NUMERICAL ANALYSIS OF DRAG AND LIFT FORCES ACTING ON LONG RANGE PROJECTILES

M. Nasir Uddin [1*], Dr. Golam Mostofa[2] & M. Sohel Rana[3]
[1,2,3]Department of ME, Military Institute of Science and Technology, Dhaka, **BANGLADESH**
Emails: * [1] nasir4725@yahoo.com; [2]mostofa@me.mist.ac.bd; [3]sohel_me@live.com

ABSTRACT

Projectiles play an important role in modern time ammunition and the development of an efficient projectile is very important. The design parameters of a projectile depend on the drag and lift force acting on the projectile. Therefore, a detail simulation is required to understand the projectile performance against the wind. In this article, an numerical investigation of long-range projectiles under different wind conditions is carried out. Three different sizes of projectiles model are used in this Study (105 mm, 122 mm, and 130 mm). The simulation was done using simulation software packages changing the Angle of Attack (AOA, 30°, 35°, 40°, 45°, and 50°) keeping wind velocity and geometry the same. The drag and lift coefficients were obtained from the measured pressure and a projected area of the projectiles. The wind flow effect on the projectiles is also analysed by Ansys. The simulation result shows that the size of the projectile is an important factor that is related with the drag force.

Keywords: Projectile, long range, CFD, Drag Force, Lift Force.

1. INTRODUCTION

Due to the large use of long-range projectiles worldwide, the study of aerodynamic characteristics of projectiles has become one of the most important focuses for its improvement. Within the field of projectile design, there has been a frequent endeavouring to increase the range and precision target hitting of projectiles [Bolokin et al., Suliman et al., and Sahu et al.,]. The increased range is achieved by the performance improvement of the projectiles. However, the forecast of drag coefficients and lift coefficients for diverse formations are crucial. However, various parameters such as fins and jets can be utilized to increase aerodynamic characteristics for better control on maneuvering projectile. Hence, several pieces of research have been carried out by using various smart technology to improve the performance of the projectiles [Sahu et al, Novak et al., and Wessam etal.,]. Self-adaptive ballistic correction is considered as an important parameter nowadays by applying numerous advanced intelligent control [Wessam et al.,]. Besides, stability and optimization of projectiles are the significant factors for the flight performance by reducing drag [Yongie et al., Linj et al.,]. They have noted that inadequate system stability outcomes in the failure of operations. Moreover, the long ranges and precise target hitting by a projectile are projected to be persistently enriched, particularly when a new projectile is developed or an existing projectile is modified or upgraded [Linj et al.,]. Consecutively, the hollow projectile known as a tubular projectile is an important parameter to be considered to improve its performance characteristics greatly [Dali et al.,]. Research on optimal hollow projectile shows that the bow shock wave in front of it results in the projectile drag coefficient reductions abruptly. Furthermore, ballistic projectile precision depends on measurement structures for the location which can be determined by using gravity vector estimation. Therefore, this study forecasts on the investigation of projectiles under different wind conditions. The demonstration is being performed to investigate the effect of design parameters on the system performance of the projectile and its effective flight path.

A. The Necessity of the Study

In recent years the hollow projectile research development has become one of the main projectiles in small calibers artillery researches. The extended firing ranges and impact precision of weapons systems are expected to be constantly improved; especially when new ammunition is developed or when existing ammunition is modified. Aerodynamic bodies such as projectiles, missiles, and rockets generally, undergo deterioration of flight performance by drag and it affects a projectile. The base drag frequently accounts for one-half, or even much more, of the total drag for large calibre ammunition. Reducing the base drag is an efficient and practical way to reduce the total drag of projectile, and increase the range of projectile for up to 30%.

Within the field of artillery techniques, there has been a continual striving to increase the range and precision of field guns. Increased range is achieved either by gun improvements, which even include modifications to propellant charges such that redesign of gun parts is required due to for example increased gas pressure in the barrel, or by improvements in the projectile performance. Improved projectile performance can be achieved in several different ways which to a certain extent can be combined in the same projectile. Base drag contributes generally to a relatively large part of the total drag and depends upon the fact that the base pressure due to the resulting wake flow in the base region is lower than the ambient air pressure.

The coefficient of drag is an important parameter in external ballistics. A 130 mm artillery shell at 943 m/s muzzle velocity in vacuum covers a maximum range of 90.7 km whereas, in the presence of air, its range reduces to 24 km. Therefore, the coefficient of drag plays a vital role in the case of range and depends strongly on the shape of the nose of the projectile. The major work of the flight engineer is to calculate the drag of flight for various speeds, altitudes, and different design configurations and try to analyze, how it can be reduced to increase the performance. It is the main drawback because to go through all that it's very difficult task forces will be different on different configurations for different parameters. For predicting the lift and drag coefficients we need a deliberate study of a projectile under different wind conditions.

