Numerical simulations over a three body launch vehicle for incompressible flows

D Siva Krishna Reddy*, Sagar Gupta, and Mayank Yadav

Department of Mechanical Engineering, SRM Institute of Science and Technology
Kattankulathur – 603203, Tamil Nadu, India.
Email: sivakriv@srmist.edu.in

Abstract: In the present work, three dimensional numerical simulations over a launch vehicle at 70 m/s are performed for 3 scale models. The vehicle has a cylindrical core body and two strap on boosters. The scaled models chosen so that they represent a wind tunnel blockage ratio of 3%, 4% and 5% respectively. Simulations are performed using commercial CFD software Ansys Fluent. Three dimensional structured mesh is generated using ICEMCFD software. The solver depends on finite volume technique, and SIMPLE calculation is utilized to address the couple continuity and energy conditions. K-w SST model is utilized for disturbance demonstration. The computed flow field is analyzed in terms of velocity, pressure and turbulent kinetic energy contours. Surface pressure acquired from the three simulation (relating to every blockage ratio) arrangements in balance plane is compared to understand the impact of blockage ratio on the stream field and drag coefficients. As the blockage is increased, it was observed that maximum velocity is increasing. Lowest blockage found to give highest drag coefficient.

Keywords: Wind tunnel interference, blockage, wind tunnel correction, drag coefficient, surface pressure.

1. INTRODUCTION

It is a rocket-controlled vehicle which is used for the movement of payload beyond earth's atmosphere. The origin of launch vehicles traces back to the era of ballistic missiles of 1950s-60s and V2 rocket is considered as the predecessor. To escape the atmosphere and reach the orbit, the minimum velocity reaches up to about M = 25 and to reach celestial bodies as moon or mars, the speed reaches up to a Mach number of 35. The first stage is used to reach the orbital velocity for a defined weight of the payload. The Indian Space Research Organization (ISRO) relies basically on three types of Satellite Launch Vehicles namely ASLV, PSLV and GSLV (Geostationary Space Vehicle). SLV-3 was the first Indian made SLV, which was used to launch the satellite Rohini in the year 1980. It was a 23m four staged solid propellant rocket with payload weight up to 40 kg. The fins are present to provide aerodynamic stability along with aerodynamic control for first stage flight. Strap on boosters on launch vehicles are used to augment the thrust during take-off. The boosters are used only for the first stage flight. In the end of the stage, the boosters are shed from the core body of launch vehicle. The strap-on boosters are effective, however, their presence results in aerodynamic instability along with the heat shield of the payload. The strap-on boosters use mostly solid propellant fuelled. While studying the fluid dynamics problems during the lift-off, the design is effective if the jet is diverted away from the launch pad. The central foundation should not reach up to the vehicle base so that it doesn’t possess any problems during lift off. However, for PSLV, the jet deflector is used to overcome the problems during lift-
off. There are two types of deflectors, namely, a double wedge type and inclined plate type. These deflectors help in withstanding the heavy thermal loads as well as proper thrust vectoring for the rocket engine.

In current serious climate there is a need to plan the satellite launch vehicles which will be equipped for dispatching significantly under some unanticipated changes in the barometrical conditions. There was one such circumstance when PSLV-C5 dispatched at the hour of substantial downpour from Sriharikota, India in Oct. 2003 [4]. Prior to dispatching, the Mobile Service Tower (MST) is detracted from the platform. It gets important to test the dispatch vehicles to air wind loads under mimicked Atmospheric Boundary Layer stream. This particular examination is one of its sort as relatively few investigations are done before on the dispatch vehicle in level design. Airflow around the dispatch vehicle model is convoluted since the Boosters, Strapons are firmly loaded with principle stage and the area resembles a compound.

