FINAL YEAR PROJECT II: DISSERTATION

MODELLING OF HYDRAULIC FLIP INSIDE A SCALED-UP DIESEL INJECTOR NOZZLE

PREPARED BY:

SHATHISWARAN A/L UTHRAPATHY

14998

SUPERVISOR:

DR. RAHMAT ISKANDAR KHAIRUL SHAZI

Preliminary report submitted in partial fulfilment of the requirement for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

JANUARY 2015

Universiti Teknologi PETRONAS,

32610 Bandar Seri Iskandar,

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

MODELLING OF HYDRAULIC FLIP INSIDE A SCALED-UP DIESEL INJECTOR NOZZLE

BY

SHATHISWARAN A/L UTHRAPATHY
14998

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL)

Approved by,

____
(DR. RAHMAT ISKANDAR)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

JANUARY 2015

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

SHATHISWARAN A/L UTHRAPATHY

ABSTRACT

The ignition process in diesel engines is mainly depending on the quality of mixing process of highly compressed air and fuel. The quality of mixing process here refers to how well the air atoms and fuel atoms are in contact with each other. Mixture of air and fuel particles can induced ignition, but the time frame for a particular ignition is depending on the surrounding parameters. These parameters either delay or furtherance the time frame of ignition. Atomization process that took place in the nozzle and is one of the parameters that enhances the ignition process. Atomization is a process where the fuel in the form of liquid is broken into tiny small particles. Good atomization is required for a better ignition process. The efficiency of atomization is affected by several condition. This paper mainly focused on two conditions. Cavitation and hydraulic flip are those two conditions. Cavitation can enhance atomization process while hydraulic flip reduces the effectiveness of atomization. Hydraulic flip begins when the cavitation bubble goes beyond the super-critical cavitation condition. Eventually, the downstream ambient air which is usually at higher pressure compared to vapor saturation process will enter into the cavitation domain to produce a layer which separates the liquid and the nozzle wall. This phenomenon has a great impact on the structure and atomization of liquid jet. This study is mainly on designing the nozzle which can prevent hydraulic flip. The design of Martynov (2005) was drawn in a computer aided software (SOLIDWORKS, 2014). The validation and simulation of the created nozzle design were performed using computational fluid design software (ANSYS 15.0). Modification of nozzle done by attaching trip wire at the throat of the rectangular nozzle. Trip wire alter the characteristic of the separated shear layer by changing the periodic shedding behavior which dependent on the formation and coalescence of micro-vortex cavity at separated shear layer and re-entrance motion. Shedding frequency increases due to attachment of trip wire. Higher frequency relates to shorter length of cavitation area.

ACKNOWLEDGEMENT

First and foremost, I would like to thank my supervisor, Dr. RAHMAT ISKANDAR for all his support for me in completing this project. Without his coaching and supervision, I would not have made it this far.

I would also like to thank my university, Universiti Teknologi PETRONAS, UTP for giving me an opportunity to carry out this project successfully.

I am also very grateful for all the support given by my family who were always there by my side. I would also like to thank my family, friends and others who have helped me in completing my final year project as a mechanical engineering student in UTP.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1:

INTRODUCTION

1.1 BACKGROUND

Diesel engine is effective for heavy duty tasks such as load carrying lorries. This is because the energy density extracted from diesel is much higher than the energy density extracted from gasoline. Ignition process in diesel engines is initiated when fuel is injected into the piston which contains compressed hot air. For a better ignition, fuel spray needs to be in fine particles. Fine fuel particles have a greater surface area in contact with the air particles.

Atomization is the process where the fuel in the form of liquid is broken down into fine particles. Atomization of fuel takes place in the diesel injector nozzle. Diesel injector nozzle sprays the fine fuel particles in the piston. Fine particles are not produced once the downstream air enters the nozzle and forms a layer that separates the nozzle wall and the fuel. This phenomenon is termed as hydraulic flip. Hydraulic flip is a condition that is not highly recommended for atomization.

This project investigates cavitation in diesel injector nozzle. Cavitation in the high pressure nozzle is related to hydraulic flip (Sou, 2007). When the cavitation bubble goes beyond the super-critical cavitation condition, hydraulic flip begins. Eventually, the downstream ambient air at higher pressure compared to vapor saturation pressure will enter into the cavitation domain to produce a layer which separates the liquid and the nozzle wall. This phenomenon has a great impact on the structure and atomization of liquid jet. The layer formed in the nozzle reduces the area in which fuel is flowing through. The velocity of the fuel flow in nozzle increases since fuel flow area is reduced. Furthermore, there will be no friction between nozzle wall and fuel since the layer of downstream air separates the two interface. Turbulence is reduced and the resultant atomization quality drops.

1.2 PROBLEM STATEMENT

Hydraulic flip in a condition where the structure and the break-up of the fuel from the nozzle is affected. The hydraulic flip condition will prevent the liquid in the jet from touching the wall of nozzle. This will eliminate the turbulence production at the wall of nozzle. Hydraulic flip is highly not favorable in the diesel injection nozzle. The problem statement of this project is to simulate the diesel injection nozzle using simulation software to understand the internal flow characteristic specifically cavitation behavior.

1.3 OBJECTIVE

The objectives of this project are:

i. To study the parameters and conditions which contribute to cavitation and hydraulic flip
ii. To model Sergey Martynov’s design in SOLIDWORKS
iii. To modify and simulate Sergey Martynov’s design using ANSYS

1.4 SCOPE OF STUDY

This project mainly will cover the topic from Martynov (2005) on the hydraulic flip. Study of the hydraulic flip and condition that affect hydraulic flip is studied. Martynov (2005) design was modeled in SOLIDWORKS 2014. The created design was validated in ANSYS 15.0. The result obtained from the simulation was compared with the Roosen (1996) and Martynov (2005) result. Once the validation process in completed, modification of the nozzle is modeled before simulating it to obtain the final result.

CHAPTER 2:

LITERATURE REVIEW

2.1 CAVITATION

The fuel spray behavior depends on the fuel properties, geometry of injectors, upstream flow condition and also condition at the downstream of the injector nozzle. Process of formation of voids in a liquid due to a sudden drop pressure, when the local pressure drop, when the local tension exceeds the tensile strength of the liquid is termed as cavitation (Brennen, 1995).

Cavitation is the development and collapse of vapor phase in liquid when the local pressure drops below the saturation pressure at given temperature. Cavitation can be divided into two categories. The first is termed as hydrodynamic cavitation where the cavitation is initiated because of the reduction of static pressure inside the nozzle by the hydrodynamic motion of liquid. The second is called as acoustics cavitation where the cavitation occurs by the propagation of pressure wave inside the nozzle (Martynov, 2005).

A cavitation number describes the nature of the flow. It is a parameter that relates the pressure drop to the local static pressure. Critical cavitation number marks the inception of cavitation and allows the flow to be classified as non-cavitating or cavitating (Martynov, 2005). Cavitation inception is the term for the flow condition when the hydrodynamic cavitation first appears in the nozzle; cavitation desinence is the term used for the condition when the cavitation vanishes from the flow inside the diesel injector due to minor changes in the parameter of the flow. Equation 2.1 shows the cavitation number introduced by Bergwerk (1959):

$$CN = \frac{P_1 - P_2}{P_2 - P_v} \qquad (2.1)$$

where P_1 is the pressure at the inlet of the nozzle, P_2 is the pressure at the outlet of the nozzle and P_V is the vapour pressure, usually associated with the saturation pressure in the liquid.

Experimental studies of cavitation flow in nozzles with different shapes have revealed there is relationship between the cavitation number and the extent of the cavitation region. The result of the relationship mentioned above is classification of cavitation flow regime depending on the cavitation number, namely: incipient, developed (sub-cavitation and transitional cavitation) and super-cavitating (Sato & Saito, 2001). At a certain value of cavitation number CN_{inc}, inception of cavitation occurs in the vena contracta region i.e. zone of recirculation flow downstream the nozzle entrance. During sub-cavitation stage cavitation bubbles fill the separation region, and changes in CN have little affect on the length of cavitation region. Further increase in CN enlarges the cavity length (transitional cavitation) and at a certain point (CN_{super}, supercavitation number) cavitation zone rapidly extends to the outlet of nozzle (supercavitation). At high cavitation number CN> CN_{super} cavitation zone exceeds the nozzle hole and forms jet cavitation when the nozzle is submerged in a liquid (Sato & Saito, 2001).

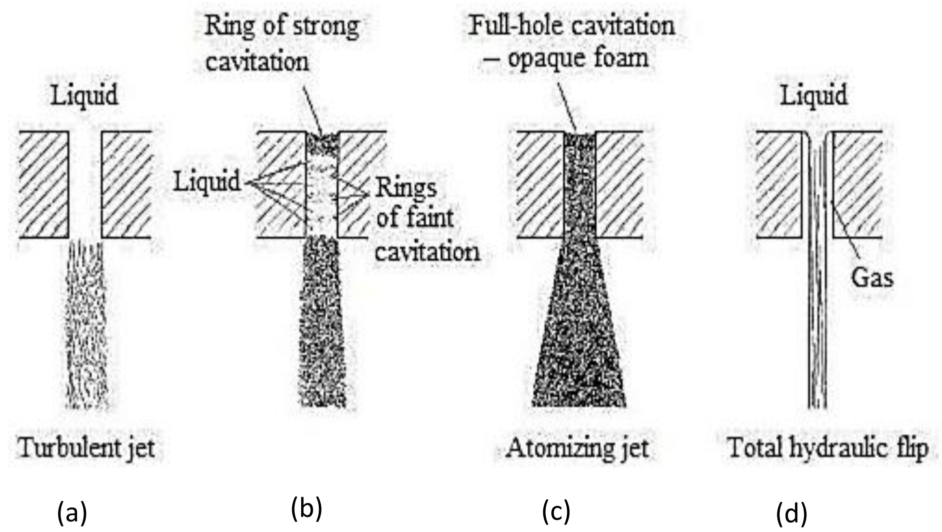


Figure 2.1: Characteristic types of spray formation while flowing through the large-scale circular nozzle (Soteriou, 1999)