Nowadays, both the studies with models and projectiles are being performed to compare the result for verifying the validity of the former. But full-scale experiments are both costly and difficult to perform. For the present study with different projectiles, full-scale experiments will not only be complex and costly but also it would be difficult to record reliable pressure distribution simultaneously on the single as well as group of the projectile as there will be a variation of speeds and direction of the wind with time. The flow around projectiles in the actual environment is very complex and formulation of a mathematical model to predict the flow is almost impossible. Thus, the model study is a must and the results obtained under the simulated condition in the laboratory are found to be quite satisfactory for practical purposes.

B. The Objective of This Research

In the present investigation different types of hollow shape projectile like 105 mm, 122 mm and 130 mm have been taken into consideration. The objectives of the research are as follows:

i. Numerical investigation of the lift and drag force, lift and drag Co-efficient for wind slow over the different types of projectiles.

ii. Recommendation for modification of existing projectile design.

The results will be expressed in non-dimensional parameters, so that it may be applied for different types of prototype projectiles. The findings will enable the engineers to design different special projectiles effectively.

2. SAMPLE PROLECTILES AND ITS CONDITIONS

The research is to be carried out based on a theoretical analysis for a small scale projectile along with the formulation of the mathematical model and numerical analysis. The mathematical model is to be formulated by understanding the wind nature, analyzing the mechanics of projectile-wind interaction, and the interaction of air-flow-aerodynamics. The selected projectile is to be then used for simulating the system performance, optimizing the design parameters, and effective range. The optimized design parameters will be used for developing a numerical model and relationship between lift coefficients and drag coefficients with free stream velocities and angle of attacks with the help of ANSYS software.

Table 1: Conditions for Different Projectiles.

Projectile Size (mm)	AOA (°)	Air Velocity (m/s^2)	Tapping Points
105	30, 35, 40, 45, 50	4.7	30
122	30, 35, 40, 45, 50	4.7	30
130	30, 35, 40, 45, 50	4.7	30

3. MATHEMATICAL MODEL AND SIMULATION

The calculation procedure of finding the lift and drag, for a lift and drag coefficients has been described briefly in this chapter. The air velocity of the air stream is obtained from the anemometer and the projected area of the segmented part of the projectile is calculated through Solidworks modeling. Then the drag (C_D), lift (C_L), and Pressure Coefficient (C_P) are calculated.

Figure 1: Wooden projectiles For Test Consideration

The pressure is measured at the tapping is measured by using Equation 1.

$$P = \Delta h_k \rho_k g \quad (1)$$

Where

P= Pressure

Dh_k = Manometer reading

ρ_k = Density of Kerosene

g= Gravitational Acceleration

The acting force on a single segment (assuming segment 1) is calculated from Equation 2.

$F_1 = P * A_{Projected\,1}$ ....(2)

Then the Total Force acting on the Projectile will be

$F = F_1 + F_2 + F_3$ ....+F_{30} ....(3)

As the air is coming at an angle, therefore, the Total forces will be divided into Horizontal and Vertical direction. If the Angle of Attack is 'α' then the drag and lift force is calculated from Equation 4 and 5.

$F_D = F\cos\alpha$......................(4)

$F_L = F\sin\alpha$......................(5)

The Drag Coefficient (C_D), Lift Coefficient (C_L), and Pressure Coefficient (C_P) is calculated from Equation 6, 7, and 8.

$$C_D = \frac{2 * F_D}{A_{Total} * \rho_{k*} U_\infty^2} \text{ .... } (6)$$

$$C_L = \frac{2 * F_L}{A_{Total} * \rho_{k*} U_\infty^2} \text{ .... } (7)$$

$$C_p = \frac{\Delta P}{\frac{1}{2}\rho_{air} U_\infty^2} \text{ .... } (8)$$

Where, $\Delta P = P - P_0$

P = Static pressure on the surface of the projectile

P_0 = The ambient pressure

ρ_{air} = the density of the air

U_∞=the free stream velocity

A_{Total}= Total Active Projected Area (A_1+A_2+A_3+.........+A_n)

The calculation of the projected area for each segment is calculated from the Solidworks model. The projectiles are segmented into 30 strips and then a 3D drawing was drawn to connect the top and bottom of the projectiles. Then a 2D drawing was drawn on a plane facing the air stream. The area of all segments was not calculated as they can not be seen from the front of the plane facing the wind. Therefore, the pressure difference in those areas is fluctuating due to turbulence and not dependable. Figures 5, 6, and 7 show the projected area measurement by SolidWorks.