2. LITERATURE REVIEW

Selvi Rajan [1] processed a simulation over the surface mounted cubic model with the k-ε and LES (Large Eddy Simulation) models. An average launch vehicle model of scale 1:50 was picked. To expand the phone thickness an extraordinary component called “installed matrices” was utilized. There were around 32,000 hexahedral cells in every calculation area. The mathematical examination utilized mass and force preservation conditions in two measurements for consistent state, incompressible stream. A third order QUICK plan utilized for demonstrating the convective terms of the energy equations. The model has been exposed to different stream speed of 2.6, 6.9, 7.4, and 10.3 m/s. To endorse CFD results, a lone part load cell was arranged and produced to gain force coefficients. As a result the value of force coefficients for the first and second case came out to be 0.23 and 1.18.

J. Anthoine et al. [2] from The von Karman Institute, Romania performed an aerodynamic investigation to better understand the wind effect on the drag of octagonal poles. It experimentally investigated the effect of the rounded edge radius and of the surface roughness on the drag coefficient of the poles. The drag coefficient is measured for Reynolds numbers, based on the cylinder diameter, ranging from $8 \times 10^4$ to $1.3 \times 10^5$. The maximum velocity is 50 m/s and the test section has a height of 2.36 m, corresponding to a maximum blockage ratio of 17%. Different formulations for correcting blockage effect on drag evolution are compared for each flow regime based on drag measurements of circular cylinders performed in a large wind tunnel for three different blockage ratios. None of the rectification models known in the writing is substantial for all. To obtain the most accurate correction different models should be combined depending upon flow regime.

Mihai-Victor PRICOP et al. [3] worked on a unique method which allowed an approximate corrections for different blockage ratios by utilizing minimal resources. The mathematical model adopted here is the potential model. The numerical solution is based on the panel method, where the source, vortices and doublet are distributed along the configuration surface. For Two-Dimensional case, a panel-based method was selected that uses linear vortex distributions on the outline. For Three Dimensional case method of distribution of the linear intensity source and a quadratic doublet distribution over the panel was selected. For the two-dimensional case, the distribution of the pressure coefficients on the airfoil contour at 2 degrees incidence and the dependence of the lift coefficients on the incidence were calculated. For the three-dimensional case, distribution of the pressure coefficients on the wing surface at an incidence of 2 degrees and the dependence of the lift coefficients on the angle of attack was analyzed.

In the distribution curve of the pressure coefficients small differences were observed. So, in order to compare the results a correction coefficient was introduced representing the ratio of the lift coefficient of the profile in the free flow and that of the profile in the wind tunnel. After noting down the values it was observed that relative percentage error doesn’t exceed 5% between the two. Hence it was concluded that the method used can be applied with confidence to estimate wall corrections in exchange for a high fidelity computation method for an inviscid flow. As a result it can be stated that the panel method (using potential models) combined with mirror image methods can estimate the tunnel corrections to a certain extent.

A. Maheshwari [4] worked on showing the effect of blockage on the stream and heat transfer around a enclosed sphere and of 3 equally spaced spheres placed in a same line in the domain of Reynolds number ranging from 1 to 100 for two specific values of Prandtl number. An organized matrix was made with three-sided cells utilizing GAMBIT. The force and thermal power conditions were addressed mathematically utilizing the
isolated solver module of FLUENT for a scope of conditions as determined. With the end goal of approval. The stream past a sphere was simulated by utilizing the concerned upstream and downstream lengths respectively and the upsides of drag coefficient and distribution length were determined for a scope of qualities and for verification of these values the equation of Average Nusselt Number given by Kaviany [5] was used. The two values are seen to be within 1.5% of each other.

Suresh Krishnan [5] worked at hydrodynamic and heat transfer assessment for streamline flow fluid stream past a spherical object put halfway in a line using CFD softwares to determine the relationships between Nusselt Number and drag coefficients and furthermore to examine the impacts of blockage when the spherical object is constrained at the focal point of a line. In the first place completely created laminar stream speed profile was determined at the gulf of the line and blockage proportion was accounted for. Reynolds number was differed up to a limit of 500 while the worth of Prandtl number was fixed at 5.12 (Water System). The BR’s were changed from 0.02 to 0.5. Every one of the simulations were acted as 3-D stream to keep away from any deficiency of data which may occur because of progress from 2-D stream to 3-D stream. Likewise the computational area is isolated into upstream, circle and downstream and sub areas. The arrangement of mathematical conditions was completed utilizing the limited second request Euler. For the approval of the CFD work results were contrasted and the connections referenced by Kramers (1946). The drag coefficient esteem was discovered to be inside 2% of the worth as indicated in the standard bend by Kramers. Additionally the Nusselt number got from CFD simulations were approximately 3% as compared with the values corresponding to Kramers.