Figure 2.1 illustrates the type of spray produced at different cavitation number. In image (a) cavitation does not occur. As a result, the exit flow is a turbulent jet. Image (b) shows the exit flow in the presence of cavitation and the result is a fairly atomizing jet. Image (c) illustrates the exit flow as an atomizing jet. This atomizing jet is produced when the cavitation number is at CN_{super}. Image (d) shows the exit flow as total hydraulic flip jet. This type of exit flow is produced when the cavitation number exeeds the $CN_{super.}$

2.2 SIMILARITY CRITERIA AND SCALE EFFECTS

According to Ashley (1997), the standard modern diesel injector typically has a nozzle diameter of 0.2mm and injection pressure of 2000 bar. He also stated that the injection speed ranges between 200-400 m/s. Since the nozzle size is too small, it is difficult to visually observe the flow inside the nozzle. To overcome this difficulty, application of special methods and equipment are needed. Scaling theory is applied, where a larger model of the nozzle can be used. The result from the large scale experiments can then extrapolated to real-scale flows (Soteriou, 1999). Parameters that were found to be very vital for the description of the cavitation flow are densities and coefficient of dynamic viscosity of liquid and vapor phases (ρ_l, ρ_v, μ_l, μ_v) coefficient of surface tension (σ), speed of sound in liquid (c), spatial scale of the flow ($l\infty$), pressure upstream and downstream the cavitation region (P_1 and P_2) and velocity scale of the flow ($\mu\infty$) (Lecoffre, 1999)

Cavitation number, CN and Reynolds number, *Re* is the most important criteria that describe the similarity in the experimental studies of cavitation flow in a real size nozzle (Bergwerk, 1959). Usually *We* and *Fr* number are neglected in high speed cavitation flow with prevailing inertia force. In practice, to make two cavitation flows similar, the scale effects associated with the liquid quality and viscous nature of the flow should be minimised.

2.3 HYDRAULIC FLIP

Hydraulic flip is an atypical conduct that can happen in short, ideal, nozzle yet is not thought to be regular in real injector nozzle. Separation of the flow occurs at the inlet corner and an undisturbed flow is achieved through the nozzle. Downstream gas surrounds the liquid in the nozzle and kept the liquid away from the nozzle wall. In short, sharp-edged nozzles, the vena contracta affects the atomization of the nozzle (Karasawa, 1992).

Figure 2.2 illustrate the vena contracta phenomena. Vena contracta is a reduction of flow area that happens because of the flow separation at the nozzle walls. Nozzle effect on the spray is determined by the contraction inside the nozzle.

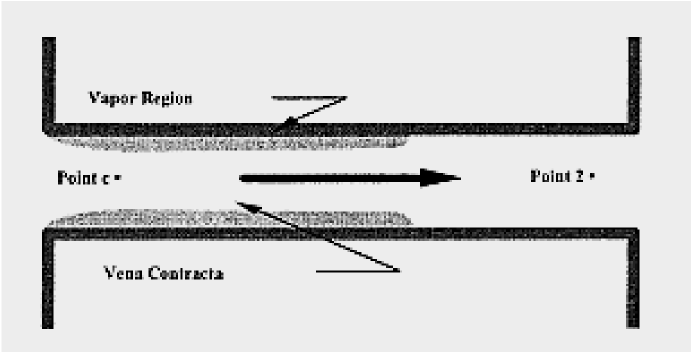


Figure 2.2: Schematic of an axisymmetric cavitating nozzle.

Difficulty to prove hydraulic flip increases as the experiment scale decreases (Bergwerk, 1959). The changes in the nozzle could not be interpreted accurately as the nozzle scale decreases.

Poor atomized sprays were produced in the flipped nozzles. This is because wall shear and pressure gradient are not present in the flipped nozzle (Soteriou, 1999). Eventually, the turbulence is overwhelmed. Smooth and continuous liquid jet is produced from the nozzle with the absence of initial disturbance. Partial hydraulic flip is where the cavitation bubble and downstream gas are present in the nozzle at the same time. Cavitation occurs on one side of the nozzle and the other side is covered with downstream gas.

Nozzle behavior depends on upstream conditions, for an example, the flow around the needle and sacs (Soteriou, 1999). A strong asymmetry exit flow jet is produced in the presence of needle. The needle causes the flow to separate only at one side of the nozzle. Flip of flow will occurs with the absence of needle. Sometimes the separation would be an asymmetric hydraulic flip. On the other hand, presence of sac tip causes flip of flow in the injector. This flow flip will take place with or without the presence of a needle. The entrance flow to the nozzle openings is influenced by the presence of the needle valve surface and the angle between the sac dividers and the hole center-line. Due to manufacturing limitation, even single-hole injector can't be axisymmetric. This has beneficial side-effect of preventing hydraulic flip.

Nurick (1976) manipulated the downstream and upstream pressure and the L/D ration of his nozzle to experiment cavitation flow and hydraulic flip. Low static pressure in the high speed nozzle flow near a sharp inlet causes the formation of cavitation bubble. When the edge of the inlet is sharp enough, separation of flow tends to occur and formation of vena contracta inside the nozzle begins. This contraction at the inlet tends to decrease the area where the liquid flow through. Velocity of the flow increases as the area is reduced. The vena contracta contraction causes acceleration in flow and eventually causes a pressure depression in the throat of the nozzle. Cavitation happens once the pressure inside the throat decreases beyond vapor pressure of liquid. Cavitation bubbles start to build around the nozzle walls.

2.4 EFFECT OF TRIP WIRE ON CAVITATION BEHAVIOUR IN NOZZLE

Attached cavity and shedding cavity are the two type of cavity vortices found in a cavitation flow inside a nozzle. The attached cavity begins at the throat of the nozzle. According to Sato & Saito (2001), the presence of the attached cavity and re-entrant jet shed a cloud-like cavity in the direction of flow. This cloud-like cavity is termed as shedding cavity. The fixed cavity type (attached cavity) generates the shedding type of vortex cavity (shedding cavity).

Sato & Saito (2001) conducted experiment on long circular-cylindrical orifices of various throat lengths including with a trip wire. The experiment was conducted about five different orifices; orifice A, B and C with different throat length; orifice D with trip wire attached and orifice E with different throat diameter. Trip wire was attached on the front step surface of the round orifice to disturb the flow inside the orifice. High-speed video camera system was used to observe the bubble behaviour inside the orifices. Table 2.1 summarises the experimental result obtained.

Table 2.1: Experimental result from Sato & Saito (2001)

Orifice	A	B	C	D	E
L (mm)	100	70	50	100	70
d (mm)	22	22	22	22	15
Trip wire	×	×	×	O	×
Tw (K)	292-298	293-296	293-296	291-296	294-296
β (mg/ℓ)	4.1-6.7	4.6-5.4	4.1-5.4	2.9-4.4	3.9-4.8
Re ($\times 10^5$)	2.06-2.38	2.21-2.26	2.11-2.26	2.01-2.26	1.59-1.66
σ	0.94	0.94	0.95	0.83	0.97
U (m/s)	9.65	9.65	9.65	9.65	10.4
F (Hz)	257	242	252	365	294
L_m(mm)	22	22	22	21	16.5
S_{tm}(=F x L_m / Uc)	0.36	0.34	0.36	0.49	0.29
L_0 (mm)	14.6	16.1	16.7	13.6	10.9
S_{tL}(=Fx L_0 / U)	0.39	0.40	0.44	0.51	0.31

For this particular project, result of orifice A and D was compared to study the effect of trip wire on the behaviour of flow inside the orifice. Orifice D has trip wire attached just in-front the throat inlet. The length of throat (L), diameter of throat (d), and inlet velocity (U) is the same for both orifice A and D. The only manipulated variable for the study was presence of trip wire inside the orifice. Table 2.2 gives a clear picture of the variables associated with the study.

Table 2.2: Variables associated with the study of effect of trip wire inside a long circular-cylindrical orifice *(Sato & Saito, 2001)*

Orifice	**A**	**D**
L (mm)	100	100
d (mm)	22	22
U (m/s)	9.65	9.65
Trip Wire	X	/

The orifice A and D was tested using same inlet velocity and the experimental result as in Table 2.3 was obtained.

Table 2.3: Experimental result with and without trip wire attachment

Orifice	**A**	**D**
Re (10^5)	2.06-2.38	2.01-2.26
Cavitation Number	0.94	0.83
F (Hz)	257	365
L_m	22	21
L_o	14.6	13.6
S_{tm}	0.36	0.49
S_{tl}	0.39	0.51

F= Shedding frequency of cloud like cavity

L_m= Time average cavity length (floating bubble)

$L_{o=}$ Length of attached cavity (bubble attached to wall)

$S_{tl=}$ Strouhal number based on cavity length, L_o and average throat velocity, U

$S_{tm=}$ Strouhal number based on L_m

From Table 2.3, due to the attachment of trip wire, shedding frequency of cloud like cavity increases significantly. Unsteadiness of the flow increases and cause the length of the attached cavity to reduce. As conclusion, the periodic shedding process dependent on the formation and coalescence of micro-vortex cavity at separated shear layer and reentrance motion. Trip wire alter the characteristic of separated shear layer by changing the periodic behavior of the shedding process. The periodic shedding frequency becomes higher due to attachment of trip wire. Lastly, higher frequency of shedding relates to shorter length of cavitation area.

CHAPTER 3:

METHODOLOGY/PROJECT WORK

3.1 PROJECT ACTIVITIES

Figure 3.1 shows the flow of this project. There are three main steps. The first step is preliminary research work. The second step is project execution and lastly result analysis and documentation.

