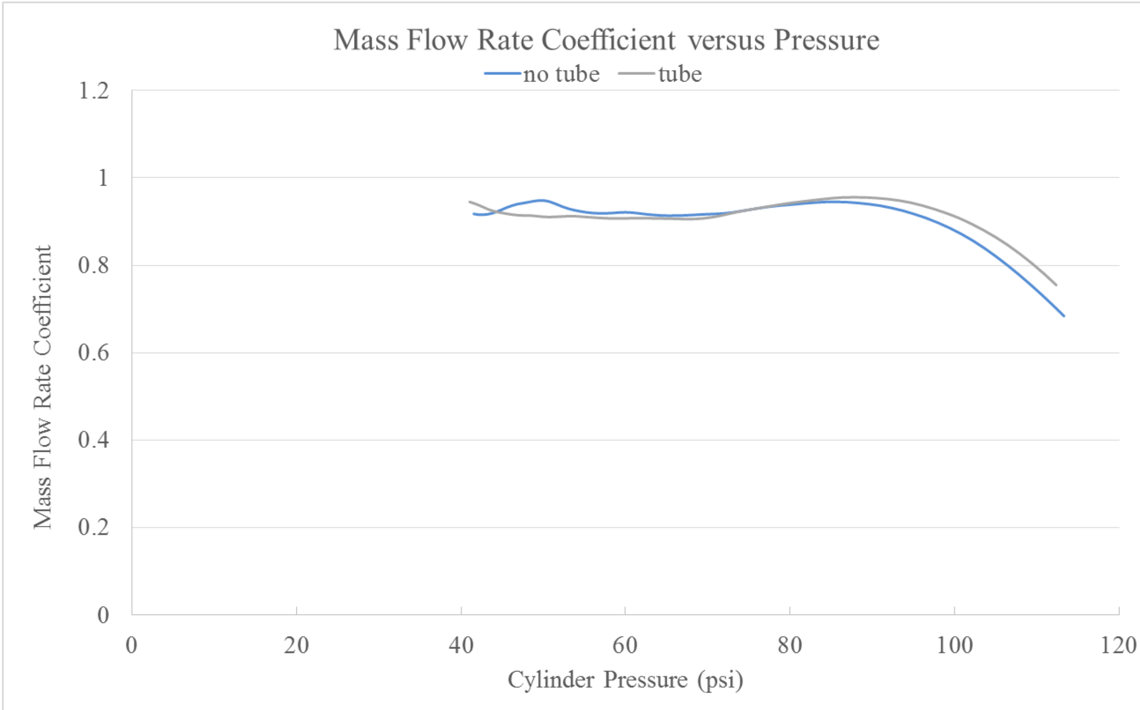

Figure 3-10 - Mass flow rate coefficient versus pressure for the sand-core port with and without extension tube

The sand-core port seems to reach the fully developed flow around 90 psi also starting from 135 psi. This slower development in flow is interesting as it could suggest that the difference in overall blowdown flow rate is caused by the speed at which the stable sonic condition develops. Below 90 psi the flow seems to be parallel to the theoretical value. There are however some slight fluctuations and slope changes throughout the range. This could potentially be caused by the fact that the flow is not sonic around the whole valve and is still in a transient state in which the shock waves and the transition to sonic flow have not settled. This would also explain why the experimental mass flow rate is significantly lower. In the region below 90 psi, the flow rate for the port with no tube is 7.8% lower than theoretical and the port with the tube is 8.5% lower.

Sand-core and Die-cast Cylinder Head Comparison

Having looked at both ports separately gave some insight into the differences in flows compared to theory. The initial comparisons have also made it clear that it is a lengthy process to fully develop the flow and reach the ideal flow rate. The flow is developing for 35 to 45 psi before reaching a slope that resembles that of the theoretical calculation. This is approximately one third of the blowdown phase at this pressure. The speed and magnitude of this development phase is affected by downstream changes in the case of the sand-core port. The flow also has not reached the theoretical value for both the ports. Figure 3-11 presents the comparison of both ports.

Figure 3-11 presents the absolute difference between both flow rates and Figure 3-12 presents the mass flow rate coefficient which is normalized versus the theoretical data for each test. Both figures show there is a difference in the performance between ports. Interestingly the die-cast port which is not designed for high performance flows better than the sand-core port. The very noticeable difference in flow rate occurs when the flow has not reached a steady mass flow with a constant slope. This reinforces some of the points addressed earlier about downstream modifications making a difference to the initial phase the most.

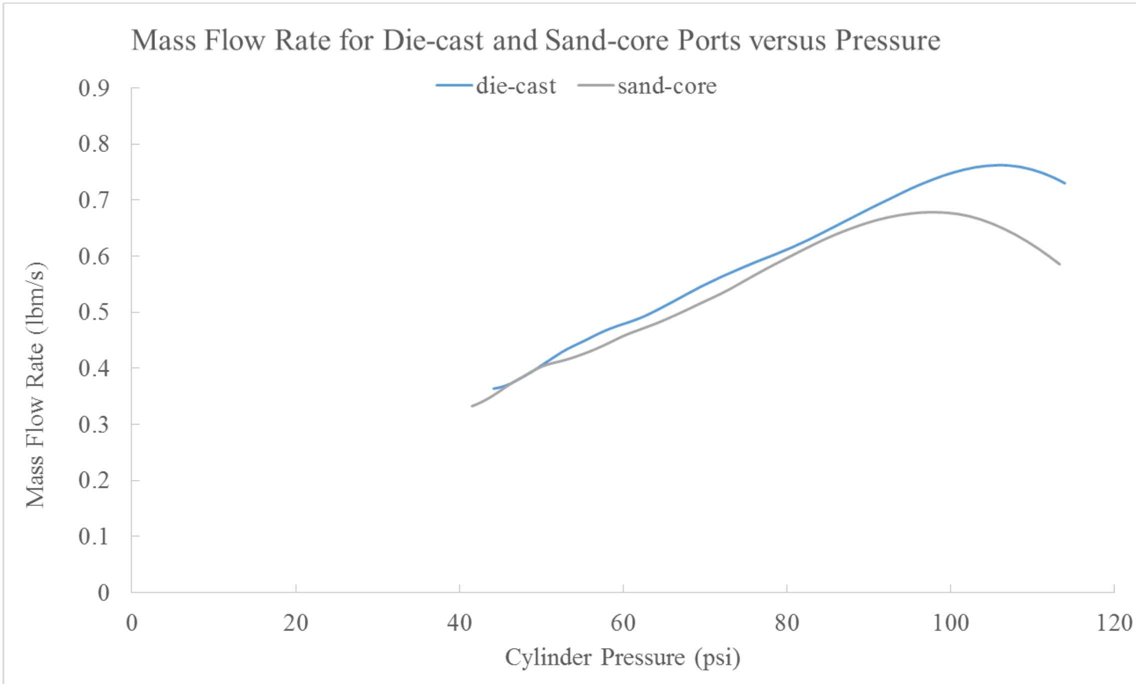

Figure 3-11 – Mass flow rate versus pressure for both ports compared

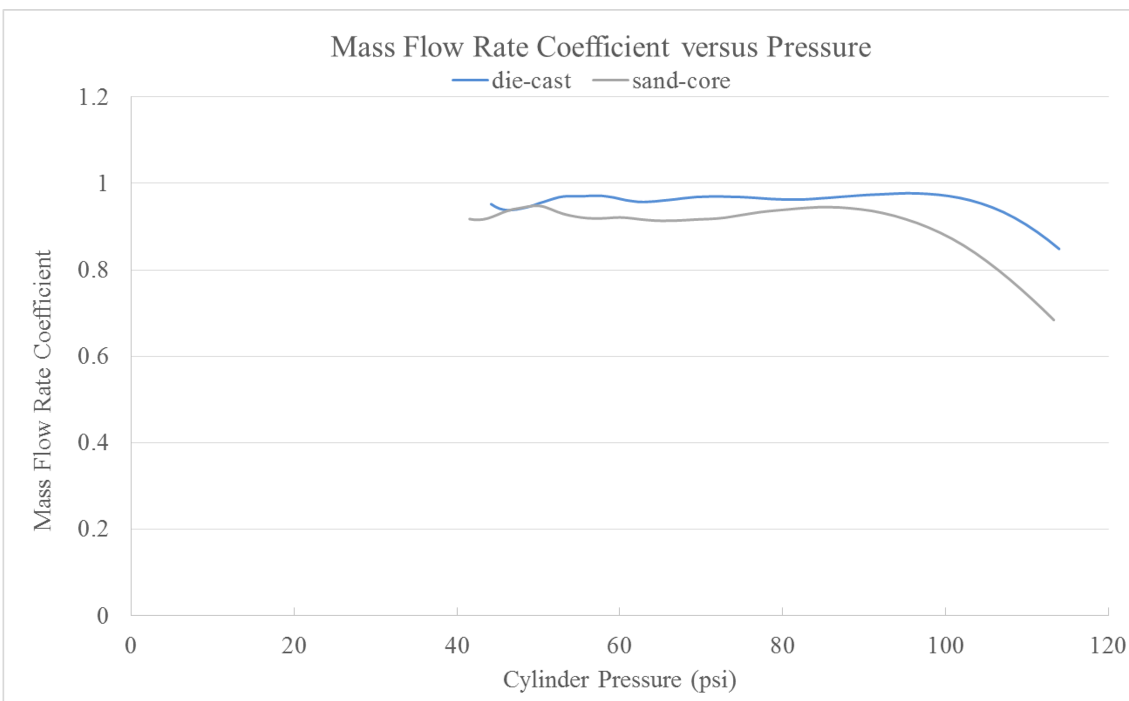

Figure 3-12 - Mass flow rate coefficient versus pressure for both ports compared

The development phase at high pressures is longer in the sand-core port. This could be due to the geometrical differences between both ports and both combustion chambers. The combustion chambers (upstream of the valve curtain) have different designs which could affect this initial phase. The valve curtain areas drawn in Figure 3-4 and Figure 3-8 also differ slightly which could also explain the differences in flow. The ports are also shaped very differently with the die-cast port being designed with a very sharp corner and expanding to a large diameter exit. The sand-core port makes a smooth turn however has a small exit diameter and is shorter in length. These geometrical differences should not affect the sonic flow however can affect the flow when the mass flow rate is still developing and below the theoretical values. There is a difference in the stabilized flow as well which needs to be investigated further and could be due to the flow not being fully developed all around the valve curtain in both cases. A few things can be concluded from the previous observations:

- The sand-core port performance has the potential to be improved dramatically to reach that of the die-cast port and the theoretical flow rate.
- The initial phase of development of the flow is affected by geometrical changes and in turn affects the overall blowdown event.
- More modifications and analysis are needed to understand the flow experimentally.

Die-cast Port – Modifications made inside the port

The testing of the die-cast port did not result in a mass flow change with the tube modification made. The mass flow rate curve is also below theoretical meaning that the flow can be improved. Further investigation into this port is necessary to understand why the flow remained unchanged and what could improve or reduce the flow rate. Changes closer to the valve inside the port are investigated in this section on the die-cast cylinder head.

Modifications were done to four areas downstream of the seat within the exhaust port and can be visualized in Figure 3-13. The four areas are the:

- Back side
- Short side
- Cylinder side
- Intake side

These four areas are shown in the diagram in Figure 3-13 and are located inside the port between the valve seat and the valve guide.

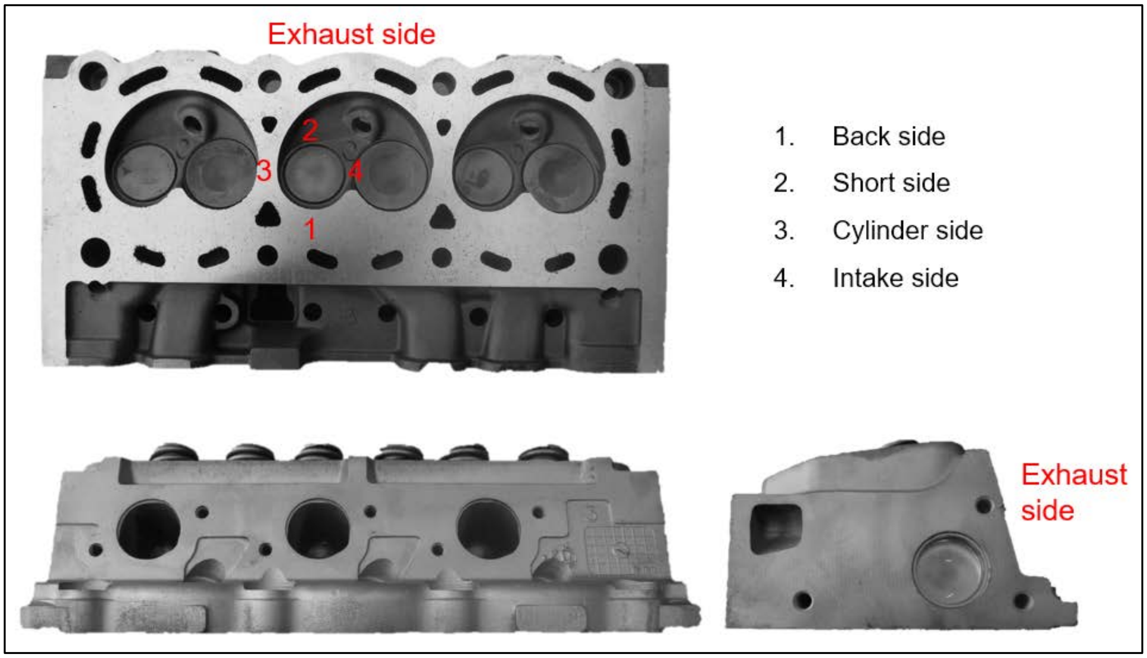

Figure 3-13 – Top, Front and Side view of the die-cast cylinder head showing the location of the 4 sides

Figure 3-14 - Pictures of the stock die-cast exhaust port (left) and modified back side port (right)

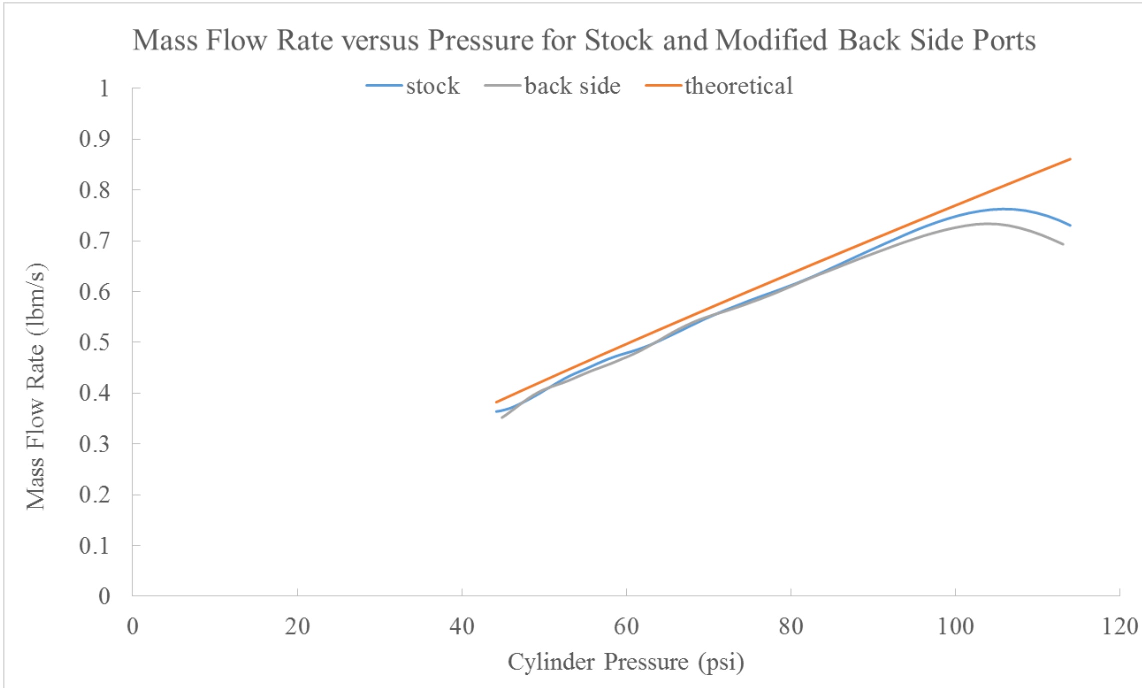

Figure 3-15 - Mass flow rate graph for the stock port and the modified back side port

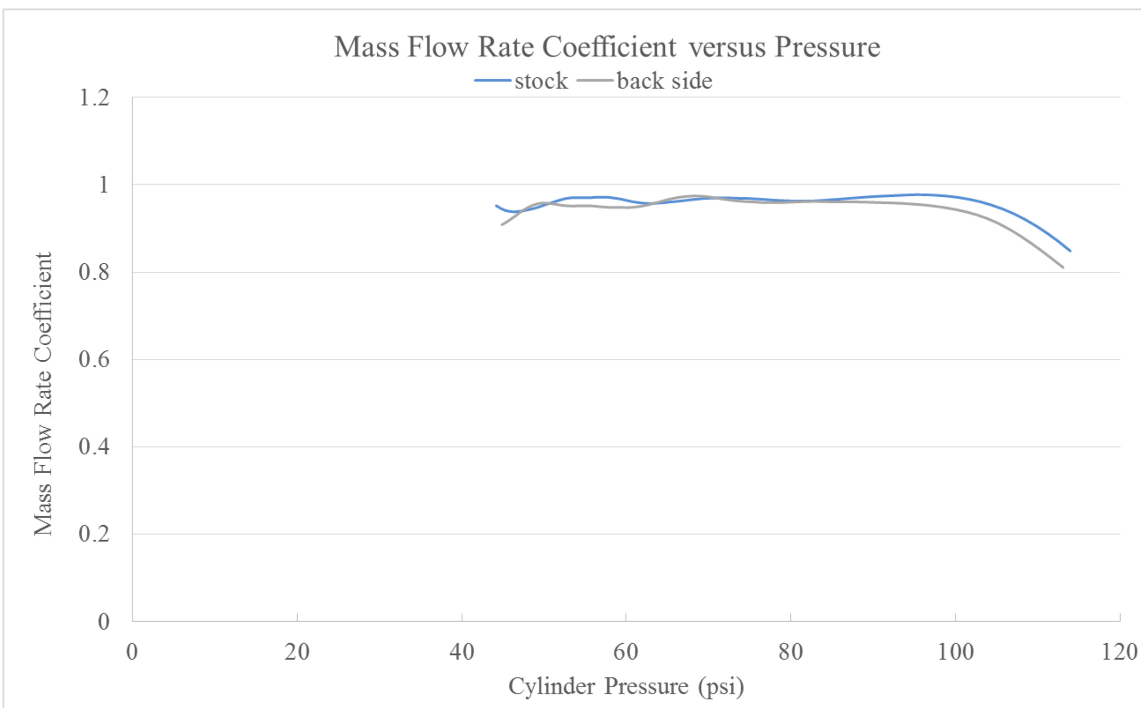

Figure 3-16 - Mass flow rate coefficient for the stock port and the modified back side port

The first modification to the die-cast port was made on the back side. It is displayed in Figure 3-14 and the comparative results are presented in Figure 3-15 and Figure 3-16. Material was removed from the port and the overall shape smoothed out. Looking at the pictures on the left side of Figure 3-14, the use of a die to cast the head causes some imperfections that are smoothed out by taking some material off. Theoretically, after the initial development region, nothing should change by modifying the backside. The results show there is an insignificant change in flow once the flow is developed however at high pressure while the flow is still forming there is a clear difference. This agrees with the sand-core tests in that the downstream modification can alter the initial part of blowdown. Once the flow is fully developed the flow rate is similar between both geometries as expected.

The second modification made to the port was to smooth the cylinder side. This has been done in addition to the back side modification. The process of going from the stock port to the cylinder side modification is displayed in Figure 3-17. Material is taken off the side of the port which increases the downstream volume slightly and creates a much smoother transition into the port. Looking back at Figure 3-14, it can be noted that the combustion chamber wall is very close to the exhaust valve on the cylinder side. This can have an effect on the flow going into the minimum area as it is squeezed by the small gap. It is unclear whether a downstream change can affect the upstream effects but this will be discussed further. Figure 3-18 and Figure 3-19 show the results for this second incremental modification made to the port. There is a clear increase in the flow rate by making this modification.

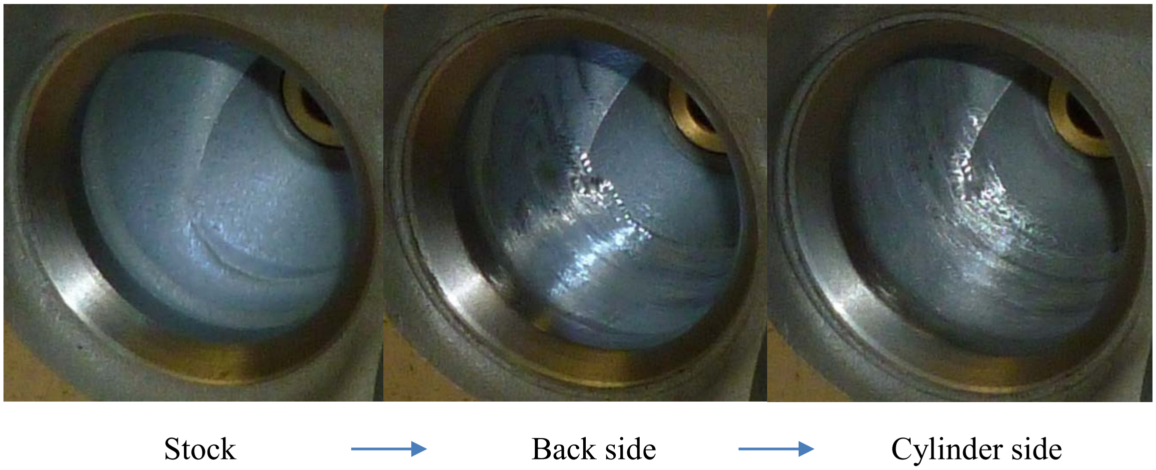

Figure 3-17 - Pictures of the stock port (left), the modified back side (middle) and the modified cylinder side (right)

The cylinder side modification increases the flow rate throughout the pressure range tested. Once again the curve can be separated into two different sections to analyze the part where the flow is still increasing to its maximum and a second instance where the flow has reached a fully developed state. The cylinder side modification seems to have had an effect on both parts of the blowdown.

The initial phase of blowdown at high pressures demonstrates that the flow rate reaches a value that is higher than the stock flow rate and the theoretical flow rate. The length of the initial phase is extended in comparison to the stock port as well. It occurs from the opening at 135 psi down to 80 psi which is a really significant portion of the whole blowdown event. This suggests that the flow is unstable and takes a long time to settle. More interestingly however, the mass flow rate passes above the theoretical line and only seems to rejoin the theoretical maximum around 80 psi. Assuming that choked flow is occurring at the minimum area this is impossible.