Miroslav Mokry [6] worked on a low request board strategy for computing divider obstruction remedies to the deliberate drag power car air streams, ¾ open, or opened divider test segments. The working methodology involves steps. Initially development of panel representation of the automobile was done. Then establishment of wind tunnel boundary conditions was completed. Finally calculation of the pressure increment which was induced by the wind tunnel over the automobile surface was done. The study was conducted on a MIRA generic notch-back model in which the wall panelling reaches out about 2.5 vehicle lengths both upstream and downstream. Examinations are conducted on the full scale model in different strong divider test areas of German-Dutch air stream and the three sub scale models in DSMA pilot air stream with walls having few slots in it. The full scale MIRA vehicle pressures were extrapolated in an infinitely large section and served as reference data. The drag coefficient required a negative correction when it is measured in the closed section and vice versa for the open section.

Xikun Wang [7] worked on the study of Particle Image Velocimetry (PIV) investigation of the two-dimensional stream past a body bound in a domain. Three common geometries of body with a similar tallness, specifically, roundabout chamber (CC), level plate, and square chamber, yet with changing BR and Reynolds number are explored. The examinations were re-enacted in a re-coursing open channel flum having a test segment with measurements The approaching stream speed was shifted as velocities of different magnitudes, and were compared to different Re separately. It was tracked down that the size of the distribution zone is an element of the shape of the body and the blockage proportion. At little blockage proportions, the distribution region of FP is the biggest, though that of SC is the least.

A.P. Roychowdhury [8] studied the mathematical simulation of air stream blockage and wall obstruction impacts at supersonic Mach numbers of a launch vehicle design. Axis-symmetric model simulations have been accomplished for two Mach numbers for various BR’s utilizing a compressible finite volume technique. In the current examination the wall blockage and obstruction impact is analysed on the vehicle The impact of blockage on the variation in pressure conveyance on the object is acquired and the divider obstruction is calculated in the concerned Mach number. Axi-symmetric model simulations were accomplished for two Mach quantities of 1.5 and 3.0 for which air stream trial estimations are accessible.

G. Lombardi [9] worked on a strategy for the remedy of divider obstruction impacts which can be applied to lift and moment coefficients, estimated over a total airplane arrangement. The pre-owned model having the scale factor of 1:32 which is the portrayal of the Mirage FI. The remedy technique utilized in the current investigation is a purported "post-test" method. In this sort of technique, test information should be given on a control surface situated close to the air stream dividers or straightforwardly on them. The trial information can be pressure, speed course or speed parts. Specifically, in the current work a "one-cluster" remedy system has been picked, in which just pressing factor information are given at certain areas on the air stream dividers.
After the model calculation is characterized, the exploratory tests are done and, other than the streamlined powers following up on the model, the pressure over the air stream wall is estimated at not many chose areas. These information are utilized as limit conditions in a mathematical reproduction of the stream around a similar calculation ("pressure given" recreation, PG). Another mathematical recreation is done in "free-air" conditions, for example with a computational area adequately huge to keep away from deceptive limit impacts. Experimental data is compared with the values obtained from simulations.

The analysis of the flow around the wind tunnel was executed. At that point, the pressure esteems got in this simulation were utilized as limit condition for the PG mathematical simulations. It was tracked down that the remaining mistake after revision increments straightly with the deviation in the pressure estimations, autonomously of the number and conveyance of the sensors. Subsequently, The lift coefficient of about 0.26 can be accepted as an average portrayal of the condition. The amendment in the lift coefficient seems huge, while the pitching second is essentially not influenced by wall obstruction impacts for the thought about arrangement. The outcomes have been contrasted and those acquired with a "pre-test" adjustment strategy, and a palatable understanding has been observed.

