NOZZLE ATOMIZATION IMPROVEMENT THROUGH CHANNEL REDESIGN

By

Athira Binti Ahmad Zolkiffli

14800

Dissertation submitted in partial fulfillment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2015

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CONTENTS

LIST OF FIGURES

LIST OF TABLES

CERTIFICATION OF APPROVAL

NOZZLE ATOMISATION IMPROVEMENT THROUGH CHANNEL REDESIGN

By

Athira Binti Ahmad Zolkiffli

14800

A project dissertation submitted to the

Mechanical Engineering Program

Universiti Teknologi PETRONAS

In partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL)

Approved by,

____

(Dr. Rahmat Iskandar Shazi Shaarani)

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.

____

ATHIRA BINTI AHMAD ZOLKIFFLI

ACKNOWLEDGEMENT

First thing first, all praise to Allah for giving me the opportunity to learn from this project and to be given the strength to finish this first phase report and to submit it before the deadline. A lot of reading needed to be done in order to complete the report especially the literature review section.

A huge appreciation to my supervisor, Dr. Rahmat Iskandar Shazi Shaarani for the guidance he provided to finish this report. With his knowledge he has helped me by giving enough explanation for me to understand this project better.

Also millions of thank you to my colleague Mr. Satishwaran whom have been helping a lot in gathering understanding the conceptual of the project and helping in further steps of the project.

ABSTRACT

The aim of this article is to study the effect of cavitation on diesel spray behavior and make few modifications to find a better atomization of the spray. Many factors can contribute to the atomization characteristics, such as the pressure at the exit, the flow rates and momentum flux. With that in mind, this research is to identify the characteristic and to come up with a method to observe the atomization and to find a way to ensure good atomization. In this paper, simulations were first validated using experimental results from past papers and then further modified to match the objective of this research. Key parameters observed are pressure, velocity and cavitation behavior of the fuel flow inside the nozzle before exiting.

The findings for the validation section is in agreement with the selected paper that runs the experimental setting of the simulation. However, the new design suggested is not following the expected result.

CHAPTER 1

INTRODUCTION

In this current world, many are racing to save the environment as they can see that human’s daily activities are harming mother earth. One small part that contribute to that is emissions from vehicles that roams the street at all times. Reduction of emissions will releases less amount of harmful waste to the world. One of the approach that can contribute to it is to make a better emission engine. A number of factors are identified in making a lower emission engine one of them is to ensure good combustion happens inside the engine.

1.1. Background

In this project, the focus is on the fuel injector, specifically at the flow channel of the fuel in the injector. It was suggested that, with the change in geometry of the channels, more optimized injection can be achieved. Some basic designs of the nozzle at the end of the injector can cause cavity between the fuel that flows through and the wall of the nozzle. The gaps might be the cause of the fuel to be injected in a high speed jet-like flow, rather in small droplets that have better air fuel ratio mixture that can lead to better performance of the engine.

Fuel injection is a system to supply fuel into an internal combustion engine. It replaced the former fuel supply system, carburetors. The major difference between diesel and gasoline is the way these explosions happen. In a gasoline engine, fuel is mixed with air, compressed by pistons and ignited by sparks from spark plugs. In a diesel engine, however, the air is compressed first, and then the fuel is injected. Because air heats up when it's compressed, the fuel ignites. Fuel injection generally increase engine fuel efficiency with the improved cylinder-to-cylinder fuel distribution of multi-point fuel injection, less fuel is need for the same power output. Exhaust emissions are cleaner because, the more precise and accurate fuel metering reduces the concentration of toxic combustion byproducts leaving the engine.

The system is controlled electronically using the Engine Control Unit (ECU) that will determine how much fuel needed to be supplied to achieve certain amount of power. From there, the ECU will signal the fuel injector to open the valve and spray the needed amount of fuel by controlling the period of opening the valve. Previously, in a diesel engine, there will be a separate chamber shaped just to swirl the compressed air and improved combustion, it was known as the indirect injection.

Many research has previously been done to investigate fuel injectors C. Baddock (1999), F. Payri et al. (2004) and R. Payri et al. (2008). Here, the main perspective is to look at how the cavitation affect the exit flow of fuel and connect it with atomization.

1.3. Problem statement

A diesel engine depends on compression of the fuel to reach certain temperature to ignite and cause movements, however to ensure a smooth combustion, the chamber must have a good air fuel mixture. It can happen when the fuel droplets have the most surface contact with the air. To achieve that, the fuel droplets must be spherical as it is the geometry with largest surface area. In order to get the most contact with the surface, the droplets coming out from the nozzle must be as small as possible.

A non-suitable channel for the flow of the fuel from the fuel rail to the injector nozzle can cause the fuel to detach itself from the wall of the flow channel. When they are flowing out from the nozzle the pressure will drop and the nozzle will not be able to produce a desired mist size that will produce a single jet-like output that will significantly result in better engine performance.

Figure 1. 1 Comparison between a single jet-like flow and a better executed mist

1.5. Objective & Scope of Study

Here are the objectives to be achieved upon completion of this project:

- To observe in simulation the effect of the nozzle atomization with changes in geometry of the flow channel
- To model flow inside a fuel injector
- To identify any other geometry of the nozzle for better atomization

All assumptions and results obtained from this project are limited to the study of nozzles only. All parameters for calculations for fluid flow will be assumed with respect to diesel only. There is no experimental method done, only simulation. All experimental values are obtain from previous research and are used to validate the simulations only.

CHAPTER 2

LITERATURE REVIEW & THEORY

2.1. Atomization

According to definition from a paper written by Karim (2000) atomization of the liquid fuel in diesel sprays is the disintegration process of a high pressure liquid jet exiting from a small circular orifice of the injector nozzle.

2.2. Effect of cavitation in nozzle flow

With cavitation along the flow of the fuel from the fuel rail to the nozzle, it can create certain amount of turbulence that can leads to lowering the coefficient of discharge (C_d) thus create cavity between the wall of the fuel channel and the fluid flow. This cavitation will cause a less smooth combustion that will of course affect the exhaust emission as mentioned by Payri (2008). Cavitation does not only controls but also enhances atomization with the bubble collapsing process identified as the main factor responsible for the liquid core disintegration.

To obtain a better combustion process, it is desired to have good spray characteristics that are strongly influenced by the flow inside the injection nozzle as the paper by Payri et al. (2005). The study of the injector nozzle included many perspectives such as the nozzle, the flow channel or even the choke valve used to control the flow of fuel. In this research, it will be focused on the design of flow channel. All the turns and geometry in the channel are vital in ensuring how the fuel will flow.