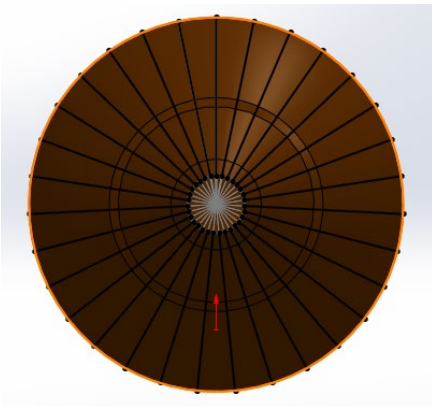


Figure 2: Plan view of the projectile from air direction

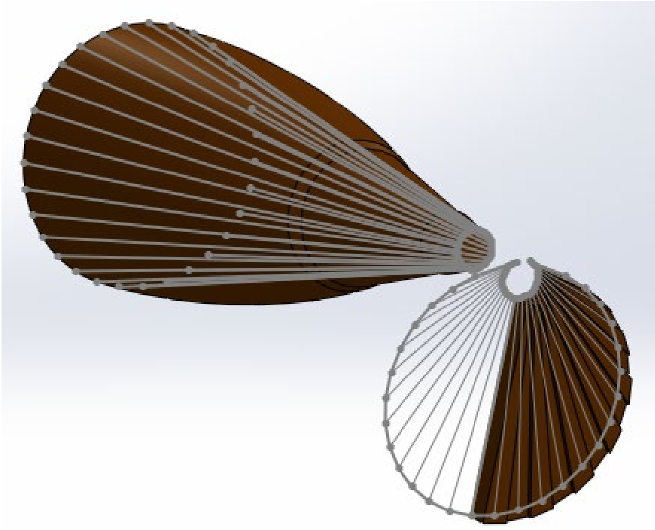


Figure 3: Area drawn from the projection of the segments

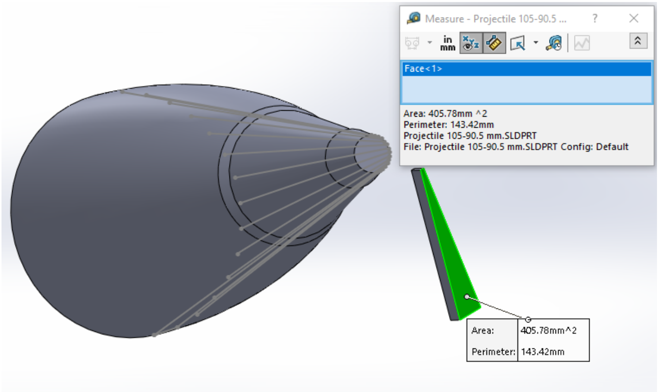


Figure 4: Measurement of the projected area

Computational fluid simulation is done on all the projectile according to the experimental conditions. Ansys software is used to analyse the CFD model. The SolidWorks model made for measuring the projected area is used for simulation. The projectile is considered as a solid domain and outside of it is considered as air domain. The k-e turbulence model is used for solving the problem. The inlet condition was 4.7 m/s air and outlet condition was atmospheric condition similar to experiment. The rest of the surface is considered a wall. Figure 8 shows the geometry of the 105, 122, and 130 mm projectiles, and the mesh file for simulation is shown in Figure 9. The geometry of the Shells with the same dimension was put forward to simulate with scale 1:1. The geometric model of Shell is shown in Figure-8. The Shell model was sketched on the Solid Works 2017 then imported to ANSYS Geometry Module where the computational domain was respect of the model as shown in Figure number 8. The boundaries are chosen at the front, right, left, top and rear up to 10D, 10D, 10D, 10D and 15D respectively from the surface of the model where D is the downstream radius of Shell. In the problem, the platform of pre-processing ICEM, CFD 16.0, unstructured grid was adopted. After that sphere of influence approach was employed to densify gird around the model, so as to improve numerical precision, as shown in figure-9. In the figure-9, the mesh generated is 2143025 elements, 2871030 nodes. The results obtained during the grid dependency tests a finer mesh, made of 2143025 elements, 2871030 numbers of nodes are compared with a course mesh of 192994 elements and 64138 numbers of nodes. The pressure co-efficient found for the finer mesh varies with the course mesh by 0.57% which is concurrent to the independency test.