Taha Ahmed Abdullah [10] worked on amendment strategy on the coefficient of pressure dispersion around an airfoil with the end goal of the analysis of air stream information, recieved from model pressure estimations. He utilized the vortex boards with straight vorticity strength variatiation to adjust the shape of the airfoil in the test segment of an air stream. Pressure variation over the airfoil is determined both within the sight of the dividers, and without them. The distinction in the coefficients of pressure between free-stream mathematical arrangement and mathematical arrangement with dividers is determined, and superposed to unnormalized pressing factor dispersions, estimated in a real air stream. Consequences of this estimation model were contrasted and some standard revision systems. The pressure variation appropriation about the airfoil was determined with the expectation of complimentary stream and in the test area. The distinction in the coefficient of pressure between free-stream mathematical arrangement and mathematical arrangement with dividers was then deducted from the pressing factor coefficient conveyance estimated in the air stream, at the comparing focuses. Pressing factor circulation got this way is incorporated to acquire coefficient of lift. The validation of here introduced computation technique that has been performed by looking at mathematically acquired lift curve with those got by two notable old style strategies, and awesome arrangements have been gotten for a few average experiments.

Pankaj Kumar [11] studied the flow structure at the wake of the bluff body with altered blockage ratio (BR) keeping the fixed aspect ratio in which the CFD analysis of the bluff body was subjected to subcritical Reynolds number ranging from 5000 to 15000 considering blockage ratio as well. The aim of the study is basically an attempt to observe effects of blockage ratio at lower subcritical Reynolds number range 5000–15,000. The simulation is executed on a 2D mesh construction of the cylinder with varying blockage ratio having foremost anxiety for pressure, drag and kinetic energy analysis. A two-dimensional incompressible transient flow over a circular cylinder is considered for the simulation. The developed structured 2-D mesh was exported for all conditions and simulated in ANSYS Fluent. Results suggested that the drag force is maximum at low Reynolds number and it decreases with an increase in Reynolds number. Also the variation of the drag coefficient with different aspect ratio and Reynolds number was observed from the graph. It can be seen that the drag coefficient gradually decreases with an increase in Reynolds at aspect ratio 0.75.

3. METHODOLOGY

The project aims to investigate the flow field over the Launch Vehicle and detect the impact of the presence of incompressible flow field. We start by creating the CAD model of the launch vehicle using the software SolidWorks. ICEM CFD is used to generate a structured mesh for the simulation and then ANSYS Fluent is utilized for conducting the simulations of the stream field over the Launch Vehicle with Finite Volume Method (FVM). The incompressible RANS conditions are addressed with pressure-based calculation. A “Shear Stress Transport” turbulence model is utilized to calculate turbulence parameters. Simulations are carried out on a structured mesh generated around the walls of the launch body vehicle which were generated using the ICEM CDF software. This project uses Computational Fluid Dynamics (CFD) as the method to calculate the various flow properties along the flow field and on the launch vehicle. The Navier Stokes equations are the governing equations of CFD which vary from case to case.
4. RESULTS AND DISCUSSIONS

The results are shown for the full configuration of the launch vehicle with strap on boosters. The results show the pressure contour, velocity contour, drag coefficients, surface pressure plots in which we can observe the effect of the blockage parameter along the length of the launch body vehicle. The surface pressure data is also plotted on the symmetry plane. The effect of Blockage parameter 2%, 3% and 5% is studied for the core body and the full configuration with the strap on boosters. The wake development is also studied to show the impact of asymmetry on the vortices near the vehicle. The aerodynamic coefficients are compared and the trends show that there is a lot of fluctuation in the values in the beginning and later a constant steady value is observed, the aerodynamic coefficients decreases.