From Gannipa (2001), the setting of the experiment was done in a transparent scaled up diesel nozzle and the results was obtained through observations on the measurement devices and camera placed in the settings to understand the cavitation. In this experiment, the manipulated variables are the angles of the exit nozzle and they are expecting to see how the flow behaves when encountered a different angle.

It is important to correlate the results and develop an effective means to predict cavitation inception inside an injector. Under cavitation condition, the cavitation number proves to be a more useful tool than Reynolds number in relating the phenomenon with the discharge coefficient of a nozzle as mentioned in Sukantha (2014). The flow is more closely associate with the Reynolds number under non-cavitation conditions.

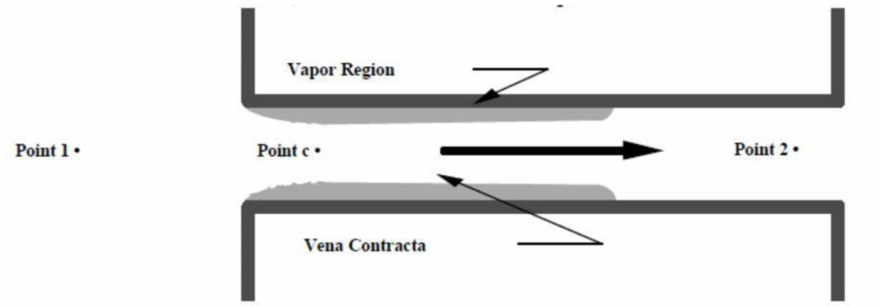


Figure 2. 1 Schematic drawing of a cavitating nozzle

Cavitation number (K) suggested by Nurick (1976) is given by the Equation 2.1. Point 1 is the far upstream condition, point 2 denotes the downstream of the nozzle and point c denotes the narrowest point of contraction. P_v is the vapour pressure of the fluid. The cavitation number given by Nurick (1976) was later used by many other research.

$$\boldsymbol{K = \frac{P_1 - P_v}{P_1 - P_2}} \qquad \textbf{Equation 2.1}$$

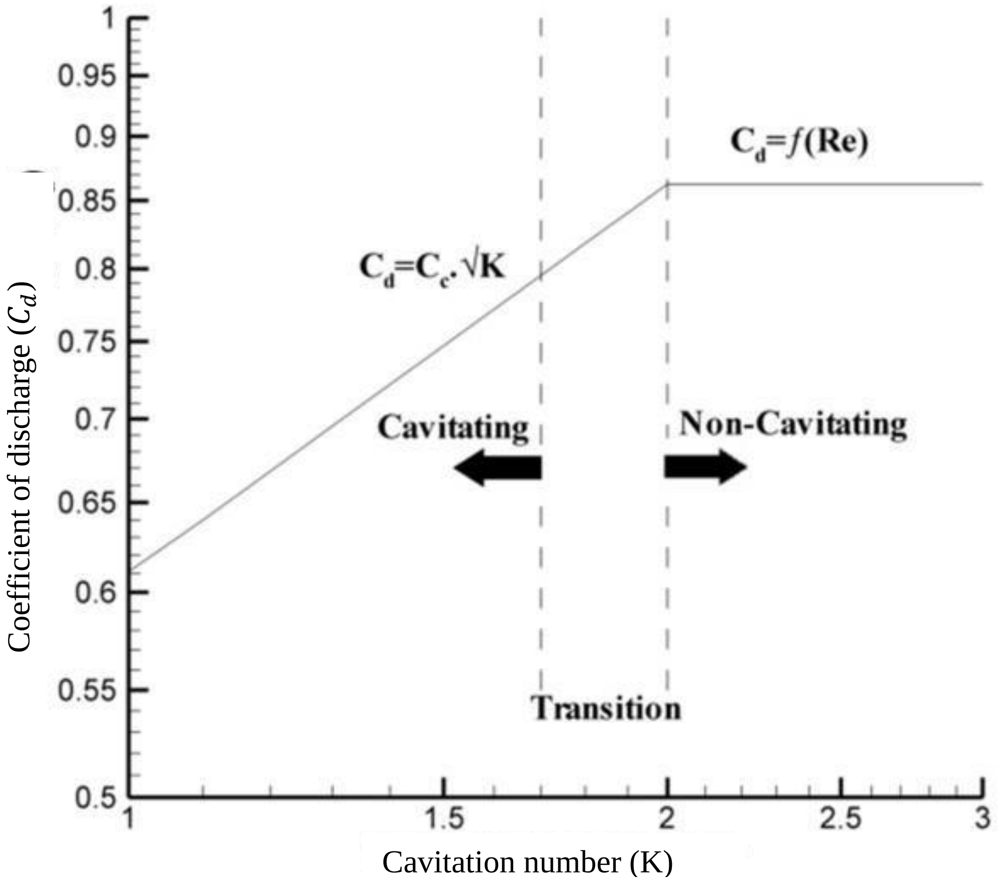
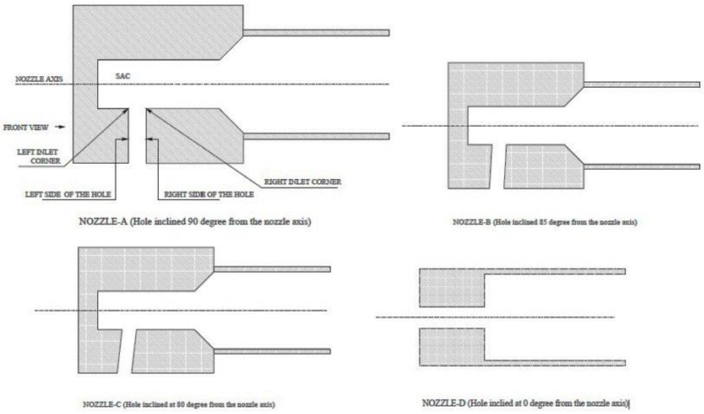


Figure 2. 2 Cavitation numbers against the coefficient of discharge

In the Figure 2.2, the cavitation numbers are plotted against the discharge coefficient. The cavitating region is correlated by the Nurick's theory whereas the non-cavitating region is a function of Reynolds number. The transition of cavitating to non-cavitating phase varies with different nozzle sizes and different fluid properties. The transition of cavitating to non-cavitating regime occurs at lower cavitation number for the low temperature fluid than the higher. According to Nurick's study (1976), the coefficient of discharge of a cavitating nozzle is directly proportional to the square root of the cavitation number (K) and is shown by the Equation 2.1. Here, C_c is the contraction coefficient of the nozzle at a point c which is calculated to be π/(π+2) for sharp nozzles.