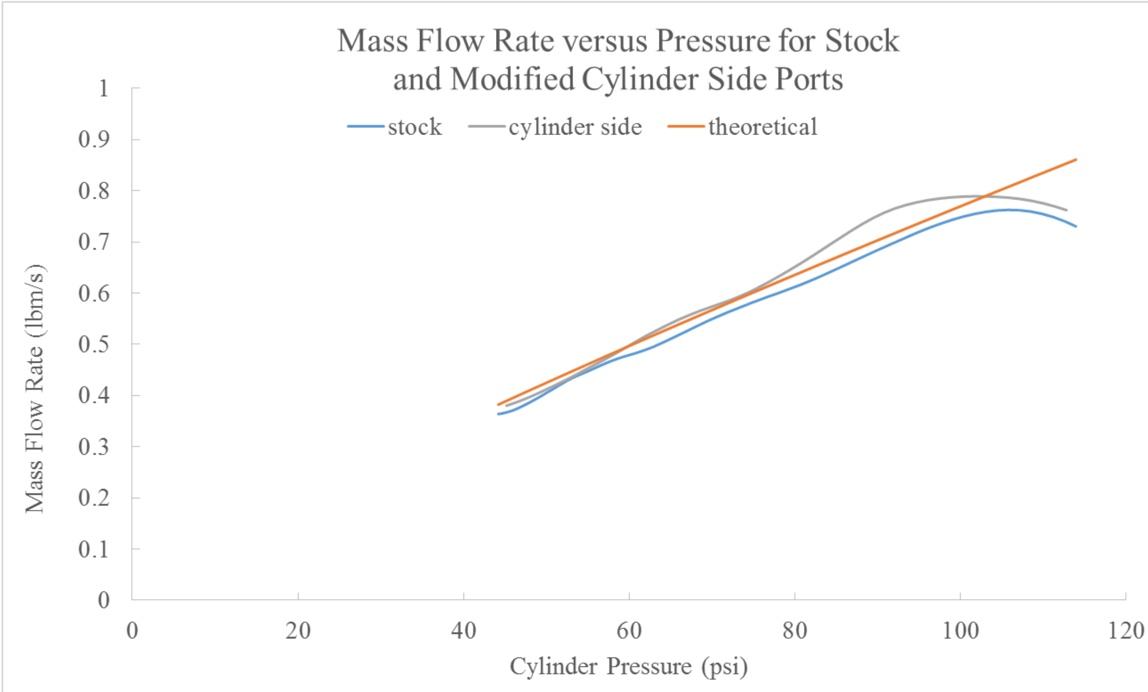

Figure 3-18 - Mass flow rate graph for the stock port and the modified cylinder side port

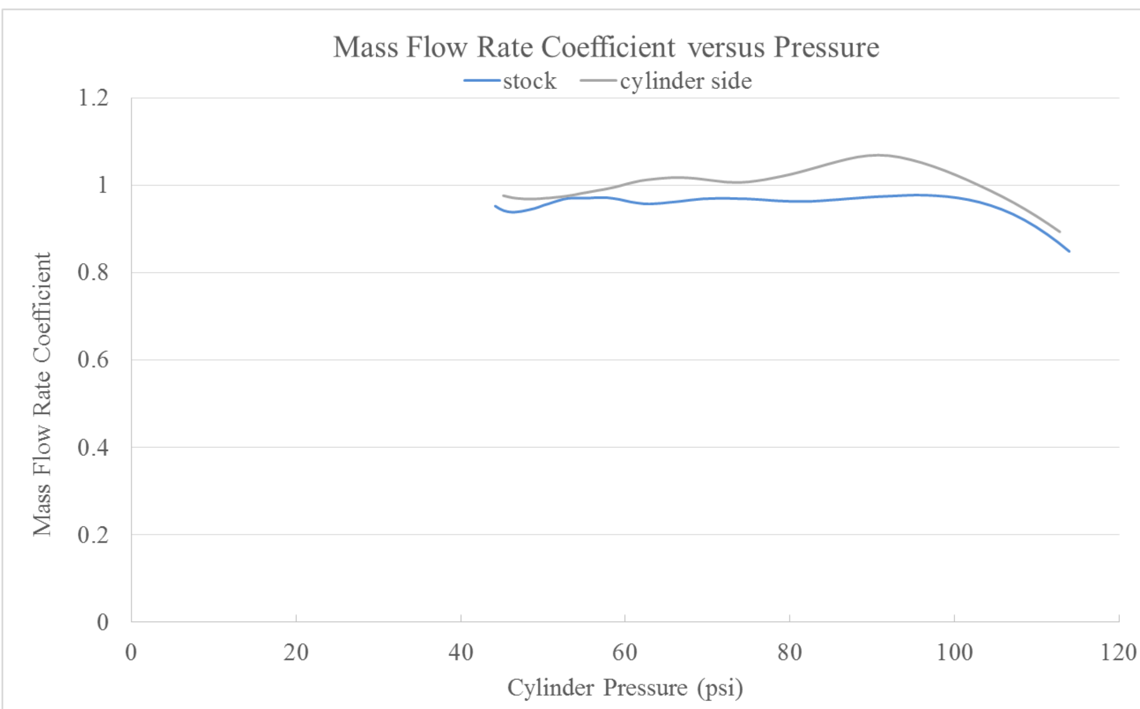

Figure 3-19 - Mass flow rate coefficient for the stock port and the modified cylinder side port

This impossibility is explained by the fact that the flow has not reached quasi-steady state and the line at which the flow transitions from subsonic to supersonic is shifting due to the highly transient nature of the flow initially. If this line is moving slightly downstream of the minimum area, the effective flow area would be increased hence allowing the flow to surpass the theoretical flow rate. The flow is transient in nature and this means that there can be fluctuations in the position of the transition. This is further investigated in Chapter 6 where the study of a sonic nozzle is discussed. The theoretical calculations use linear equations and steady flow assumptions but this is a type of flow that can be highly non-linear due to the rapid changes in local temperatures and densities when opening the valve. The shape of the sonic transition can also be bowed increasing the area at which the flow is transitioning.

It is also very important to put this experimental setup in context with the operation of a real engine. Currently, the experimental setup allows opening of the valve and keeping it open at one constant valve opening. This is different than the engine operation where the valve is constantly moving which could mean the flow never actually settles to a quasi-steady state where the mass flow rate follows the slope of the theoretical calculation. This will be discussed further in the discussion of this chapter once all the testing methods have been reported.

The flow rate under 80 psi is higher than the stock port and is very close to the theoretical flow rate. This is encouraging as it shows that changes in the flow rate in the fully developed stage of the flow are occurring due to downstream changes as well, not just in the transition region above 80 psi. This potentially means that the flow around the valve was not sonic in all parts around the valve in cases such as the stock design and downstream modifications allow for the flow to be fully sonic around the whole valve.

The next modification made to see if the flow rate would change further was done to the intake side of the port. Once again, this modification was done following the back side and the cylinder side modifications. The modification is displayed in Figure 3-20 where a smoothing of the untouched side was performed. This gets rid of all the lines and curvatures present in the stock port and increases the volume of the port slightly more. The results from this test are presented in Figure 3-21 and Figure 3-22. The results show an overall higher mass flow rate for the third modification compared to the stock port. This is confirmed by the statistical analysis done using the Nash-Sutcliffe method. The duration of the development phase above 95 psi is similar in length to the one seen in the stock port however it reaches a higher mass flow rate once fully developed.

Figure 3-20 - Pictures of the stock port (left) and the modified intake side port (right)

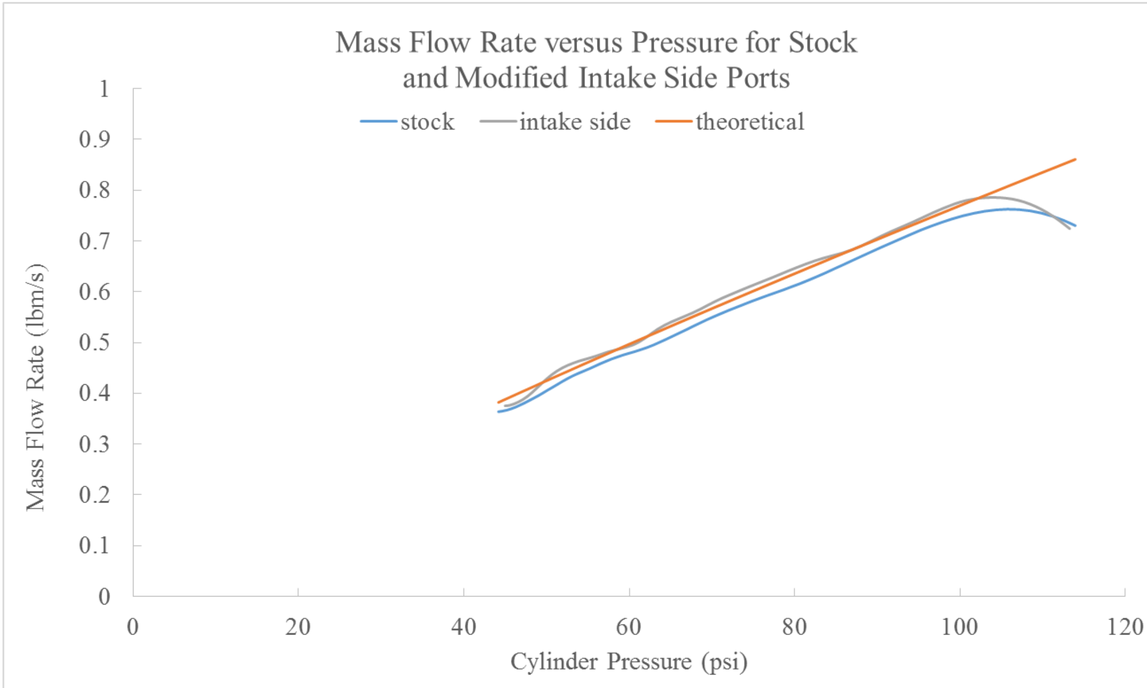

Figure 3-21 - Mass flow rate graph for the stock port and the modified intake side port

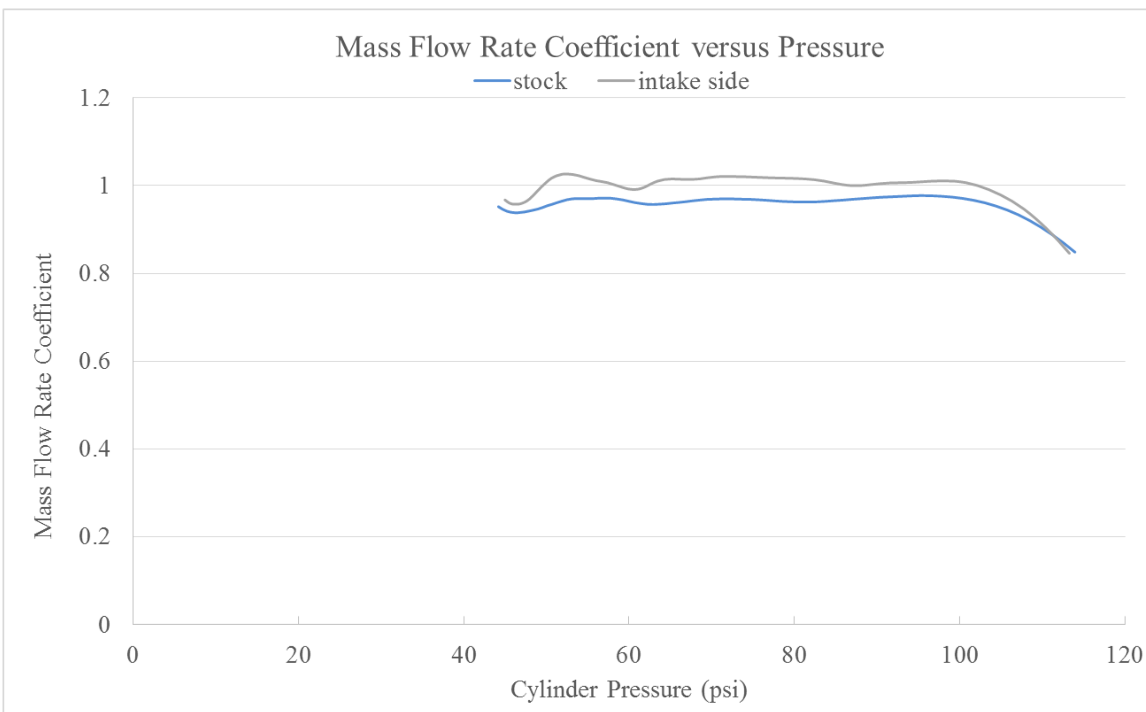

Figure 3-22 - Mass flow rate coefficient for the stock port and the modified intake side port

In the region below 95 psi the mass flow rate is very close to the theoretical value suggesting that the flow is sonic around the whole valve at the minimum area. This was already the case with the modified cylinder side port at low pressures. The intake side modification cancels the effect that was seen at high pressures with the cylinder side modifications. The development of the flow is once again faster and does not go above the theoretical value. Interestingly, a modification on the other side of the port affects the general behavior seen for the previous iteration of design. This demonstrates that the formation of the flow is very sensitive to any minor change made.

The next modification made was to the short side radius of the port as displayed in Figure 3-23. On the left side the picture shows the port unmodified with a straight edge turn in the port due to the die cast manufacturing method. This straight edge was modified to a smoothed turn which resembles the sand-core port design. At low pressures following the blowdown phase this should improve the flow through the exhaust port significantly. The biggest geometric difference with the sand-core port in stock form is this straight edge. Since the sand core port does not flow as well during blowdown as the die-cast port in its stock configuration it is important to see what effect this modification has on the overall performance of the port.

Figure 3-24 shows the results that suggest that the modification has hurt the high pressure flow before being fully developed. Below 100 psi, the flow is however maintained at the higher flow rate which is in line with the theoretical values. The modification has not changed the behavior when the flow is fully developed however greatly hinders the flow when it is still in transition. This is unwanted compared to the previous modifications.

Figure 3-23 - Pictures of the stock port (left) and the modified short side port (right)

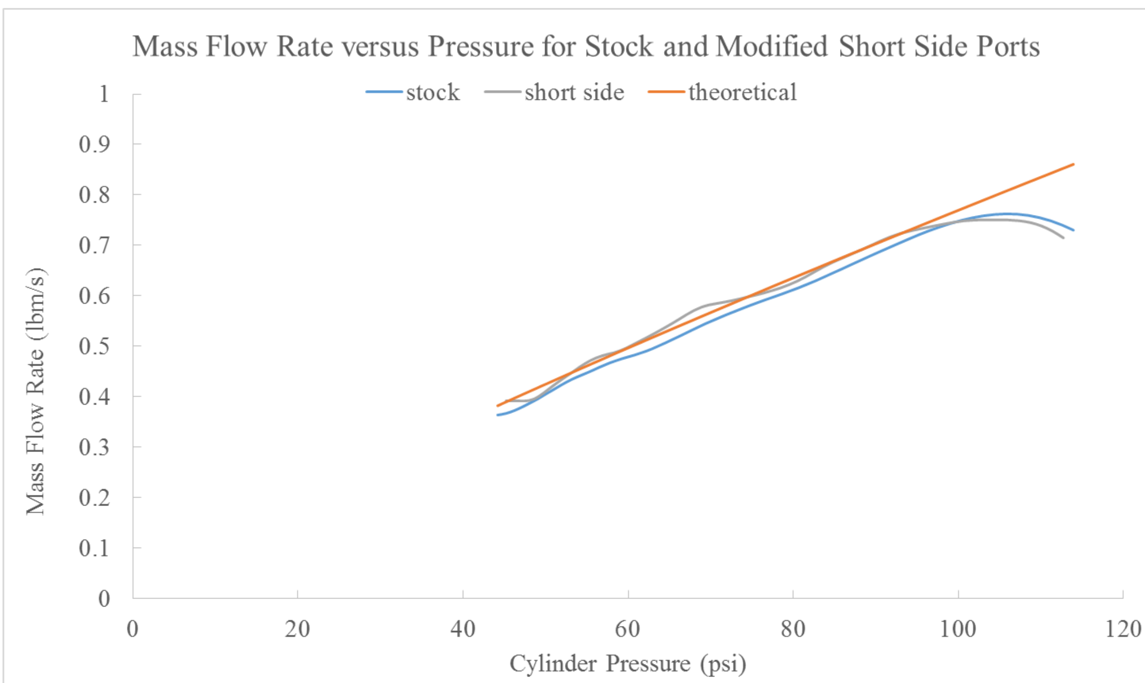

Figure 3-24 - Mass flow rate graph for the stock port and the modified short side port

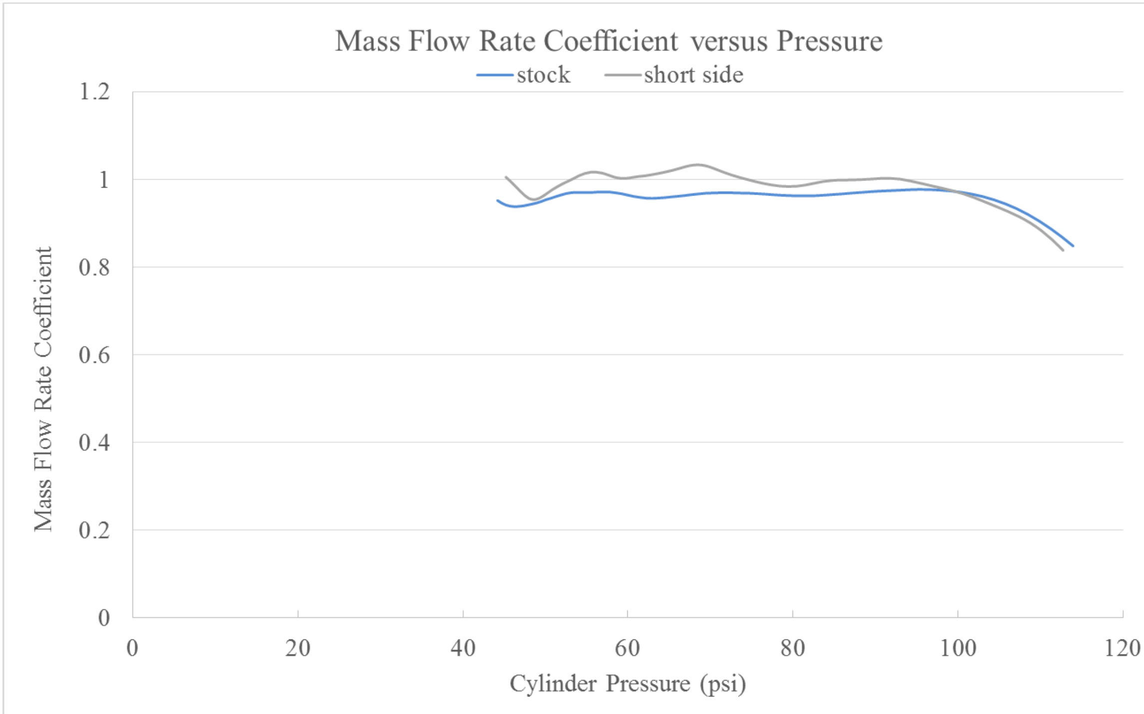

Figure 3-25 - Mass flow rate coefficient for the stock port and the modified short side port

A lot of valuable information is learned in this part of the investigation of the die-cast port. The effect of slight changes made to the port shape can result in significant changes in mass flow rate. The change in this port that increased the flow rate the most overall was done to the cylinder side. The experimental data matched the theoretical values once the flow was developed. The flow at high pressures before being fully developed alters the overall average flow rate most dramatically during a blowdown event. Getting the shape of the port correct for this developing phase is therefore critical. The cylinder side modification once again had the largest mass flow rate for this initial phase.

Die-cast Port – Comparing the Port Modifications with a Second Port

The study of the die-cast modifications reported in the previous section was done on one port out of the three on the cylinder head. In order to verify if the same behavior occurs in the other port a second study was done on one of the other ports. The first port used is named port 1 and the second port used is named port 2. The goal of the study for port 2 will be to understand if the modifications are done in different order whether or not the changes in mass flow rate will be the same. The order for the modifications of port 2 will start with the cylinder side and will follow with a modification of the back side. This is mainly to see if the cylinder side alone can affect the flow with no other modifications.

The first thing that has to be done is to measure the flow rate in port 2 in its stock form and compare it to port 1. The ports are all slightly different due to manufacturing defects and the die to manufacture the head could be slightly different between each port. Each seat may also differ slightly and could possibly affect the flow. The results from this comparison are shown in Figure 3-26 and Figure 3-27.

As is clear from the figures, port 2 shows an increase in flow rate throughout the pressure range tested. This is surprising as a much smaller change in values between ports was expected. Looking at the results in comparison with the theoretical data port 2 agrees with the theoretical values in the developed flow range. In the initial phase of the blowdown process at high pressures the mass flow rate is higher for port 2 than port 1 as well.

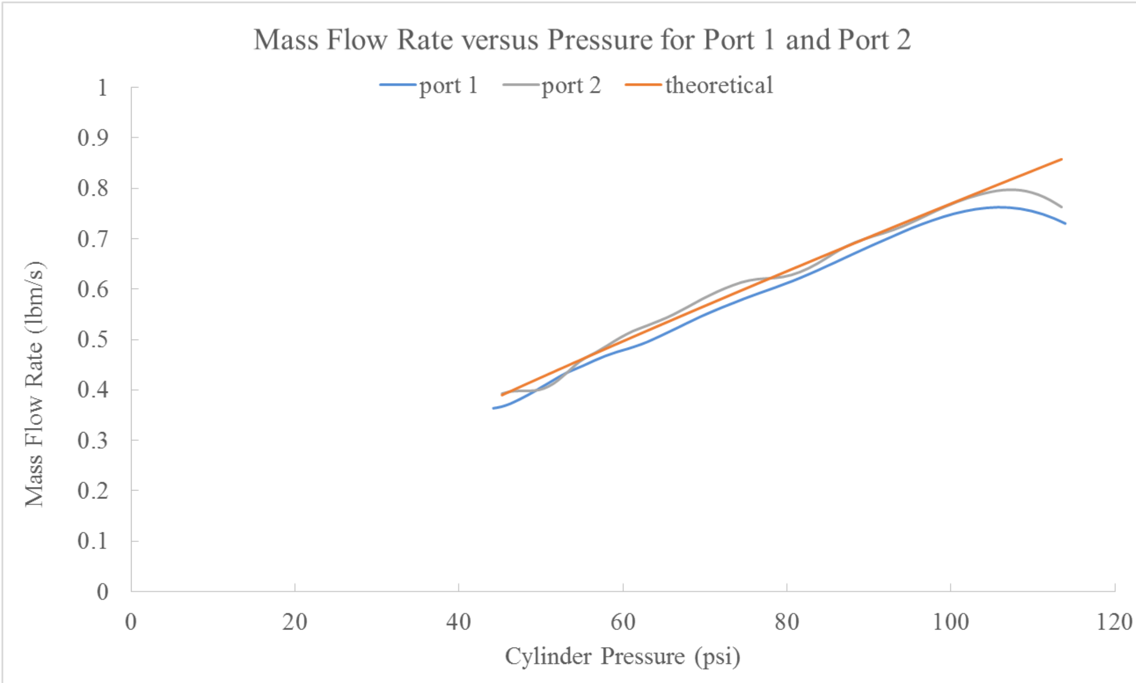

Figure 3-26 – Mass flow rate comparison for Port 1 and Port 2 in the stock form

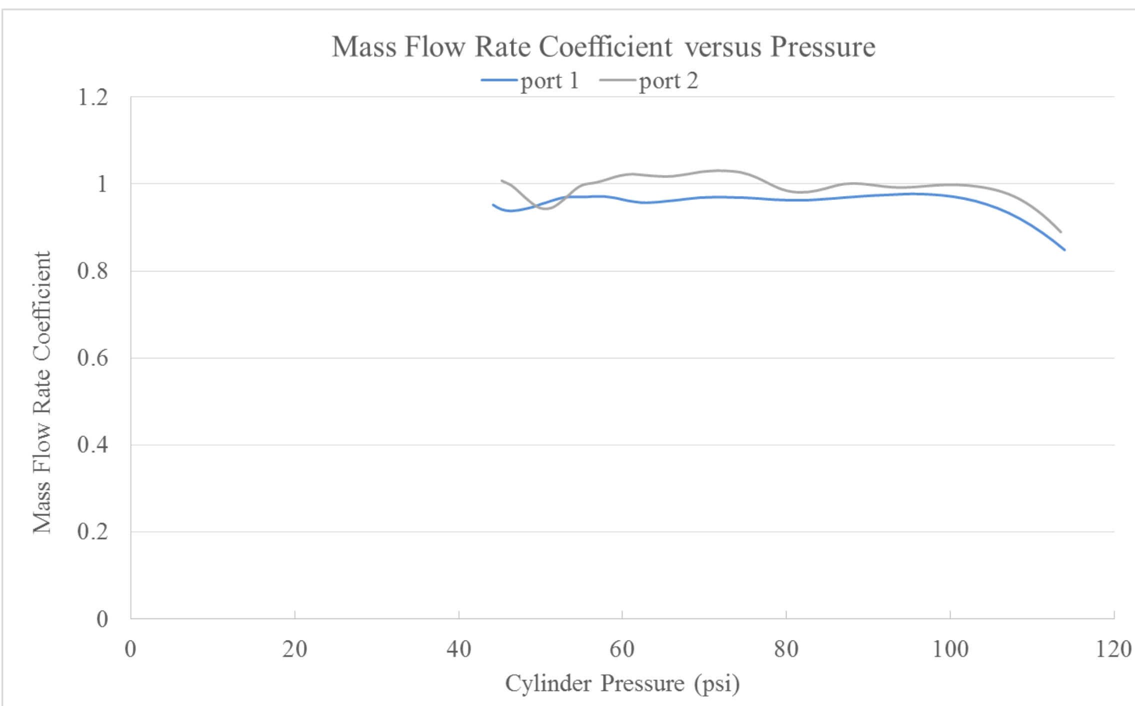

Figure 3-27 - Mass flow rate coefficient comparison for Port 1 and Port 2 in the stock form

A modification to the cylinder side was done first to see if the same changes in flow as port 1 can be observed. The expectation is for the mass flow rate to increase in both phases of the flow however the magnitude of the increase might be smaller due to the fact that the stock port starts at a higher overall flow rate. For this modification, the cylinder side was segmented into two parts: the back half and the front half. The back side was smoothed first and the results from this are presented in Figure 3-28. The front side was then smoothed so the whole side resembles the cylinder side modification made in port 1. The results for this are presented in Figure 3-30. All the modifications to the ports are made using a technique called "porting" which involves grinding off metal by hand therefore there can be differences between the modifications done to each port. This technique is used to modify ports which are then tested on flow benches and it is important to note that very minor changes in geometry can have very big effects on the flow rate [8]. Both cylinder sides were modified to be as similar to port 1 as possible considering this is a non-automated process.

Figure 3-28 shows that the modification of the back part of the cylinder side does not improve the flow overall and the only significant differences are a slightly higher flow rate at low pressures and a lower flow rate in the development region above 100 psi. This seems to suggest that this modification is not beneficial. This also indicates that the flow increase in port 1 was caused by the front side of the alteration of the cylinder side. This is confirmed by the results in Figure 3-30 for port 2. The whole cylinder side modification (front and back) compared to stock shows an overall increase in mass flow rate. This increase is not as significant as the one perceived in Figure 3-18 for port 1. This is expected as the initial conditions that were recorded for the stock port 2 were closer to theoretical initially.