4.1. Grid convergence study

![Figure 1: The predicted $P_w/P_\infty$ along the length of the launch body for 3 mesh cases.](image)

**Table 1:** $P_w/P_\infty$ obtained at the nose end of the launch vehicle on three successively refined grids.

| Grid            | Number of Cells | $P_w/P_\infty$ |
|-----------------|-----------------|----------------|
| Coarse Mesh     | 1600683         | 1.022268       |
| Medium Mesh     | 1982002         | 1.025397       |
| Fine Mesh       | 2655247         | 1.025419       |

Resolving flow gradients requires the access of the grid point density and for this purpose a convergence study is performed. In all the three grids the near wall spacing is taken to be constant. The number of study points are first reduced and then increased by 25% and then the corresponding simulations are performed. The Table 1 depicts the $P_w/P_\infty$ obtained from the three grids. The grid point density was decreased by 25% and then we observed change in $P_w/P_\infty$ by approximately 0.3%. Indications appear confirming the fact that when the mesh was refined beyond the medium grid significant changes are not observed. Therefore the intermediate grid was preferred for the upcoming simulations because it utilizes a shorter memory time when compared to the refined mesh.
4.2. Blockage Ratio = 0.03:

4.2.1. Velocity, Pressure and Turbulent Kinetic Energy Contours:-

![Detailed Representation of velocity contours, pressure contour and Turbulent Kinetic Energy at Z plane.](image)

*Figure 2*: Detailed Representation of velocity contours, pressure contour and Turbulent Kinetic Energy at Z plane.
Figure 2 shows the velocity contours, pressure contour and Turbulent Kinetic Energy over the 3 body launch vehicle at Z plane. The free stream velocity is 70m/s. On studying the point concerning the nose stagnation region, stream retards and achieves a speed below the free stream speed at stagnation point. When the fluid flow takes place beyond the stagnation point, development happens and brings about the expansion of velocity to approximately 86.4m/s at end of spherical portion. Downside of the boat tail region, the velocity decreases to 62m/s.

Since it is an incompressible fluid flow, the equation given by Bernoulli is clearly applicable which states that the total energy, i.e. the pressure head, kinetic head and potential head of the system remains constant. As a result, the points where the velocity decreases, correspondingly the pressure increases as can be seen from the contour. At the nose stagnation point region where we had observed a drop in the velocity here we can observe a sudden increase in the pressure to $1.04 \times 10^5$ Pa. When the flow progresses below the stagnation point we observe an expansion in the flow which is responsible for the decrease in the pressure to atmospheric pressure of $1.01 \times 10^5$ Pa at end of spherical portion. Stream again isolates at the highest point of center body. It has been observed that, the stagnation pressure in the wake is lower than the stagnation pressure toward the end of the launch body vehicle. Which implies that difference in pressure is small likewise, making the stream staler and less vigorous.

Turbulent Kinetic Energy is the mean active energy related with the whirlpools. It is showed by estimated root mean square (RMS) speed variations. Mostly there is a yellow region (Depicting the highest magnitude) inside which green part can also be seen corresponding to slightly lesser magnitude. Both the regions are enveloped by the light blue region which is of least magnitude among the three.

4.2.2. Surface Pressure Distribution for Wall Conditions:

Surface pressure parameter along the wall of the body in the mid Z plane, is appeared in Figure 3. The curve length of the point on nose stagnation point is standardized by the core body dimensions, and the surface pressure is standardized by the atmospheric pressure at X and Y axis respectively. At nose stagnation point, $P_{w}/P_{\infty}$ is 1.0177. Pressure progressively increases in the cylindrical portion. After boat-tail, pressure increases till end point of the launch vehicle. A steep pressure rise is also observed in the nose region of the strap-on boosters with the value corresponding to about 1.0081 times $P_{\infty}$.

![Figure 3: Standardized surface pressure along the body in the mid Z plane.](image-url)
4.2.3. Validation:

Mach number, M is calculated by

\[ M = \frac{70}{\sqrt{\gamma RT}} \]

Where, 70 is the free stream velocity (m/s), R is the gas constant (287 J / Kg-K), and T is the atmospheric temperature taken of the earth’s atmosphere (298 K).

On substituting the values, Mach number came out to be 0.202, which clearly indicates that it is an incompressible fluid flow.