Shown below in Figure 2.3 are the designs they come up with,



NOZZLE-A (Hole inclined 90 degree from the nozzle axis)

NOZZLE-B (Hole inclined 85 degree from the nozzle axis)

NOZZLE-C (Hole inclined at 80 degree from the nozzle axis)

NOZZLE-D (Hole inclied at 0 degree from the nozzle axis)

Figure 2. 3 Model nozzle of various hole angles

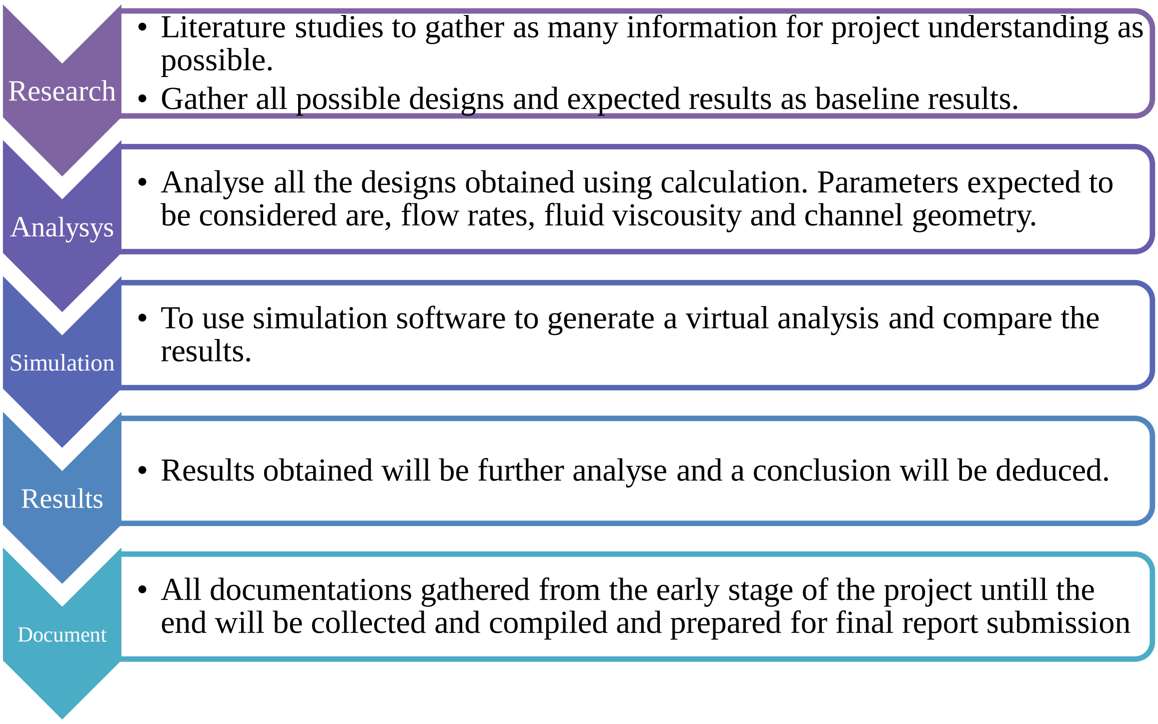
They have concluded that the cavitation (bubbles) are growing in the developed to a bigger cloud like structures.

CHAPTER 3

METHODOLOGY

In this section, the methods used will be explained. All the steps taken to complete the project are documented in this section as well as the suggested timeline explained in the Gantt chart.

3.1. Research Methodology



- Literature studies to gather as many information for project understanding as possible.
- Gather all possible designs and expected results as baseline results.

- Analyse all the designs obtained using calculation. Parameters expected to be considered are, flow rates, fluid viscousity and channel geometry.

- To use simulation software to generate a virtual analysis and compare the results.

- Results obtained will be further analyse and a conclusion will be deduced.

- All documentations gathered from the early stage of the project untill the end will be collected and compiled and prepared for final report submission

Figure 3. 1: Project Flowchart

3.2. Assumptions and procedures

Fuel used for this study is diesel fuel and all the properties of diesel fuel are listed in the following table, the values will later be used in simulation of the flow

Table 3. 1 : Test Fuel Properties

Item	Value	Item	Value
Compressibility of vapour	$6.53 \times 10^{-7}\ s^2\ /\ m^2$	Vapour pressure	5400 Pa
Compressibility of liquid	$6.1 \times 10^{-7}\ s^2\ /\ m^2$	Saturated liquid pressure	113499 Pa
Saturated vapour density	$0.35\ kg\ /\ m^3$	Dynamic viscosity of vapour	5.953×10^{-6} Pa-s
Saturated liquid density	$818\ kg\ /\ m^3$	Dynamic viscosity of liquid	2.372×10^{-3} Pa-s

All modeling and simulation activities were done using Solidworks software and ANSYS. However, other meshing software are used sometimes to validate meshing techniques. The mesh fineness can be chosen based on previous researches. The approximation of number of cells needed to be 500000. Payri et al. (2005). As mentioned earlier, the fluid that will be used in this simulation is diesel and all parameters are as in_Table 3.1. To ensure that the model is validated, CFD simulation results from Winklhofer (2001). Baseline simulation will be conducted using all the dimensions that are stated in the paper.

To validate the simulation, existing experiments was used to model. Using data from paper written by Winklhofer (2001) the model was made and created. Below is the dimensions provided from the research.