The air enters into the domain with the velocity of 4.7 m/s. The density of air was 1.225 kg/m3 and viscosity about 1.7894e-05 kg/m-s. At the outlet, pressure outlet condition is applied at the domain. Steady and incompressible flow of air is considered in this Analysis. The solution procedure adopted to solve the CFD model using FVM solver. The default solver settings are selected because pressure based solver is used to solve the steady state problem. An atmospheric pressure is maintained at outlet therefore use default value (0 Pa for gage pressure). In these calculations, the second order upwind scheme based on multidimensional linear reconstruction approach is used. The SIMPLE algorithm for pressure velocity coupling with second order upwind discretization scheme is used to obtain solution for the equations of Momentum, Turbulence Kinetic Energy and Turbulence Dissipation Rate. The target of all discretization techniques in FVM is to develop mathematical model to convert each of terms into an algebraic equation. Once implemented to complete control volumes in a particular mesh, we attain a full linear system of equations that requires to be solved. These computations are carried out using FVM

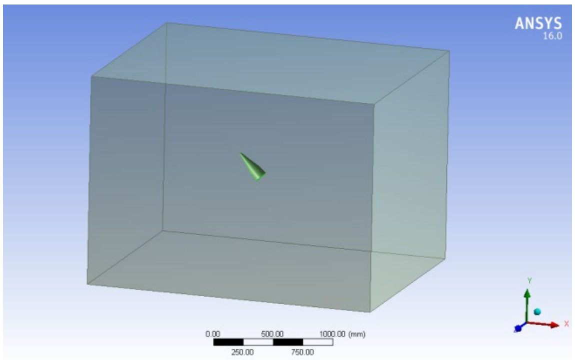


Figure 5: Geometry files of 105 mm projectile for CFD simulation

solver (ANSYS FLUENT 2016), a commercial CFD package with a 3D double precision configuration. The default convergence criterion in FLUENT is

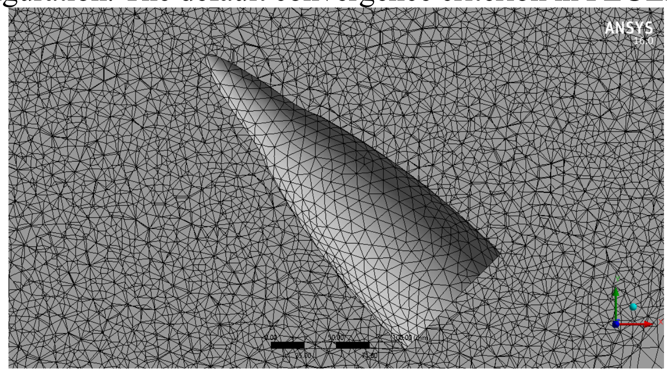
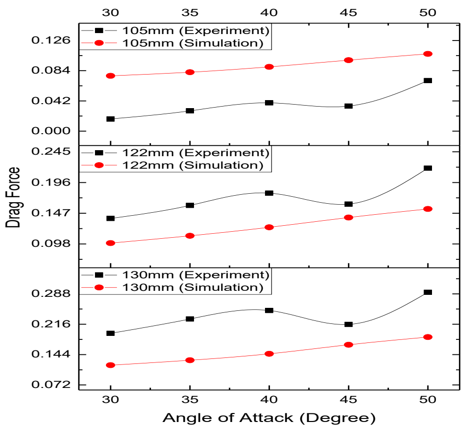


Figure 6: Mesh of the CFD Simulation for 105 mm Projectile

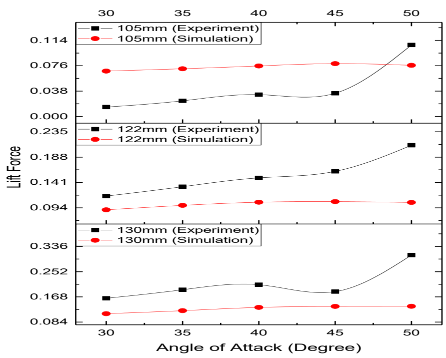
4. Results and Discussion

The Lift, Drag, and pressure Coefficients are calculated.

The simulation show that the drag force and lift force is increasing as the angle of attack is increasing for constant air velocity. The drag force and lift force is increasing as the size of the projectile is increasing. Angle of Attack (AOA). The data for the corresponding plots are shown in Table 2 and Table 3



(a)



(b)

Figure 7: (a) Drag Force (b) Lift Force at different Angle of Attack (Experiment vs Simulation).

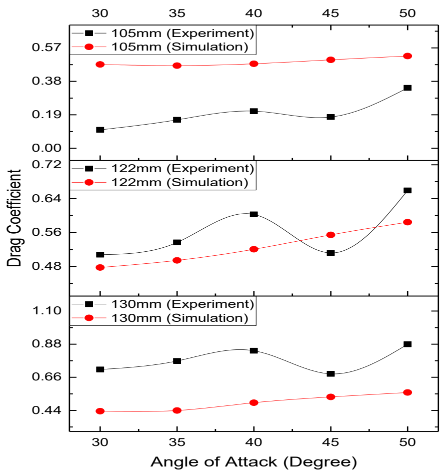
Table 2: Simulation and Experimental Drag Forces on 105, 122, and 130 mm Projectiles at Different AOA.