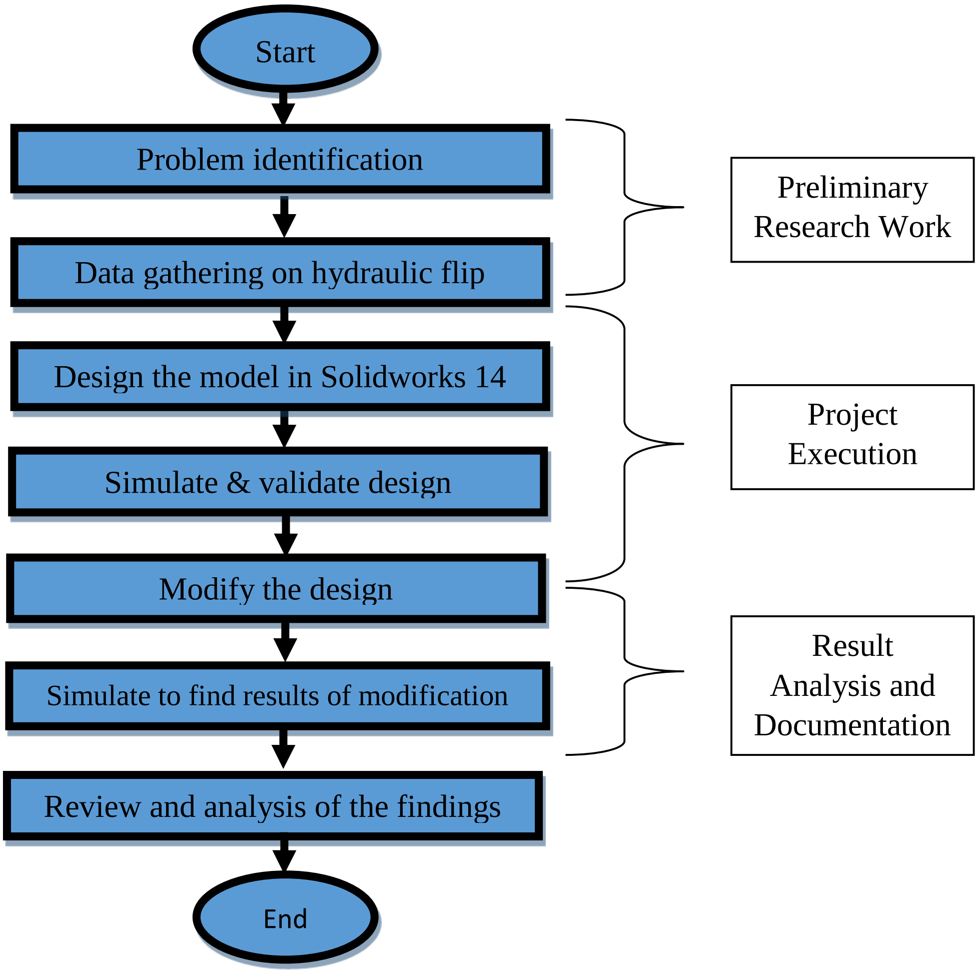


Figure 3.1: Workflow of project.

3.1.1 PROBLEM IDENTIFICATION AND DATA GATHERING

Prior research which covers on cavitation and hydraulic flip is necessary to get a clearer view of the problem. The data for this project was carried out on various previous studies initially. Once the problem has been identified, focus was on Martynov (2005) to frame the scope of study. This paper is the main reference for this project.

3.1.2 DESIGN OF MODEL

Reference measurement was taken from Martynov (2005) and figure 3.2 shows design data provided. The diesel is supplied to the nozzle at the inlet and goes through the larger section of supply tube first before reaching the nozzle inlet. Diesel then exits the nozzle at the outlet as spray.

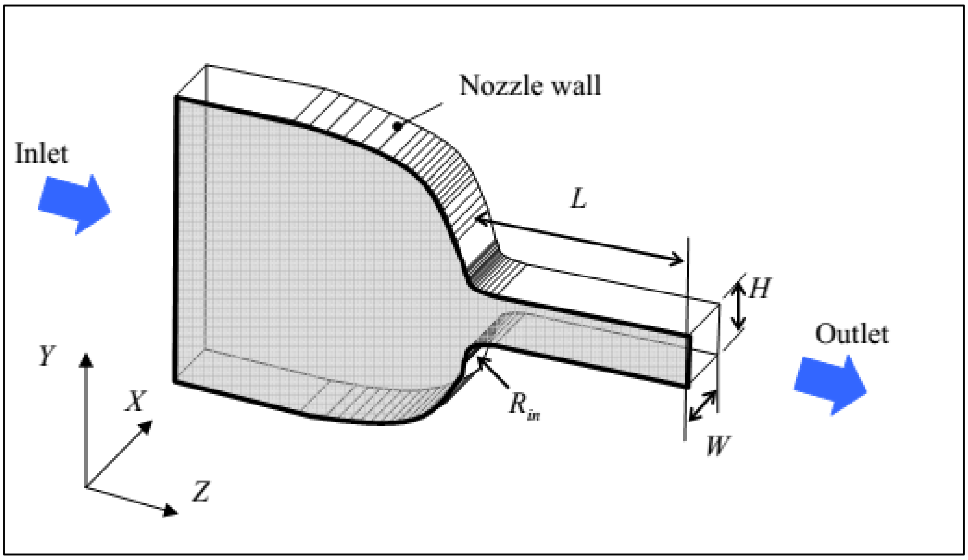


Figure 3.2: Design of diesel injector nozzle *(Martynov, 2005)*

The nozzle has a length L=1mm, width W=0.2 mm, and height, H=0.2 mm and the radius of inlet corner R_{in}=0.028 mm. The flow from inlet to outlet is in the positive Z-direction.

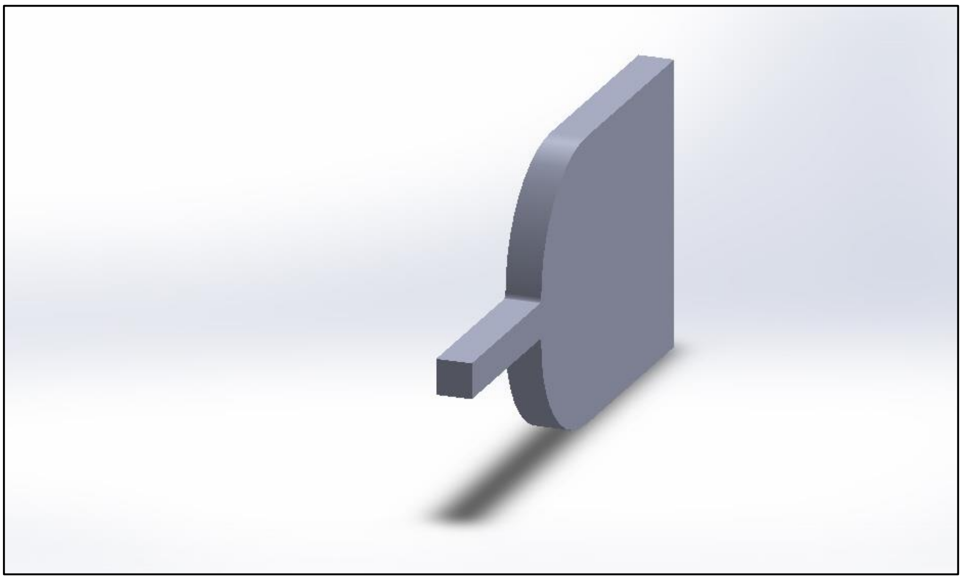


Figure 3.3: Full diesel injector nozzle designed in SOLIDWORKS 2014

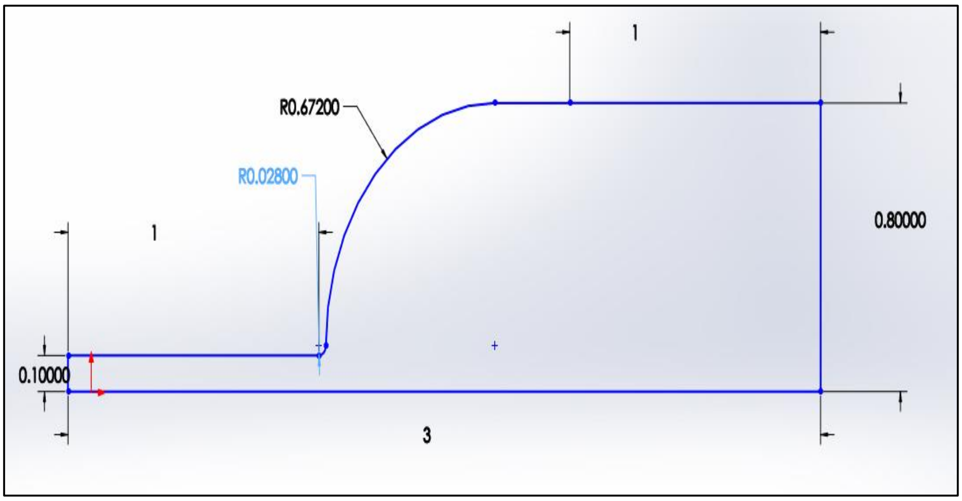


Figure 3.4: Sketch of diesel injector nozzle.

Figure 3.3 is the diesel injector nozzle model designed in the SOLIDWORKS 2014 according to Martynov (2005). Figure 3.4 shows the sketch of quarter of the diesel injector nozzle which is then extruded to 0.2 mm. The width of the diesel injector nozzle is 0.2 mm.

Figure 3.5 shows the quarter section of the diesel injector nozzle. Due to the symmetrical characteristic of the diesel injector nozzle only quarter section of the nozzle was used to perform simulation. Figure 3.6 illustrate the technical drawing of quarter section of diesel injector nozzle.

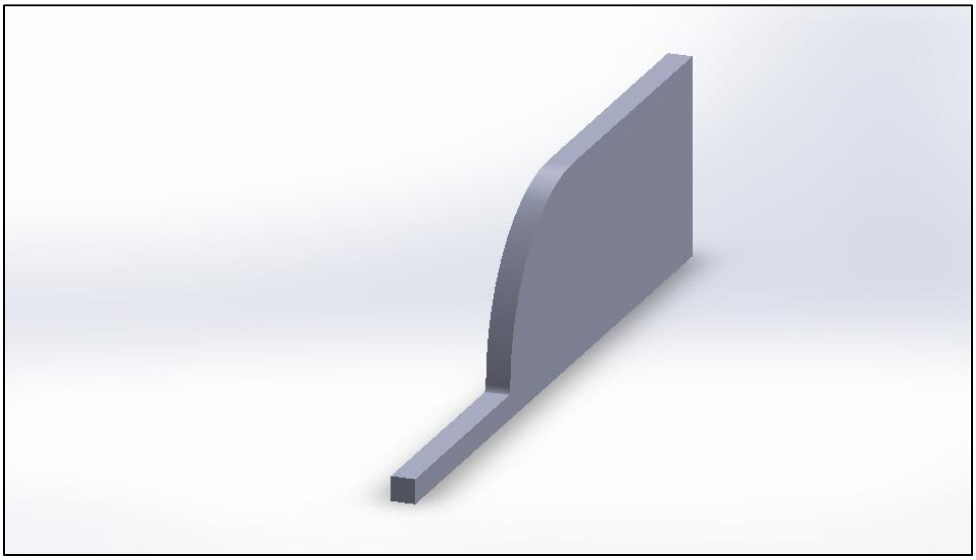


Figure 3.5: Quarter section of Diesel injector nozzle.