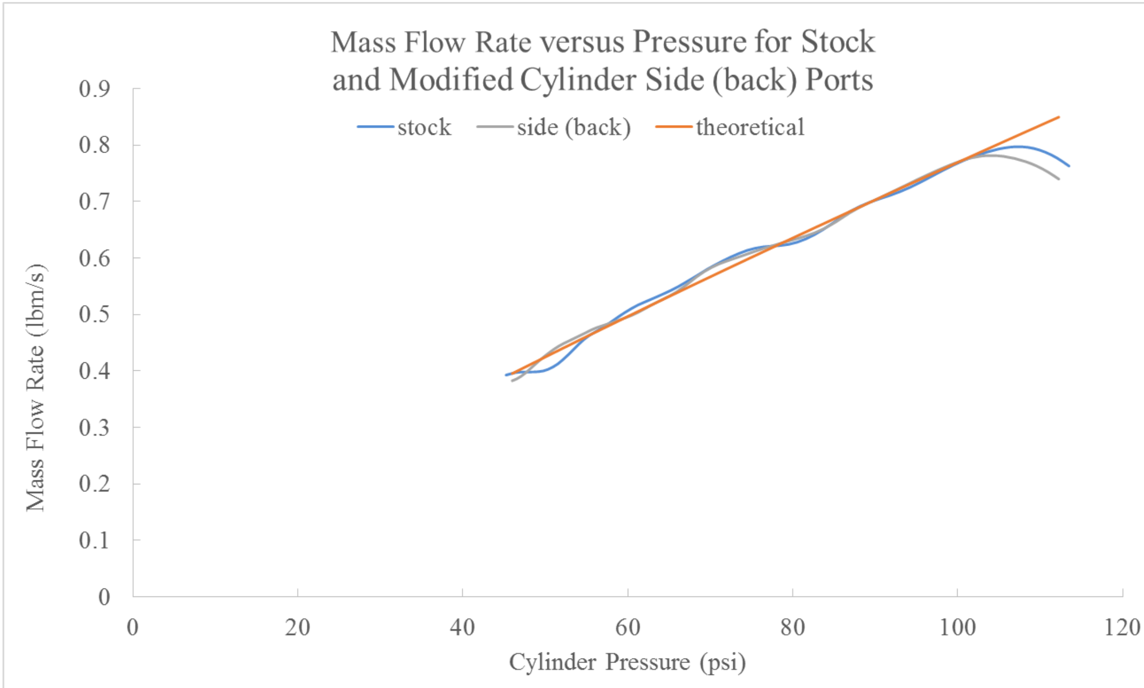

Figure 3-28 - Mass flow rate graph for the stock port and the modified cylinder side (back)

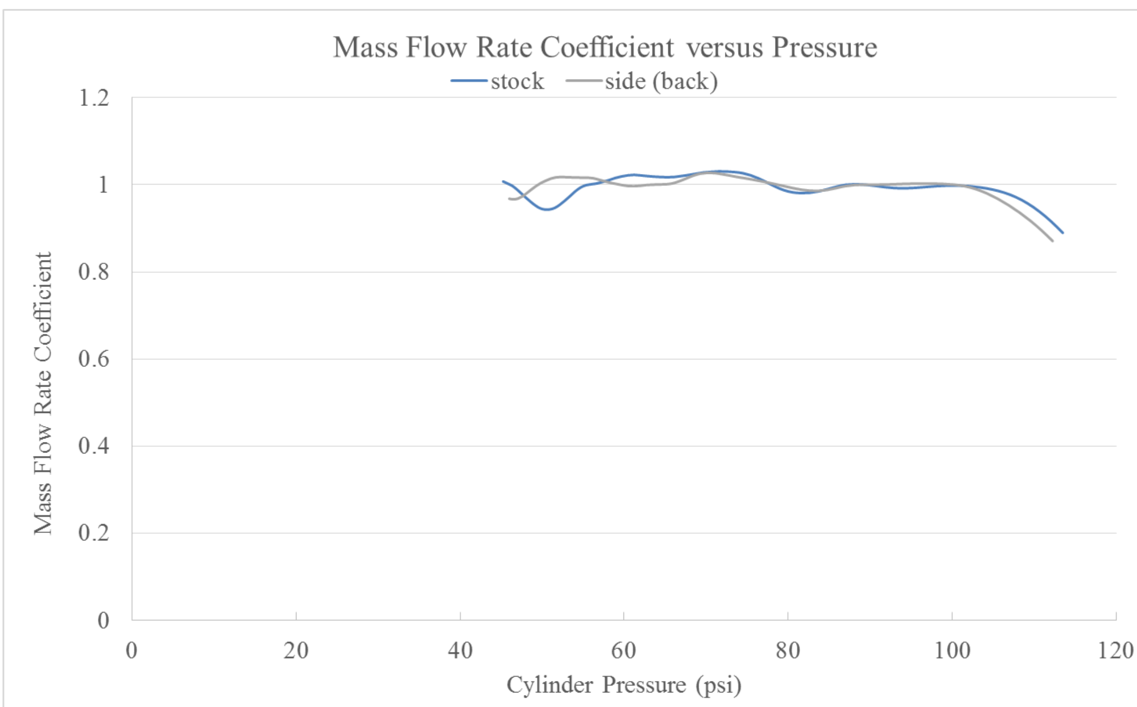

Figure 3-29 - Mass flow rate coefficient for the stock port and the modified cylinder side (back)

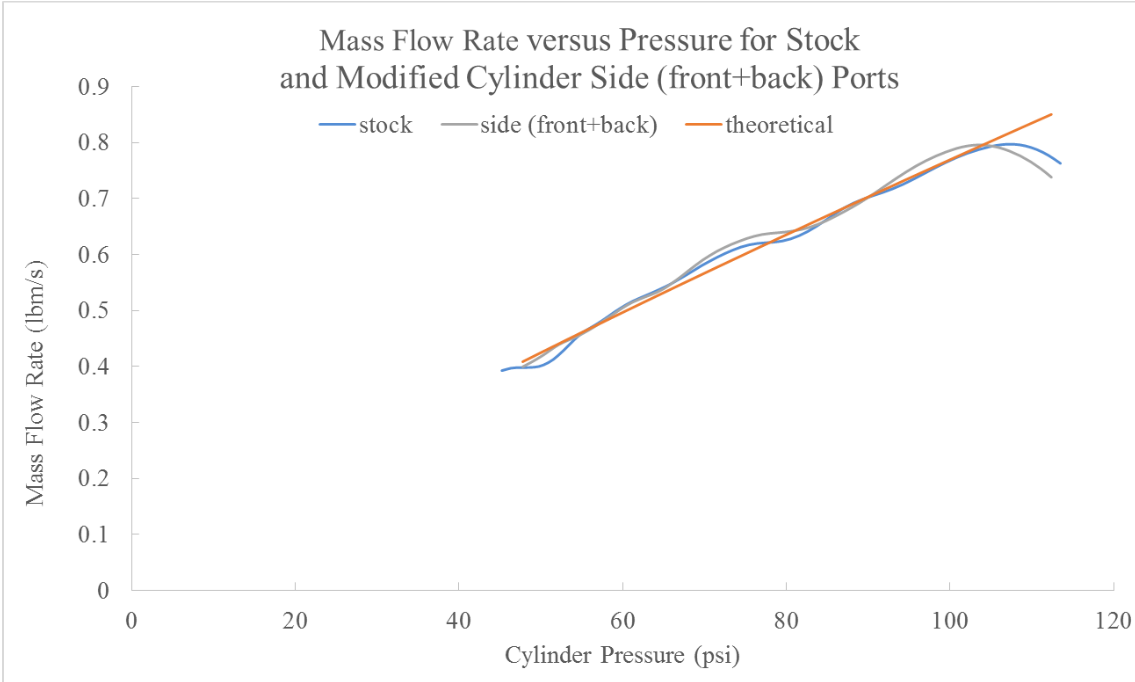

Figure 3-30 - Mass flow rate graph for the stock port and the modified cylinder side (front and back)

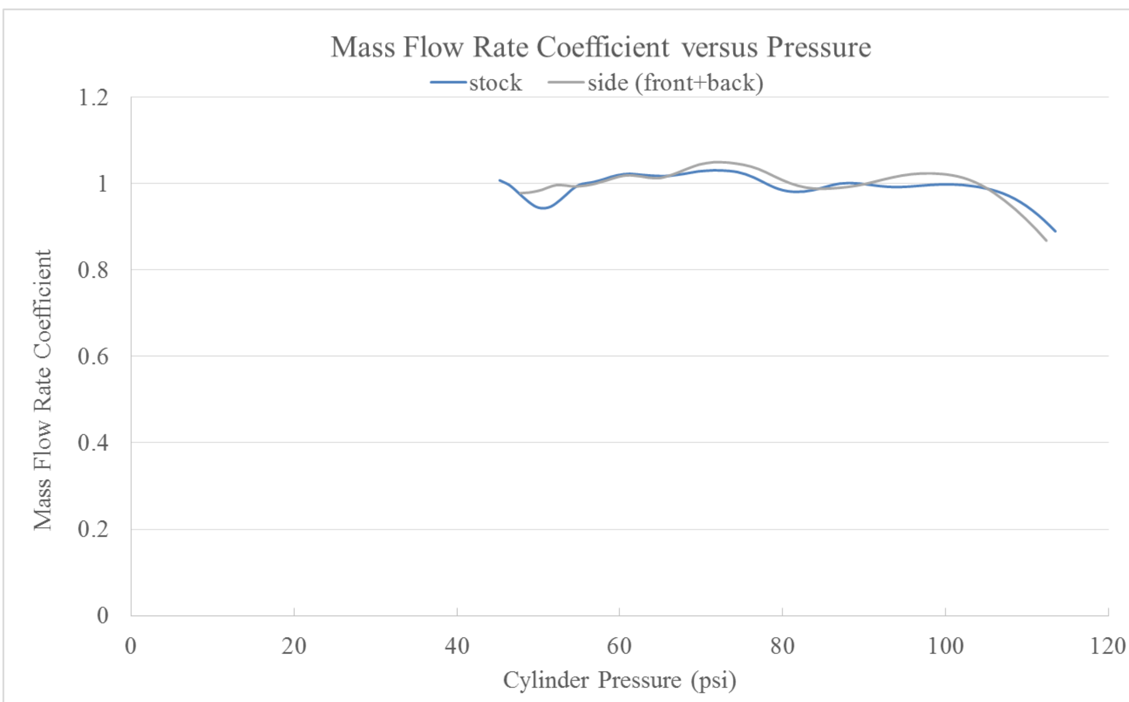

Figure 3-31 - Mass flow rate coefficient for the stock port and the modified cylinder side (front and back)

Looking at Figure 3-30 more closely, the development phase which occurs from 135 psi down to 90 psi, is extended in comparison to the stock port. It also reaches over the theoretical line which has been discussed for port 1 already and will be discussed further in the chapter. The flow rate below 90 psi is also slightly increased throughout the range of pressures.

Following this positive change to the overall flow rate, the next modification done increases the depth of the material taken off on the cylinder side. Removing a small amount of material to the side increases the flow rate therefore the thought process is that further expanding the volume of the port in this place could yield even better results. The zone that was already modified was further ported. This is the second iteration of the cylinder side being modified and is presented in Figure 3-32 and Figure 3-33. The result of this is an overall decrease in the flow rate both when the flow is developing and fully developed. This indicates there is a very high sensitivity to the modifications made on the cylinder side of the port. There is a limit as to how much the expansion of the volume on that side of the port is beneficial to flow especially in the development region.

The next step is to verify if the back side of the cylinder head has any effect on the flow after the cylinder side has been modified. Previously the back side had been altered first for port 1 and had only shown a bettering of the flow at high pressures when the flow had not reached its fully developed state. The results for the back side modification are presented in Figure 3-34 and Figure 3-35.

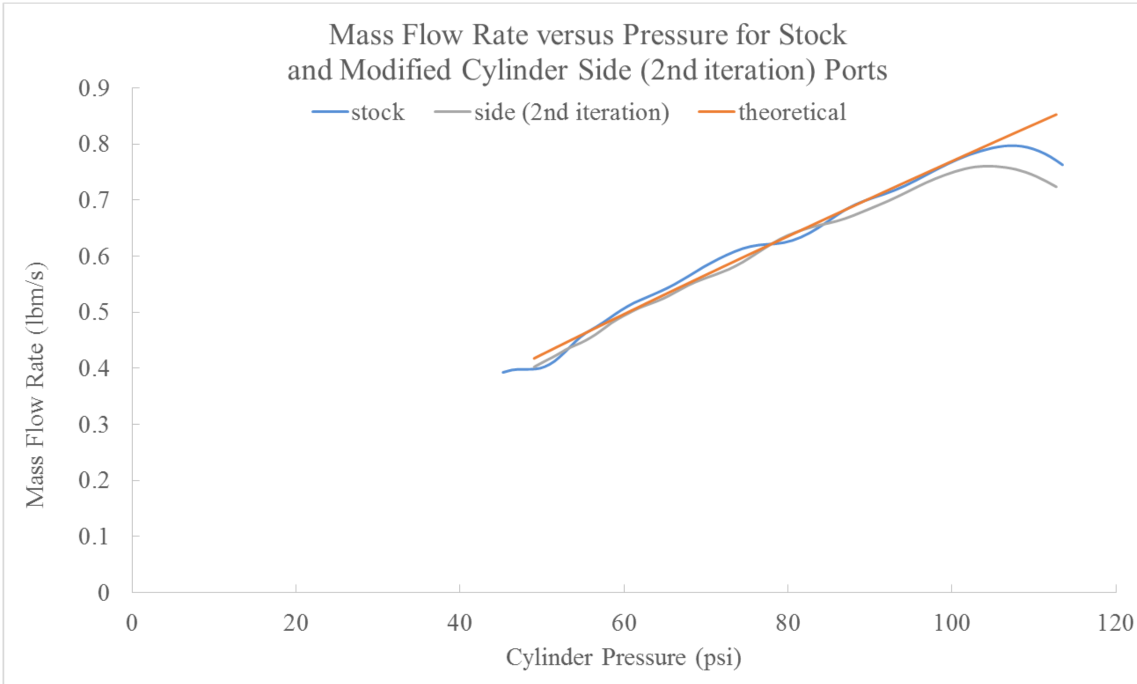

Figure 3-32 - Mass flow rate graph for the stock port and the modified cylinder side (2nd iteration)

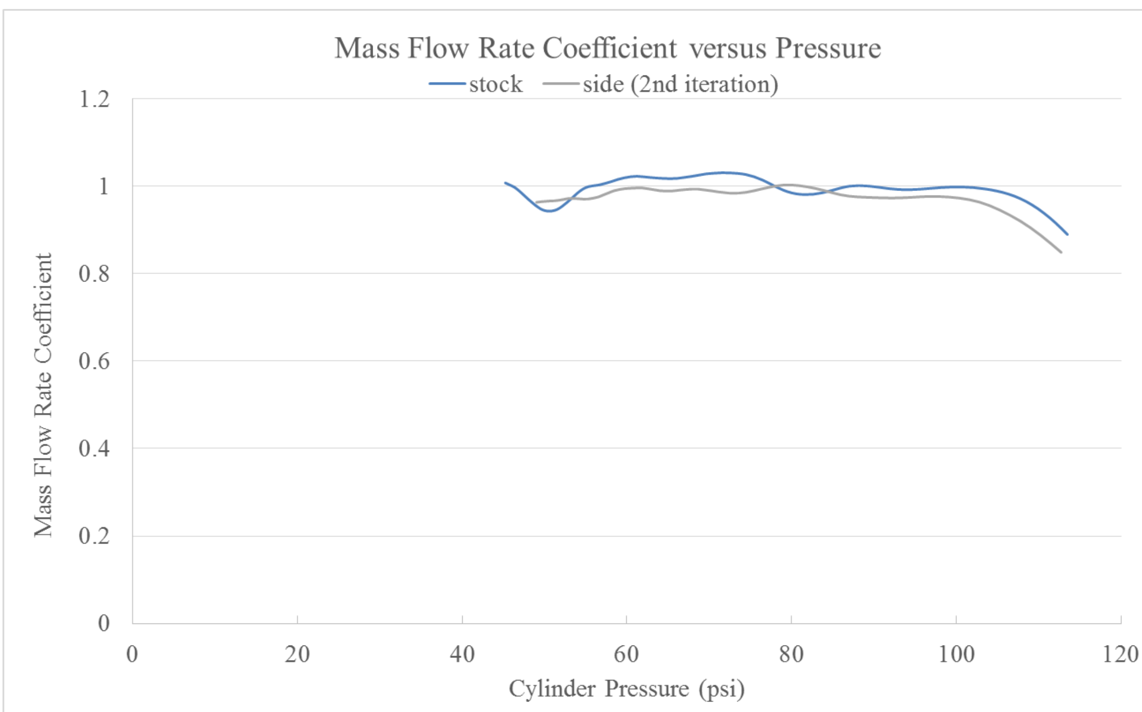

Figure 3-33 - Mass flow rate coefficient for the stock port and the modified cylinder side (2nd iteration)

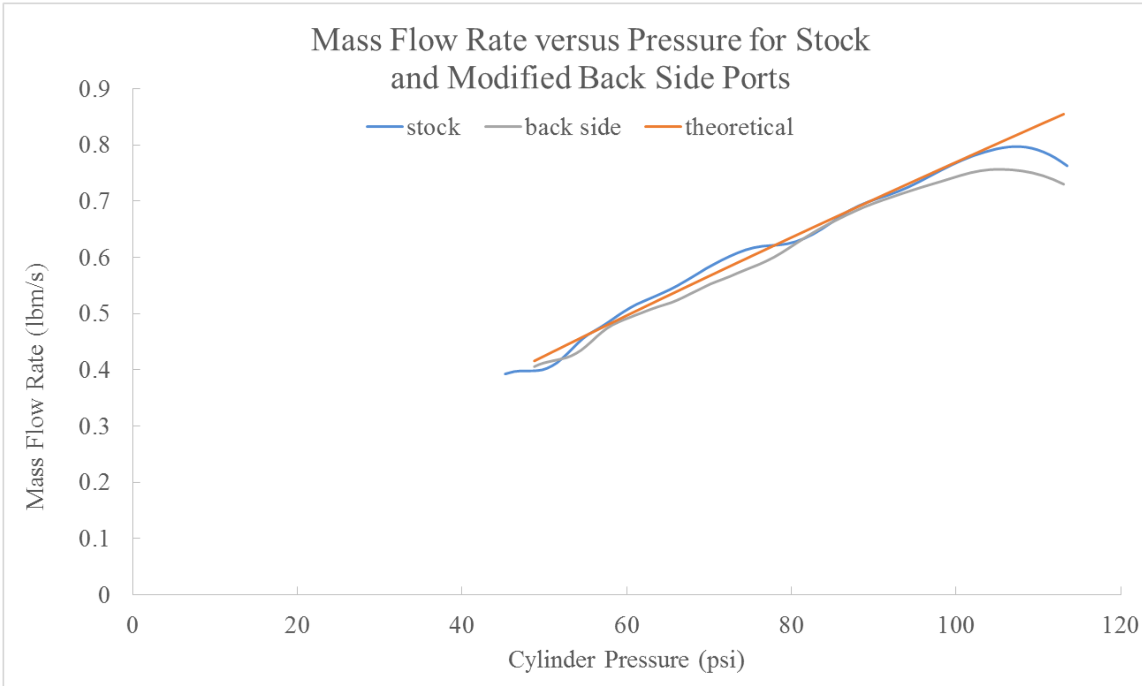

Figure 3-34 - Mass flow rate graph for the stock port and the modified back side

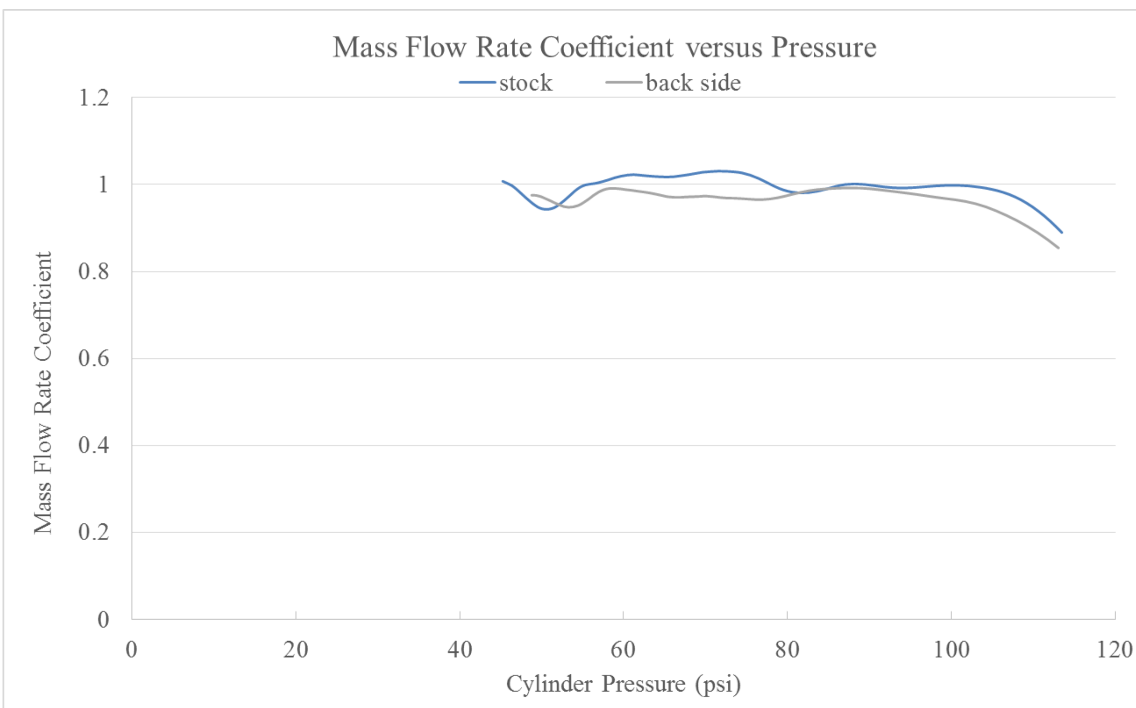

Figure 3-35 - Mass flow rate coefficient for the stock port and the modified back side

The combination of the cylinder side modified in two iterations and back side combined yields mass flow rates that are the lower than the stock port and the port with just the side modifications. This shows that the further increasing the volume just downstream of the valve seat does not help the flow rate. This is contrary to the effects seen with port 1 but is likely due to the overexpansion of the cylinder side cut in this case.

Port 1 and port 2 show somewhat different results overall however the expected trends in the results were met when the cylinder side was modified to the same depth in both cases. New observations have been gathered from port 2, namely that the back side does not seem to have an effect on the cylinder side modification. If the volume downstream of the valve seat is increased too much in this specific type of port, the mass flow rate is hurt in its development phase and throughout the whole blowdown event.

The Effect of Valve Seat Angles on Mass Flow Rate

The next phase of testing on a second die-cast cylinder head involves testing different valve seat angles and valve angles. This die-cast cylinder head is very similar to the one used for the initial testing however differs slightly in geometry. The part number for this cylinder head is RF-F7ZE-6090-A22A. The modification of seat and valve angles is done to understand how these changes can affect mass flow rate. This test is modifying the minimum area of the valve curtain which should change the flow rates dramatically. The flow area is directly proportional to the sonic mass flow rate as given in equation 5. It is evident that theoretically, a greater flow area will better the blowdown performance. This is due to the fact that sonic flow is majorly affected by the minimum area and not affected much by other geometry changes. It is of interest to see which angle and which shape performs the best compared to theoretical.

At lower pressure differentials, when the flow is subsonic, the larger area might not translate to higher performance. This is due to the fact that to have a higher area in the valve curtain, the angles in the valve and valve seat are lower and the gases have to make a sharper turn to exit the cylinder. Figure 3-36 illustrates the valve and valve seat geometry and clearly demonstrates that if the valve seat angle is smaller the gases will have to make a bigger turn to get to the port. The seat angle and valve angle have to be the same in order for the valve to seal when closed.

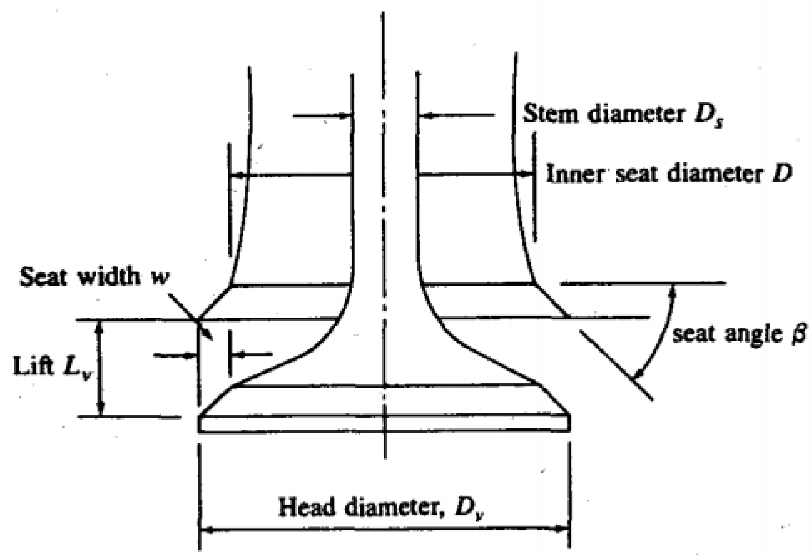

Figure 3-36 - Parameters defining valve geometry including the seat angle [9]

It is possible knowing the angle of the seat and the valve to calculate the minimum flow area using simple geometrical relations. The flow area is given in equation 8.

$$A_f = \pi \cdot D \cdot L_V \cdot \cos\beta \text{............} \quad (8)$$

Where D is the average diameter of the valve curtain, L_V is the valve lift and β is the seat angle.

It follows from the equation that the lower the value of β the higher the value of A_f. For blowdown a low angle will increase the mass flow rate. Four different valve designs and angles were chosen to be tested using a die-cast cylinder head with very similar ports. New valve seats and valves were custom made and fitted to the cylinder head. 30º, 45º and 55º degree seats were tested and compared.