Angle of Attack (°)	105 Sim	122 Sim	130 Sim
30	0.0769	0.0994	0.1189
35	0.0819	0.1110	0.1306
40	0.0893	0.1246	0.1458
45	0.0986	0.1402	0.1673
50	0.1074	0.1538	0.1856

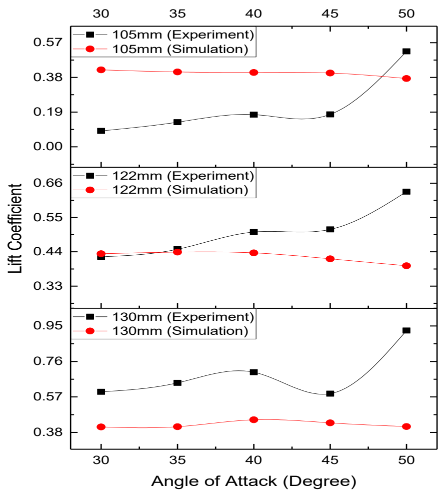
Table 3: Simulation and Experimental Lift Forces on 105, 122, and 130 mm Projectiles at Different AOA

Angle of Attack (°)	105 Sim	122 Sim	130 Sim
30	0.0680	0.0903	0.1119
35	0.0715	0.0986	0.1221
40	0.0757	0.1044	0.1330
45	0.0792	0.1056	0.1362
50	0.0766	0.1040	0.1367

The Drag and Lift Coefficients are calculated from simulation . This on a and lift coefficients for the 105 mm projectile . The limitation of our simulation is the air speed which is in the subsonic zone. The Simulation results will make more sense if the experiment can be done in Supersonic air velocity. the drag and lift coefficient plot at different attack angles and Table 4 and Table 5 shows the corresponding data for drag and lift coefficients.



(a)



(b)

Figure 8: (a) Drag Coefficient (b) Lift Coefficient at different Angle of Attack (Experiment vs Simulation).

Table 4: Simulation and Experimental Drag Coefficients on 105, 122, and 130 mm Projectiles at Different AOA

Angle of Attack (°)	105 Sim	122 Sim	130 Sim	130 Exp
30	0.4760	0.4775	0.4352	0.7116
35	0.4695	0.4943	0.4394	0.7687
40	0.4799	0.5205	0.4909	0.8366
45	0.5023	0.5545	0.5298	0.6827
50	0.5239	0.5845	0.5591	0.8792

Table 5: Simulation and Experimental Lift Coefficients on 105, 122, and 130 mm Projectiles at Different AOA

Angle of Attack (°)	105 Sim	122 Sim	130 Sim
30	0.4211	0.4339	0.4095
35	0.4099	0.4391	0.4107
40	0.4071	0.4366	0.4477
45	0.4039	0.4175	0.4313
50	0.3740	0.3953	0.4116

The simulation pressure and velocity gradient are shown for 105 mm, 122 mm, and 130 mm projectiles in Figures 19, 20, 21, 22, 23, and 24. The pressure contour shows that the pressure is more felt at the front of the projectiles for all sizes. However, the velocity streamline plot shows that the streamline is flowing over the 105mm projectile. The 122 mm and 130 mm projectile does not show any streamline flowing over them. Therefore, the drag forces should be higher for larger projectiles

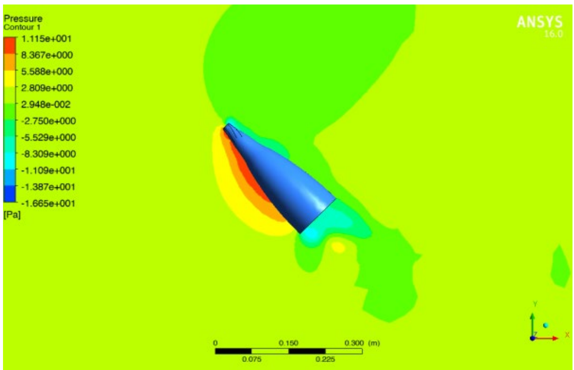


Figure 9: The pressure contour for 105 mm projectile at 45°AOA.