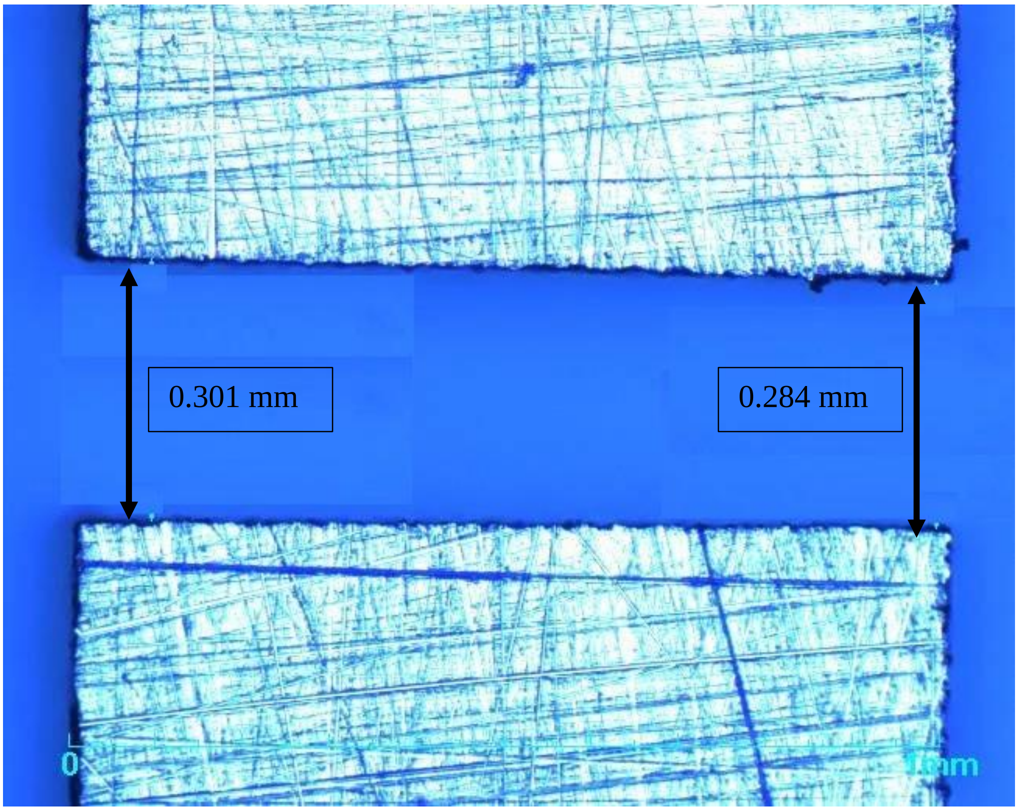


Figure 3. 2 Experimental geometry from Winklhofer

The view if from the XY plane and was assumed one cell thick in the Z direction. After the model is validated, further changes to the model such as the internal geometry of the flow channel will be introduced and same boundary conditions will be applied. However, the same aspect from the experimental data will be taken into account that is, according to Payri et al. (2005), the mass flow rate for different pressure condition and the cavitation number. The simulation will also be done by assuming that the needle is in its maximum lift position.

One of the factors that we are looking at is the cavitation of the fuel flow in the channel and is partly contributed by the coefficient of discharge (C_d) that can be given by

$$\boldsymbol{C_d = \frac{\dot{m}_f}{(A_o)(\sqrt{2(\rho_l)(\Delta P})}} \qquad \textbf{Equation 3.1}$$

$$\boldsymbol{C_v = \frac{u_{eff}}{u_{th}}} \qquad \textbf{Equation 3.2}$$

$$\boldsymbol{C_a = \frac{A_{eff}}{A_{geo}}} \qquad \textbf{Equation 3.3}$$

Where,

$$\Delta\boldsymbol{P = P_{inj} - P_{back}} \qquad \textbf{Equation 3.4}$$

ρ_l = Density of liquid-fuel density.

$$\boldsymbol{\dot{m}_f = \int_{A_o}^{\infty} u.\rho.dA} \qquad \textbf{Equation 3.5}$$

$$\boldsymbol{u_{eff} = \frac{M_f}{\dot{m}_f}} \qquad \textbf{Equation 3.6}$$

$$\boldsymbol{A_{eff} = \frac{{m_f}^2}{\rho_f M_f}} \qquad \textbf{Equation 3.7}$$

3.3. Gantt Chart & Key Milestone

Table 3. 2: Project Gantt chart

No.	Detail/work	Last Semester	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Selection of Project topic	■															
2	Preliminary research work on nozzle atomization	■															
3	Research work on cavitation on the flow of fuel	■															
4	Submission of Extended Proposal	■															
5	Proposal Defense	■															
6	Modeling and validation of model	■															
7	Submission of Interim Draft Report	■															
8	Submission of Interim Report	■															
9	Simulation for Validation Starts						■	■	■	■	■	■					
10	Submission of Progress Report										■						
11	Simulation of New Design												■	■	■	■	■
12	Pre-Sedex Poster Presentation													■			
13	VIVA presentation																■
14	Dissertation Submission																■

Table above suggest all deadlines of the first part of the project. Since there will be two parts of the project, all research are expected to be done during the first part of the project so that all project execution can be done in the second part next semester.

The simulation for the new design could is still on going and are not done because it takes more time than expected.

Table 3. 3: Latest Project Milestone

No.	Detail/work	Last semester	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Selection of Project topic	🟥															
2	Proposal Defence	🟥															
3	Submission of Interim Draft Report	🟥															
4	Submission of Interim Report	🟥															
5	Submission of Progress Report										🟥						
6	Pre-SEDEX Poster Presentation													🟥			
7	Dissertation draft submission																🟥
8	VIVA Presentation																🟥
9	Final Dissertation Submission																🟥

Current point of the project. It highlights all important period of part one of the project. As the project commence, the milestone table will be update accordingly.

CHAPTER 4

RESULTS AND DISCUSSION

This section will discuss about the execution of the project where the modelling and simulation started.

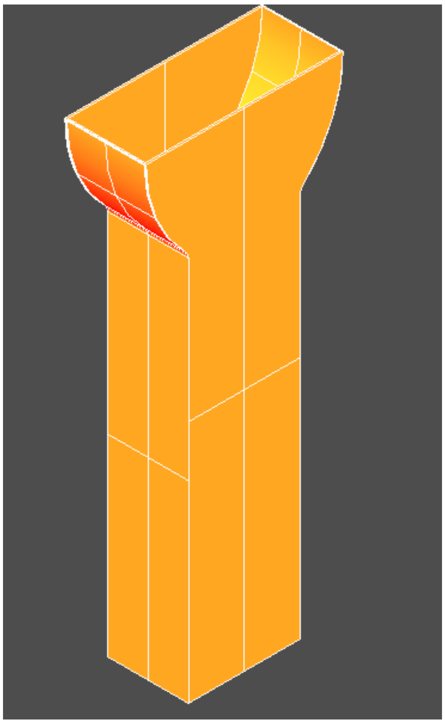


Figure 4. 1: First draft model in Solidwork

Further models are not available as there was some problem to export the models from the modeling software. Meshing attempts using ANSYS also did not achieve satisfactory as there are quite a number of functions that are not working in the software version used. Earlier when the project started, it was aimed to have the validation done by the submission of this report. However due to time constraints and a few hick ups here and there, the validation of the simulation was not available as of now.