Figure 3-37 - Pictures of different angle valves

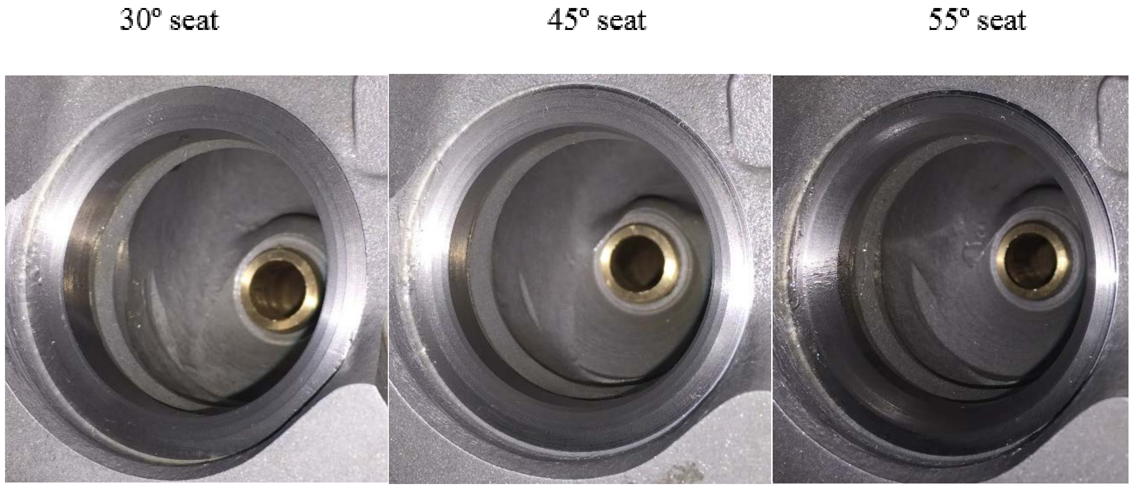

Figure 3-38 - Pictures of different angle valve seats

The diagram in Figure 3-36 is simplified showing one angle for the whole seat. In practice the seats are cut at a few different angles along the seat to create a rounded edge effect with only part of the seat sealing with part of the valve. Typically 3 different angles are cut into the seat. When the valve is opened at 0.1 inches the seat and valve angle can be misaligned causing the minimum area to be greater than that of the seat angle. This is the case for the 45º seat. Two different valves were tested on the 45º seat as a consequence of this. Figure 3-37 shows pictures of the different valves used in the test. Figure 3-38 shows pictures of the different seat angles that were cut. From the pictures the multiple angled cuts are visible with one of these cuts being the sealing surface with the valve. Drawings of each valve curtain area were made in order to analyze the differences between each port and find the minimum area of the ports. They are shown in Figure 3-39, Figure 3-40, Figure 3-41 and Figure 3-42 for each respective valve and seat combination.

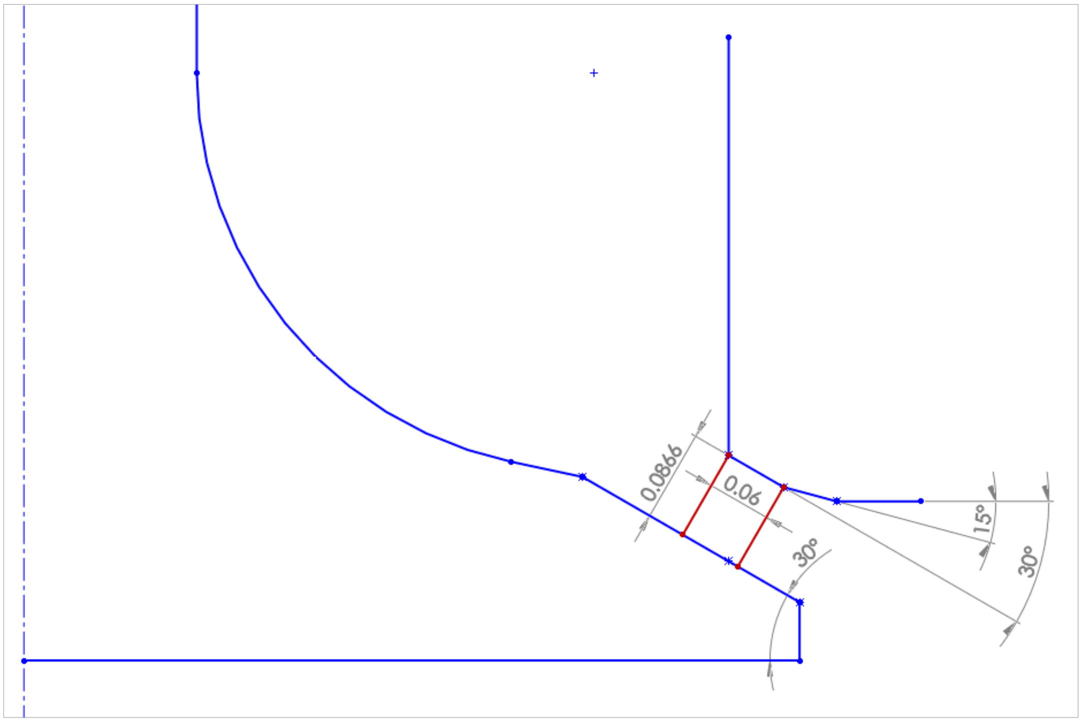

Figure 3-39 - Drawing of the 30º valve and seat opened at 0.1 inches

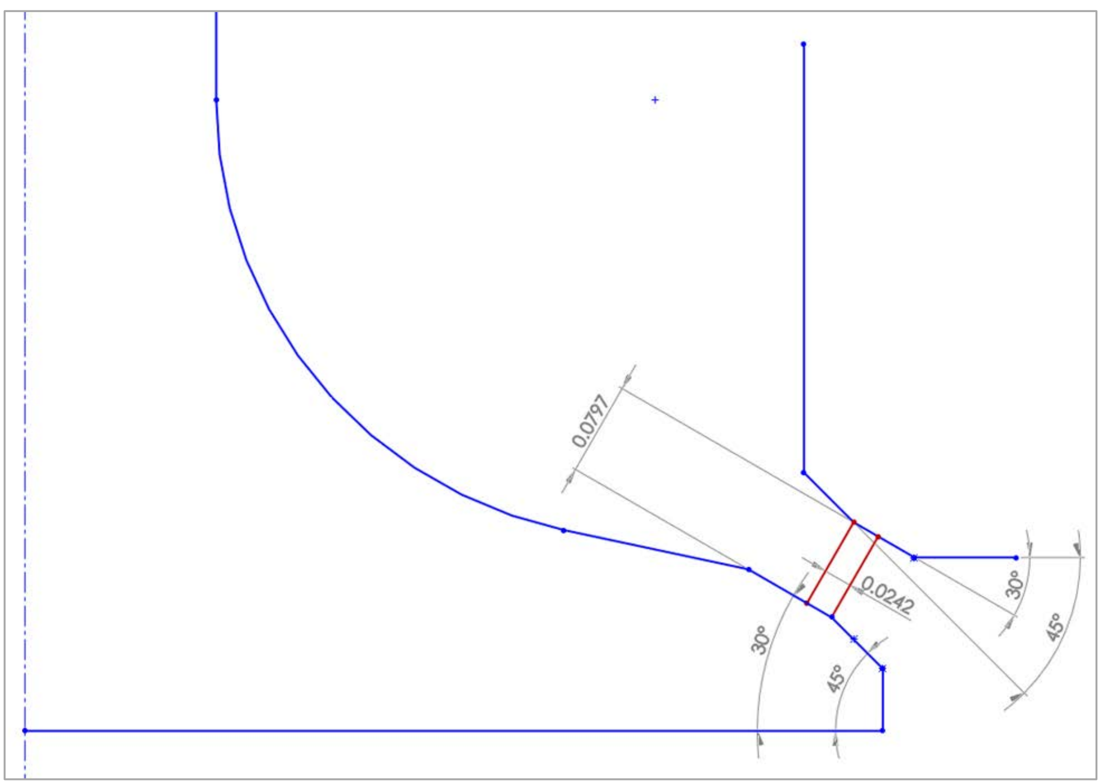

Figure 3-40 - Drawing of the 45º(30º) valve and seat opened at 0.1 inches

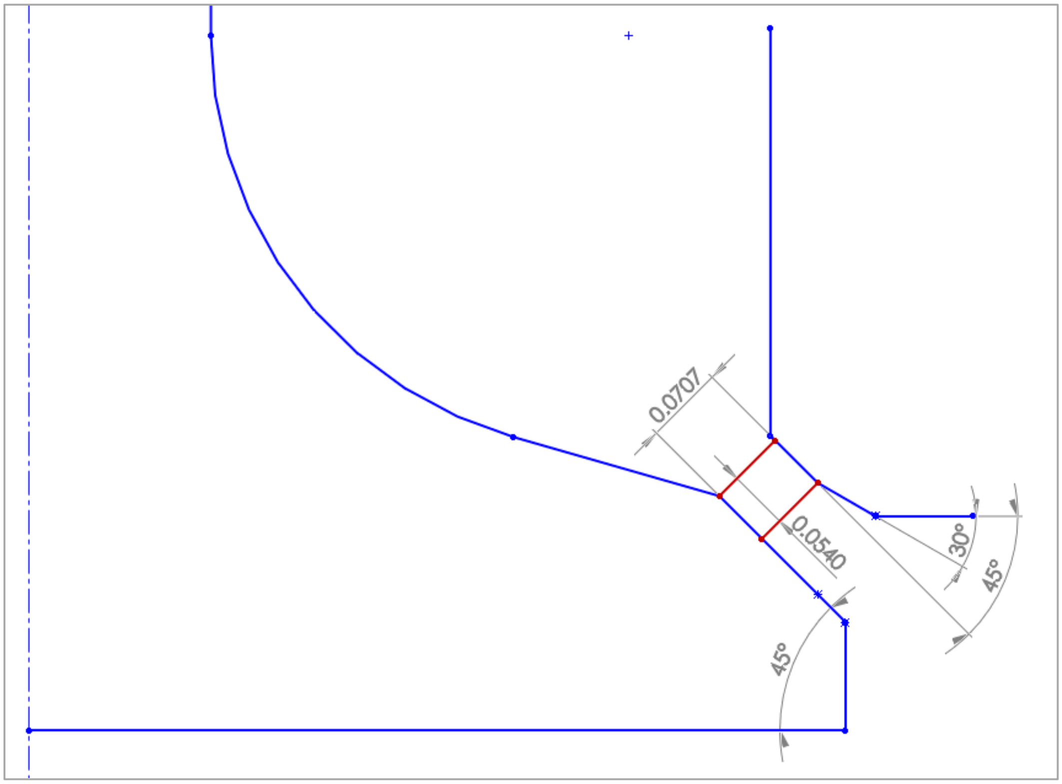

Figure 3-41 - Drawing of the 45° valve and seat opened at 0.1 inches

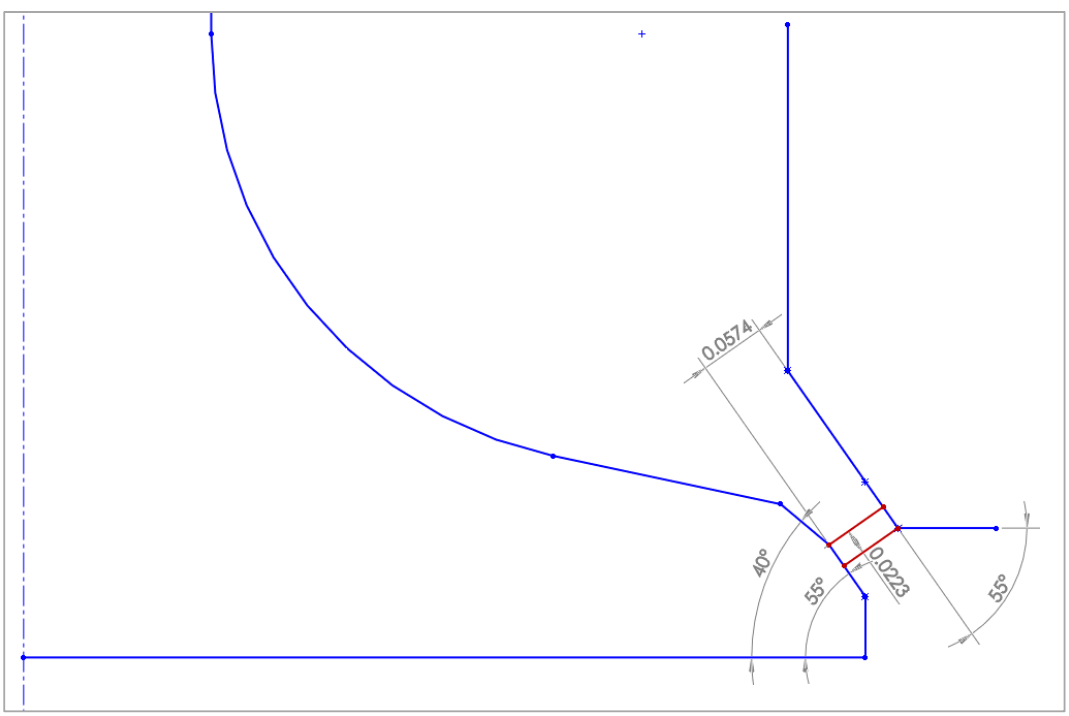

Figure 3-42 - Drawing of the 55° valve and seat opened at 0.1 inches

The drawings of each valve and seat allow to visualize the shape of each new seat and the minimum distance when the valve is opened 0.1 inches. Each valve design is different in shape and the placement of the minimum area differs. The 30º valve and seat combination has a two angle seat and a single angle valve. The seat is cut with a maximum angle of 30º and immediately dishes into the port at 90º which is a very abrupt change of direction for the flow. This abrupt change of direction occurs where the minimum area is, this could have an effect on the flow as discussed further in this section.

The 55º valve has two cuts to it with the outermost cut being 55º and the inner cut being 40º. Contrarily to the 30º seat the 55º seat is cut in one long section. This means the minimum area is this time placed very close to the combustion chamber. The 55º cut is an abrupt transition from the chamber to the minimum area.

The 45º(30º) valve and seat combination illustrated in Figure 3-40 has a two angle valve and a two angle seat. This is more typical of what would be seen in an engine so as to improve the low pressure flow that would occur after blowdown. Machinists making valves know that the 30º overcut to the valve will improve flow at low pressures. The two angles smoothen the transition from a large to small area and back to a large area. The outside cut on the valve is a 45º cut which is matched to the inside 45º cut on the seat when the opening is shut. When the valve is opened however the minimum distance between both surfaces is not placed at the 45º but rather at the 30º cut. This causes the distance and area to be greater than if the 45º cuts faced each other at 0.1 inch lift. The minimum area is located in the middle of the seat far from any abrupt angles upstream or downstream and resembles the design of a sonic nozzle or venturi more.

The 45º valve and seat combination illustrated in Figure 3-41 uses the same seat as the 45º(30º) design however has a one cut valve at 45º. This allows for the minimum area to be located between the two 45º surfaces very close to the transition in the port similar to the 30º design. This does not resemble a venturi design and has an abrupt transition into the port. The flow areas for each seat angle are presented in Table 3-1.

Table 3-1

Seat Angle	**Effective Flow Area (square inches)**
30º	0.3504 *(+22.4%)*
45º(30º)	0.3460 *(+20.9%)*
45º	0.2861
55º	0.2613 *(-9.1%)*

The results of the testing of the three seats are presented in Figure 3-43 and Figure 3-44. The three ports with different angled seats yield results that are different than what has been seen so far. Two of the three experimental curves lie above the theoretical calculation for the given area of flow. The development phases for the 30º and 55º seat curves are much longer due the design of these new seat and valve combinations. Compared to the first die-cast cylinder head used for the previous tests, there is a dramatic increase in flow rate relative to theoretical values.

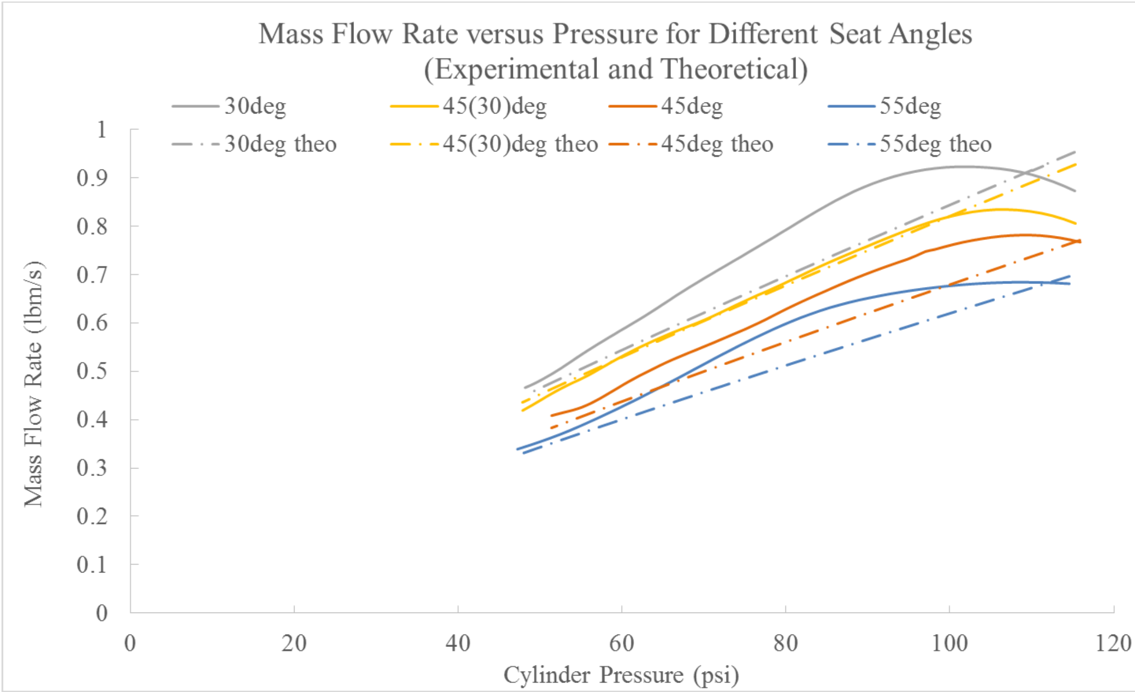

Figure 3-43 - Mass flow rate graph for the die-cast port at four different seat angles

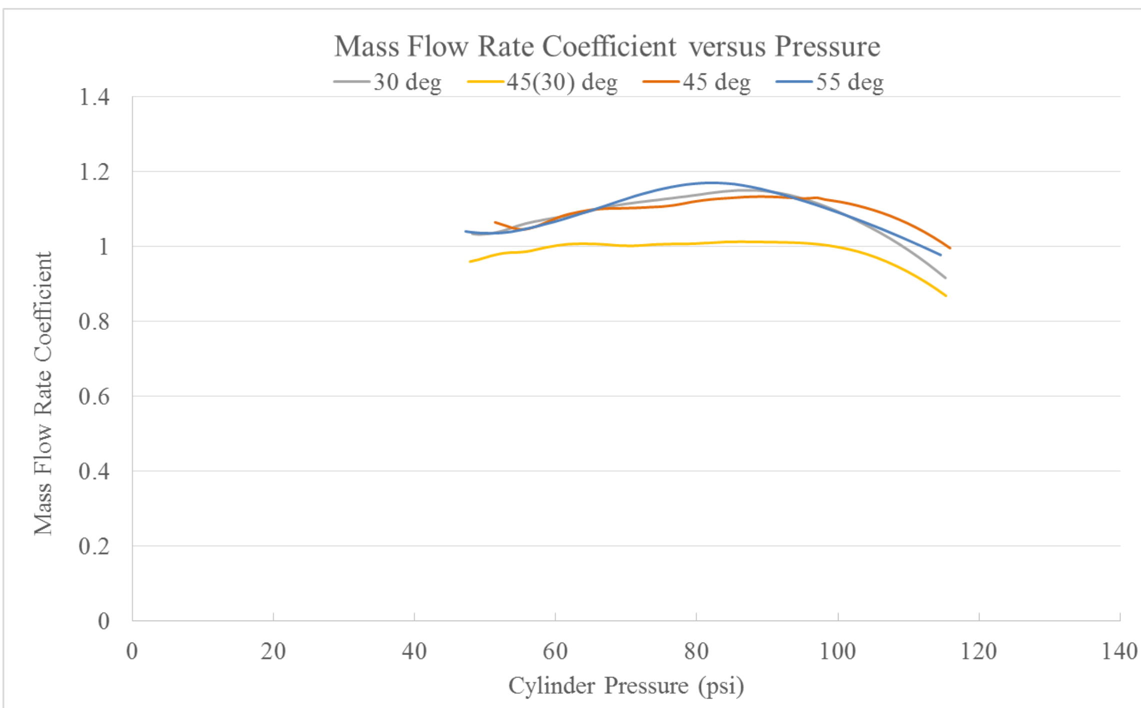

Figure 3-44 - Mass flow rate coefficient for the die-cast port at four different seat angles

From Figure 3-43, the 30º seat performs the best in absolute terms as predicted and it seems like the development phase of the flow occurs from opening down to 50 psi which had not previously been seen. This very long development phase is thought to be caused by the way the valve and seat were cut and where the minimum area occurs. The minimum area, as seen in Figure 3-39, is close to the part where the seat transitions into the port where the turn is very abrupt. The area downstream of the minimum expands rapidly which could cause the instability that is needed for the development phase to improve to higher than theoretical. This is possibly the case due to the fact that the transition from subsonic to sonic could be bowed allowing the area and flow to effectively increase above theory. The 45º exhibits a very similar behavior in comparison to theory to the 30º seat due to their very similar design features. This is shown in Figure 3-44 where both curves reach a maximum of 15% above theoretical and rejoin the theoretical flow as pressure decreases.

The 45º(30º) seat follows theory very closely overall and shows a behavior very similar to the results seen in the previous die-cast experiment for the stock cylinder head. The valve and seat design for this specific port test were very similar to the one shown in Figure 3-4. The valve and seat make smooth transitions from the combustion chamber to the port. The results from the stock die-cast test and the 45º(30º) seat test are compared in Figure 3-45. The curves all show similar properties. The initial development phase occurs under theoretical flow rate and reaches it rapidly and stabilizes close to the theoretical value. This is done by having multiple cuts on the seat which causes the minimum area to be placed far from the abrupt changes in angle that are located at the extremities of the valve curtain area. This is unlike the 30º, 45º and 55º tests.

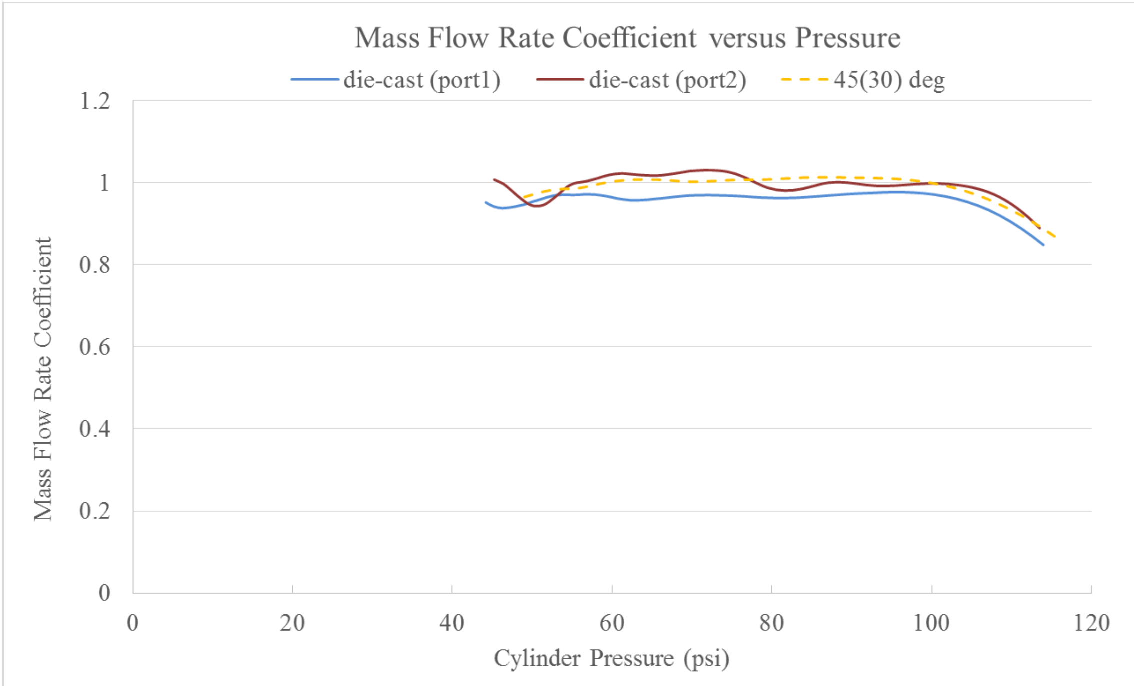

Figure 3-45 - Comparison of die-cast and the 45º(30º) mass flow rate coefficient

The 55º seat has an extremely lengthy development phase where the flow surpasses the theoretical flow rate for the minimum area by the largest percentage. It seems to come back down to theoretical values around 50 psi similarly to the 30º seat. This is believed to be triggered by the fact that the minimum area is very close to a very abrupt change in volume upstream. The shape of the valve and seat do not have the nozzle like shape required for the flow to develop fast and reach the minimum area. This is an advantage for blowdown on an engine as it increases the flow rate in the developing phase of the flow. It is clear that it is desirable to have transitions from high to low volume and back to high volume that are abrupt and not gradual as is the case of the 45º(30º) case.

The other die-cast head never showed such long development phases and the flow would settle under or on the theoretical line much quicker. The seats were designed in different ways for this test compared to the previous head and this explains the phenomenon. In terms of designing a cylinder head that works well in blowdown on an actual engine, the type of valves and seats used for the 55º test would function the best as the flow rate is high in the initial phase of blowdown and stays high.