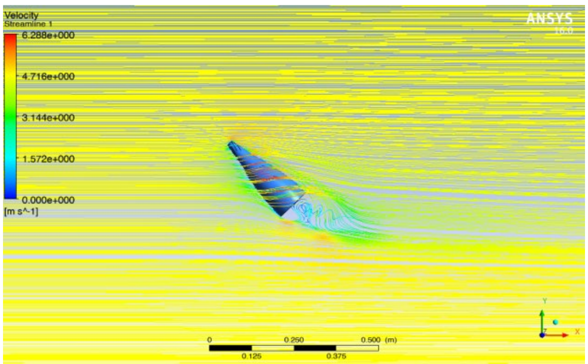


Figure 10: The velocity contour for 105 mm projectile at 45°AOA.

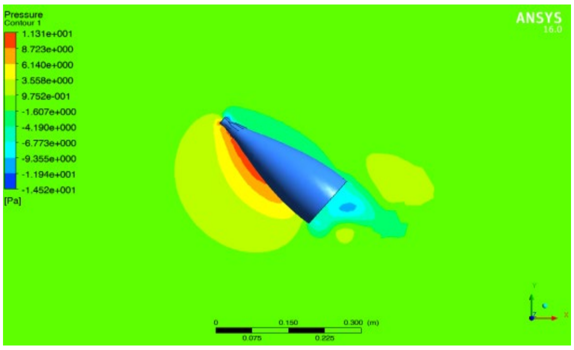


Figure 11: The pressure contour for 122 mm projectile at 45°AOA.

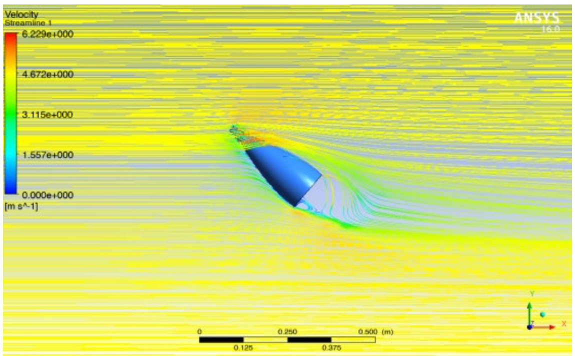


Figure 12: The velocity contour for 122 mm projectile at 45°AOA.

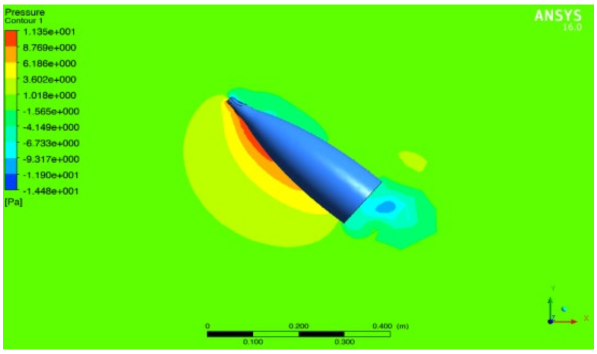


Figure 13: The pressure contour for 130 mm projectile at 45°AOA.