When further research and practice of the software was done, it was realized that the 3D model is complicated to be meshed and it does not show expected result. It was then switched to a 2D asymmetrical model based on the model above.

By using experimental values from Winklhofer (2001) research, the validation of the simulation was started.

4.1. Validation of Simulation

The model obtained from the research paper selected was remodeled in Solidwork and later meshed in FLUENT software. There is one model used in the paper and dimensions are as stated in Chapter 3

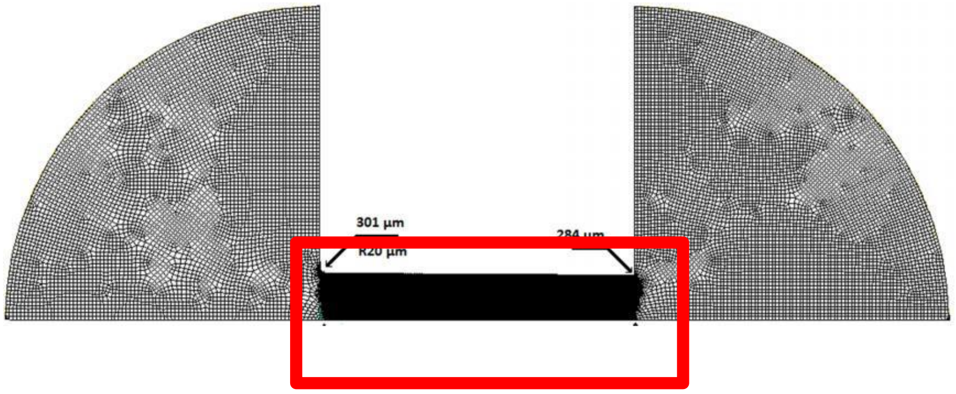


Figure 4. 2 The meshed geometry

The red box specify the critical part that are observed in this experiment because from Winklhofer (2001), the paper had only the images from that section of the model. So it is fit to have only this section being meshed extensively to get more accurate result.

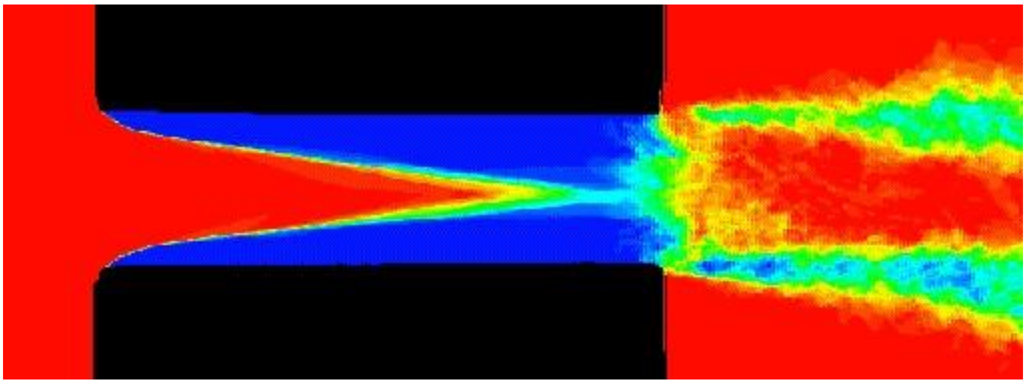


Figure 4. 3 Experimental vapor generation image

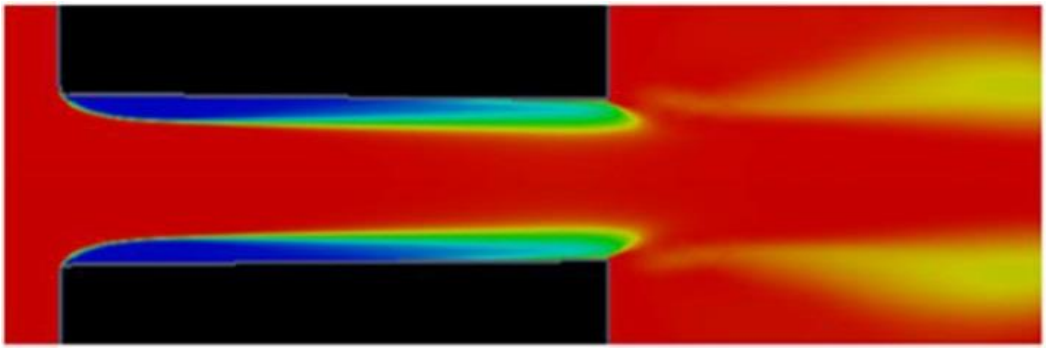


Figure 4. 4 Vapor generated in the simulation

The void fraction contour obtained from the simulation is compared with the cavitation probability distribution obtained from the experiments and shown in the Figure 4.3 and Figure 4.4. The vapour prediction is close but experimental measurements have shown a greater amount of vapour. One of the reasons for the lower vapour prediction can be due to the difference in fuel temperature and properties. The properties of diesel used in this study are at 313K as the operating temperature is not mentioned in the experimental investigation.

