Faculty of Mechanical Engineering

PREDICTIVE STUDY OF FLUID FLOW THROUGH AN INTAKE SYSTEM RESTRICTOR OF A SINGLE-SEATER RACING CAR

Ahmed Z. I. Zaqout

Master of Mechanical Engineering (Energy Engineering)

2018

DECLARATION

I declare that this thesis entitled "Predictive Study of Fluid Flow Through an Intake System Restrictor of a Single-Seater Racing Car" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature	:	
Name	:	Ahmed Z. I. Zaqout
Date	:	

APPROVAL

I hereby declare that I have read this dissertation/report and in my opinion this dissertation/report is sufficient in terms of scope and quality as a partial fulfillment of Master of Mechanical Engineering (Energy Engineering).

Signature	:
Supervisor Name	: Dr. Ahmad Kamal Bin Mat Yamin
Date	:

DEDICATION

To

My Mother And Father

Who taught me to trust in Allah, believe that hard work always pays off and for supporting and encourging me to believe in myself

My Brothers

For being my guardians during my educational Journey

Along with all hard working and respected
Teachers

ABSTRACT

The society of Automotive Engineers conducts an annual student design competition referred to as Formula SAE (FSAE). Teams are limited to a 610 cc engine that is restricted by a 20 mm air restrictor. The restrictor limits the power output by reducing mass flow rate flowing to the engine. Thus, the aim of this project is to design and predict the flow behaviour of a flow restriction device to be installed in the single seater race car. The design consists of a converging part (nozzle) and a diverging part (diffuser) with the 20mm throat mandated by the FSAE rules being in the middle. The nozzle design was based on the work of T. Morel on axisymmetric wind tunnel contractions as a way to explore different aspects of the restrictor geometry criterion. While the diffuser geometry was obtained from previous optimizations. The simulation was carried out with the academic version of Ansys 17.2 Workbench which was conducted in 2D-axisymmetric geometry with isothermal process. Nozzle design comparison between contraction ratios of 16 and 9 were performed. The simulation applied the local mesh with appropriate controls in order to obtain correct predictions of the flow behaviour near the walls. Velocity and pressure contours, velocity vectors and surface monitors showed that the novel design with contraction ratio 16 displays superior pressure recovery characteristics than that of contraction ratio 9 with pressure difference of 12610 Pa and 15758 Pa respectively. However, there is always more room for improvements considering the design is in the development stage. Furthermore, based on the design criterion of the restrictor, separations are avoided completely for both designs.

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my supervisor Associate Professor Dr. Ahmad Kamal Bin Mat Yamin from the Faculty of Mechanical Engineering Universiti Teknikal Malaysia Melaka (UTeM) for his essential supervision, support and encouragement towards the completion of this thesis.

Special thanks to all my peers, my mother, beloved father and siblings for their moral support in completing this degree. Lastly, thank you to everyone who had been to the crucial parts of realization of this project.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF FIGURES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.0 Background

The performance and emissions characteristics of a spark-ignition (SI) engine are improved from the perspective of the intake system when a fuel-air mixture is efficiently delivered and uniformly distributed to each of the combustion chambers over an RPM band. This involves designing an effective intake system.

The society of Automotive Engineers conducts an annual student design competition referred to as Formula SAE (FSAE). Universities from all around the world strive as design teams to produce a wining automotive race car. To encourage creativity, the design teams are limited to a 610 cc engine that is restricted by a 20 mm air restrictor. The restrictor limits the power output of the by reducing mass flow rate flowing to the engine. Thus, a restrictor should be efficiently designed to allow maximum possible air flow and maintain minimum pressure difference across the restrictor.

The typical 600 cc four-cylinder four-stroke Formula SAE engine has an output of about 80 horsepower with an intake restrictor installed and is fuel injected. Air travels through a throttle body and intake plenum before reaching individual runners that feed each cylinder. Figure 1.1. An optimized intake system will net the engine more airflow and more power; however, designing, machining, and testing several intake systems can be costly. The design approach is essentially a process of trial and error.