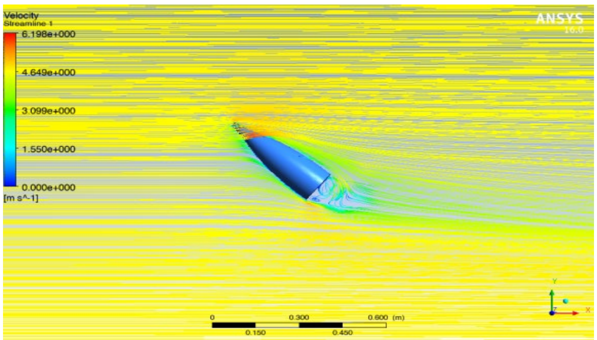


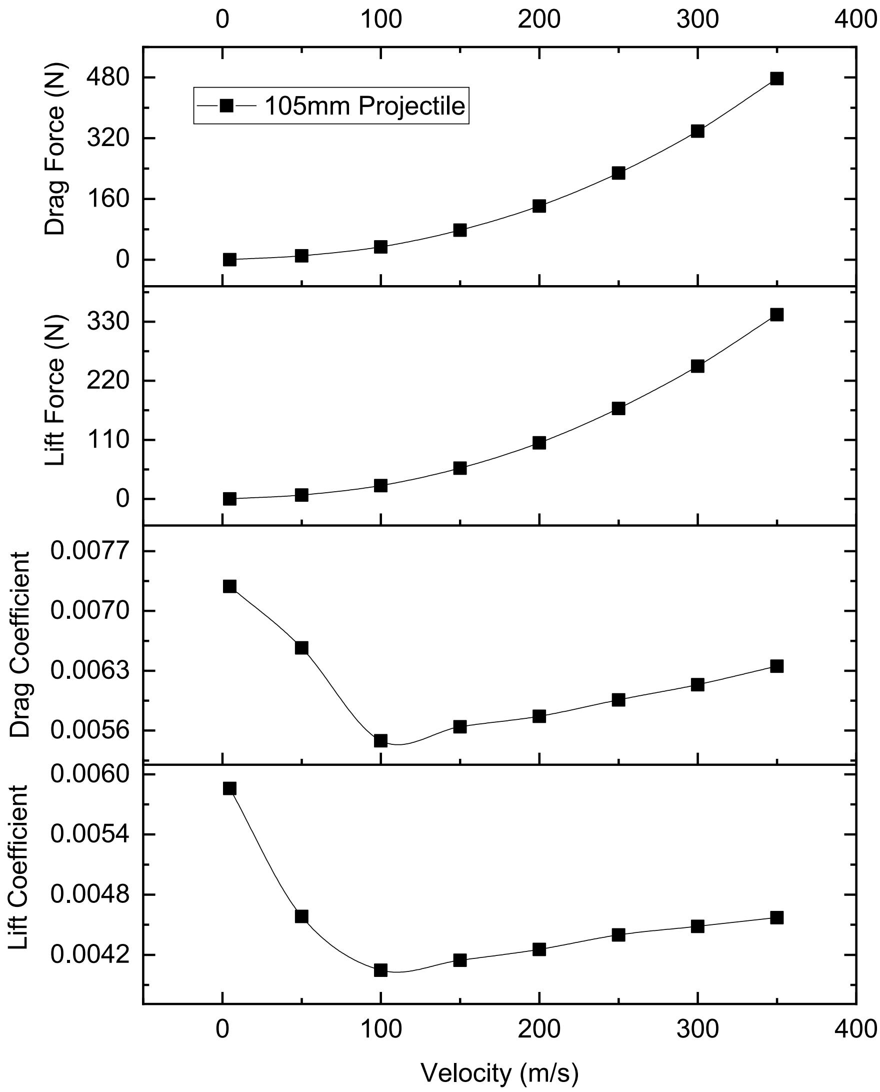
Figure 14: The velocity contour for 130 mm projectile at 45°AOA.

The simulation was performed for higher air speed to investigate the drag and lift forces. The simulation in supersonic speed is not same as the subsonic speed therefore the comparison of the simulation result was different. However, the trend was familiar as the lift and drag coefficient changes near our Experimental speed is almost negligible.

Table 6: The lift and drag coefficient of 105 mm projectile at 45° AOA for various design points.

Table of Design Points

	A	B	C	D	E	F	G	H	I
1	Name	P1 - AOA	P2 - Inlet Velocity	P3 - Drag Force	P4 - Lift Force	P5 - Drag Coefficient	P6 - Output Parameter	Retain	Retained Data
2	Units	degree	m s^-1	N	N	kg m^-1	kg m^-1		
3	DP 0 (Current)	45	4.7	0.098586	0.079262	0.0072864	0.0058582	☑	✓
4	DP 1	45	50	10.055	7.0162	0.0065663	0.004582	☑	✓
5	DP 2	45	100	33.564	24.786	0.0054799	0.0040467	☑	✓
6	DP 3	45	150	77.762	57.115	0.0056426	0.0041444	☑	✓
7	DP 4	45	200	141.27	104.21	0.005766	0.0042533	☑	✓
8	DP 5	45	250	228.06	168.36	0.0059574	0.0043979	☑	✓
9	DP 6	45	300	338.27	247.16	0.0061364	0.0044835	☑	✓
10	DP 7	45	350	476.72	342.9	0.0063536	0.0045701	☑	✓
*								☐	



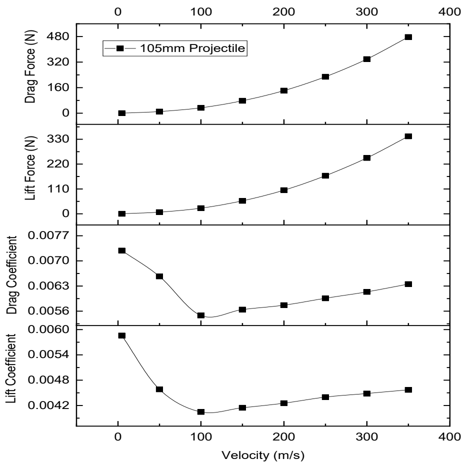


Figure 15: The lift and drag coefficient of 105 mm projectile at 45° AOA for supersonic speed.