Excluding the area change, the shape that yields the best result for blowdown is the 55º seat and valve as presented in Figure 3-44 where the data is non-dimensional. The flow relative to theory is best at this angle and suggests that having the minimum area close to the combustion chamber is the best solution by having a one cut seat and then a quick increase in volume downstream of the valve curtain area. The least nozzle like shape is what works best and simplifying the design of the seat and valve is considered the most efficient for best blowdown performance. This design philosophy could be applied to any desired seat angle.

Looking at the experimental data in isolation the advantage rests in using the smallest angle possible for blowdown, namely 30º in this test specifically. Improvements can however be made to this angle seat to make it perform even better compared to theory. This was tested on the regular flow bench as well as it is necessary to see if using a lower angle seat has a detrimental effect on the flow that would outweigh the advantages of using it for the blowdown phase. This is reported in Chapter 4. Some very useful insight is gained by researching the different seat angles and comparing them between themselves. The minimum area alteration and shape choice has a significant effect and should factor in the design choice for the optimal port.

Discussion

The experimental testing of cylinder heads in many different configurations has allowed to gain a better understanding of blowdown going through an exhaust port. There are generally two parts in the process for this following the valve opening. The first part occurs at high pressures right after the valve opens and is a phase when flow is transient and still forming. This is a phase that sees the mass flow rate increasing over time and eventually reaching a quasi-steady state flow. Interestingly, this phase allows for the flow rate to reach above the theoretical maximum for a minimum valve curtain area. The second phase of the flow occurs at lower pressures and is characterized by the experimental data agreeing with the slope of the theoretical curve. Depending on the downstream setup and port used, the data in this quasi-steady state phase can be in agreement or below theoretical which is likely due to the flow not being sonic around the whole valve curtain. This is likely due to the asymmetry of the port and the clearance volumes in some areas upstream and downstream of the flow. From the analyses made on all the different configurations it is clear that the changes in downstream shapes affects the initial developing phase of blowdown the most. This has the highest influence on changing the average mass flow rate. This is highly important as it can explain why the engine performs better when this part is improved.

Putting this experimental setup in context with the operation of a real engine it is noted that the valve is not moving in the experimental setup. It is set at a height that represents approximately one fifth of lift near bottom dead center of the total opening of the valve during an exhaust cycle. The valve is constantly moving in the engine which suggests the flow will never reach the quasi-steady state configuration seen in the blowdown tests.

This is further confirmed by measuring how long the development phase of the blowdown takes at 0.1 inch lift in comparison to the exhaust event in an engine blowdown process. The development phase of the flow for the stock die-cast cylinder head takes 0.0172 seconds. Comparatively a full exhaust event at 5000 rpm occurs in just 0.006 seconds and the blowdown phase takes 0.0013 seconds. The development phase in the experiment takes 13.2 times longer than the whole exhaust event. Improving the results from the initial phase of blowdown will have an effect on the efficiency of the exhaust stroke. This comparison reinforces the fact that engine designers assuming sonic flow through the exhaust port for blowdown is erroneous as the flow never actually reaches a quasi-steady state point in the engine.

Ideally, modifying the sonic flow bench to have a moving valve as opposed to a static one after opening would allow to capture the mass flow rate as a function of time and displacement of the valve. Practically, this is however extremely difficult to achieve because the opening has to occur only once in time and the accelerations necessary for this are extremely high. It is easy to do when the system is spinning at 5000 rpm, however more complicated to do for one event with no inertia in the system prior.

It is clear that with such a short length of time for blowdown being of importance, increasing the performance of the port right after the opening is the most beneficial. In the case of the die-cast cylinder head this has occurred in the case of the cylinder side being modified keeping the same valve seat. The seat angles were also investigated suggesting that a low seat angle should be used with very abrupt transitions upstream and downstream. The 30º seat angle increases the area of flow the most and it will be important to compare it to the regular flow bench results. The flow at subsonic velocities at higher lifts may be affected negatively by a low seat angle which might not make it the best option overall.

The investigation of the seat angles gave some further insight into what shapes for the valve and seat should be used. It is concluded from the drawings and results that in order to promote a high mass flow rate in the development phase, the design has to place the minimum area close to abrupt volumetric changes. This means there needs to be a very sharp transition from the combustion chamber to the valve curtain area. This also means that the area downstream of the valve curtain needs to be expanded as much as possible. This also explains why removing material from the port slightly downstream of the seat in the case of the first die-cast cylinder head can increase the flow rate above theoretical in this development phase. This effect can also be achieved by flattening the cone angle of the valve downstream of the seats. The cone angle is the surface immediately downstream of the valve seat on the valve.

Chapter 4 Regular Flow Bench Testing

Regular flow bench testing is done to analyze the effects that modifications made to cylinder heads may have on the subsonic part of the exhaust stroke. Improving the flow rate through a port in the blowdown phase does not necessarily mean that it will improve overall on the engine. When designing a port it is important to take both tests into account. Figure 2-3 illustrates the way the regular flow bench works at a pressure differential of 28 inches of water. The low pressure bench illustrated in Figure 2-3 was used for testing the three cylinder heads to get a better understanding of the overall performance of the ports. The die-cast port produced higher flow rates than the sand-core port when compared on the sonic flow bench. The results for the low pressure testing will show which cylinder head flows better at subsonic speeds. The fully modified die-cast port that is smoothed out to look more like a sand-core port was also tested to see the added benefits of porting. The second die-cast cylinder head with the different valve and valve seat angles was also tested to provide insight into what seat would be best for the exhaust stroke as a whole.

Die-cast and Sand-core Port Testing

On regular flow benches the data is recorded as a steady state reading at different valve lifts. The data recorded is a pressure differential across an orifice which can then be converted to volumetric flow rate. Five valve lifts are recorded from 0.1 to 0.5 inches of lift at even intervals. Figure 4-1 reports the results for the sand-core (SC) and die-cast (DC) cylinders tested on the low pressure bench. Three exhaust ports were tested on the sand-core head and two of them on the die-cast head. Cylinder 1 on the die-cast head has been fully modified and was tested in the configuration shown on the right side of Figure 3-23. Volumetric flow rate is very sensitive to the smallest of changes in a port therefore the three sand-core ports were tested as there can be some discrepancies in flow due to minor changes in geometry [8].

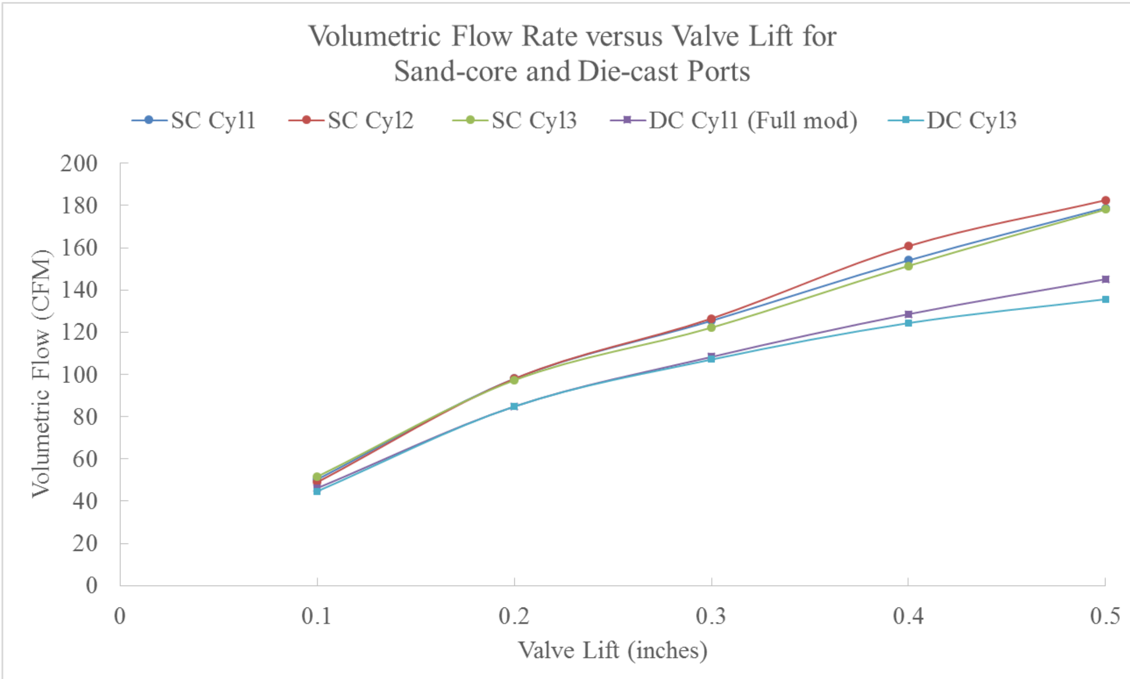

Figure 4-1 - Volumetric Flow versus valve lift for 3 sand-core ports (SC) and 2 die-cast ports (DC) (one fully modified)

The results from the low pressure testing show some predictable results for the two ports tested. Clearly the sand-core port outperforms the die-cast port due to it being much better suited for subsonic velocities. The shape is smoother, makes less of an abrupt turn around the short side radius and the port is shorter. This port is designed using this type of flow bench for high performance. The air stays attached to the walls of the port much more easily when the turns are smooth and less abrupt. The die-cast port is hurt by its abrupt angles inside the port and its high turn angle from the valve to the exit of the port. These angles have for effect to detach the air from the walls of the port which hurts performance greatly as extra turbulence and flow disruptions are introduced.

The modified die-cast cylinder performs better at higher valve lifts above 0.3 inches than the stock shape. This is due to the fact that the short side radius was smoothed causing less of a detachment in air flow on the surface of the port allowing for better overall flow rate. The detachment is probably moved to further down the port which allows for a better effective flow area downstream inside the port. The flow under 0.2 inches is not really dependent on the far-field downstream geometry but more by the valve and seat geometry hence why the changes in flow only start occurring at 0.3 inches in lift.

Valve Seat Angle Investigation

The same tests were run for the different valve seats that were investigated for blowdown. This can be compared to the results achieved in blowdown. The drawings of the shapes of each valve curtain area will help understand the results of the regular flow bench test. At low lifts under 0.2 inches the flow is mostly dictated by the area available for the air to pass through. At higher lifts however the shape of the seat and the smoothness of the seat are the deciding factor in the results. It is expected that the 30º seat will outperform the 45º(30º) and 55º at low lifts. At higher lifts the smoothest shape should be best and this should occur for the 45º(30º) seat. The drawings shown in Chapter 3 previously are presented from Figure 4-2 to Figure 4-5 in this section to refer to during the analysis. The results for the three different seats are presented in Figure 4-6.

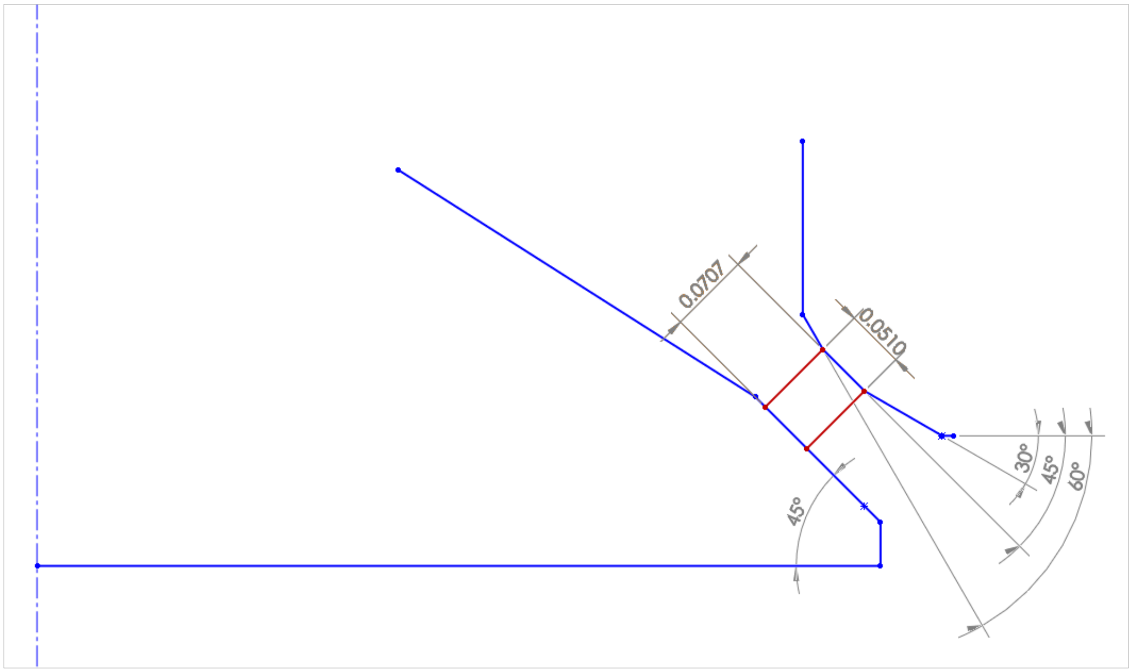

Figure 4-2 - Drawing of the die-cast port valve curtain area, including the valve outline (bottom), seat outline (top) and the location of the minimum area (red)

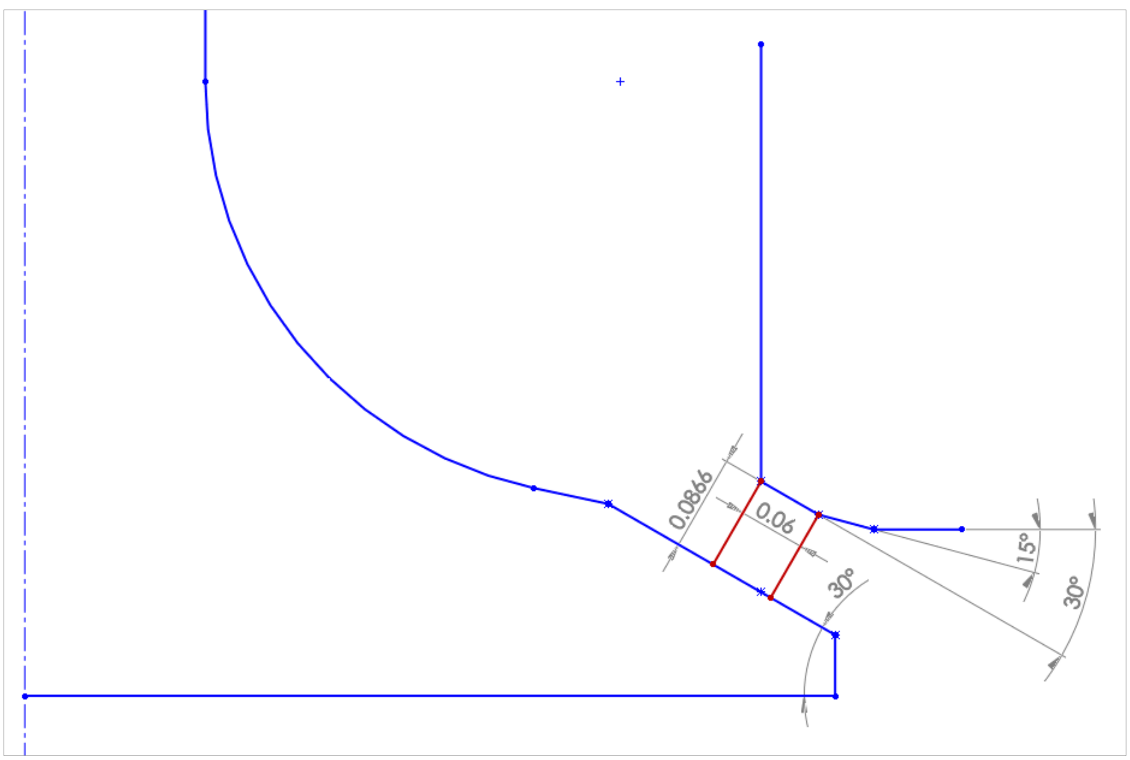

Figure 4-3 - Drawing of the 30° valve and seat opened at 0.1 inches

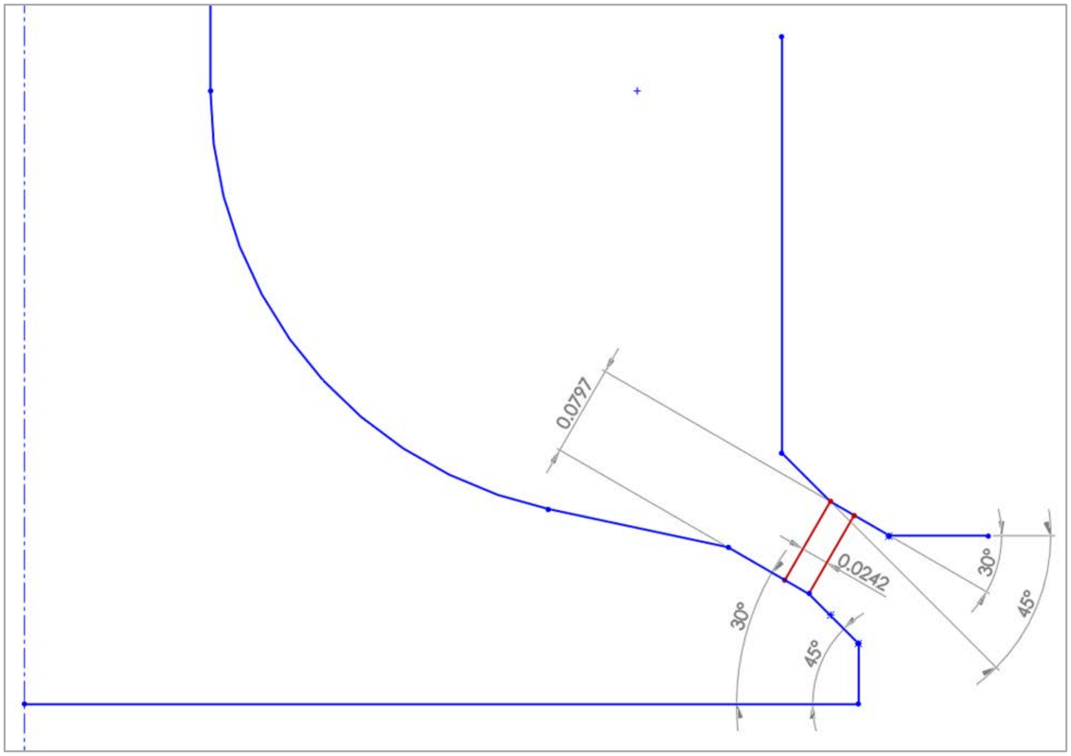

Figure 4-4 - Drawing of the 45º(30º) valve and seat opened at 0.1 inches

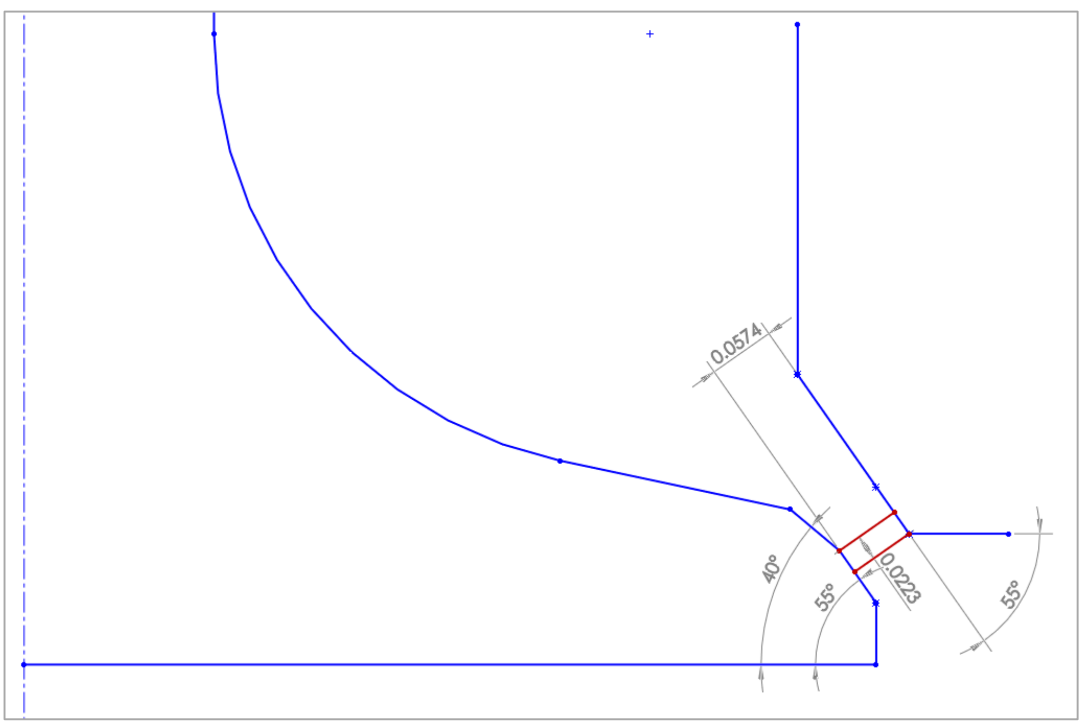

Figure 4-5 - Drawing of the 55º valve and seat opened at 0.1 inches

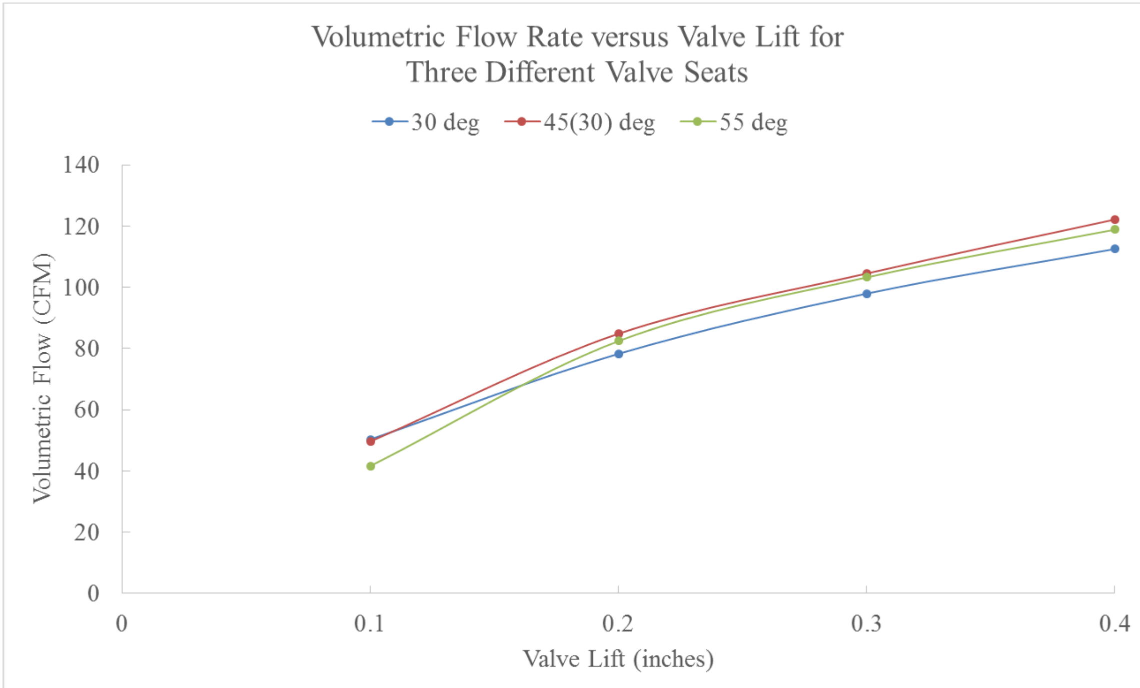

Figure 4-6 - Volumetric Flow versus valve lift for three different valve seats

Looking at the three curves in Figure 4-6, it is immediately clear that the best overall flow rate is achieved by the 45º(30º) seat and valve design. At 0.1 inch lift the best performing port is unsurprisingly the 30º port closely followed by the 45º(30º) port. The flow areas differ by 1.3% causing the difference in flow rate to be negligible. The 55º port is well below the others due to the much smaller area through the port. At 0.2 inches lift, the results are surprising for the 30º in which the flow is lower than the other two ports. This is due to the seat transitioning from 30º to 90º very close to the minimum area causing a very large separation at the wall which is known to hurt the flow a lot. This separation continues to occur throughout the range of lifts tested.

The 55º port shown in Figure 4-5 gets closer to the values of the 45º port at 0.2 inches lift and above due to its higher angle causing the transition from the cylinder head to the port to be smoother than at 30º for example. At 0.3 and 0.4 inches lift the air goes around the curvature smoothly for both ports as they have smooth transitions downstream from the valve. The flow does not detach from the wall following the valve curtain area in both cases. Overall it clear that the 45º seat and valve combination is the best option at low pressures which in an engine would occur after the blowdown phase has completed. This seems to be due to the smoothness of the transition that occurs just downstream of the valve and valve seat.

Looking at this 45º(30º) seat in comparison to the original die-cast cylinder head it is possible to compare both valve geometries drawn in Figure 4-4 and Figure 4-2 respectively.