Computational fluid dynamics (CFD) flow modelling software such as ANSYS offers an alternative to the experimental method of design. CFD is based on computer simulation, where the governing equations of fluid motion (Navier-Stokes equation) are solved numerically. CFD allows designers to simulate a variety of intake profiles and flow conditions without having to machine multiple prototypes for practical flow testing. The cost savings found through CFD can be substantial, as the manufacturing cost of a single intake system restrictor, after material and machining time, can be overpriced.

Over the years of the Formula SAE project, the air intake system had seen many improvements, pushing the engine to produce increasing amounts of power. However, there is always room for innovation. Unlike previous researchers, the venturi-type nozzle design will be replaced by a nozzle with a wall contour constructed of two matched cubic arcs.

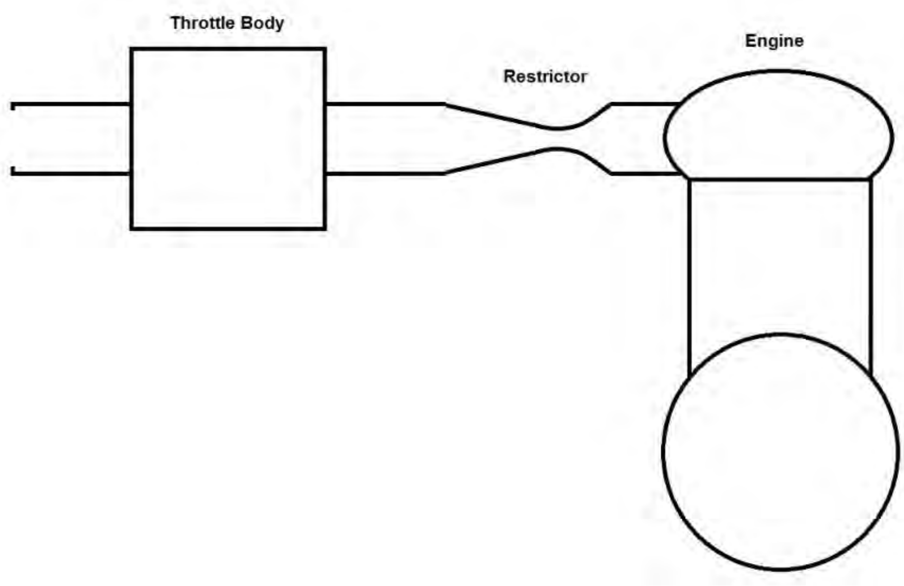

Figure 1.1 Intake System Restrictor

1.1 Problem Statement

An engine requires proper air-fuel ratio to work as per its design. Due to 20 mm restrictor rule, the designed intake reduces to 20 mm for all four cylinders. This extreme modification reduces flow of air mass to the engine. When engine is running at low rpm of about 3000 rpm this required mass flow rate is compensated by increase in velocity of air through the nozzle. At such high speeds engine require much more air for combustion and thus mass flow rate should increase, but due to restrictor area being less air must pass with very high velocity to compensate or fill the engine with required amount of air. Thus, air tries to achieve maximum velocity through restrictor which gives rise to critical flow conditions. Thus, mass flow rate is a fixed parameter for 20 mm restrictor, which is used for calculations further for optimization of the nozzle.

1.2 Statement of Purpose

The purpose of this research is to simulate the fluid flow through an air restrictor consisting of a nozzle-diffuser design. The restrictor will be designed to give satisfactory pressure recovery and diminsh flow separations along the diffuser. In addition, The simulation will apply the local mesh with appropriate controls in order to obtain correct predictions of the flow behaviour near the walls. Velocity and pressure contours, velocity vectors and surface monitors will be demonstrated and discussed comparatively.

CHAPTER 2

LITERATURE REVIEW

2.0 Background

The rules of the FSAE competition confines the engine's power with a component designated as ''restrictor'' which is a 20mm diameter opening. all the air entering the engine must pass through this restrictor.