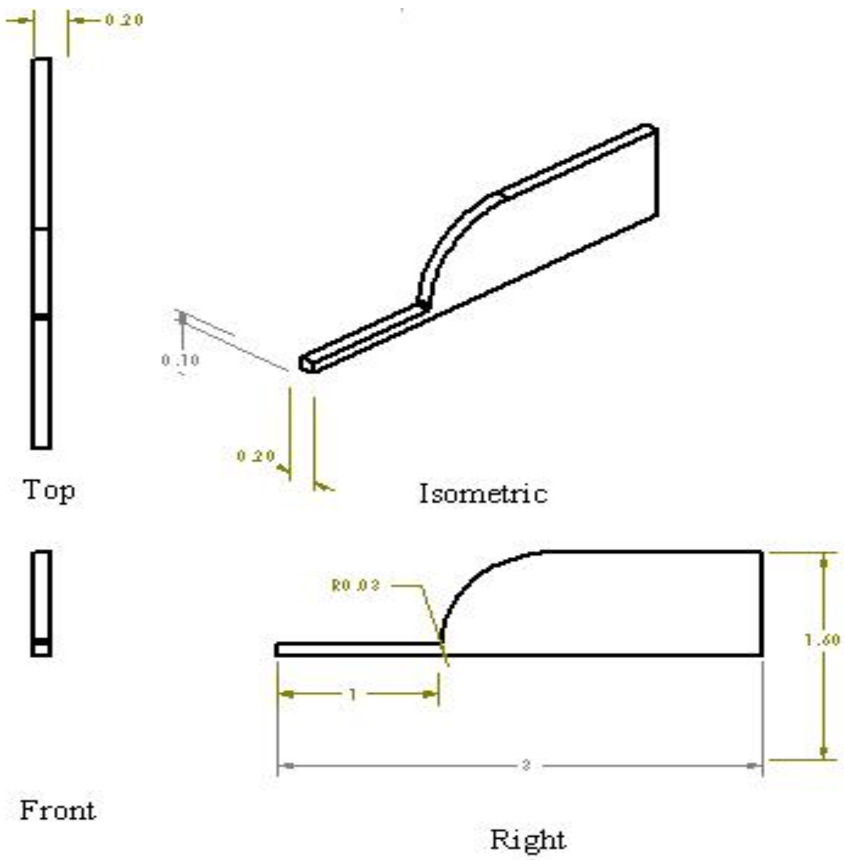


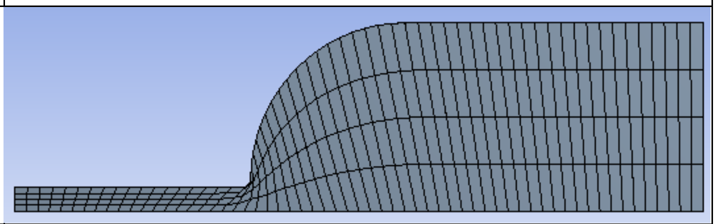
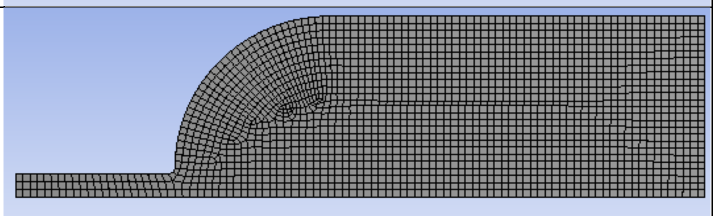
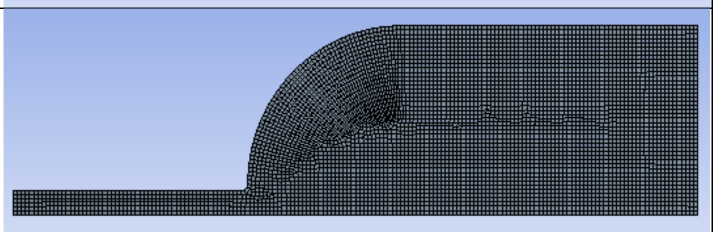
Figure 3.6: Technical drawing of Diesel injector nozzle

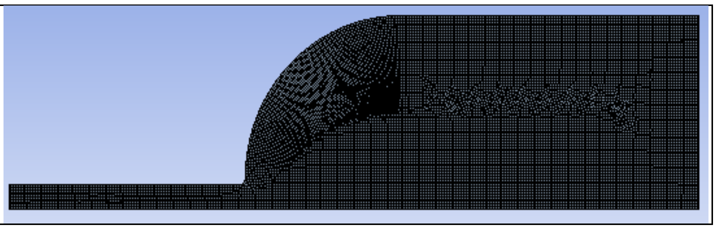
3.1.3 SIMULATION AND VALIDATION OF MODEL

All simulation in this project was done in ANSYS 15.0. Meshing of the geometry was done before simulate the flow problem. Meshing is a process where the geometry is divided into small equivalent cell to solve the flow problem. The size of the cell is variable. Fine mesh cells produce an accurate result but the time taken to solve the problem is long compared to a moderate mesh element. A balance between the mesh element size and accuracy of result should be maintained. Martynov (2005) provides four mesh element sizes. All the mesh element sizes were followed and simulation was done based on the input value given in Martynov (2005). Velocity at the inlet (14 m/s) is the input value given. Pressure drop (P_1-P_2), minimum pressure at nozzle throat (P_{min}-P_2), discharge coefficient of nozzle are the result provided in the paper. In this study, only the minimum pressure at nozzle throat was compared with the generated result to find the percentage difference.

Table 3.1 shows the mesh element size, number of cells and the corresponding generated mesh of the nozzle geometry.

Table 3.1: Grid use in the study of mesh dependence.

Mesh	Mesh Element Size (mm)	Numbe r of Cells	Mesh Generated
1	0.05	480	
2	0.03125	5922	
3	0.01786	31788	

4	0.01042	155050	

Simulation was performed for all the meshes with an input velocity of 14 m/s at inlet. All the four meshes have skewness, but the skewness metric is less than 0.3. According to Ansys, optimum skewness is below 0.9. Detailed explanation of skewness is presented in Appendix A. Table 3.2 shows the minimum pressure at nozzle throat obtained from the meshing (actual), minimum pressure at nozzle throat taken from Martynov (2005) (theoretical), percentage error and CPU time spent for computation using different meshes.

Table 3.2: Actual, theoretical value, percentage error and time taken by CPU for computation using different meshes

Mesh	Actual Value (MPa)	Theoretical Value (MPa)	Percentage Error (%)	CPU time, seconds
1	-3.46	-3.24	6.64	6.549
2	-4.98	-5.19	4.05	7.578
3	-6.08	-5.90	3.02	25.372
4	-6.17	-6.06	1.78	57.482

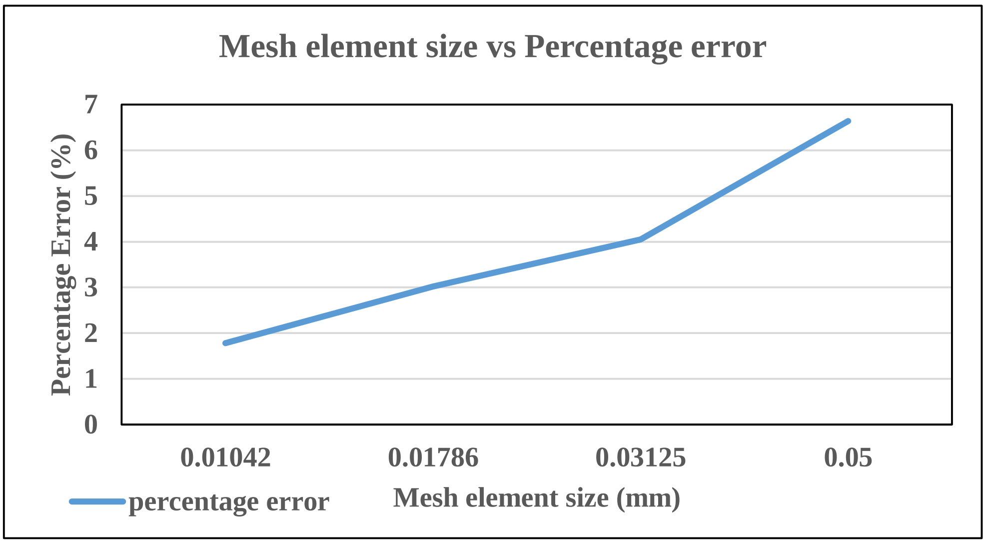


Figure 3.7: Effect of mesh element size on percentage error.

Figure 3.7 shows the percentage difference increases as the mesh element size increases. In order to evaluate grid dependence of the solution, the minimum pressure at nozzle throat was plotted as function of mesh element size. Finer mesh results in more accurate result. Mesh 2 has a percentage error of 4.05%. This meshing is sufficient for simulate further modification since the percentage error is less than 5%, since 5% is the threshold value for this study. On the other hand, mesh 4 has a percentage error of 1.78% which is lesser than mesh 1, but the time taken by CPU to solve the problem is longer compared to mesh 1. Mesh 2 is more suitable compared with all other 3 meshes and will be used for future modification process.

3.1.4 MODIFICATION TO THE MODEL

Modification done on the design of the diesel injector nozzle by attaching the trip wire at the entrance of the nozzle throat based on Sato & Saito (2001). The trip wire is a ring which has a thickness of 0.02 mm and diameter of 0.09 mm. Figure 3.8 and Figure 3.9 show the structure of the trip wire used in this project and the technical drawing of the trip wire, respectively. Figure 3.10 shows the attached position of the trip wire inside the nozzle.