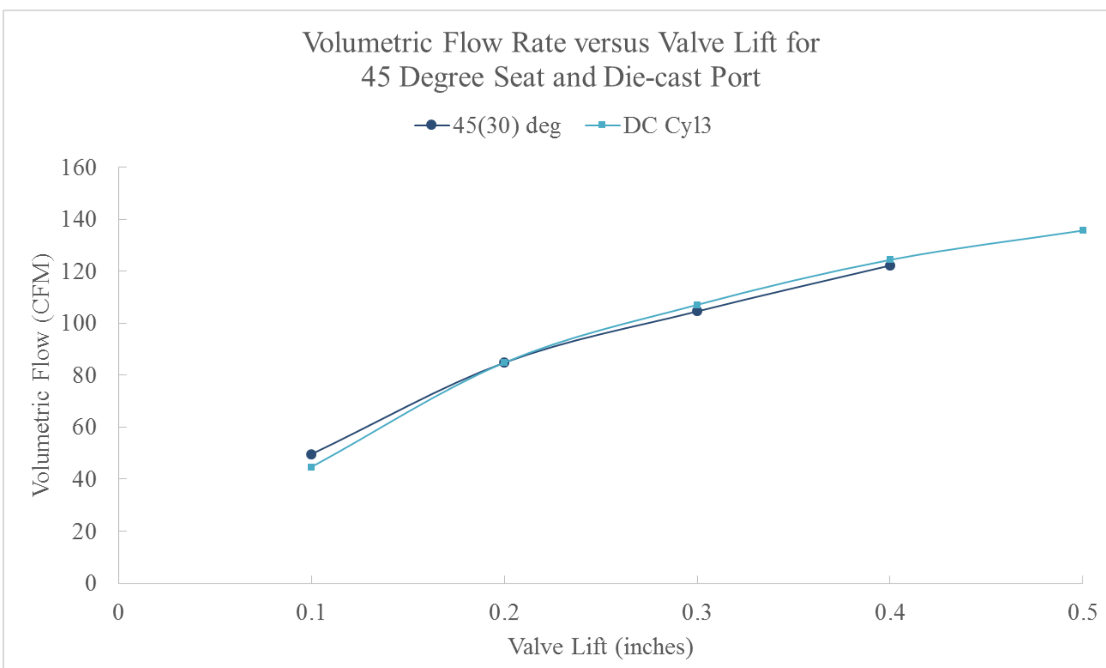

Figure 4-7 - Volumetric Flow versus valve lift for three different valve seats

The comparison between both the 45º(30º) seat for the seat experiments and the 45º seat for the initial die-cast port testing is made in Figure 4-7. The seat drawn in Figure 4-4 shows that at 0.1 inch lift the minimum distance at the valve is actually larger than what the seat in Figure 4-2 is. This is because of the misalignment of the valve and seat surface. This means that the seat test port performs slightly better than the original die-cast port at low lifts. At higher lifts however this trend is reversed by the fact that the seat and valve in the original die-cast test transition more smoothly into the port. This is due to the three cuts on the seat and the higher valve cone angle. The differences are however small and the blowdown results for both ports show a clear gain in performance by the larger minimum area seat shown in Figure 4-8.

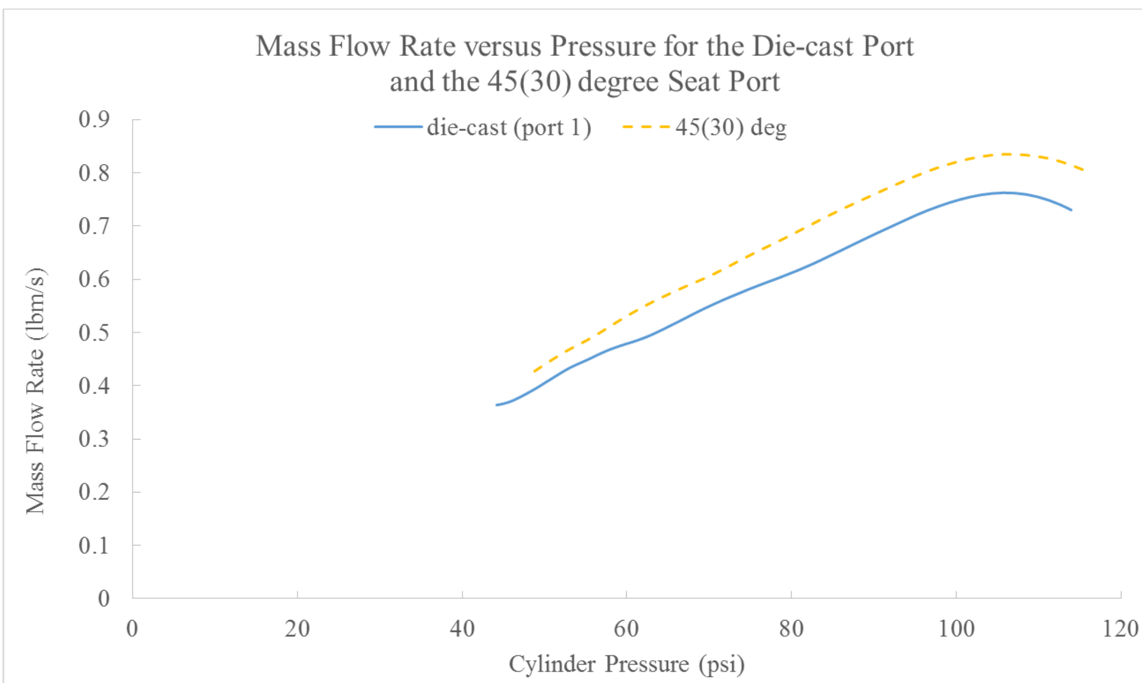

Figure 4-8 - Comparison of the die-cast and the 45º(30º) mass flow rate

The increase in blowdown performance and the very small loss in low pressure flow make the valve and seat combination tested in the valve seat investigation a better choice for the engine overall. This can be then improved by downstream modifications such as the ones made to the die-cast port originally or further improved by optimizing the valve and seat geometry for blowdown more without hurting the low pressure flow.

Using the right combination of blowdown performance and low pressure flow performance is the ultimate goal of this study but this requires extensive engine testing that exceeds the scope of this study at this time. In order to verify which test between blowdown and regular flow bench testing has the most effect on the power output of the engine many different configurations have to be tested on an actual engine. For example, it is clear from blowdown testing that the 30º seat improves blowdown significantly however on the low pressure bench the results are much lower than the 45º(30º) seat which seems to provide a good balance of blowdown performance and low pressure testing. It is not possible to tell at this time which seat shape would perform best on the actual engine. All that can be said is that the combination of modifying the downstream shape of a port and shaping the valve and seat can provide performance gains in both blowdown and low pressure testing.

Chapter 5 CFD Modeling of Exhaust Blowdown

Having learned a great amount of important information about blowdown through experimental testing, CFD modeling is a tool that can be used in parallel to better understand the process. Capturing the flow characteristics seen in experimental testing is a challenge due to the complex geometries used and the limitations presented in modeling. The method used for CFD was a process of trying the simplest approach and adding degrees of complexity. The options in preparing a problem for testing in CFD are numerous and involve a lot of different configurations. Using ANSYS Fluent as the CFD solver, it is possible to test the flow with over ten different turbulence models for the air; all calculating the results for air flow with different equation systems. The meshing of the flow field also has many different parameters to consider. All in all it is important to get a reliable solver that can model the real flow to the best ability and hopefully show some changes in flow rate due to downstream modifications.

From the blowdown results in Chapter 3 the changes downstream of the valve alter the flow in the development region the most. It also has an effect on fully developed flow but this should not occur in the engine in operation. This is because the time of blowdown is so short in the engine and the valve is continually moving meaning there never is time for the flow to settle to a quasi-steady state phase. CFD is modeled based on the experimental test and not the engine operation as it is extremely complex to create a CFD model with a moving valve and would require much more computing power than available.

CFD Workflow and Results

The CFD modeling process started by taking measurements of the valve and the valve seat from the die-cast and sand-core cylinder heads to create the geometry for CFD. Once the geometry was chosen, an initial mesh of the shape could be done. The initial mesh could be done in two main different ways: three dimensional (3D) or two dimensional (2D). 3D modeling would allow for better shape accuracy and allow the shape downstream of the flow to be exactly the same as in the experimental tests. It would also allow to take into account any asymmetries in the shape. Conversely, 2D testing would use an axisymmetric line to simulate a 3D flow. Axisymmetric 2D simulates a plane of what the 3D flow would be. This method is computationally faster and allows to use a finer mesh which can be required.

Initially, both options were considered and evaluated to get an understanding of what method would be best suited based on computational time and accuracy of results. Using a 3D mesh there was a restriction of less than 1 million elements to get the fluent solver to run a simulation in under 30 days. This time scale would be appropriate if this mesh size could capture a change in flow by modifying the downstream shape. This was not a certainty and would require too much time to test all configurations and solvers available. It was vital to get a faster time between the start and end of a simulation to test many different designs. In addition to this, 1 million elements for supersonic flow through a valve this size would not be enough to capture any differences in flow due to the mesh being too coarse. This is known because mesh convergence was not met at this element size. Mesh convergence occurs once the mesh is made smaller and the results don’t change between mesh sizes anymore. This means the results are independent of the mesh. This was not the case at the 1 million element limit.

The decision was made to use the axisymmetric 2D simulation to fine tune the simulation and be able to make the solver robust and converge reliably. The axisymmetric 2D simulation is also limited by mesh size but this time the mesh is only on a 2D cross section which means the elements can be made a lot smaller than the 3D elements. The smaller the elements the better they accurately capture the fluid behavior but the longer the computational time. Using a transient simulation was chosen initially to try to duplicate the experimental test. This was done by the release of a high pressure fluid from the chamber into the port.

This is a computationally intensive process but can be done within a three day period with a 200,000 element mesh. This is the realistic mesh size limit for this type of modeling using the computational power available. No supercomputer was available to do this simulation work. With this limit, the element size could be made 10 times smaller than the 3D mesh which increased the possibility to capture the flow accurately. Using these limits set by the computational time two different downstream configurations were modeled initially. One port would be a straight exit port and one would have a downstream tube added to it just like in the experimental phase of this project. The two configurations are shown in Figure 5-1. This duplicates the initial experiment done.

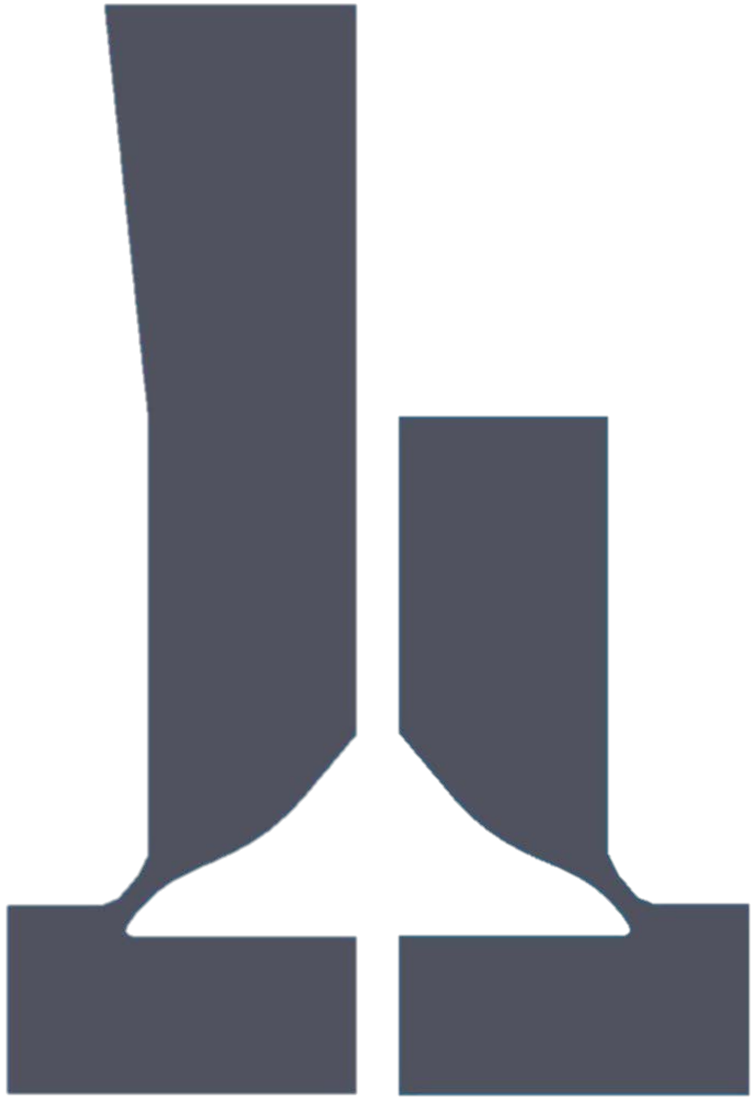

Figure 5-1 - Comparison of extension design (left) tested and straight tube (right) tested in the transient simulation

The goal was to see a change between both ports although it was possible that no change would be seen just like in the die-cast port experimental example. Different mesh sizes and solvers were investigated. In order to know whether the mesh was correctly simulating the flow, different size meshes were tested starting with a coarse mesh and refining it. Once it is fine enough, two meshes with different number of elements should give the same results in terms of flow and flow pattern.

The pressure change over time was recorded and it was deemed that in terms of flow rate a 70,000 element mesh could capture the pressure drop well enough and any refinement above that gave the same flow rate results. Using this as a mesh sizing for the simulation the two different configurations were tested and showed no change in pressure drop however the shock patterns downstream of the valve curtain area changed. This is possible according to theory and will not change the mass flow rate through the opening.

Three different solvers were initially tested and all gave slightly differing results which is expected due to the different nature of each solver. The three solvers tested are Realizable k-epsilon, k-omega SST and Transition SST. The three different models were chosen due to their potential to simulate the flow correctly. The standard k-epsilon and k-omega models are two equation models which are the most widely used [10] [11]. They are the most developed and have the most literature available to work with. The standard k-epsilon model is very good for many different applications of flow however is bad at predicting flows with adverse pressure gradients and strains accurately [12]. The Realizable model was used instead due to its better estimation of viscosity using an eddy viscosity equation and improvement in flow prediction at high Reynolds number [13].

The Shear Stress Transport (SST) k-omega model by Menter [14] combines advantages of the k-epsilon and k-omega models in predicting aerodynamic flows. It predicts the boundary layer with strong pressure gradients very well which is necessary for this type of testing. This model has been validated against experimental data with very good results [14].

The transition SST model is similar to the k-omega SST model except it is a 4 equation model combining the k-epsilon and k-omega solvers together with a new set of equations. The four equation model increases computational time and could be used in the future Using the literature, the initial results and the robustness of the solver for sonic flow a k-omega SST solver was chosen to run simulations.

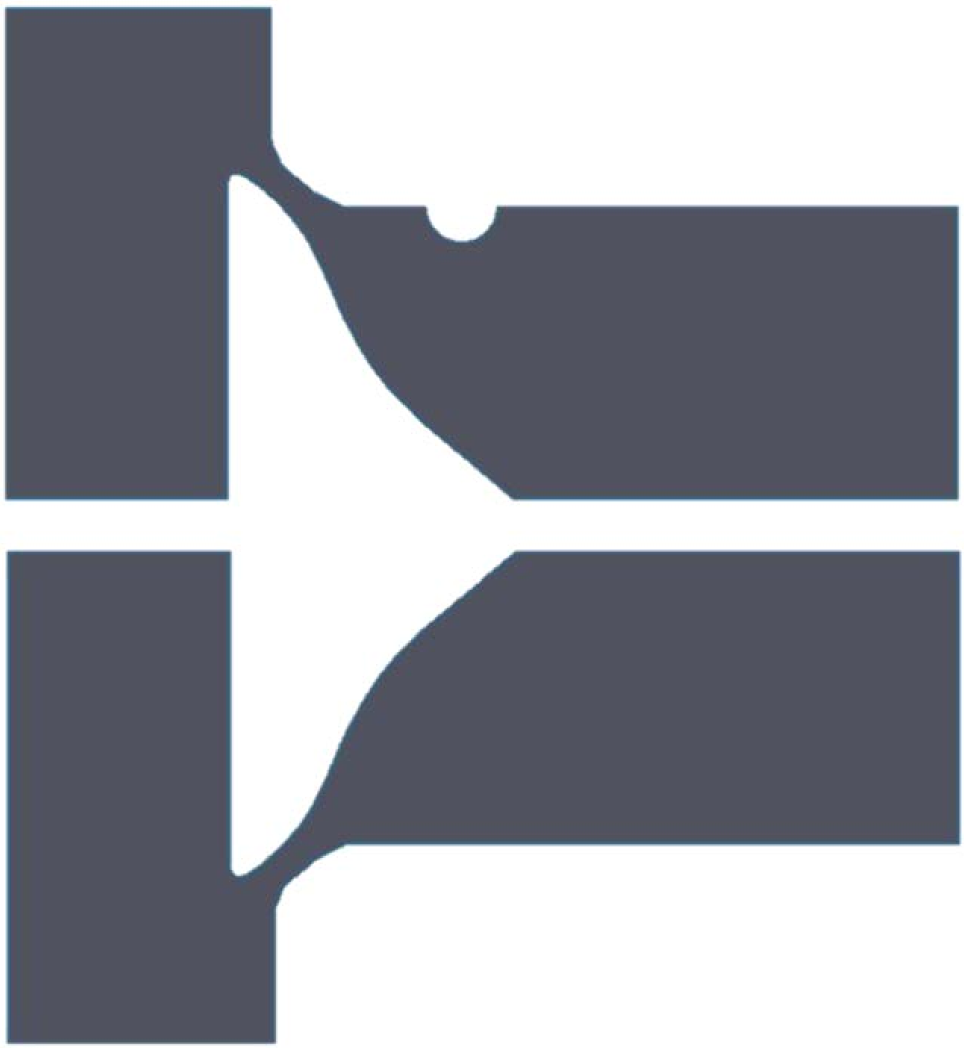

Figure 5-2 - Comparison of notched design (top) tested and straight tube (bottom) tested in the transient simulation

More testing was needed to see if a modification closer to the valve seat inside the port could affect the flow. A change in geometry was created by adding a notch close to the valve seat in the port to see if any change in flow could be captured as shown in Figure 5-2. The addition of a notch is an extreme modification to the downstream shape that would never actually be done in a real port. It is done in order to try to provoke a change in flow if at all possible. The pressure in the upstream chamber as a function of time is reported for both these tests in Figure 5-3. Clearly, both pressures in the chamber are nearly exactly the same. The ends of the notched port curve were cut in order to show that the regular port data is exactly the same. These results suggest there are no changes in mass flow rate between the two designs using CFD.

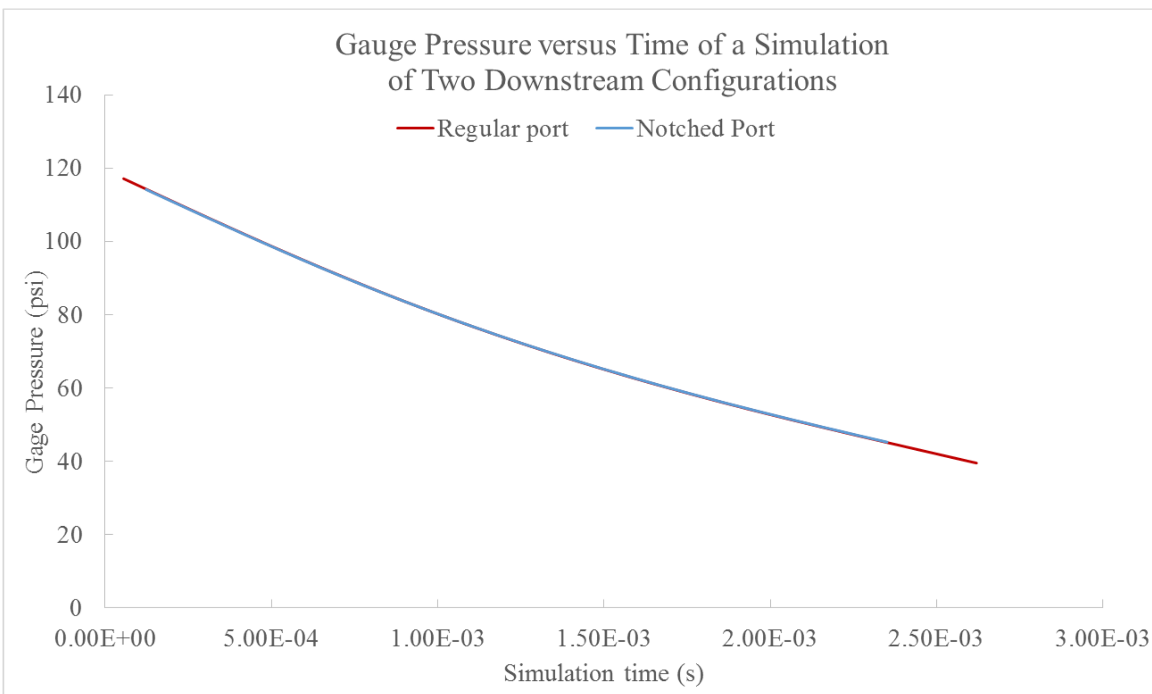

Figure 5-3 - Gage pressure versus simulation time for two different port designs

The fact that there is no change in flow even in the early stages of development suggested that more work needed to be done on the mesh or the solver used.

At supersonic velocities, the boundary layer of air on the surface is very small and this has to be modeled correctly. A prism layer needs to be added to the model in order to do so. It allows the mesh to have a very high number of elements very close to the boundary surface of the flow. It is a stack of elements all of the exact same size and increasing aspect ratio in layers. It allows to compute the boundary layer flow and the increase in size of the boundary correctly as the flow follows the path of the port and valve. This needs to be set up correctly as it is the only way to model separation accurately and results can vary greatly if this feature is not well prepared.

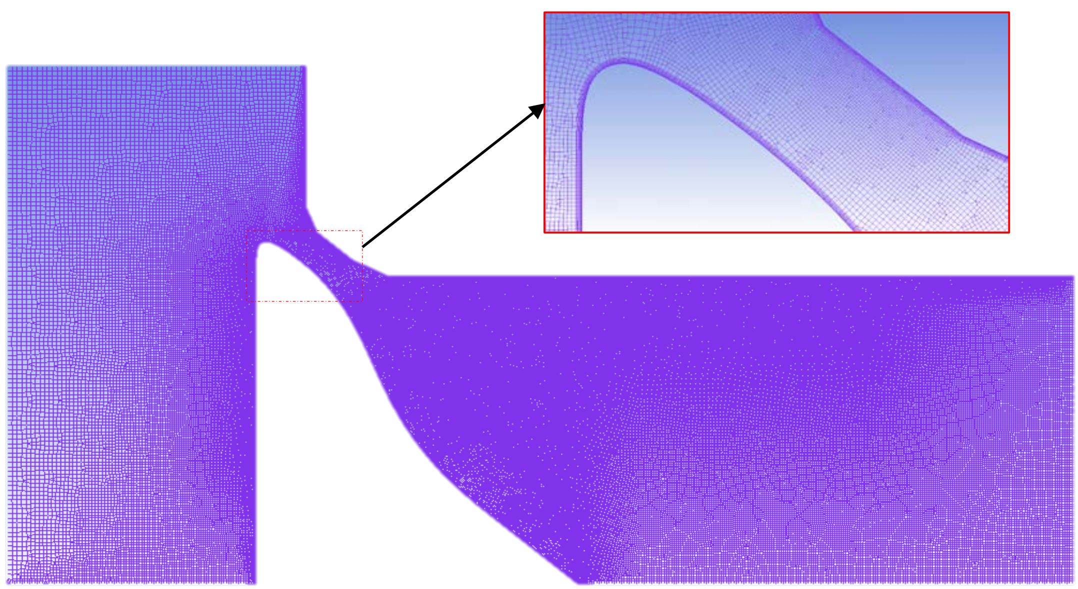

Figure 5-4 - Mesh of the port with an inflation layer

Fluent yields a y+ value which allows to see if this prism layer is set up correctly. The idea is that the y+ value has to fall within a certain range in order to calculate the boundary layer thickness correctly. The range depends on the wall function used in Fluent. The prism layer is shown in Figure 5-4 inside the red box. This was originally added to the 70,000 element mesh however was too large to capture the flow accurately on the surface of the valve and valve seat. The y+ value was outside the recommended range.

In an attempt to understand if a finer mesh was needed, looking back at the initial tests between 20,000 and 200,000 elements, the downstream shocks were different for every test. This was the case above 70,000 elements showing that a higher mesh density could still yield differences in flow. Modeling at a higher number of elements was therefore necessary to evaluate the shocks correctly and allow the mesh to be paired with a small enough inflation layer. The prism layer has an effect on the rest of the mesh size due to the fact that there needs to be a smooth transition between that layer and the general mesh. For example, a 25,000 element mesh could not be used with an extremely small inflation layer as the interface between both sizes would create a large error. The transition between inflation and general mesh needs to be smooth as shown in Figure 5-4. To achieve an appropriate inflation layer for this problem, a 350,000 element mesh is needed. This is more than the 200,000 size which would solve within 7 days. This new size would be solvable in 30 days. This is a timescale that does not allow for fast design of a port.

The solution to this is to simplify the problem and get away from the experimental requirements. Due to the symmetry around the axis, the experimental data is not duplicated in CFD for now but different shapes and their effects can still be tested. The modeling was transient up to this point however it can be modified to run at steady state at one given pressure. Instead of having a chamber that releases gas at high pressure, a pressure-inlet used that would supply a constant pressure of air upstream of the valve and the mass flow rate can be recorded depending on the geometry. This setup is shown in Figure 5-5. This simplification eliminates the transient effects caused by the cylinder emptying that are needed to understand the process of blowdown in an engine. This change therefore is used to see if the CFD solver will yield some differences in flow rate at steady-state. This should not be the case according to theory as long as the valve curtain has sonic flow all around. This will most likely be the case as this is a symmetric setup.