The air intake system plays a critical role on the performance of the engine. It is known that increasing the amount of air drawn into the combustion chamber increases the volumetric efficiency with the consideration of the range of RPMs.

With a restrictor placed early in the air intake system, engine performance is being significantly compromised, as it is proportional to the volumetric efficiency of the engine system.

Consequently, the air restrictor design must maximize the airflow that can be passed through it for the cylinders to take in as much air as possible.

2.1 Restrictor Geometry

The shape of a restrictor can be as simple as a plate with the 20mm hole machined into it, and placed anywhere along the air intake system. But, a lot of pressure will be lost downstream of the restrictor, if a simple orifice plate is placed. and the end influence is a reduced efficiency along the line of airflow.

In observing the flow of air through an orifice plate using a CFD tool, there is a point in the air stream where the diameter of the stream is the least, and fluid velocity is at its maximum, known as a *vena contracta*. Figure 2.1.

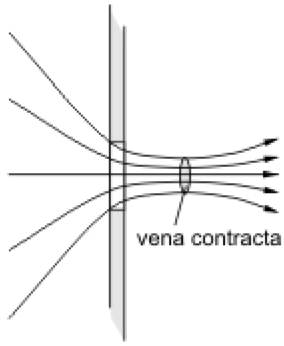

Figure 2.1 Vena Contracta

This convergence leads to a restriction of air flow that is smaller than the orifice plate opening itself. Therefore, there is a need to guide the air into the restrictor, instead of creating a sudden step that would have been in the case of the orifice plate. Thus, came the common design of a Convergent-Divergent Nozzle that replaces the orifice plate as an air intake restrictor.

The CD Nozzle is a tube which, on one end, is exposed to the environmental atmosphere, tapers into the mandated restriction diameter, and then tapers out into the manifold chamber, and in doing so seeks to reduce the pressure loss across its length as much as possible.

In numerous researches and literature, the recommended shape for the convergent part of the restrictor is an elliptical curve heading to the minimum diameter point, while a 3° to 7° taper on the divergent side of the restrictor would allow the air to recover the pressure lost as air flows into the contraction.

2.2 Simulation Research

A paper by Pranav Anil Shinde [1] regarding the optimization of a FSAE car engine intake air restrictor compared two options to restrict air flow using a 20mm diameter constriction and these are both a simple orifice and a converging diverging nozzle.

Table 2.2a: Differences between Orifice and C-D nozzle

Parameters	Orifice	Nozzle
Coefficient of Discharge	0.06	0.975
Pressure Loss	Medium	Low
Viscosity Effect	High	High
Accuracy (% of full scale)	3	1
Cost	low	Medium
Manufacturing	Easy	Difficult

The comparison between these two restrictions, judging by the efficiency it was found that the converging diverging nozzle is the optimal choice. Consequently, the C-D nozzle with a throat diameter of 20 mm mandated by the competition was used in conjunction of a 38mm inlet and outlet diameters based on the widely used throttle body diameter.

For boundary conditions at outlet of venturi we can have either pressure, velocity or mass flow rate. Calculating pressure and velocity at outlet of venturi involves complex procedures and thus gives rise to some errors. Mass flow rate at outlet can be easily calculated by using choked flow equation. The mass flow rate was calculated to be 0.0703 assuming no losses in friction and turbulence. This result is supported with several papers previously made. [1]

Ansys Fluent and Solidworks Softwares were simultaneously used to test the result of the CFD analysis. Since the parameters of the C-D nozzle were established, dimensions of venturi which will provide minimum pressure drop across the venture was investigated by assuming some dimensions of diverging and converging angles via basic knowledge of functioning of venturi. CAD modeling was done using Solidworks 2014 and then analyzed in Flow Simulation for following boundary conditions:

Inlet: Total Pressure = 101325 Pa

Outlet: Mass flow rate = 0.0703 kg/s

Iterations carried out on converging and diverging angles are as indicated in below Table 2.2b with pressure difference:

Table 2.2b: Iterations on C-D angles with pressure difference results

Iteration No	**Converging angle (degrees)**	**Diverging Angle (degrees)**	**Pressure Difference (Pa)**
1	12	6	8560.24
2	14	6	9161.78
3	16	6	9256.88
4	18	6	10009.65

Based on the research objective of Pranav Anil Shinde [1], maximum pressure recovery is achieved at converging angle of 12 degrees and diverging angle of 6 degrees. Diverging cone angle was set to 6 degrees as it was found that any increase or decrease in angle caused streamline disturbance and drop in pressure at downstream side.

The work of Anshul Singhal, Mallika Parveen [2], which is similar objective wise to [1], also agrees that the best general design to achieve maximum possible mass flow rate, minimize the pressure loss through the flow restriction device is to use the Venturi design. From the data gathered through the numerous simulations, they found that the values for converging angle and diverging angle of the Venturi are 18 degrees and 6 degrees respectively, which is in turn supports the result obtained on the diffuser 6 degrees half angle.

Mark Claywell and Donald Horkheimer [3] paper studied the effects of different diffuser geometries and plenum dimensions using WAVE, and then a sequence of different diffuser angles were simulated using WAVE-VECTIS. The restrictors in the simulations utilized 7°, 5.5°, 4°, and 3° diffuser half-angles. These diffuser half-angles were investigated in WAVE using the Conical-Spline Intake figure 2.2a. Several geometry configurations were tested, including changing diffuser half-angle while using the same plenum. The restrictors were then inspected by looking at items such as volumetric efficiency along the diffuser. The paper consisted of several cases, however, the interest is on two cases only.

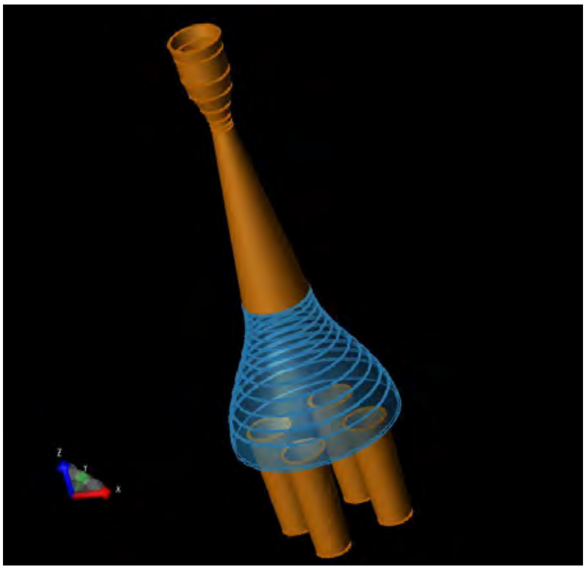

Figure 2.2a: Conical-Spline Intake – Basic Simulation Model

In Case 1, the exact same plenum was used for all the diffuser half-angles simulated. Since same plenum was used, and all the diffusers must couple to the same plenum, this made the diffuser outlet diameter a fixed dimension.

One geometric effect of different diffuser half-angles is as the diffuser angle is increased from 3° to 7°, the length of the diffuser decreases. The plenum volume and plenum length are held constant. Both the total length and total volume after the throat changes for each restrictor angle, but the change is caused through using different diffuser angles. The relative dimensions are noted in Table 2.2c below.

Table 2.2c: Dimensions for intake restrictor case 1

Diffuser half angle	3°	4°	5.5°	7°
Diffuser exit diameter (mm)	72.90	72.90	72.90	72.90
Diffuser length (mm)	504.7	378.2	274.7	215.4
Diffuser volume (L)	0.95	0.71	0.52	0.40
Plenum volume (L)	2.26	2.26	2.26	2.26
Plenum length (mm)	160.5	160.5	160.5	160.5
Total length from throat to plenum bottom (mm)	665.2	538.7	435.2	375.9
Total volume after throat (L)	3.21	2.97	2.78	2.66

The simulation results in Figure 2.2b, shows the change in Volumetric Efficiency as the diffuser half angle is modified, using the same plenum. There are significant shifts in the VE curve across the rev range of the engine. As the RPM increases, the VE profiles tend to come together.