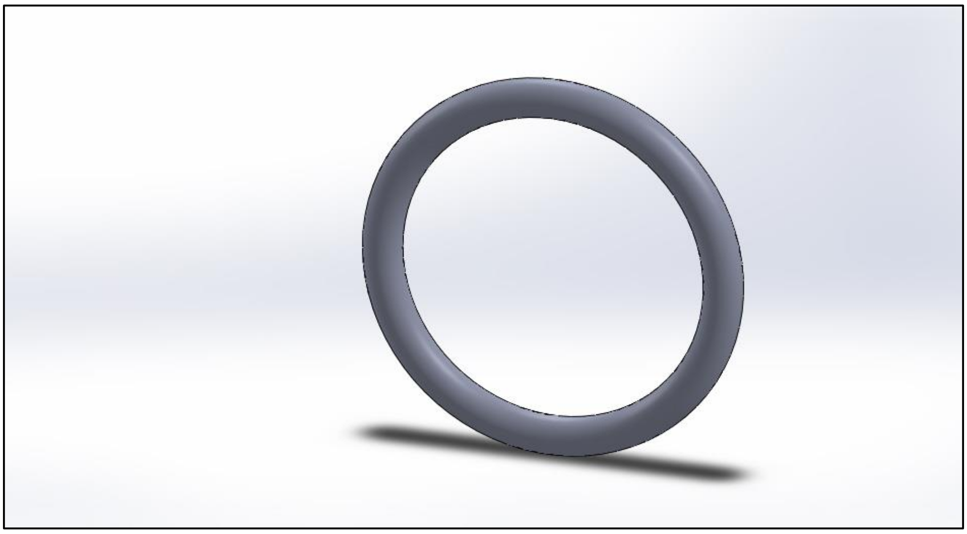


Figure 3.8 Trip wire model designed in SOLIDWORKS 2015

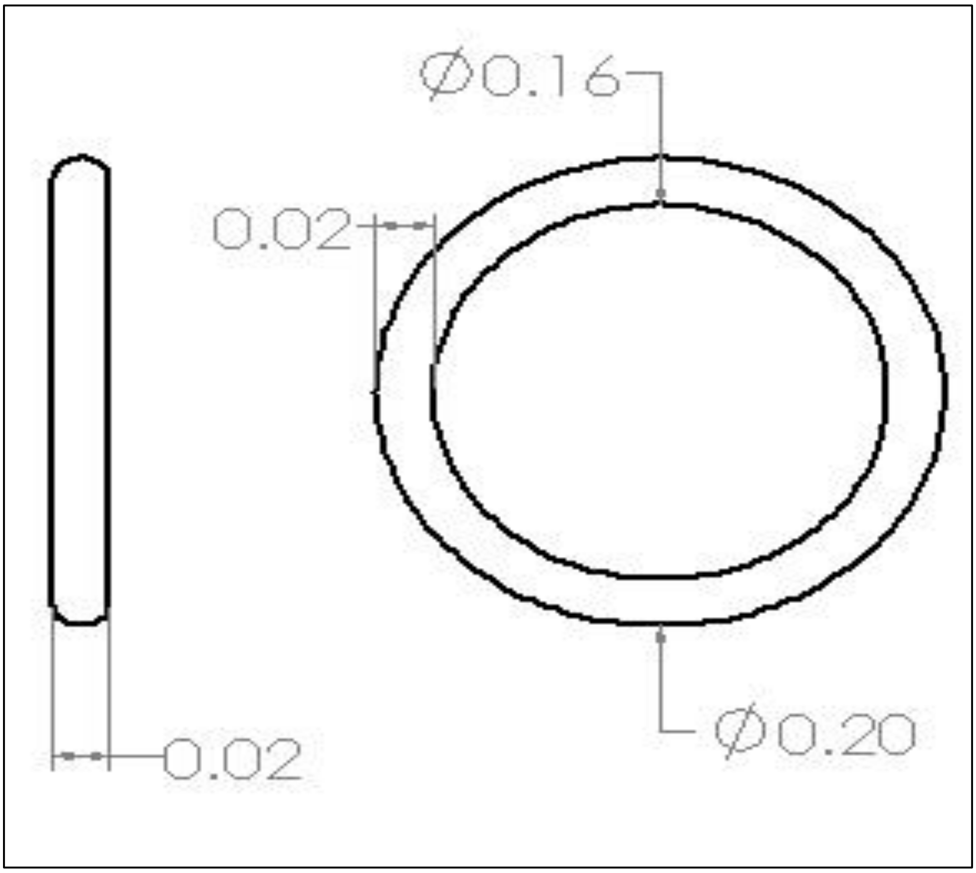


Figure 3.9: Technical drawing of trip wire

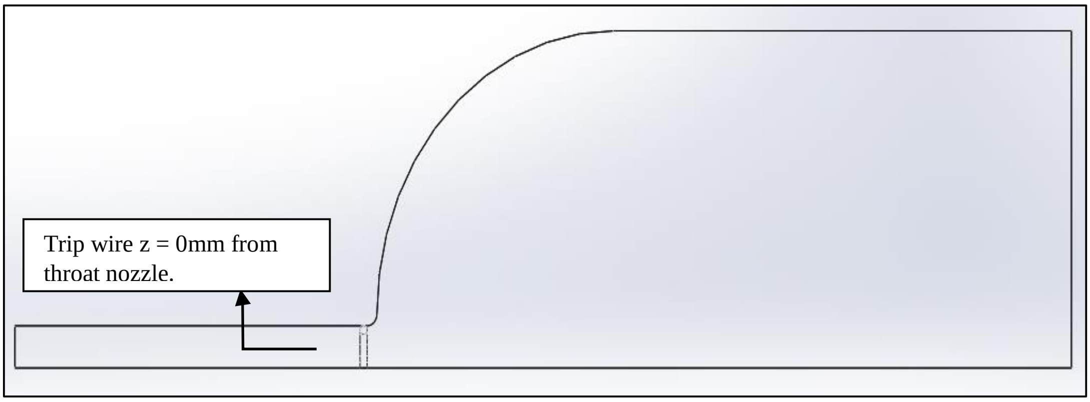


Figure 3.10: Trip wire attached at the entrance of the nozzle throat

3.1.5 SIMULATION SETUP

Simulation of Diesel injector nozzle model was performed first. Simulation for the modified Diesel injector nozzle model was done using the same simulation setup as for the unmodified Diesel injector nozzle model. For both model, two simulation were performed. The first is for simulating the inlet cavitation flow and second for simulating super-cavitation flow. For both flow, the injection pressure, P_1, and inlet velocity were same. The only varying parameter was the outlet pressure, P_2. The boundary conditions for the simulation for unmodified Diesel injector nozzle model and modified Diesel injector nozzle are presented in Table 3.3.

Table 3.3: Boundary conditions for simulation of Diesel injector nozzle

Boundary Condition		Inlet Cavitation	Super Cavitation
Inlet	Normal Speed	14 m/s	14 m/s
	Water Volume Fraction	1	1
	Water vapour Volume Fraction	0	0
Outlet	Static Pressure	21 bar	11 bar
Symmetry		Right and bottom	Right and bottom
Wall		No slip	No slip

Few input parameters were inserted for the cavitation flow. The mean diameter of the bubble (average diameter of cavitation bubble in a flow) was set to default value (2.0e-6 m). The saturation pressure of water at 25 degree Celsius was set to 2340 Pa and the cavitation condensation coefficient was set to 0.01.

3.1.6 REVIEW AND ANALYSIS OF THE RESULT

There are several comparison steps to analyse the result obtained from the simulation. Figure 3.11 summarises comparison steps.