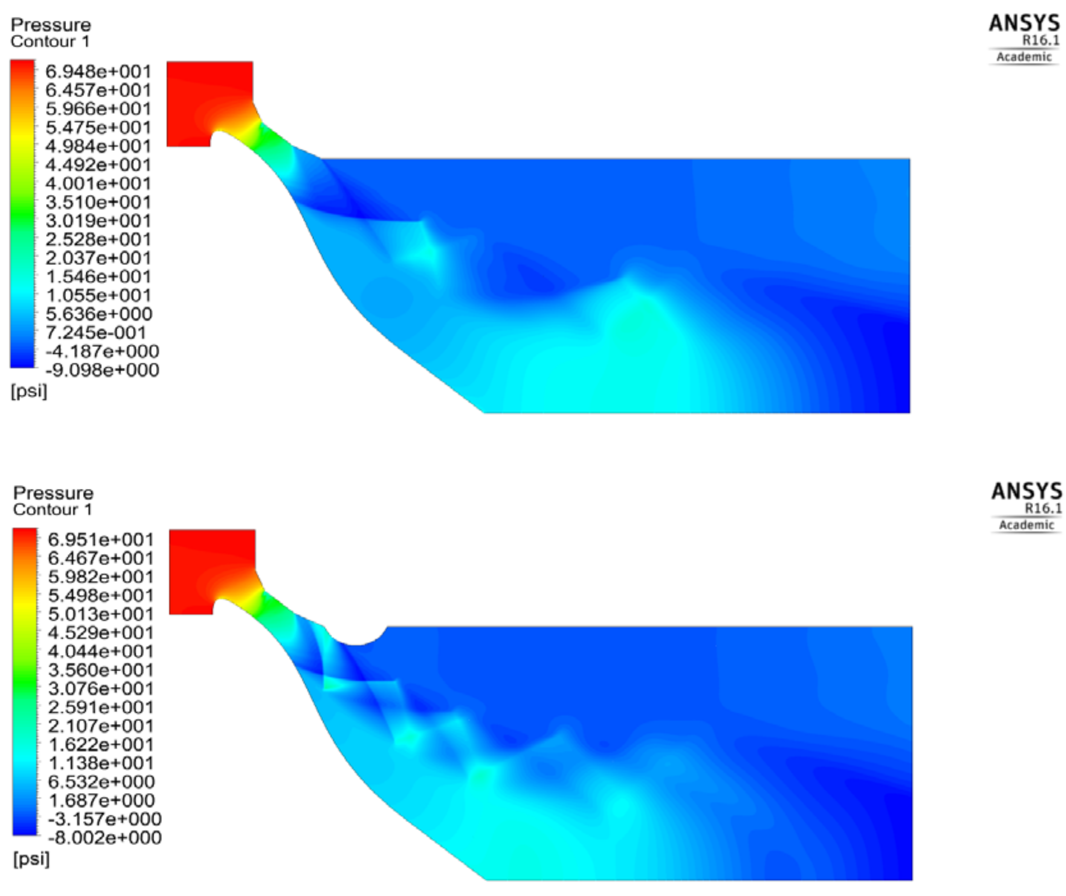

Figure 5-5 - Steady state testing of port without notch (top) and with notch (bottom) at 69.5 psi inlet pressure

This modeling is much faster due to the fact that only one moment in time is being investigated and a 350,000 element mesh solves in 12 to 24 hours. A notch in the port close to the valve seat was investigated with different initial pressures and showed no significant changes in mass flow rate across the valve curtain.

As seen from Figure 5-5, the notch has an effect on the pattern of shock and pressure downstream of the valve. The mass flow rate is however within 1.5% along the range of pressures from 120 to 40 psi. The pattern of shocks downstream of the valve have no effect on the flow through the actual valve curtain. The maximum of 1.5% difference between both ports could be due to error between two slightly different meshes due to the notch or could also be due to numerical inaccuracy.

CFD uses flow residuals as the metric to know if a simulation has converged to an acceptable solution. Residual convergence is the change in values calculated between two iterations. Currently the convergence of the flow residuals is set to 10e-3 which is standard for CFD but can be lowered further to 10e-4 if the problem requires even more accuracy. In an ideal case once the change in values reaches zero between two consecutive iterations the solution is converged but this does not happen. A criterion of convergence is therefore set. The convergence of the mass flow rate per iteration is presented in Figure 5-6 and Figure 5-7 for both cases respectively. Initially the flow develops to form into the final steady state shape it will have hence the high fluctuations at low iteration numbers. Once the solution is more and more stable the mass flow rate through the valve curtain stabilizes.

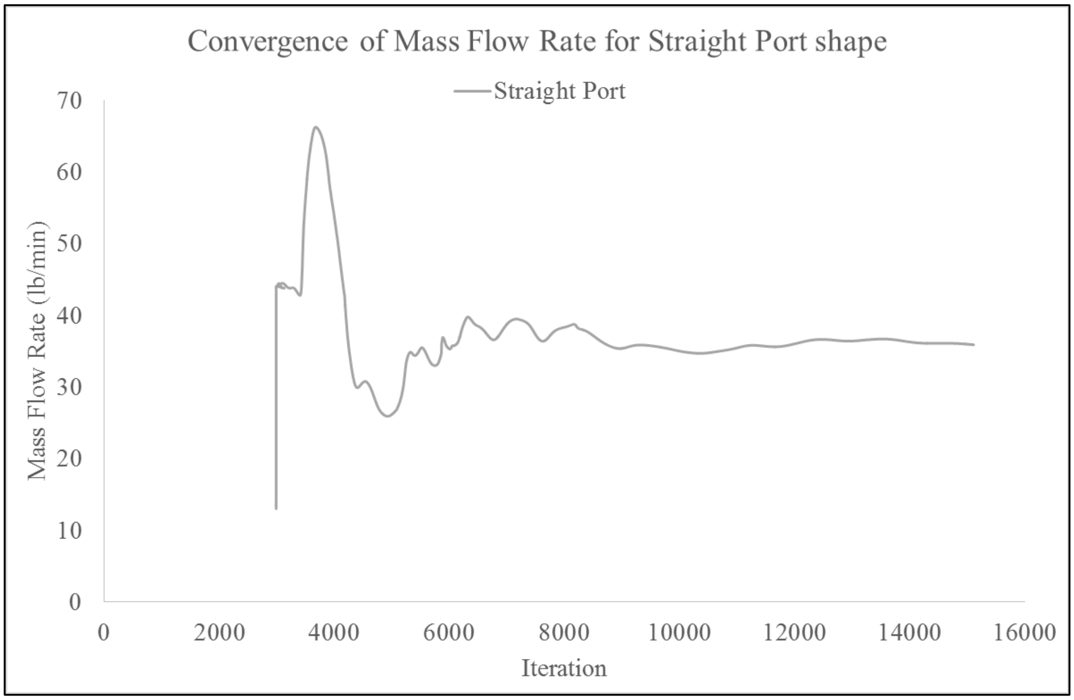

Figure 5-6 - Convergence of the mass flow rate for the straight port

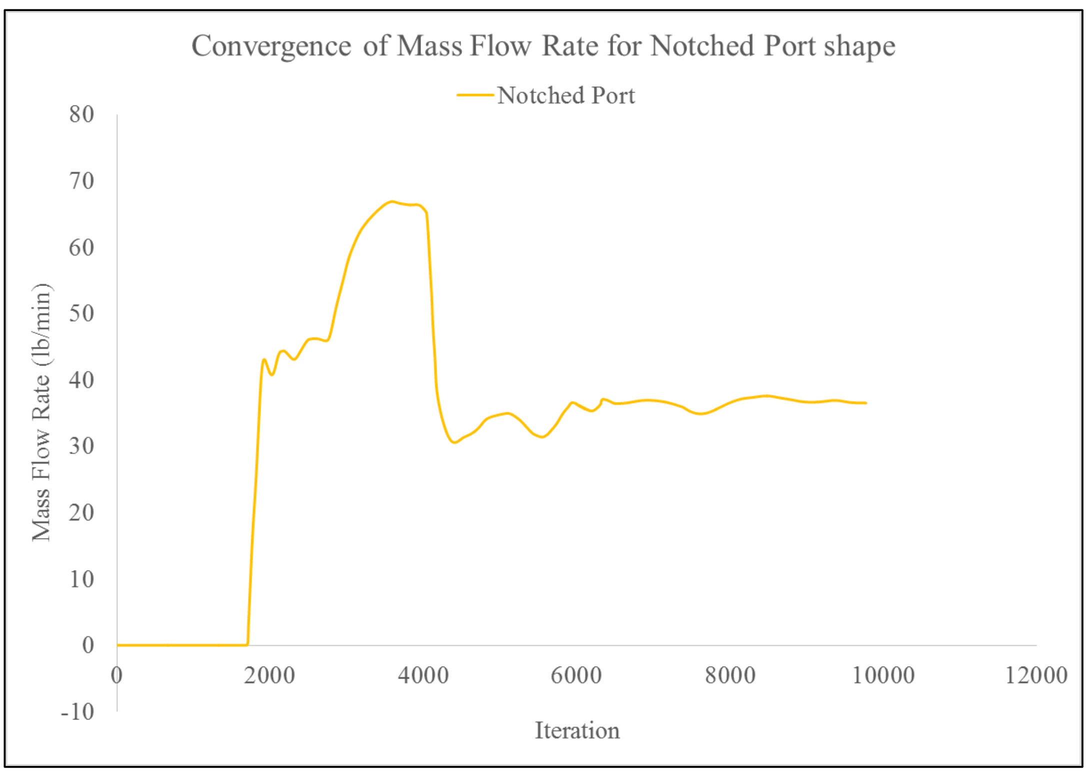

Figure 5-7 - Convergence of the mass flow rate for the notched port

Discussion and Significance of CFD Modeling

CFD modeling was used to try to understand the phenomena discovered through experimental testing. It is clear that to this date this has not been achieved due to numerous reasons. The major reason for this is the limitation in computing power. It seems clear that it is necessary to simulate in three dimensions and in a transient way. The use of a steady state solver is not useful for the engine problem as the mass flow rate changes that are important occur due to transient effects. Doing this requires a great amount of computing power and access to an extended ANSYS Fluent license. The second limitation of CFD is the fact that errors can be large when solving for supersonic flows mostly dependent on the meshing and the way the solvers deal with shock waves. This is still an area of development that needs further investigating.

Using only experimental testing and trying to understand the flow in that manner was possible due to the setup used and the additional comparison of a cylinder head to a sonic nozzle. These steps of comparison of many different geometries experimentally was done to eliminate the need for CFD to understand the mechanisms of a blowdown event. The initial thought was that CFD would allow to visualize what was happening to the flow when different shapes were tested but it became clear that this would be a greater challenge than initially imagined. Experimental tests were designed in parallel to the CFD modeling to reinforce the understanding of blowdown which was not achieved by the modeling. CFD might allow in the future to design the optimal shapes for blowdown, however, the use of a sonic flow bench and porting today is the method that will yield tangible results by focusing on bettering the transient development phase when the valve is initially opened.

Chapter 6 Blowdown Testing of a Sonic Nozzle

Introduction, Setup and Methodology of Testing

In order to supplement the testing of the cylinder heads on the sonic flow bench it was necessary to analyze the blowdown phase for a sonic nozzle. The device is used as a flow meter for sonic flows to regulate mass flow. Sonic nozzles give very accurate results compared to theoretical calculations. The shape of a sonic nozzle is shown in Figure 6-3 and is optimized upstream and downstream of the minimum area to flow consistently within a certain range of pressures. Typically a sonic nozzle is used in a steady state or quasi-steady state condition. This means that it is very good at regulating flow when the pressure upstream is constant or changing slowly. The nozzles are calibrated using constant pressure measurements of flow rate. In a blowdown event the testing of a sonic nozzle will be subject to transient effects. These effects are not taken into account in the design process of the nozzle. This testing will therefore allow to see if there any transient effects that affect the flow. The setup of the apparatus is the same as described in the diagram in Figure 2-5 except the cylinder head is replaced by the sonic nozzle assembly.

The sonic nozzle assembly is shown in Figure 6-1 below. The nozzle is attached to a plate that is bolted to the chamber of the flow bench. It is attached to the plate using adapters and a ball valve is attached above the nozzle and ends the assembly. The system is sealed when the ball valve is closed. The opening of the chamber is done by rapidly opening the ball valve. The nozzle used is a Cox Series 220 model with a diameter of 0.125 inches [15]. The nozzle is pictured in Figure 6-2. No engineering drawing is available for the sonic nozzle from Cox. In order to visualize the shape of a sonic nozzle, another model from Flow Systems Inc. with available drawings is presented in Figure 6-3 [16].

Figure 6-1 - Sonic nozzle test piece set up on the blowdown flow bench

Figure 6-2 - Cox Series 220 sonic nozzle [15]

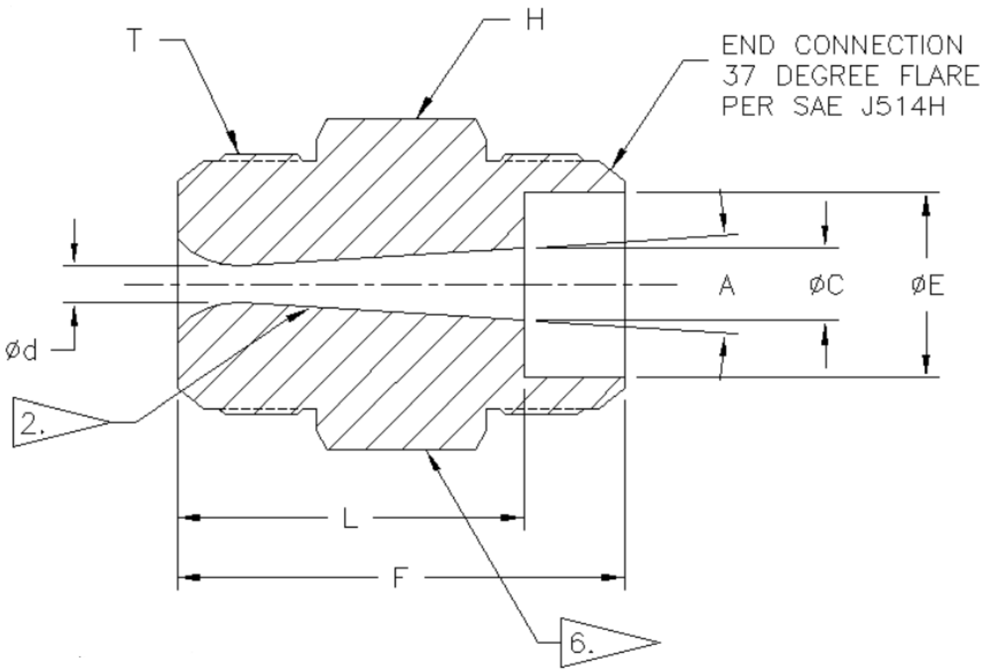

Figure 6-3 - Sonic nozzle drawing provided by Flow Systems Inc. [16]

The nozzle from Flow Systems is very similar in design to the Cox sonic nozzle used for the project. It uses the same exterior threads shown in Figure 6-2 and the overall shape is very similar. The main features of the nozzle are a flow straightening inlet with a radius of curvature before the minimum diameter. A particularity of a nozzle is there is only one place where the minimum area occurs unlike the valve curtain area. The nozzle diverges after the minimum area in a straight line at approximately 3º from the centerline on each side. This ensures the flow stays attached to the wall of the nozzle. Nozzles are typically used with flow straighteners upstream of the minimum area in order to condition the flow to be as uniform as possible when passing through that area. If theory is followed the flow rate should be the same if the flow straightener is used or not when the flow is steady. In transient flow this might be a source of error however is expected to be very small. It was decided to use the nozzle without a straightener as this would mimic the experimental setup with the cylinder head better. In the case of the engine the flow is even more disturbed due to the asymmetries in the geometry. The error for the nozzle provided by the manufacturer is ± 3% compared to the theoretical calculation of the flow rate.

The nozzle was tested in blowdown with a starting pressure that is the same as the cylinder head in order to be able to picture the effects at a realistic cylinder pressure occurring for blowdown. The first test of the nozzle was completed starting at around 135 psi. The blowdown process is a lot slower using the sonic nozzle as the effective flow area is much smaller due to the small diameter used. The flow area is 0.324 in^2 for the cylinder head and 0.0123 in^2 for the sonic nozzle. This is a big difference but should provide a result that will allow to compare both blowdown events qualitatively and using the coefficient of mass flow rate.

Results

Figure 6-4 presents the results of the test compared to the theoretical calculation of mass flow rate against pressure starting at 135 psi. The blue line shows the experimental trace with added 3% error bars accounting for the possible error quoted by the manufacturer. The first look at the graph shows that the blowdown process is similar to the test of the cylinder head. There seems to be two phases throughout the blowdown event: developing flow and fully developed quasi-steady state flow.

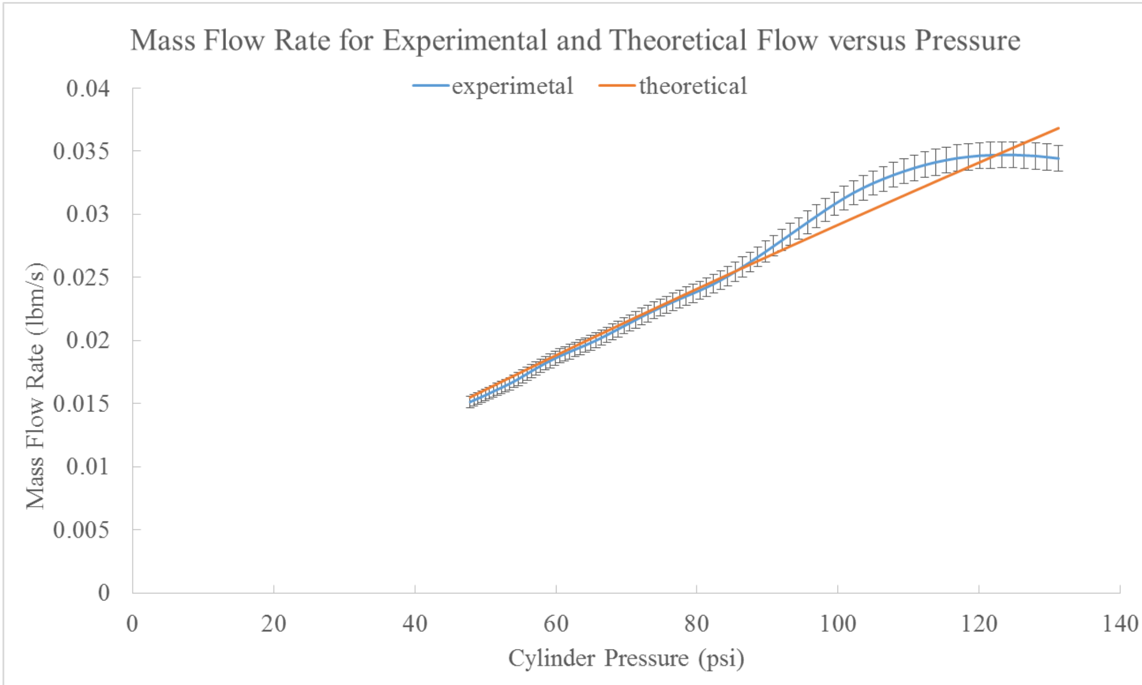

Figure 6-4 - Mass flow rate graph for the sonic nozzle tested at 135 psi starting pressure

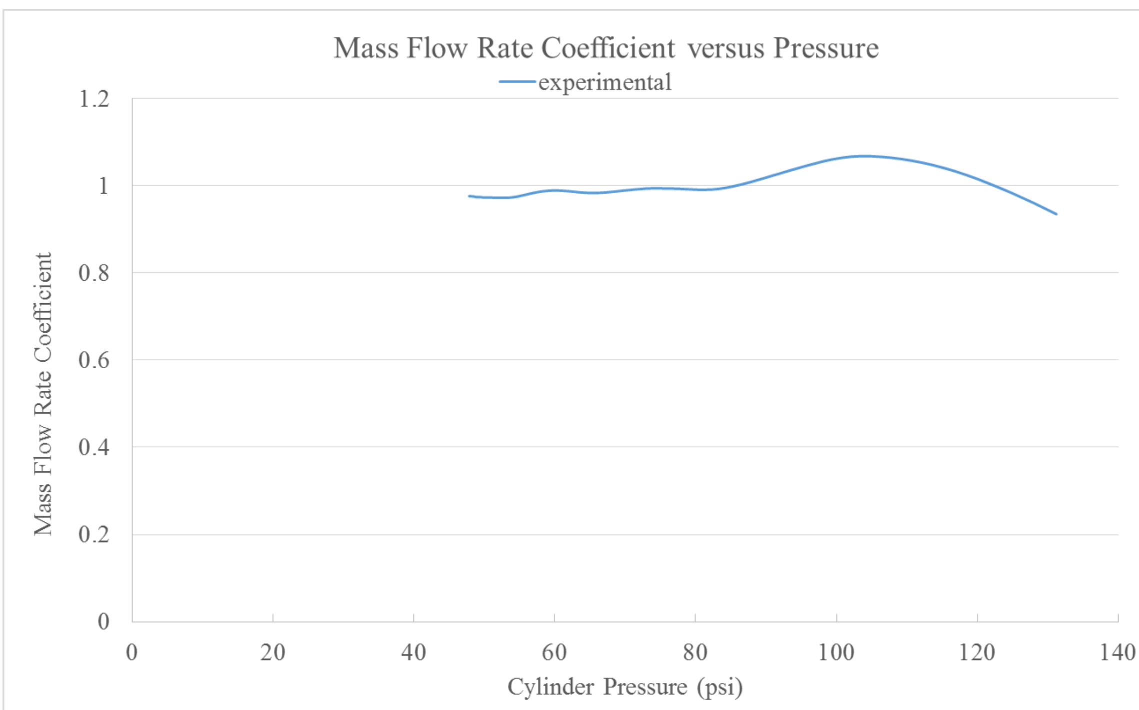

Figure 6-5 - Mass flow rate coefficient graph for the nozzle tested at 135 psi starting pressure

From 130 to 85 psi in the initial phase of blowdown the flow rate does not agree with the theoretical calculation. In a similar way than some of the tests run on the cylinder heads, the flow seems to be higher than theoretical and this is due to the fact that the transition from subsonic to supersonic flow has not stabilized to the minimum area. This allows the flow rate to be above theory and similarly to the cylinder head testing. The sonic nozzle shows there is a significant development phase until the flow has reached quasi-steady state matched to theory. Under 85 psi the data from the testing agrees within the error margin with the theoretical data showing that the sonic nozzle is in fact functioning in the ideal way. This also proves that the blowdown flow bench is an effective tool at measuring flow rate through any orifice as it is essentially calibrated correctly against the nozzle which is widely used as a flow meter.

The length of time for this blowdown event is increased by an order of magnitude and the flow still takes a very large pressure range to settle to the theoretical flow rate. This reinforces the thought that the most important phase of blowdown is actually the part where the flow is still developing. The area where the transition is occurring has to be downstream of the minimum area in this development phase. The larger area where the transition occurs is the only way that the flow rate can be increased above the theoretical maximum. This can occur due to the bowing of the shape of the transition which would increase the effective flow area. This bow shape would push the transition area slightly downstream of the minimum diameter and resorb back to a straight line once the flow reaches quasi-steady state. This then matches the minimum area and the theoretical curve. The transient effects at high pressures can cause the flow to be asymmetric along the axis of the nozzle. This asymmetry is caused by local density and temperature changes within the control volume.

In addition to this, the way the valve is opened using the ball valve is creating an asymmetric release of pressure initially. This can contribute to the fact that the flow is not immediately in line with the theoretical curve. Figure 6-6 shows how the ball valve is opened asymmetrically.

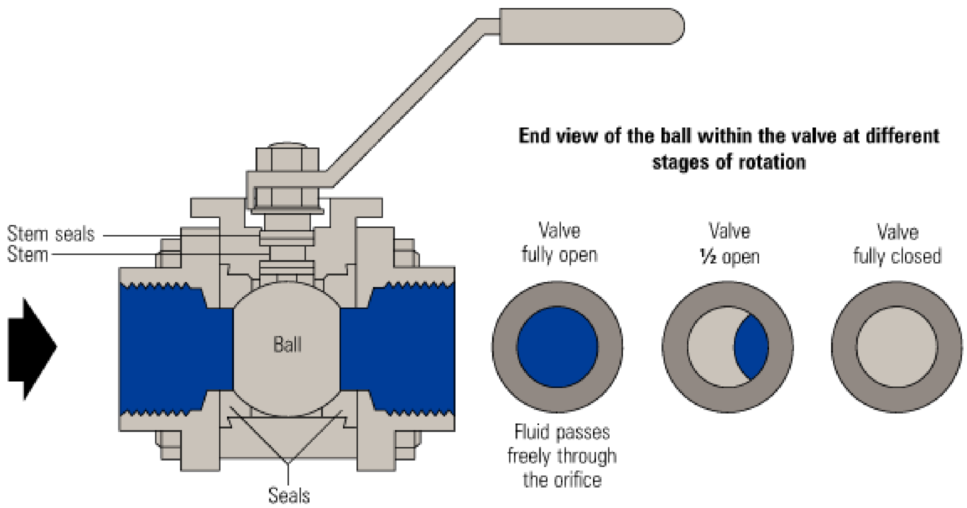

Figure 6-6 - Diagram showing the asymmetric opening process of a ball valve [17]

The testing at this pressure has agreed with how a sonic nozzle should release gas from a chamber but has also shown that transient effects are still present in the initial phase after opening. In order to verify the results at 135 psi starting pressure more tests were run at 160 psi and 80 psi. The test at 160 psi is to verify that the pressure peak in the previous test is a result of the opening and formation of the flow, in which case the peak will be shifted slightly higher in pressures. It is done to verify that the peak is not an anomaly happening between 120 and 90 psi. The results of these tests are presented in Figure 6-7 and Figure 6-8.