For the validation of the pressure ratio we calculated the pressure ratio at stagnation point from the Gas Table using the formula,

\[ \frac{P_o}{P_\infty} = \left[ 1 + \frac{\gamma - 1}{2} M^2 \right]^\frac{\gamma}{\gamma - 1} \]

Where,

\( P_o \) is stagnation pressure and \( P_\infty \) is atmospheric pressure, \( \gamma \) is the adiabatic index for air.

For the calculated Mach number and gamma value 1.4, the theoretical pressure ratio \( P_o / P_\infty = 1.028 \).

The results we obtained from the CFD analysis was 1.0177 so, the percentage error obtained for this case is 1.01%.

4.3. Blockage Ratio = 0.04:

4.3.1. Velocity, Pressure and Turbulent Kinetic Energy Contours:-

\[ \text{Imagery of velocity, pressure and turbulent kinetic energy contours.} \]
Figure 4: Detailed Representation of velocity contours, pressure contour and Turbulent Kinetic Energy at Z plane.

Figure 4 shows the velocity contours, pressure contour and Turbulent Kinetic Energy over the 3 body launch vehicle at plane, $Z = 0$. The free stream velocity is 70m/s. On studying the point concerning the nose stagnation region, stream retards and achieves a speed below the free stream speed at nose stagnation point. As the stream proceeds beyond the stagnation point, development happens and brings about the expansion of velocity to approximately 91.7 m/s at end of spherical portion. Downstream of the boat tail region, the velocity decreases to 62m/s. Stream detachment is observed at the beginning which further leads to the development of separation bubble. Velocity of the flow in the separation bubble spans from 9.17 m/s to 27.5 m/s.

For an incompressible fluid flow, the equation given by Bernoulli is clearly applicable which states that the total energy, i.e. the pressure head, kinetic head and potential head of the system remains constant. As a result, the points where the velocity decreases, correspondingly the pressure increases as can be seen from the contour. A pressure increase is observed at the nose stagnation point region corresponding to a dip in velocity as discussed earlier. A high value of pressure is obtained at the nose end of the launch body vehicle with a value of $1.04 \times 10^5$, then successively near the cylindrical and boat-tail region the pressure is nearly atmospheric. The pressure at the nose end of the strap-on boosters increases a bit. Further there is a decrease in pressure at the end of portion due to flow expansion. Which implies that difference in pressure is small likewise, making the stream staler and less vigorous.

Turbulent Kinetic Energy is the mean active energy related with the whirlpools. It is showed by estimated root mean square (RMS) speed variations. Mostly there is a yellow region (Depicting the highest magnitude) inside which green part can also be seen corresponding to slightly lesser magnitude. Both the regions are enveloped by the light blue region which is of least magnitude among the three.
4.3.2. Surface Pressure Distribution:

![Figure 5](image)

Figure 5: Standardized surface pressure along the body in the mid Z plane.

Surface pressure parameter along profile of the body in the mid Z plane, is appeared in Figure 5. At nose stagnation point, \( \frac{P_w}{P_\infty} \) is 1.0201. Pressure progressively increases in the cylindrical region. After boat-tail, the pressure increases till the end point of the launch vehicle. A steep pressure rise is also observed in the nose region of the strap-on boosters with the value corresponding to about 1.0478 times \( P_\infty \).

4.3.3. Validation:

The stagnation pressure obtained for Blockage ratio of 0.04 is 1.0201 and for the calculated Mach number and gamma value of 1.4, the theoretical pressure ratio \( \frac{P_0}{P_\infty} = 1.028 \). So, the percentage error obtained for the theoretical and experimental value is 0.768%.

4.4. Blockage Ratio = 0.05:

Simulations are performed for blockage ratio of 0.05. The distribution of velocity, pressure and turbulent kinetic energy is similar to those presented to above for the cases of blockage ratio 0.02 and 0.03, and hence not presented for the sake of brevity.