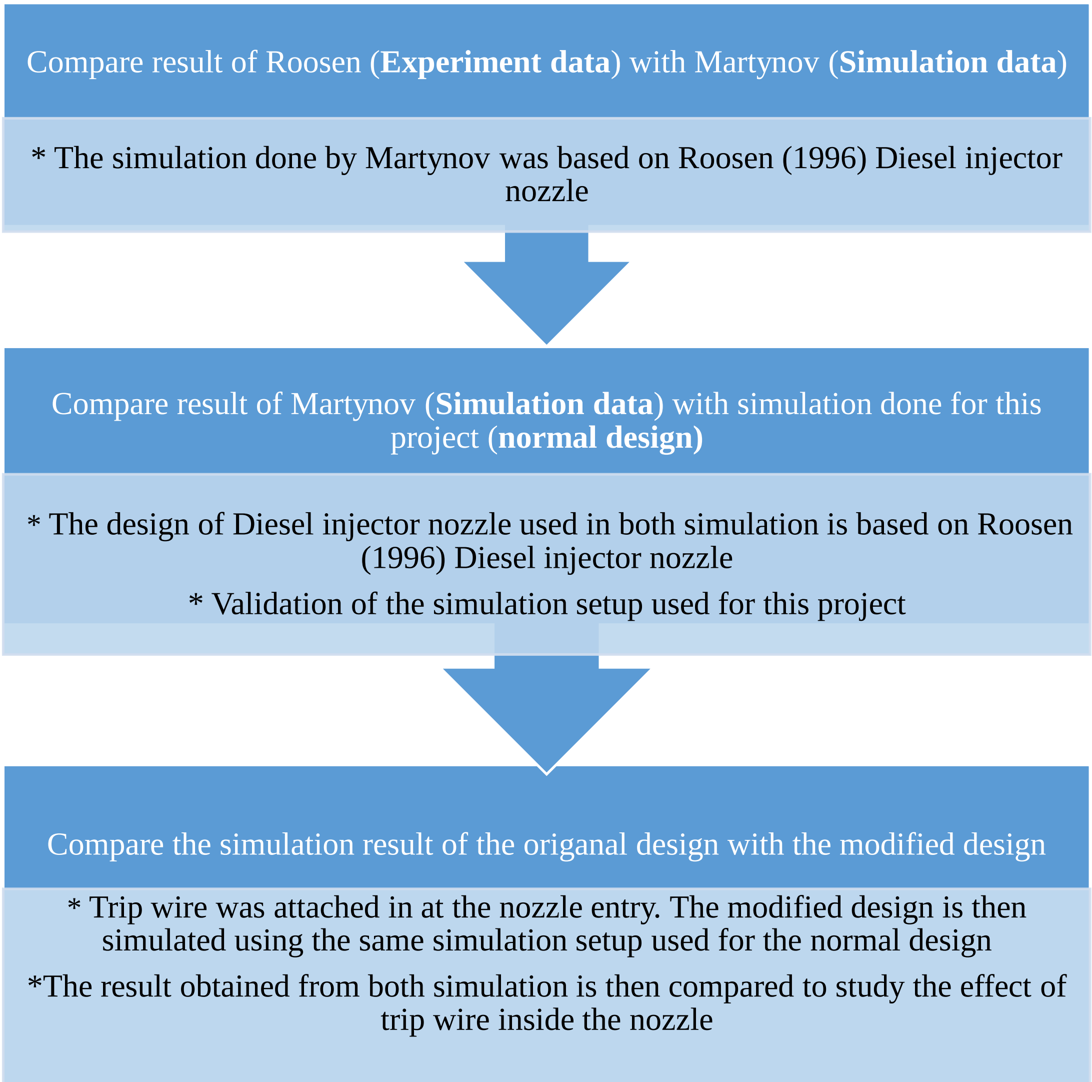


Figure 3.11: Steps of analyzing data

3.2 KEY MILESTONE

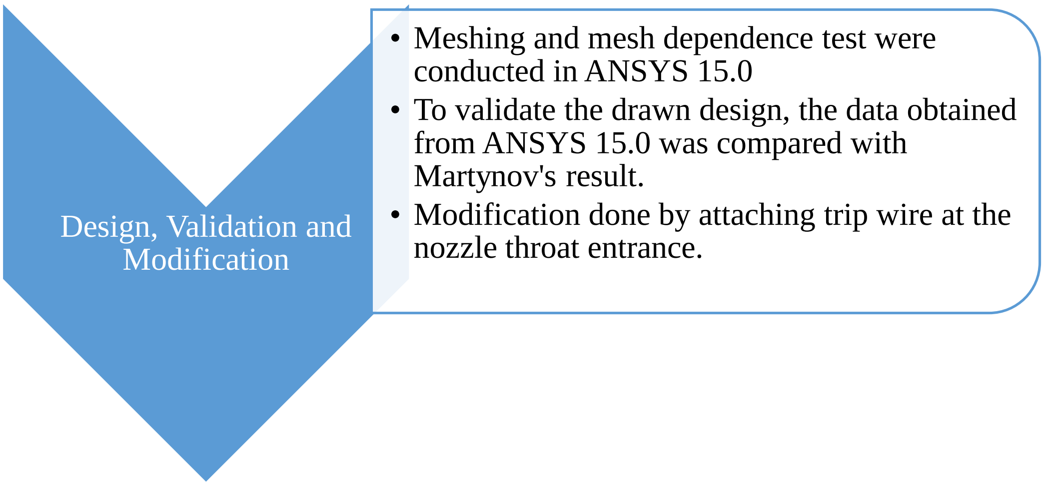


Figure 3.12: Key Milestone.

3.3 GANTT CHART

Table 3.1: Gantt chart.

No	Details	Weeks													
		1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Title and Supervisor Allocation														
2	Preliminary Research Work														
3	Preparing Extended Proposal														
4	Submission of Extended Proposal														
5	Proposal Defense														
6	Project Work Continues														
7	Submission of Interim Draft Report														
8	Submission of Final Interim Report														

CHAPTER 4:

RESULT AND DISCUSSION

4.1 OVERALL RESULT SUMMARY

4.1.1 RESULT FOR UNMODIFIED DIESEL INJECTOR NOZZLE MODEL

The working fluids for the simulation were water at 25 degree Celsius and water vapour at 25 degree Celsius. Boundary conditions as mentioned in methodology part were used for the simulation. Figure 4.1, Figure 4.2 and Figure 4.3 show the result obtained from the simulation for inlet cavitation flow for the unmodified Diesel injector nozzle model.

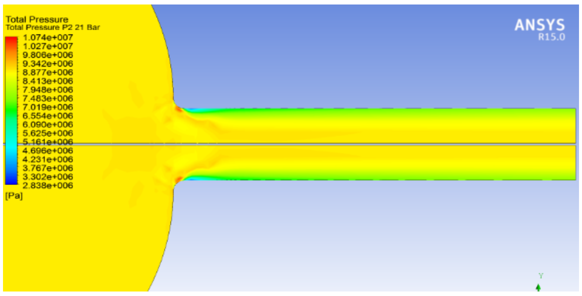


Figure 4.1: Total pressure for inlet cavitation flow (unmodified)

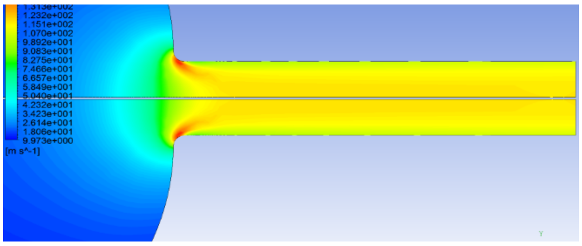


Figure 4.2: Water velocity for inlet cavitation flow (unmodified)

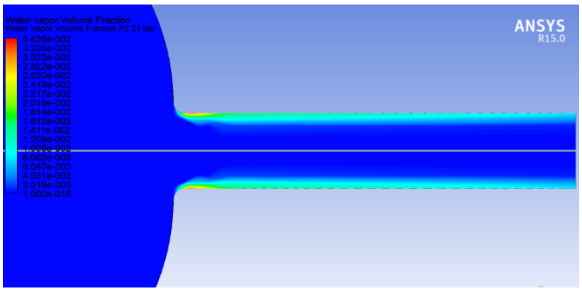


Figure 4.3: Water vapour volume fraction for inlet cavitation flow (unmodified)

The same simulation setup was used for the super-cavitation flow. The only varying parameter for the super-cavitation flow is the outlet pressure, P_2. Figure 4.4, Figure 4.5 and Figure 4.6 show the result obtained for the super-cavitation flow of the unmodified Diesel injector nozzle model.

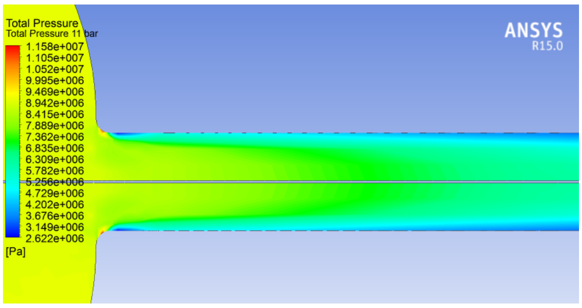


Figure 4.4: Total pressure for super-cavitation flow (unmodified)

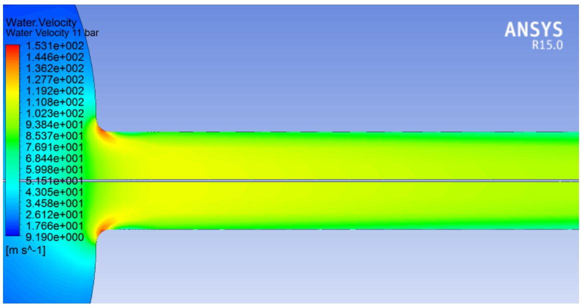


Figure 4.5: Water velocity for super-cavitation flow (unmodified)

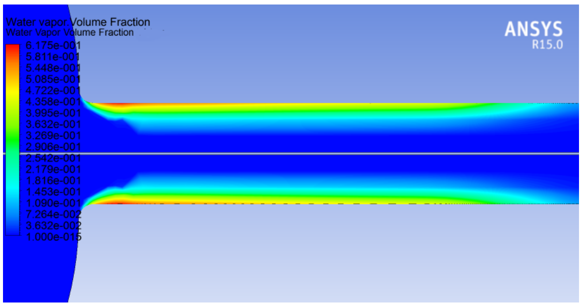


Figure 4.6: Water vapour volume fraction for super-cavitation flow (unmodified)

4.1.2 RESULT FOR MODIFIED DIESEL INJECTOR NOZZLE MODEL

The working fluid for the modified Diesel injector nozzle model were water at 25 degree Celsius and water vapour at 25 degree Celsius. The same simulation setup as previous model was used to simulate the modified model. Figure 4.7, Figure 4.8 and Figure 4.9 show the result obtained from simulation of modified Diesel injector nozzle model for the inlet cavitation flow.

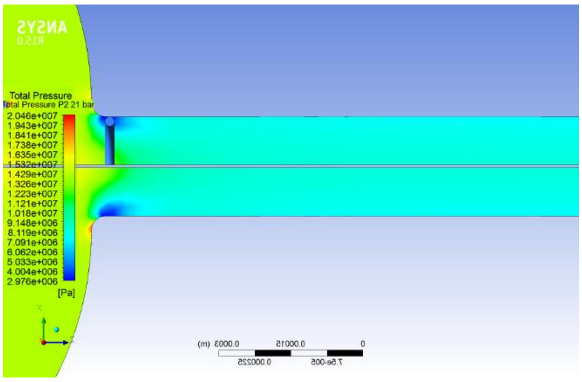


Figure 4.7: Total pressure for the inlet cavitation flow (modified)

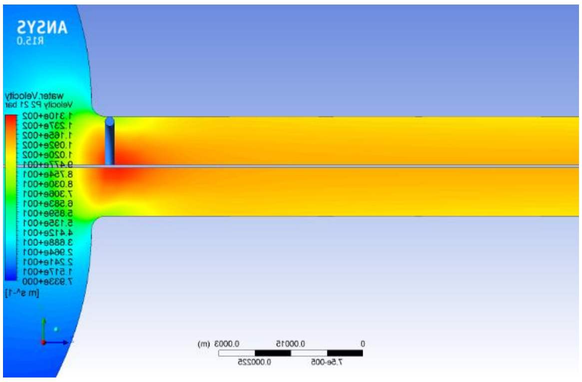


Figure 4.8: Water velocity for inlet cavitation flow (modified)

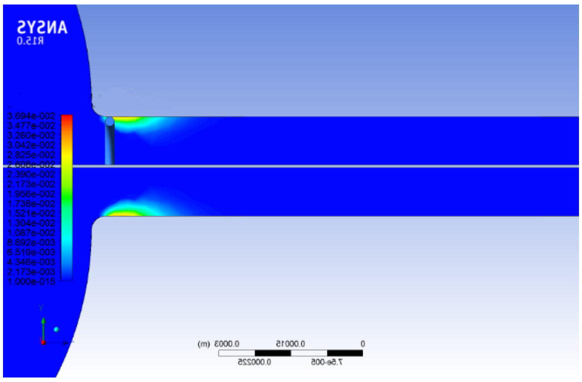


Figure 4.9: Water vapour volume fraction for inlet cavitation flow (modified)

For simulate super-cavitation flow in modified model, the same simulation setup used for similar flow in unmodified model was used. Figure 4.10, Figure 4.11 and Figure 4.12 show the result obtained for the super-cavitation flow in modified Diesel injector nozzle.