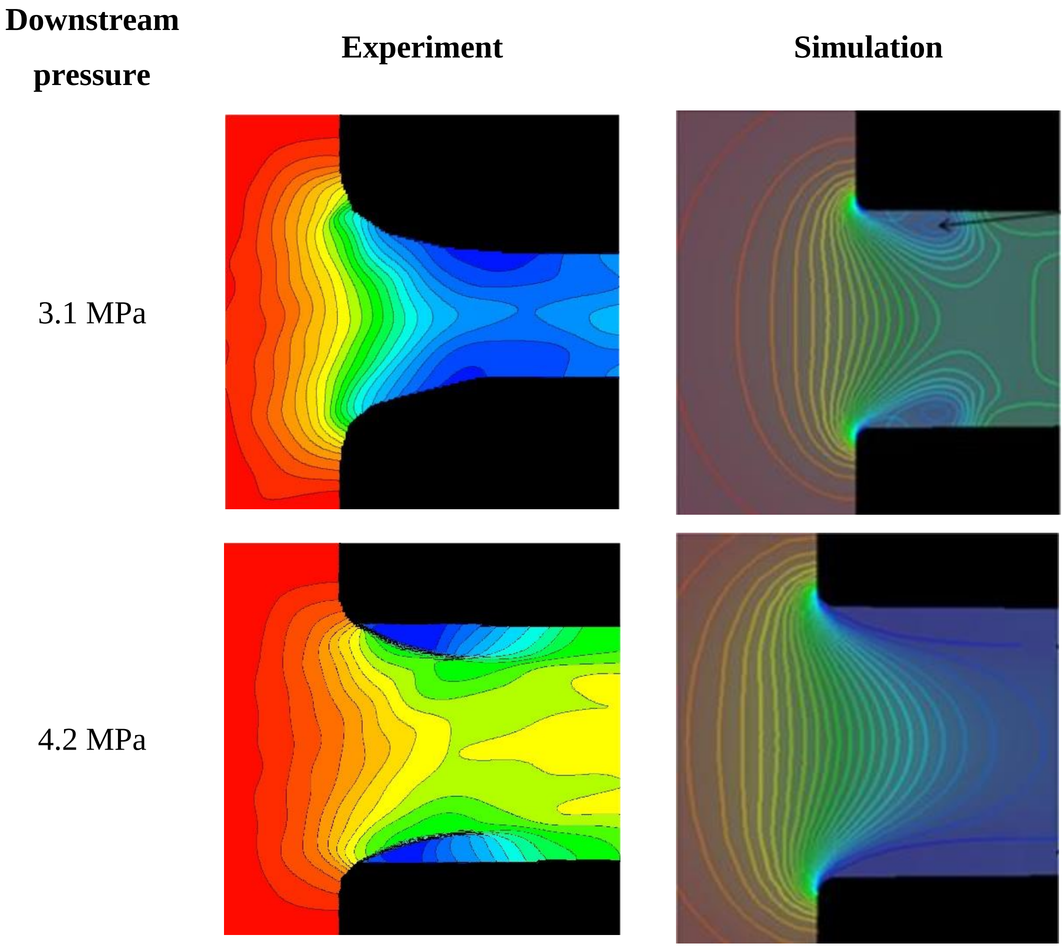


Figure 4. 5 Comparison of pressure contour

The Figure 4.5 describes the cavitation inception and fully developed cavitation cases. In the top figure, formation of a low-pressure region near the inlet is visible and limited to the inlet of the nozzle. This region creates a recirculation zone near the wall due to the drop in pressure inside the nozzle. This drop in pressure below the saturated liquid pressure generates a two-phase cloud near the wall and is termed cavitation inception. At this point, the cavitation is limited to the inlet area of the nozzle and the cavity length is minimal. With a further decrease in downstream pressure, the recirculation zone extends to the outlet of the nozzle and creates a cloud of cavitation attached to the wall. In the bottom part of the Figure 4.5, the low pressure region is visible and extends along the nozzle showing a fully developed cavitation region. The pressure contours obtained from the simulations are in good agreement with the experimental images.

4.2. Proposed design for better atomization

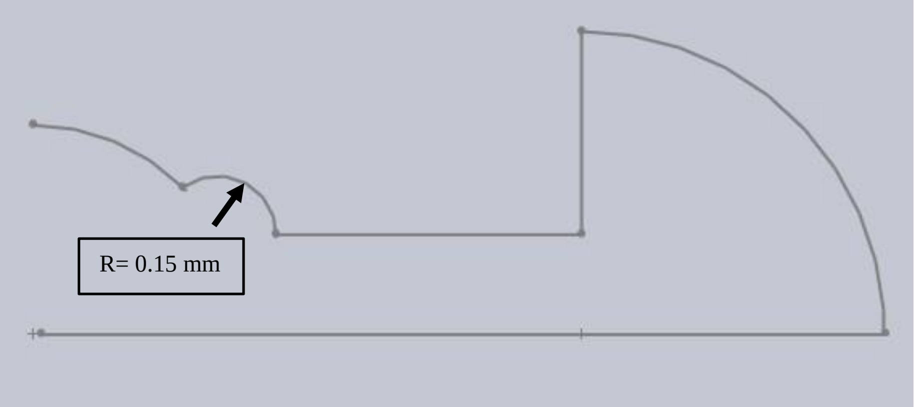


Figure 4. 6 Proposed design to observe the behavior of cavitation

In Figure 4.6, shows the proposed design to help cavitation occurs and in consequence better the atomization. The design showed above is design in such manner with expectation to shift the cavitation region to further back to have the cavitation region as far to the exit of the nozzle but still not causing hydraulic flip.

However, as mentioned in chapter 3, the simulation could not be finished due to meshing problem and the first results obtained from the simulation is way out of the range of results expected to be.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1. Conclusion

The two-dimensional simulations performed with the nozzle given by Winklhofer (2001) were in close agreement with the simulations. The pressure contours and cavitation probability distributions from the numerical results were in close agreement with the experiments. The comparison of the velocity profiles were slightly over predicted in simulation but the nature of the flow was closely captured. The mass flow rate from the simulations was very close to the experimental values and the incidence pattern of cavitation is accurately captured in the simulations.

The simulations performed in this study obtained grid independent and accurate results. A higher order scheme used in the simulations was able to capture the shock behaviour inside the nozzle. The two-dimensional simulations were in close agreement with the theory and the experiments. These simulations were close to capturing the behaviour inside the real size injectors and can be useful in developing future fuel injectors.

5.2. Recommendations

1. The hexahedral mesh is difficult to generate for complex domain as compared to square mesh but they converge faster and produce more stable results. The current solver issues with high-density ratio and compressibility and instability due to poor mesh was evident. Improvement in stability with a wide range of meshes can be significant in producing efficient results.

2. The thermo physical properties for a single component fuel at equilibrium are easily available from the NIST online resources but the multicomponent fuels like diesel and gasoline are not present in open literatures. Adding thermo physical database for multicomponent fuels will enable further investigations of effects due to temperature and transport properties in internal nozzle flows.

3. A needle motion adds significant turbulence to the incoming flow and can produce string cavitation. Implementing needle motion to the existing study could add significant value to the solver and can be used more closely to develop fuel injectors for future diesel engines.

4. To further analyse the atomization, a simulation to see the size of droplets of the fuel can be generated. With that better perspective of the atomization can be obtain.