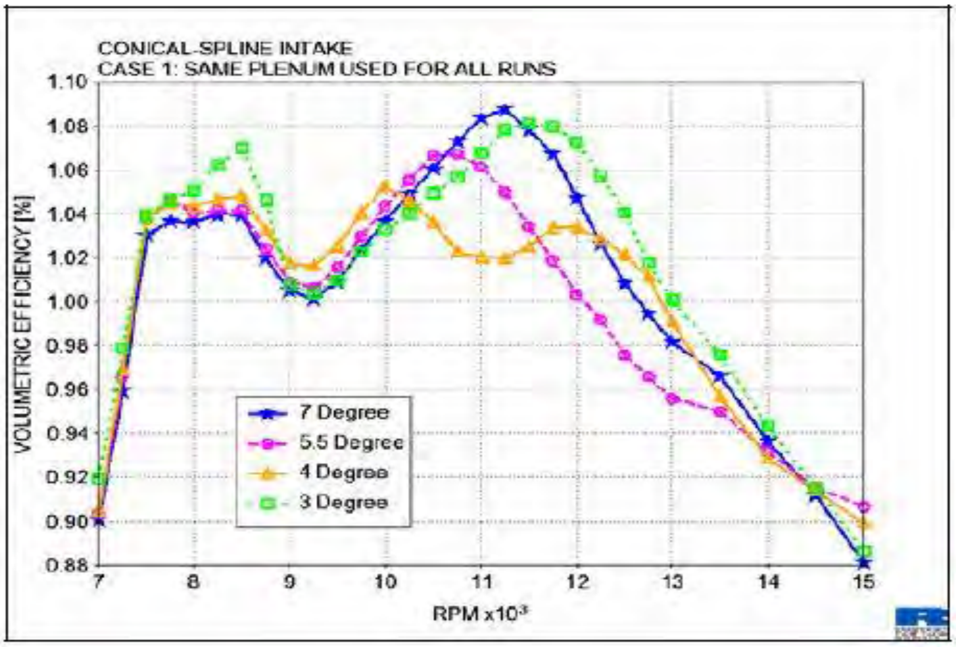

Figure 2.2b: Volumetric Efficiency for Various Diffuser Half-Angles with Constant Exit Diameter

The 5.5° restrictor has more volume than the 7°, despite that, the 5.5° shows a much lower VE from 11,000 to 14,000 RPM. The 4° restrictor also exhibits a similar behaviour, it has lower VE than the 7° restrictor from 12,250 to 13,000 RPM. This points that there is more to the determination of a VE curve than plenum volume or the total volume after the throat. In addition, the fact that the length is changing with different half angles is not negligible considering the VE profile.

In case 2, the length of the diffuser was not allowed to change, set to 216.4 mm from the throat to the diffuser exit. The plenum length was also fixed to the same length as in Case 1, at 160.5 mm. In Case 2, the diffuser exit diameter was determined by the diffuser angle and the pre-set diffuser length of 216.5 mm. As the exit diameter changed for each case, the plenum was customized to some degree to keep the plenum volume constant at 2.25 to 2.26 litres for each case. While the distribution of volume (i.e. the cross section) along the length of the plenum will differ slightly for each diffuser angle case, the plenums are very alike. The base diameter of the plenum was also not changed. Table 2.2d, shows the comparative dimensions for Case 2.