Note: In my simulation work I have done some simulation keeping the projectile fixed in a fluid domain at a particular angle while varying the speed of the fluid. Again I have tried to do another simulation. Where in the fluid domain both projectile and fluid is moving. Because in real scenario while firing with Gun both projectile and air is moving. For this I have created a separate mesh for the projectile and fluid domain. But in this condition the projectile in motion in the moving fluid domain, there is no build-in function present in fluent solver. Moving towards the user-defined functions, in the 3D simulation user-defined function cannot be applied.

5. Conclusions

In this thesis, an simulation investigation was performed on the projectiles model. Similar conditions were applied for numerical simulation to investigate the further parameters that are not possible to measure or visualize in real-time. However the conclusions are drawn from this Study are as followings:

a. The simulation was done focusing on the drag force and lift force acting on the projectiles. The reason for focusing those two parameters as they are responsible for the flight of the Projectiles and precision of hitting the target. In the real time scenario, the flight of the projectiles takes place at supersonic speed and not in subsonic speed. But we have conducted our Simulation at 4.7 m/s provides the initial flight scenario and the related drag and lift forces which can be used to investigate the projectiles at supersonic speed.

b. The simulation results found in this investigation show that the drag force and lift forces are increasing with respect to increasing angles of attack. The drag and lift force also increases as the size of the projectile is increased.

c. The study found that, the lift force increasing rate is lower than that of drag forces with the increasing angle of attack. Therefore, the drag forces and lift forces acting on the projectiles are a combined effect of the size, shape, and angle of attack

d. The simulation reveals that there is almost no air streamline is flowing over the 122 mm and 130 mm projectile that may increase the drag forces significantly. The pressure contour shows more pressure at the front of the projectiles that come in contact with the air directly.

6. Recommendations:
Some recommendations are given below for the future study.

a. In future an numerical investigation of projectiles can be done where the behaviour of the projectile can be measured at different points of the projectile trajectory.

b. In real time scenario both projectiles and fluid are moving. So in future study both experiment and simulation can be done with moving projectiles at different air speed.

c. Both experiment and simulation can be done with optimization of the projectiles by changing the shape.

REFERENCES
Journal articles:

Bolonkin, A., (2013). Long Distance Bullets and Shells. *International Journal of Aerospace Sciences*, vol 2 (2), pp. 29-36.

Dali, M. A., and Jaramaz, S., (2019). Optimization of Artillery Projectiles Base Drag Reduction using Hot Base Flow, *Thermal Science*, vol. 23 (1), pp. 353-364.

Hemateja, A, B., Ravi, Teja., Dileep K.A., Rakesh, S.G.Influence of Nose Radius of Blunt Cones on Drag in Supersonic and Hypersonic Flows. *Material Science and Engineering*, 225 (2017) 012045.

Novak, L., Bajcar, T., Širok, B., Orbanić, A., Bizjan, B., (2018). Investigation of Vortex Shedding from an Airfoil by Computational Fluid Dynamic Simulation and Computer-Aided Flow Visualization. *Thermal Science*, vol. 22 (6B), pp. 3023-3033.

Lijin, J., and Jothi, T. J. S., (2018). Aerodynamic Characteristics of an Ogive-nose Spinning Projectile. *Sadhana*, vol. 43:63, pp. 1-8.

Sahoo, S., Laha, M. K., (2014). Coefficient of Drag and Trajectory Simulation of 130 mm Supersonic Artillery Shell with Recovery Plug or Fuze. *Defence Science Journal*, vol. 64 (6), pp. 502-508.

Conference Proceedings

Alexey, M.L., Stanishlav, A.K., Ivan, G.R., (2017). Optimization of Aerodynamic Form of Projectiles for Solving the Problem of Shooting Range Increasing. AIP Conference Proceeding 1893, 030085.

Suliman, M. A., Mahmoud, O. K., Al-Sanabawy, M.A., Abdel-Hamid, O.E., (2009). Computational investigation of base drag reduction for a projectile at different flight regimes. 13th International Conference on Aerospace Sciences & Aviation Technology, Paper ASAT-13-FM-05, PP: 1-13.

Sahu, J., (2003), Unsteady numerical simulations of subsonic flow over a projectile with jet interaction. 41st Aerospare Sciences Meeting and Exhibit, AIAA 2003-1352, pp. 1-10.

Wessam, M. E., Huang, Z., and Chen, Z., (2014). Aerodynamic characteristics and flow field investigations of an optimal hollow projectile. Proceedings of the 5th International Conference on Mechanical Engineering and Mechanics, pp. 181-186.

Yongjie, X., Zhijun, W., Guodong, W., Jianya, Y., Shouli, P., (2015), Ballistic characteristics of rocket projectile with deflection nose. International Power, Electronics and Materials Engineering Conference (IPEMEC 2015), pp. 405-411.