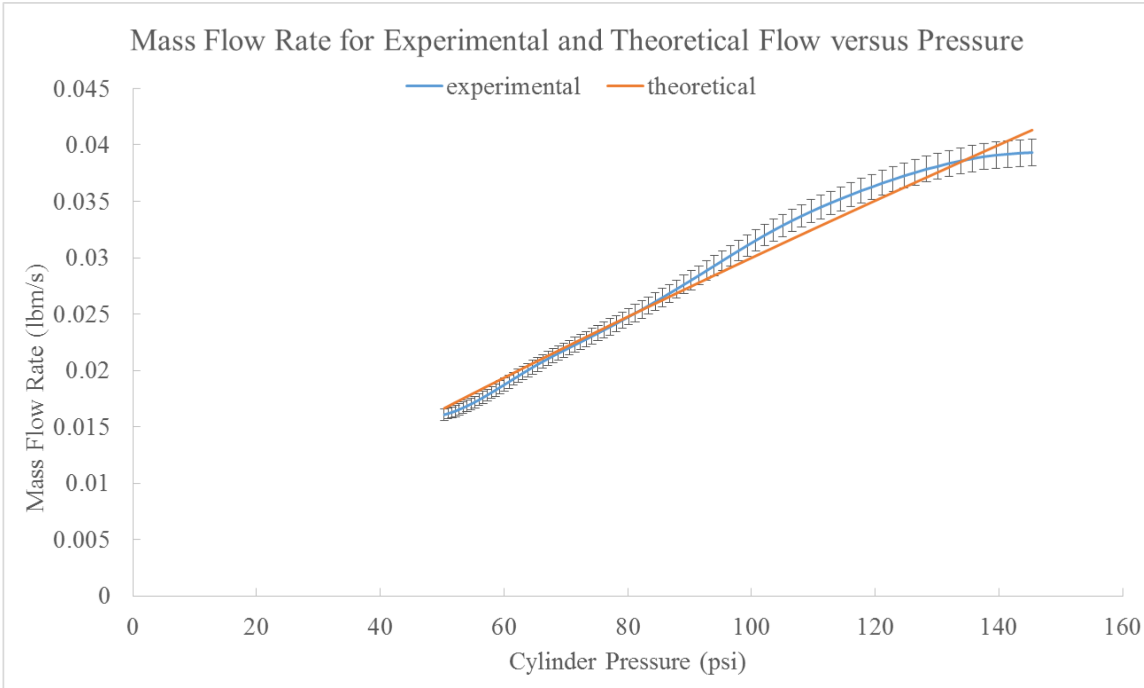

Figure 6-7 - Mass flow rate graph for the sonic nozzle tested at 160 psi starting pressure

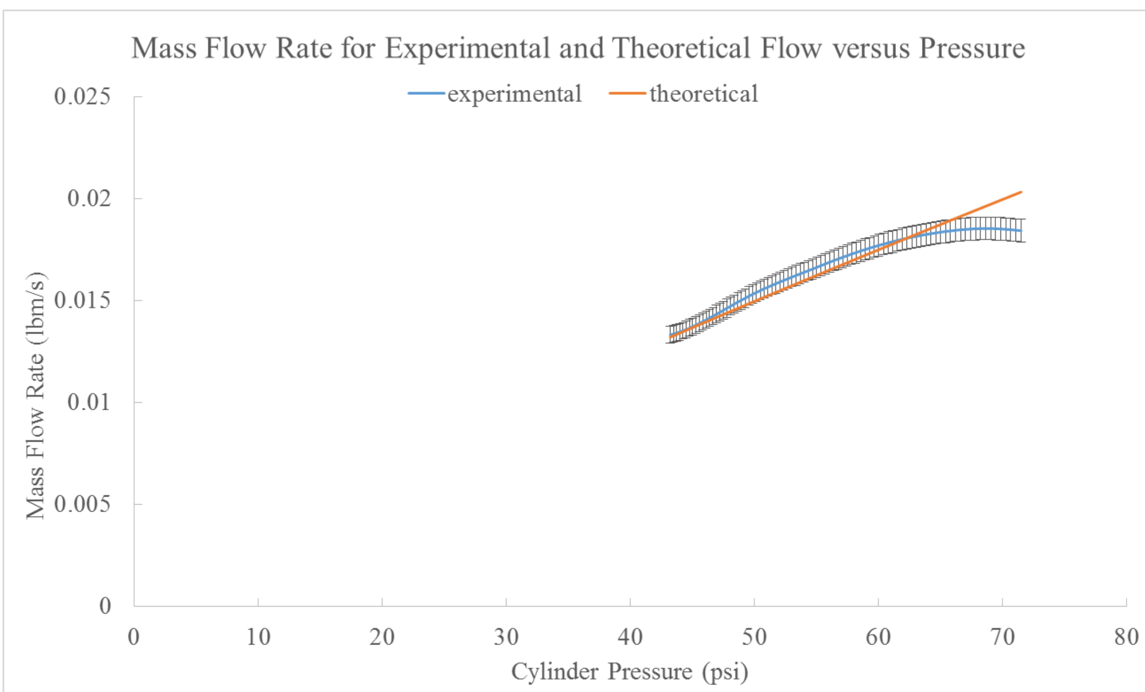

Figure 6-8 - Mass flow rate graph for the sonic nozzle tested at 80 psi starting pressure

Figure 6-7 presents the results for the test done starting at 160 psi. The behavior is very similar to the one seen at 135 psi with a development phase that reaches higher mass flow rates than theoretical and then the once the flow is developed agrees very closely with theory. The initial phase occurs above 85 psi which is the same as the first test done. This means that the development phase was slightly extended by opening the valve at higher pressure. The shape of the curve is very similar to the one seen at 135 psi and eliminates the idea that the flow could be altered in a region between 120 and 90 psi due to the conditions in the flow bench. The difference between both tests is seen in the magnitude of the peak above the theoretical curve. The 160 psi peak was lower than the 135 psi peak.

The 80 psi test is presented in Figure 6-8 and due to the low starting pressure seems to show that the development region is located from 80 to 45 psi. Under 45 psi limit the flow seems to be fully developed. The shape of the curve agrees with the two other tests.

Discussion

The testing of the sonic nozzle provides results that are very important for the overall understanding of blowdown using a sonic flow bench and provides further insight into the transient natures of the flow that can occur when opening a valve. The understanding of overall critical exhaust flow in engines is enhanced by knowing that the sonic nozzle and the cylinder head have a similar two phase type blowdown. This is presented in Figure 6-9 where the modified die cast seat is compared to the nozzle. Due to the time of the event being much longer for the sonic nozzle the opening of the valve has a less of an initial delay in the data reaching high mass flow rate. Both curves do however exhibit the same kind of flow where it only settles once the flow is no longer transient. The test of the nozzle validates the idea that having a flow rate above the theoretical values is possible due to transient effects. The sonic nozzle flow rate once the flow is developed is identical to the theoretical calculation which shows that the transient effects from the flow bench disappear with time in blowdown. This was expected however could not be confirmed by the cylinder head testing as different geometries could change the flow rate when fully developed. The developing phase of the flow shows that the flow rate can go above the theoretical maximum and is independent of time. If the time of the experiment is lengthened due to the area, the development phase is still only dependent on the pressure drop. In the case of the nozzle the flow rate is approximately 30 times smaller than the exhaust ports and also develops slower in time by the same magnitude.

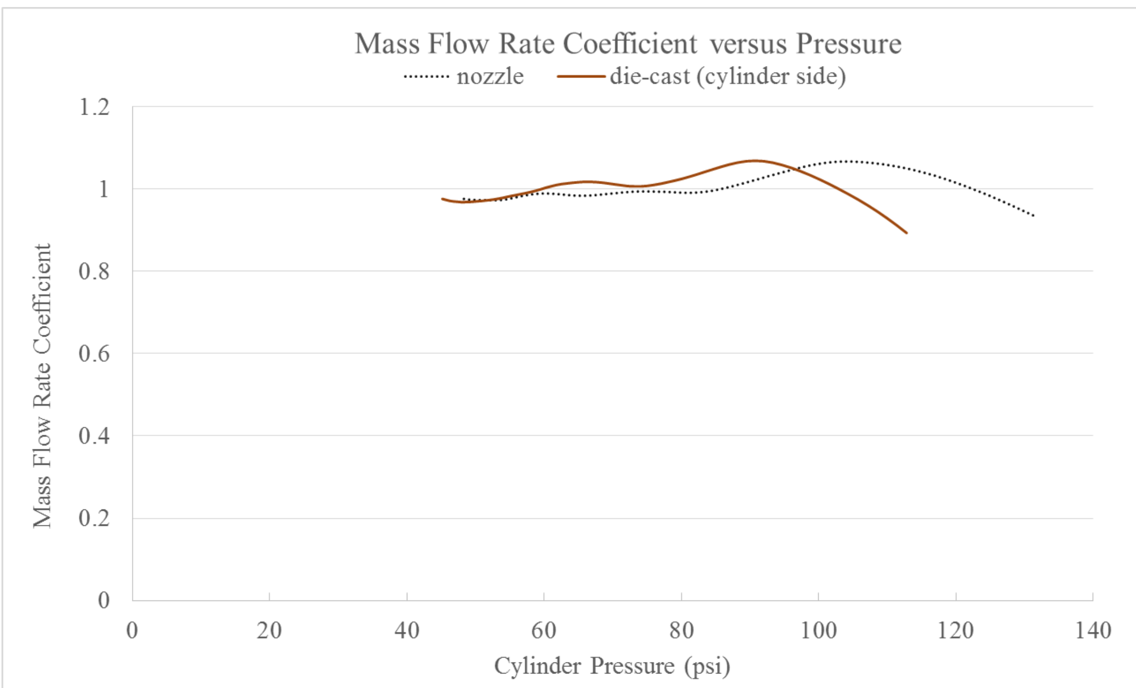

Figure 6-9 - Mass flow rate coefficient for the die-cast cylinder side modification port compared to the sonic nozzle

The results from this test will prove useful in the overall discussion of all the testing completed and will allow to make some clear affirmations on what is important in the design of exhaust ports specifically for the blowdown phase.

Chapter 7 Comparison of Testing Methods and Conclusions

Three different testing methods were used together to understand blowdown flow fully. Blowdown testing using a sonic flow bench was performed on three cylinder heads and a sonic nozzle. Low pressure flow bench testing was also performed on those cylinder heads. CFD modeling of the exhaust port was also attempted to provide some more information on this phase of engine operation. The three different methods allowed to combine what was learned to come to conclusions on how the blowdown phase works and what can be modified to optimize it.

The overall results revealed that the blowdown event could be split into two phases in which the nature of the flow was characterized differently. It was noted from the testing on the blowdown flow bench for the cylinder heads and the nozzle that the flow region right after valve opening is highly transient and flow is still developing. This is known as the development phase of blowdown where the mass flow rate starts from zero and eventually stabilizes to the theoretical value. This zone is very responsive to changes in geometry in the flow region. Some changes increased the overall mass flow rate by more than 10% overall. This phase is characterized by the flow accelerating and not reaching an equilibrium very fast with a potential to reach higher mass flow rates than what the maximum theoretical value is. This means that the area at which the flow transitioned from subsonic to supersonic was at a different location than the minimum area because of the highly transient nature of the flow, the asymmetrical development of the flow and the viscous forces on the bounding surfaces. Testing the sonic nozzle was very beneficial as it confirmed the existence of an initial transient phase where the flow rate could surpass the theoretical value due to the transition moving around the minimum area.

Once this highly transient phase was passed the flow developed into a second stage that can be called the fully developed region or the quasi-steady state region. In this region, the sonic nozzle behaved as expected by matching the theoretical values for mass flow rate at the minimum area. This is the case because the sonic nozzle is specifically designed for this to occur and the shapes upstream and downstream allow for the transition to occur right at the minimum area and not slightly downstream as can be the case due to inertial forces. This phase on the cylinder heads yielded results that allowed for improvements to be made in order to better this flow rate. The stock die-cast and sand-core ports were underperforming compared to the theoretical values and this could be altered by making downstream modifications in the port. Although this is an important phase of blowdown to understand it is of much less importance than the first phase. This is because of the fact that the engine blowdown event occurs much faster than the one recorded with this experimental setup and the valve is constantly moving. This means that the flow will never reach a fully developed state and will be transient during the real engine blowdown process. It follows that the goal is to maximize the transient phase of blowdown on a flow bench to extract the most performance out of the exhaust port.

These two separate regions of flow really explain why blowdown assumptions made by engine designers are wrong and geometry does really have an effect on the engine performance. A testing rig such as the blowdown flow bench allows to record this transient behavior and better it by making modifications to the engine. From port to port the solution for improving this transient might be found in different ways thus why it seems that the sonic flow bench is a necessary iterative tool for design.

Another aspect that affects blowdown significantly is the shape of the valve and valve seat. This has a large influence on performance due to the changes that can be done to minimum flow area and the shape. The results from testing valves and seat at different angles and with different design ideas allowed to conclude that a shape that resembles a venturi works best at low pressures however the opposite is needed for good blowdown. Abrupt transitions in shape promote the transient phase to last longer and go above the theoretical maximum. The best combination of blowdown performance and low pressure performance is provided by the 45º(30º) port tested. This will require extensive engine testing to validate.

Due to the limitations discussed in Chapter 5, CFD modeling did not provide any valuable insight into geometry changes and their effects on overall flow rate. The fact that an axisymmetric model had to be run greatly hindered the chances of capturing changes in flow rates due to minor changes in the downstream shape. With the correct computing power necessary to undertake a simulation of this complexity some insight could have been gained into why certain shapes allow for the transient development region to develop faster than others. This might be possible in the future however the time cost makes the use of a flow bench as a tool for design much more interesting currently. The errors in CFD are also usually very high due to modeling inaccuracies and small modifications to the cylinder heads may not be captured.

More investigation into the effects of the results of the positive changes made on the blowdown flow bench on an actual engine is the next step this project can take but is outside of the scope of the goals set forth. Correlating a performance increase on the flow bench with the gains on an engine is what a designer of an engine needs as a further tool for design.

Improving the average mass flow rate during an exhaust event allows for a shorter duration to empty the cylinder of gases. This means that the improvements found using the sonic flow bench will directly impact the duration of the exhaust event on the engine. By making the exhaust event shorter and more efficient, it is possible to retard the opening of the exhaust valve which allows an increase in the expansion stroke. Increasing it allows for the combusted gases to push the piston down for a longer time which increases the power output of the engine. This shows the importance of increasing the flow rate out of the cylinder using the techniques developed in this study.

REFERENCES

[1] J. D. Decker, "Critical and Steady-Flow Analysis of a High Performance Automotive Exhaust Port," University of Miami, Miami, 2013.

[2] J. B. Heywood, "Engine Operating Cycles," in *Internal Combustion Engine Fundamentals*, New York, McGraw Hill, Inc., 1988, p. 10.

[3] F. White, Fluid Mechanics, 5th edition, New York, NY, USA: McGraw Hill, 2003.

[4] R. Stone, Introduction to Internal Combustion Engines, 3rd edition, Warrendale, PA, USA: SAE, 1999.

[5] B. Massey, "Compressible Flow of Gasses," in *Mechanics of Fluids*, New York, Taylor & Francis, 2006, p. 523.

[6] R. Cunningham, "Orifice Meters with Supercritical Compressible Flow," in *Transactions of ASME*, 1951.

[7] B. Massey, "Other Losses in Pipes," in *Mechanics of Fluids*, New York, Taylor and Francis, 2006, p. 263.

[8] M. Swain, P. Filoso and E. Grilliot, "Curse of The Short-Turn Radius," *Hot Rod Magazine,* pp. 67-71, September 2000.

[9] J. B. Heywood, "Poppet Valve Geometry and Timing," in *Internal Combustion Engine Fundamentals*, New York, McGraw Hill, Inc., 1988, p. 220.

[10] B. Launder and B. Sharma, "Application of the Energy Dissipation Model of Turbulence to the Calculation Near a Spinning Disk," *Lett. Heat Mass Transfer,* vol. 1, pp. 131-138, 1974.

[11] D. Wilcox, "Reassessment of the Scale-Determining for Advanced Turbulence Models," *AIAA,* no. 26, pp. 1299-1310, 1988.

[12] K. Hanjalic, "Closure Models for Incompressible Turbulent Flows," in *Introduction to Turbulence Modelling*, Belgium, Von Karman Institute for Fluid Dynamics, 2004, pp. 1-75.

[13] T.-H. Shih, W. Liou, A. Shabbir, Z. Yang and J. Zhu, "A New k-e Eddy Viscosity Model for High Reynolds Number Turbulent Flows," *Comput. Fluids,* no. 24, pp. 227-238, 1995.

[14] F. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal,* no. 32, pp. 1598-1605, 1994.

[15] Cox Turbine Flow Meters, "Cox Sonic Nozzles," November 2015. [Online]. Available: https://www.badgermeter.com/business-lines/flow-instrumentation/sonic-nozzles/.

[16] Flow Systems, "Sonic Nozzle Schematics & Performance Charts," 1 March 2002. [Online]. Available: http://www.flowsystemsinc.com/sonic-nozzle-schematics-performance-charts.

[17] FCE Technical Services, "Ball Valves," 2016. [Online]. Available: http://www.fcets.co.za/BallValves.html.

Appendix A Number of data collections needed for statistical significance

Using the data from the blowdown testing done on the die-cast and sand-core cylinder heads with and without the downstream tube, a statistical analysis was undertaken to understand the data collected as well as determine the statistically significant number of data traces needed per cylinder head test.

The goal was to understand how many data traces need to be retrieved from an experiment to provide statistical significance. In other words, how many data points are necessary to have repeatable data?

The assumption in this experiment is that 15 data traces would be enough to provide a large enough sample size to be representative of more data. From this assumption, ANOVA testing was used to determine the minimum number of data points necessary in order to achieve the same curve within a 95% confidence interval.

In order to do this, 15 data points of one set of data will be used and reduced down to 2 data points by increments of 1. This means the ANOVA test will provide (at a specific time on the data curve) a comparison between the mean of 15 data points and 14,13,12,11,10,9,8,7,6,5,4,3,2 data points until it shows the means are different. The number of points one larger than where the mean becomes different is the number of minimum data traces needed for statistical significance.

3 different methods can be used to eliminate data points to reduce the set from 15 to 2 incrementally:

- Taking out the median one by one,

- Taking out the maximum and minimum (one each),
- Taking out only the maximums or only the minimums.

The first two ways were tested on 2 different data points of the time plot shown below.

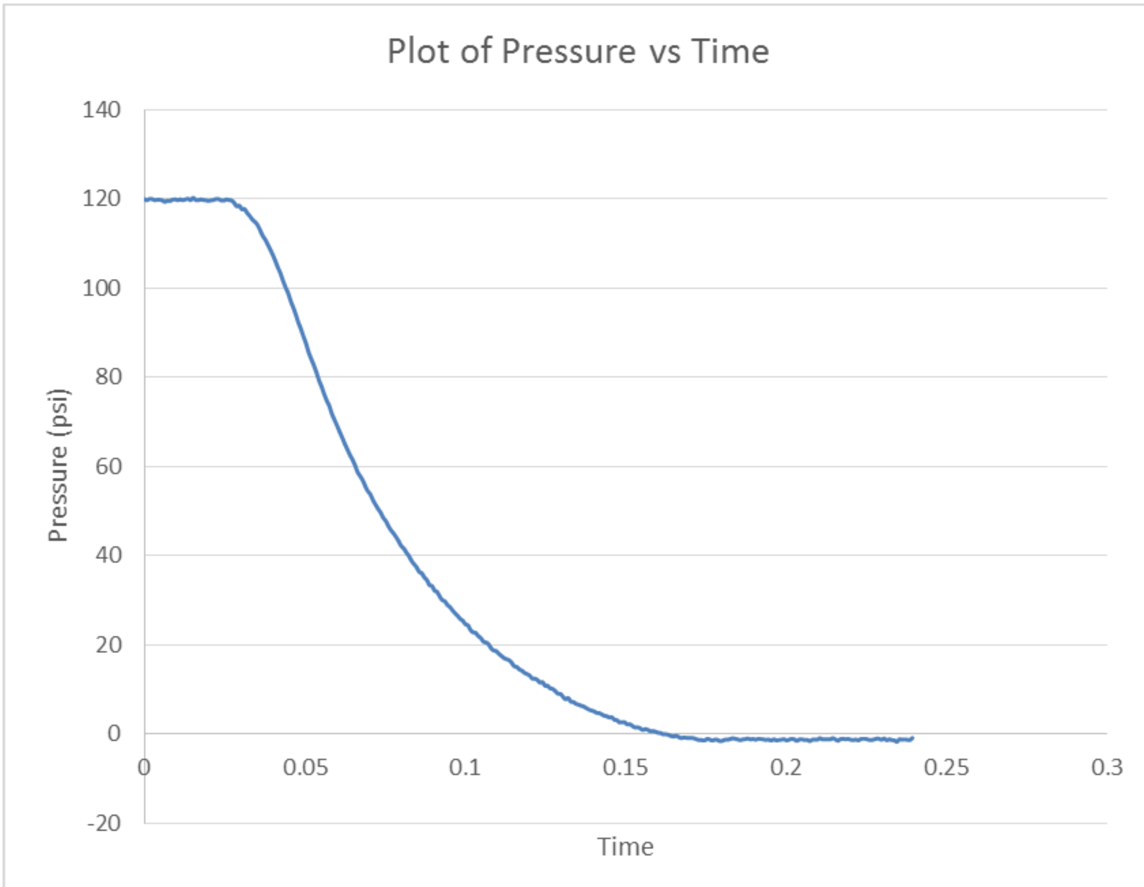

Figure A-1 – Raw data plot of pressure versus time for a blowdown event on a die-cast port

Reducing the data from 15 to 2 points (2 time points) using the median elimination or the max/min elimination method shows that 2 data points are sufficient to achieve statistical similarity. More points could be investigated, however it is deemed that a more one sided method should be used. It would be useful to cut off all the maximums or all the minimums as this would mimic picking out the most extreme data points in the set. This gives a better metric to show how many points are needed to have similar data to 15 points.

Five different time points were tested with minimum and maximum values being truncated from 15 to 2 data points and the ANOVA test provided a response for what number of data points is correct.

Table A-1

Data points (time)	Number of data points needed (MAX values truncated)	Number of data points needed (MIN values truncated)
t=0.0376s	4	5
t=0.0504s	6	6
t=0.08s	5	3
t=0.1s	7	2
t=0.124s	3	5

The minimum number of data points needed for similarity is 7 data points. The data set at t=0.1s did not behave like a normal distribution with 7 data points skewed to the minimum. This caused the ANOVA to have to pick 7 points as a minimum for statistical similarity with 15 data points.

Appendix B Nash Sutcliffe analysis on two heads

Both heads were tested with and without a downstream tube added to the end of the exhaust port. The investigation determines if there is a statistically significant difference in rate of pressure drop between having a downstream tube or not. The Nash-Sutcliffe method was used to determine the difference between two average data curves for testing. The Nash-Sutcliffe equation is shown below:

$$NS = 1 - \frac{\sum_0^t (A-B)^2}{\sum_0^t (A-A_{mean})^2} \text{............ (B1)}$$

Where A is the pressure of data set A, where B is the pressure of data set B and where A_{mean} is the mean pressure of one data set.

If two identical data sets were used for A and B the value of NS would be 1. Thus the closer to 1 the more likely the two data sets are the same. Looking back at Figure 25, the recorded data includes points when the valve has not been opened yet and points when the pressure stays constant at the end. To make the Nash-Sutcliffe work better, it has been decided to cut off the two ends of the graph.

The sand-core and die-cast ports were tested and the NS values are shown in the following table:

Table B-1

Port	NS value
Sand-core	0.999646
Die-cast	0.999843

The values are very close to 1 and this is because A_{mean} is far from the A points at the extremities of the data set. The Nash-Sutcliffe method indicated that the die-cast port seems to show similarity in data more than the sand-core. This is confirmed by inspecting the graph of the curves. Two modifications to the Nash-Sutcliffe method were investigated to use a larger range between 0 and 1 to visualize the difference between the two sets.

The first modification made is to change the weight of the bottom term of the equation. This term is used to normalize the variance between two samples.

$$NS = 1 - \frac{\sum_0^t (A-B)^2}{\left(\frac{\sum_0^t (A-A_{mean})^2}{1000}\right)} \text{............ (B2)}$$

The new results increase the working range of NS and allow a statistical difference to be seen between the test with the downstream tube or without. The problem with this method is it is difficult to evaluate if a change has been made or not to the flow.

Table B-2

Port	NS value
Sand-core	0.646493
Die-cast	0.842809

The other method that is used to have a better understanding of the data is separating the curve in different sections. This shows where the differences occur and will detail if two data sets are in fact statistically different or the same. This is deemed the most complete and comprehensive method.

The curves were split into 6 sections in order to get a NS-value for each segment in increments of 20 psi.

Table B-3

Pressure range (psi)	Sand-core Port	Die-cast Port
120 – 100	0.996588	0.995343
100 – 80	0.946994	0.995849
80 – 60	0.93669	0.996772
60 – 40	0.959827	0.995551
40 – 20	0.991932	0.994269
20 – 0	0.997927	0.990645

The sand-core port clearly shows a difference between having the downstream tube or not. It seems that this last method is the best at determining if there is a difference in pressure. The die-cast port behaves in the same way along the entire path of the curve showing that having the tube does not affect the flow. This last method agrees with the weighting method shown in Table B-2.

The section method will be used for all testing to check if changes occur in flow. The magnitude of the change and the location of the change can be easily explained using this method.