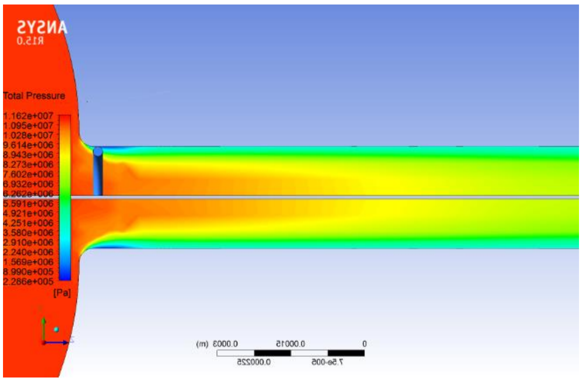


Figure 4.10: Total pressure for super-cavitation flow (modified)

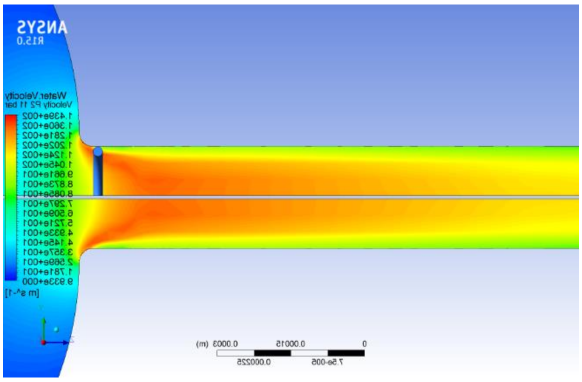


Figure 4.11: Water velocity for super-cavitation flow (modified)

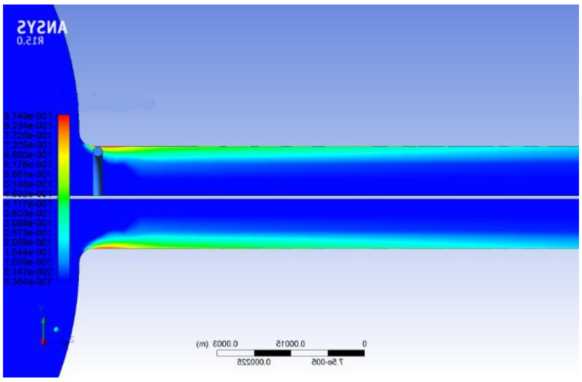


Figure 4.12: Water vapour volume fraction for super-cavitation flow (modified)

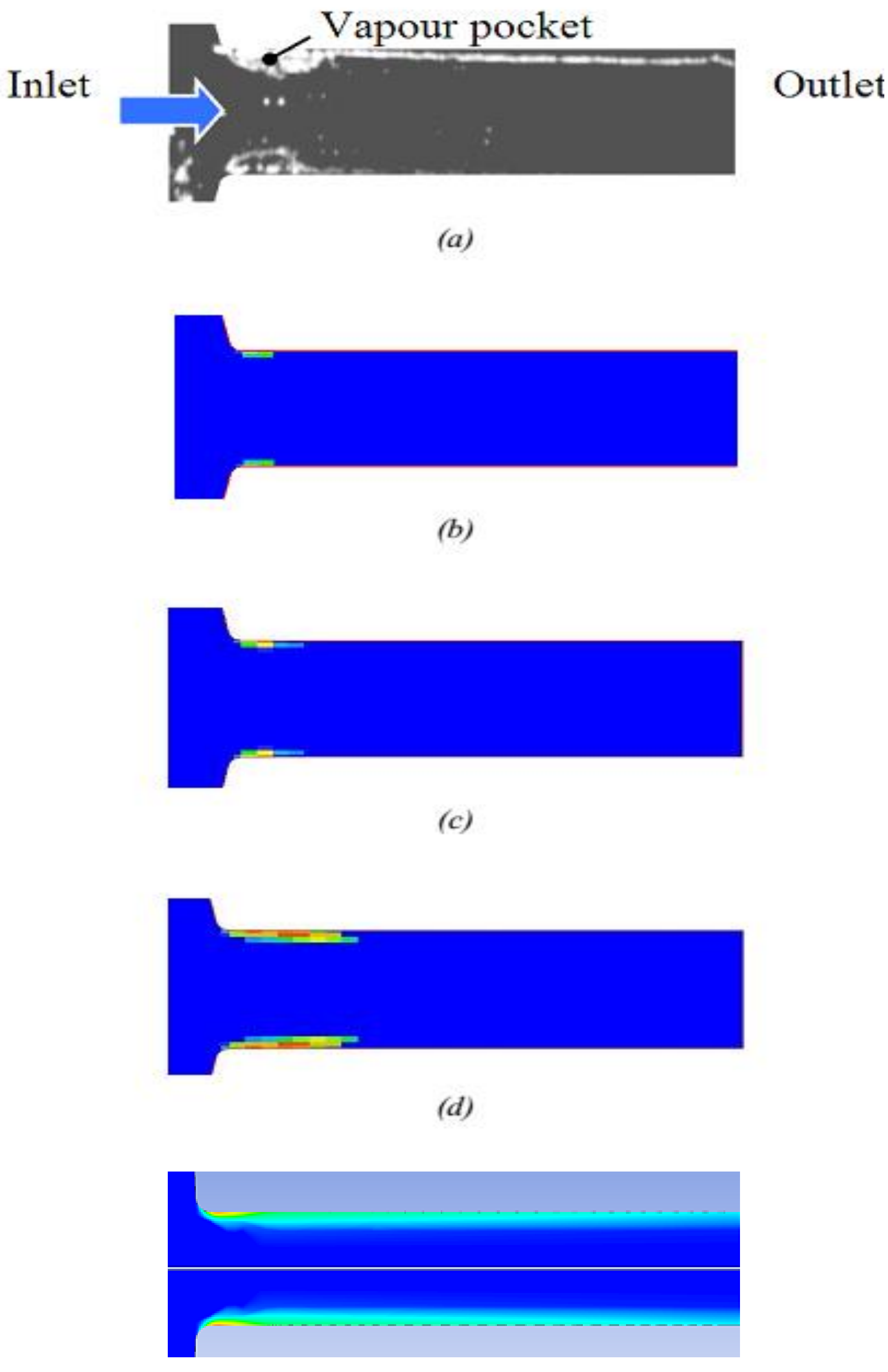
4.2 COMPARISON BETWEEN DIFFENT DIESEL INJECTOR MODEL RESULT (ROOSEN (1996) EXPERIMENTAL RESULT, MARTYNOV (2005) SIMULATION RESULT AND SIMULATION RESULT OBTAINED FOR THIS STUDY

Roosen (1996) performed experiment on rectangular model of Diesel injector nozzle. The working liquid for the experiment was tap water. Roosen (1996) observed vapour pocket at the nozzle entry (inlet cavitation) at injection pressure, $P_{1,}$ of 80 bar and the outlet pressure, $P_{2,}$ of 21 bar. According to Roosen (1996), at injection pressure, $P_{1,}$ of 80 bar and the outlet pressure, $P_{2,}$ of 11 bar, he observed super cavitation flow begins. The velocity of the flow for the both condition is 14 m/s.

Martynov (2005) modeled Roosen's (1996) diesel injector nozzle. He follows exactly the same condition as Roosen (1996) for the simulation of both flow. Specification of number density of critical cavitation nuclei was required for the model developed by Martynov (1996). Number density of cavitation bubble is number of bubble present in a cubic meter. This number specify the concentration of bubble in a flow. Several iterations were done by the author to determine the number density of cavitation bubble which could match the measured pattern of cavitation flow by varying the parameter n_* = liquid quality. He observed vapour pocket at the throat of the nozzle with the number density range from n=1.6(10^{13}) (m^{-3}) to n=2.0(10^{15}) (m^{-3}). For super-cavitation flow, he found out that number density range from n=1.3(10^{15}) (m^{-3}) to n=2.0(10^{15}) (m^{-3}).

The vapour structures predicted by the model (Figure 4.13, *b, c*, *d* and Figure 4.14, *b, c*) are thinner than the vapour pocket observed by Roosen (1996) (Figure 4.13, *a* and Figure 4.14, *a*). According to Martynov (1996), this can be explained by the accuracy of prediction of flow separation, which may result from specification nozzle geometry, model of turbulence and interaction between vapour and liquid phase in cavitation flow. Figure 4.13 *(d)* gives the best match with the length of photographed vapour pocket in Figure 4.13 *(a).* For the super cavitation flow, Figure 4.14 (*c*) gives the best match with the photographed vapour length in Figure 4.14 (*a*).

For this study, the similar simulation as to Martynov (2005) was carried out. The result for simulations match quite well to the both previous study results mentioned above. For the inlet cavitation, the photographed result of Roosen (1996) and both simulation results were not match quite well. This is because the number density of the nuclei bubble in Roosen (1996) flow was unknown and the number density in the simulation need to be varied to match the flow pattern of Roosen (1996). But for the super cavitation flow, simulation result obtained for this study matched well with Roosen (1996) flow pattern compared with simulation result obtained by Martynov (2005).



(e)

Figure 4.13: Pattern of inlet cavitation flow (photograph by Roosen (1996) *(a)* in comparison with the results of numerical predictions of vapour field; *(b)* - n=1.3(10^{14}) (m^{-3}); *(c)* - n=4.4(10^{14}) (m^{-3}); *(d)* - n=2.0(10^{15}) (m^{-3}); *(e)* water vapour volume fraction obtained for this study.