CHAPTER 6

REFERENCES

Badock, C., Wirth, R., Fath, A., & Leipertz, A. (1999). Investigation-of-cavitation-in-real-size-diesel-injection-nozzles_1999_International-Journal-of-Heat-and-Fluid-Flow. International Journal of Heat and Fluid Flow, 20.

Ganippa, L., Bark, G., Andersson, S., & Chomiak, J. (2001). Comparison of Cavitation Phenomena in Transparent. CAV 2001: Fourth International Symposium on Cavitation. Conference or Workshop Item (Lecture). California Institute of Technoloogy. Retrieved from http://resolver.caltech.edu/CAV2001:sessionA9.005

Payri, F., Bermúdez, V., Payri, R., & Salvador, F. J. (2004). The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles. Fuel, 83(4-5), 419-431. doi: 10.1016/j.fuel.2003.09.010

Som, S., Ramirez, A. I., Longman, D. E., & Aggarwal, S. K. (2011). Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions. Fuel, 90(3), 1267-1276. doi: 10.1016/j.fuel.2010.10.048

Sou, A., Hosokawa, S., & Tomiyama, A. (2007). Effects of cavitation in a nozzle on liquid jet atomization. International Journal of Heat and Mass Transfer, 50(17-18), 3575-3582. doi: 10.1016/j.ijheatmasstransfer.2006.12.033

Suh, H. K., & Lee, C. S. (2008). Effect of cavitation in nozzle orifice on the diesel fuel atomization characteristics. International Journal of Heat and Fluid Flow, 29(4), 1001-1009. doi: 10.1016/j.ijheatfluidflow.2008.03.014

E. Winklhofer, E. K., E. Kerlz, A. Morozov. (2001). Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions. Journal

Ganippa, L., Bark, G., Andersson, S., & Chomiak, J. (2001). Comparison of Cavitation Phenomena in Transparent. CAV 2001: Fourth International Symposium on Cavitation. Conference or Workshop Item (Lecture). California

Institure of Technoloogy. Retrieved from http://resolver.caltech.edu/CAV2001:sessionA9.005

Karim, Z. A. A. (2000). Diesel Injector Nozzle Cavitation, Thesis

Nurick, W. H. (1976). Orifice cavitation and its effect on spray mixing. Journal

Payri, R., Garcia, J., Salvador, F., & Gimeno, J. (2005). Using spray momentum flux measurements to understand the influence of diesel nozzle geometry on spray characteristics. Fuel, 84(5), 551-561. doi: 10.1016/j.fuel.2004.10.009

Payri, R., Salvador, F. J., Gimeno, J., & Zapata, L. D. (2008). Diesel nozzle geometry influence on spray liquid-phase fuel penetration in evaporative conditions. Fuel, 87(7), 1165-1176. doi: 10.1016/j.fuel.2007.05.058

Sukanta, R. (2014). High Speed Flow Simulation in Fuel Injector Nozzles. University of Massachusetts. Journal