Table 2.2d: Dimensions for intake restrictor case 2

Diffuser half angle	3^o	4^o	5.5^o	7^o
Diffuser exit diameter (mm)	42.578	50.126	61.485	72.898
Diffuser length (mm)	215.4	215.4	215.4	215.4
Diffuser volume (L)	0.17	0.22	0.31	0.40
Plenum volume (L)	2.25	2.25	2.26	2.26
Plenum length (mm)	160.5	160.5	160.5	160.5
Total length from throat to plenum bottom (mm)	375.9	375.9	375.9	375.9
Total volume after throat (L)	2.423	2.474	2.564	2.663

The simulation results in Figure 2.2c, in contrast to Case 1, the VE curves in Case 2 generally have the same profile and manifestation and have minor differences in VE. When volume changes are made to the diffuser, there is a necessity to evaluate if VE changes are due merely from the diffuser change or other simultaneous plenum changes.

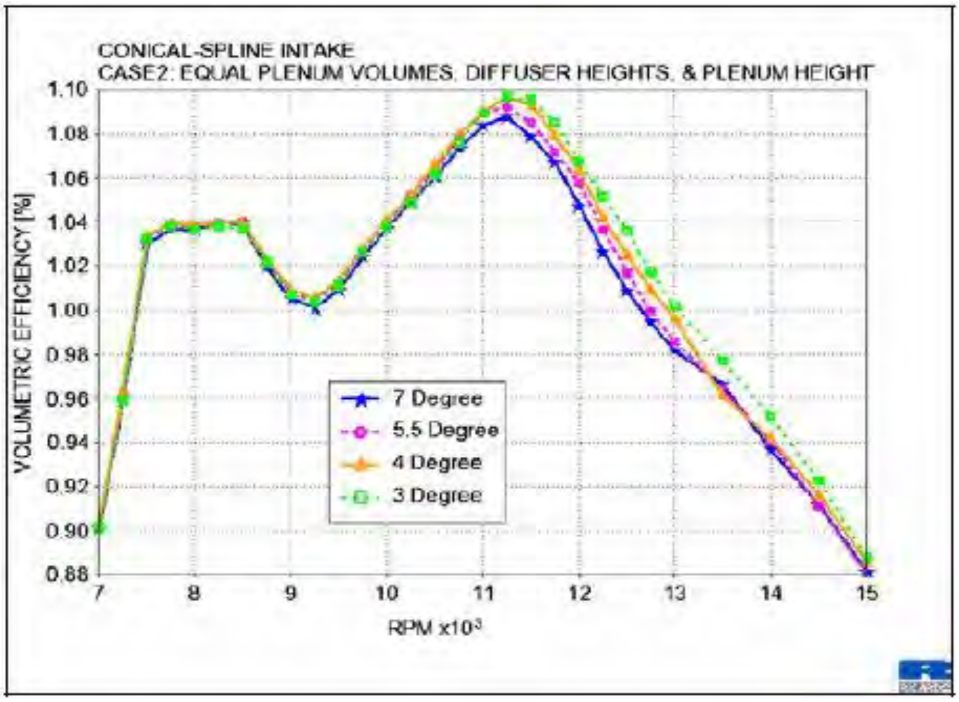

Figure 2.2c: VE of Conical-Spline Intake with Different Diffuser Angles and Constant Total Intake Height

If changes in plenum volume are considered in restrictor performance, one must also evaluate the impact of length or geometry changes to obtain that volume increase. Case 1 and Case 2 prove this point. Additionally, Case 1 and Case 2 show the difficulty in comparing multiple intakes at only a few rpm points if some concept of the entire VE curve is not known. [3]

2.3 Practical Research

In the paper of Logan M. Shelagowski and Thomas A. Mahank [4], two intake system restrictor prototypes: design I and II, Figure 2.3a. The first was designed and built based on a paper by Byam et al. [5] and a design study by Jawad et al. [6]. It features an 8° incline taper beginning at the (0.787 inch – 20 mm) throat and gradually transitioning to the outlet. The inlet shape is bell mouthed. The second restrictor was designed and built upon conclusion of this CFD study. It consists of a 6.3° incline