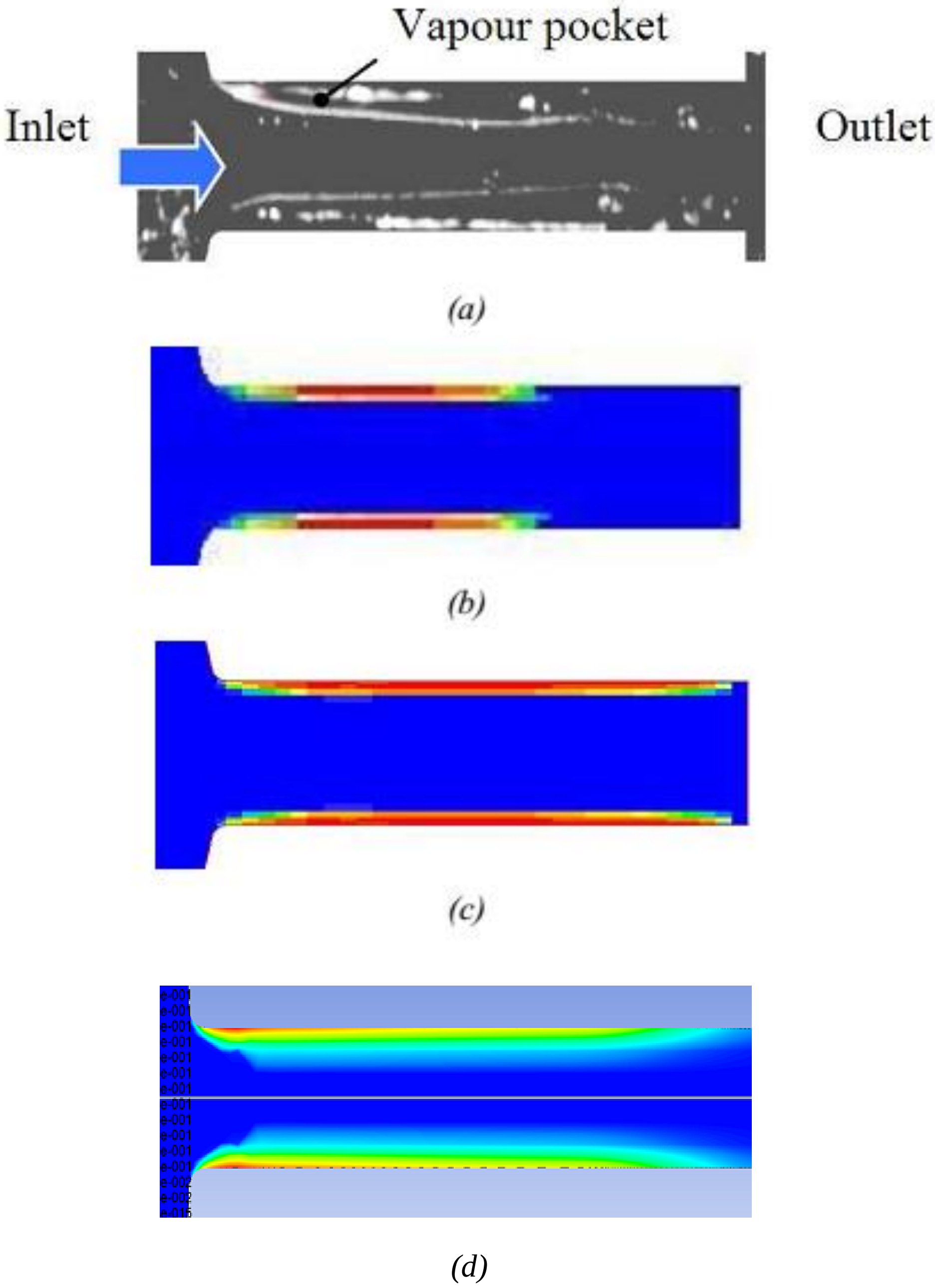
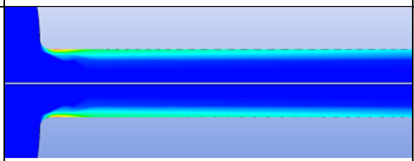
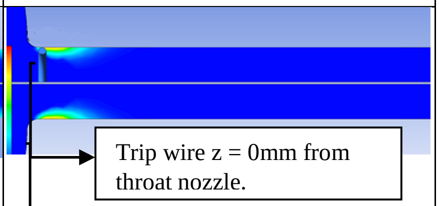
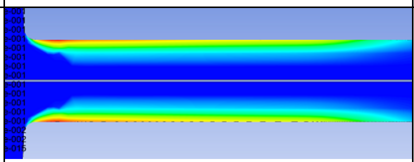
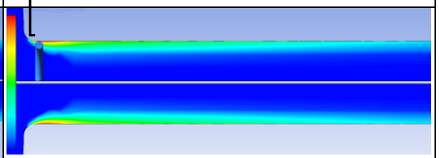


Figure 4.14: Pattern of super-cavitation flow (photograph by Roosen (1996) *(a)* in comparison with the results of numerical predictions cavitation pockets using $n=1.3(10^{14})$ (m^{-3}) *(b)*, $n=4.4(10^{14})$ (m^{-3}) *(c)*, water vapour volume fraction obtained for this study *(d)*.

4.4 COMPARISON BETWEEEN UNMODIFIED DIESEL INJECTOR NOZZLE MODEL AND MODIFIED DIESEL INJECTOR NOZZLE MODEL.

In this chapter, the modified model's simulation result was compared with the unmodified model's simulation result to study the effects of the modification. Table 4.1 shows the comparison between those two models.

Table 4.1: Water vapour volume fraction contour of unmodified and modified Diesel injector nozzle

Flow	Unmodified	Modified
Inlet Cavitation		
Super-Cavitation		

From the Table 4.1, the water vapour length during the inlet cavitation flow was reduced in the modified model, same goes to the water vapour length during super-cavitation flow in modified model.

According to Sato & Saito (2001), the trip wire attached at the throat of the circular orifice, disturbed the flow by increasing the unsteadiness of the flow. Thus, the frequency of the cloud-like cavity shedding increases. This phenomena was caused by the shortening of vortex formation interval on the separated shear layer. Furthermore, the increased frequency cause the shortening of separation bubble length.

For rectangular model of Diesel injector nozzle, the same phenomena observed. The attached cavity length reduces, due to the high frequency of cloud-like cavity shedding process caused by the trip wire attachment. The turbulence intensity at the outlet is the same as the unmodified model. Only the frequency of shedding affected, but the amount of bubble coalesced and burst at the outlet still the same for modified model. Since the length of attached cavity in the modified model reduced for both flow, the hydraulic flip can be prevented.

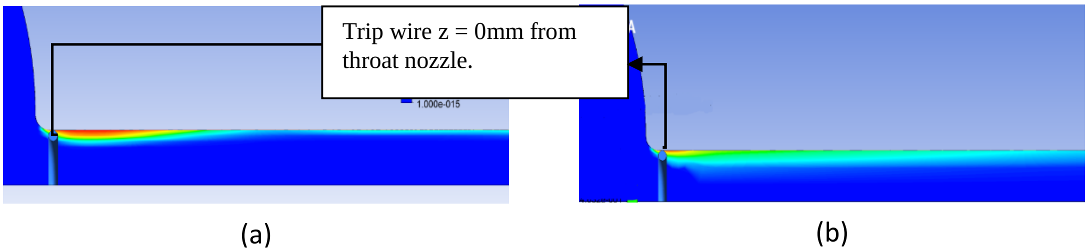


Figure 4.15: Water vapour volume fraction contour for the super cavitating flow using different mesh element sizes; (a) mesh 4; (b) mesh 2.

Contour produced by Mesh 4 has a little difference from the contour of mesh 2. The mesh 4 has a more accurate result compared to mesh 2. But the time taken to complete the simulation was nearly 10 hours.

CHAPTER 5:

CONCLUSION & RECOMMENDATION

5.1 CONCLUSIONS

As a conclusion, this project deals with the production of cavitation inside the diesel injector nozzle which increases the efficiency of the atomization process. The atomization process is very important in diesel engine. The condition and parameters that contribute to hydraulic flip is studied. To prevent hydraulic flip in the rectangular model of Diesel injector nozzle, attachment of trip wire introduced in the nozzle. The trip wire alter the characteristic of the separated shear layer by changing periodic behavior of shedding process. The shedding frequency of cloud-like cavity from the attached cavity increased by the attachment of trip wire. Higher frequency relates to shorter length of cavitation area. This study is within capability of a final year student to be executed with help and guidance from the supervisor and the coordinator.

5.2 RECOMMENDATION

Recommendation for this study will be vary the diameter and thickness of the trip wire attached in the nozzle. For the future study, the location of the trip wire attached at nozzle can be varied within the nozzle throat length. Last recommendation to carry this study in future is to conduct experimental study on effect of trip wire inside the rectangular model of diesel injector nozzle to verify the simulation result.

6.0 REFERENCES

Arcoumanis, C. F. (1999). Investigation of cavitation in a vertical multi-hole diesel injector. *SAE Paper*.

Ashley, S. (1997). Diesel cars come clean. *Mechanical Engineering, vol. 8*, p. 1.

B., B., G., C., G., H., J., A. R., & R., S. S. (2009). Cavitation and Hydraulic Flip in the Outward-Opening GDi Injector Valve-Group . *SAE International*, 1-9.

Bergwerk, W. (1959). Flow pattern in diesel nozzle spray holes. *Proc. Inst. Mech. Engrs*, 173.

Brennen, C. (1995). Cavitation and Bubble Dynamics. *Oxford University Press*.

Karasawa, T. (1992). Effect of nozzle configuration on the atomization of steady spray. *Atomization and Sprays*, 411-426.

Lecoffre, Y. (1999). Cavitation: bubble trackers. *Rotterdam*, 399.

Martynov. (2005). Numerical Simulation of the Cavitation Process in Diesel Injector Nozzle. *The University of Brighton*.

Nurick, W. H. (1976). Orifice cavitation and its effects on spray mixing. *J. Fluids Engng*, 98.

P., S. D., & L., C. M. (2001). The internal flow of diesel fuel injector nozzles: A review. *International Journal of Engine Research*, 1-20.

Ranz, W. (1958). Some experiments on orifice sprays. *Can. J. Chem. Engng*, 175.

Roosen, e. a. (1996). Untersuchung und Modeleirung des transienten Verhaltens von Kavitationserscheinungen bei ein- und mehrkomponentigen Kraftstoffen in schnell durchstromten Dusen. *Report of the Institute for Technical Thermodynamics*.

Sato, K., & Saito, Y. (2001). Unstable cavitation behaviour in circular-cylindrical oriffice flow. *CAV2001*.

Soteriou, C. A. (1999). Further studies of cavitation and atomization in diesel injection. *SAE Paper*.

Sou, A. H. (2007). Effects of Cavitation in a Nozzle on Liquid Jet. *Intl. J. Heat Mass Trans*, 17-18.

7.0 APPENDIX

Skewness is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below 0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.