CHAPTER 7

Appendix

7.1. First models before switching into 2D asymmetrical model

7.1.1. Cylindrical Nozzle

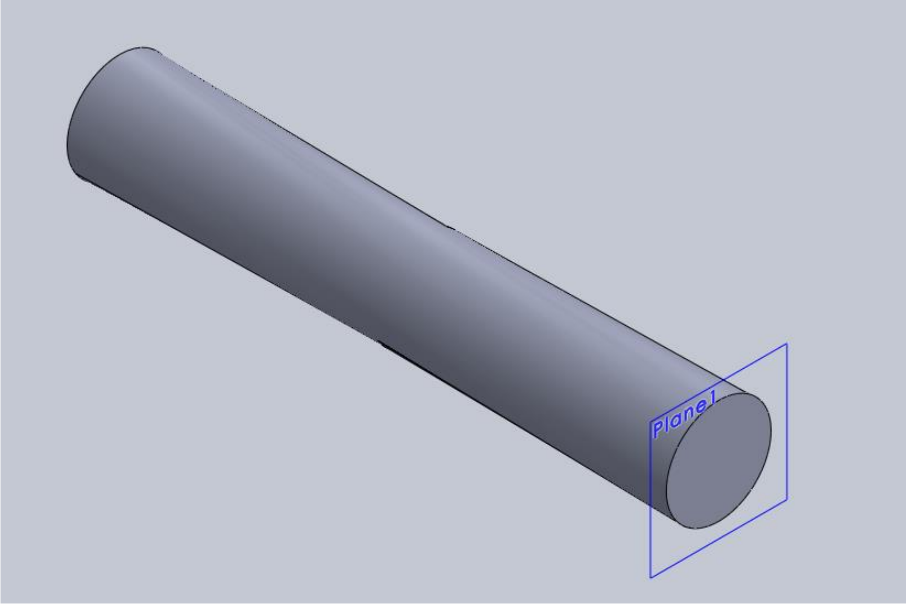


Figure 4. 7: The model of cylindrical nozzle

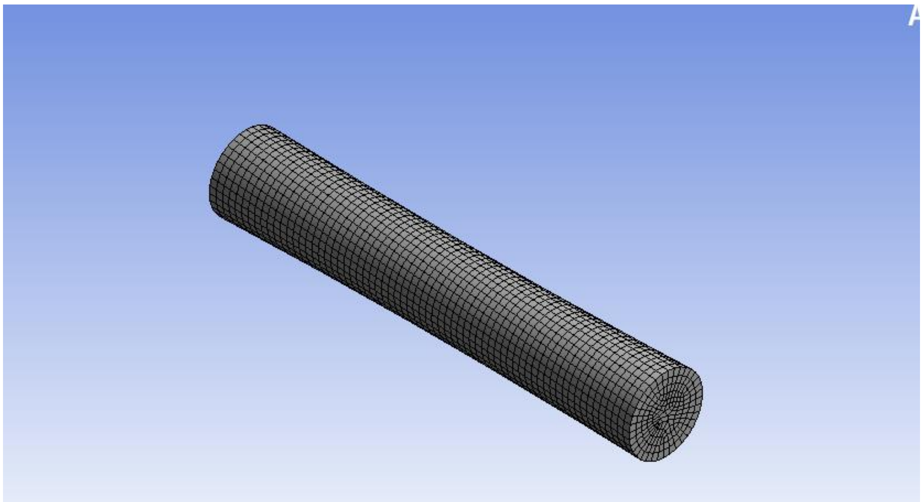


Figure 4. 8: Meshed version of the cylindrical model

7.1.2. Conical nozzle 1

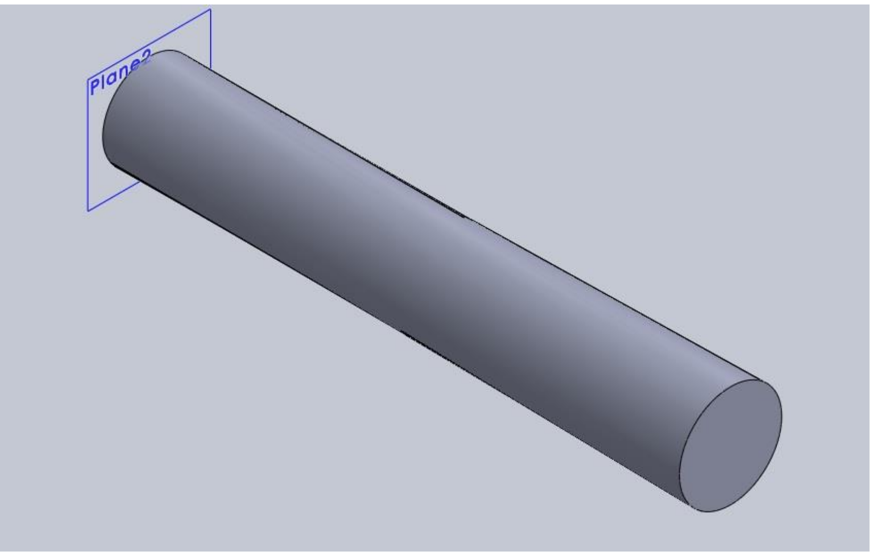


Figure 4. 9: Conical nozzle 1

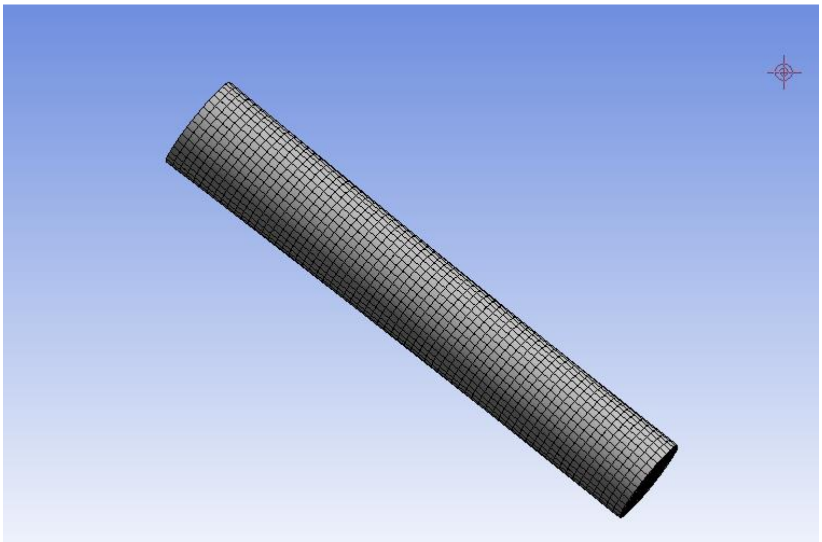


Figure 4. 10: Meshed version of first conical nozzle

7.1.1. Conical nozzle 2

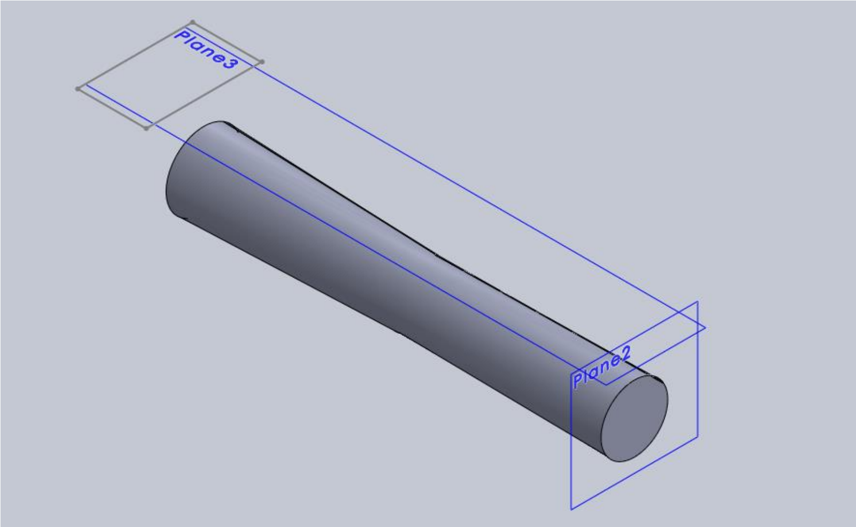


Figure 4. 11: Conical nozzle 2

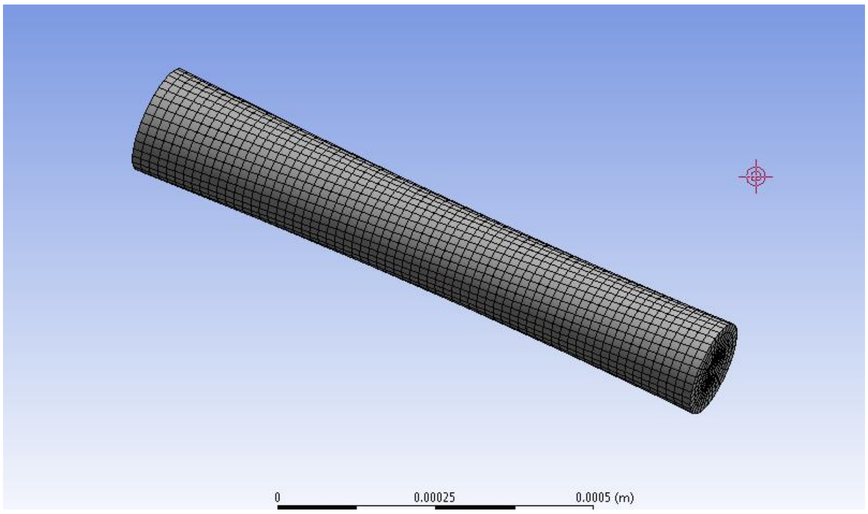


Figure 4. 12: Meshed conical nozzle 2