4.5. Study of the surface pressure plots for 3 blockage ratios

![Figure 6](image)

Figure 6: Detailed visualization in the surface pressure concerning the 3 blockage ratios.
Surface pressure parameter along the profile of the body in the mid Z plane is appeared in Figure 6. The curve length of the point on nose stagnation point is standardized by the core body dimensions, and the surface pressure is standardized by the atmospheric pressure at X and Y axis respectively. The effect of the blockage ratio is plotted for each of the 3 cases to identify the effect of the blockage parameter. At the nose stagnation point, the $P_w/P_\infty$ is different for 3 cases lowest for the blockage ratio of 0.03 having the value of 1.0172 and the value of 1.0201 and 1.024524 for the blockage ratio of 0.04 and 0.05 respectively. The difference in the stagnation pressure value obtained when compared with the BR of 0.03 is 0.285% and 0.720% respectively for BR 0.04 and 0.05 respectively. Due to further development of the flow at the spherical region, the pressure reduces to 1.06 times free-stream pressure. Due to the expansion in the conical region, decrease in the pressure to $0.9780P_\infty$, $0.9779P_\infty$, and $0.9772P_\infty$ for Blockage Ratio 0.03, 0.04, and 0.05 respectively. Pressure progressively increases in the cylindrical portion. After the boat-tail region, the pressure increases till end of the launch vehicle. Also, it has been noticed that the pressure continuously elevates beyond the boat tail region. For the BR 0.04 and 0.05, except for the nose stagnation region the pressure all along the launch body vehicle varies exactly the same. The flow expansion for along the spherical and conical region are exactly the same. For the BR 0.04 and 0.05, except for the nose stagnation region the pressure all along the launch body vehicle varies exactly the same. The flow expansion for along the spherical and conical region are exactly the same.

4.6. Comparison for the drag coefficient for 3 blockage ratios:

**Table 2: Comparison of the Aerodynamic Coefficients for 3 Blockage ratios.**

| SI No | Blockage Ratio | Drag Coefficient |
|-------|----------------|------------------|
| 1.    | 0.03           | 2.875            |
| 2.    | 0.04           | 1.02             |
| 3.    | 0.05           | 0.87             |

The Table 2 depicts the comparison of the drag coefficients for 3 Blockage ratios. Maximum drag coefficient is observed for the BR of 0.03 with a value of 2.875 and least drag coefficient of 0.87 for the BR of 0.05. The drag coefficient is observed to be decreasing with increase in the Blockage ratio.

4.7. Change in drag coefficient with blockage ratio and Reynolds number:

**Figure 7(a):** Change in Drag Coefficient with respect to Blockage Ratio (b) Change in the drag coefficient with Reynolds number.

Variation of the drag coefficient with BR is shown in figure 7(a), it is clearly observed that the drag coefficient keeps on decreasing with increase in the Blockage ratio. A sudden decrease in drag coefficient is observed from BR 0.03 to 0.04 from the value of 2.875 to 1.01, then a steady decrease from 1.01 to 0.87 is observed from BR 0.04 to 0.05.
With the further analysis of results, the drag coefficient is maximum at low Reynolds number and then it decreases with increase in the drag coefficient. Figure 7(b) shows the variation of drag coefficient with the Reynolds number.

5. CONCLUSION

- With increase in the Blockage ratios, the maximum velocity at the nose keeps on increasing.

- On comparison of the surface pressure parameter, at the nose stagnation point, the \( P_w/P_\infty \) is different for 3 cases lowest for the BR of 0.03 having the value of 1.0172 and the value of 1.0201 and 1.024524 for the blockage ratio of 0.04 and 0.05 respectively.

- The difference in the stagnation pressure value obtained when compared with the BR of 0.03 is 0.285% and 0.720% respectively for BR 0.04 and 0.05 respectively.

- The results we obtained from the CFD analysis was 1.0177, 1.0201, and 1.0245 for Blockage ratio 0.03, 0.04, and 0.05 respectively. Percentage error obtained for each case was 1.01%, 0.768%, and 0.340% respectively for the Blockage ratio of 0.03, 0.04, and 0.05 respectively.

- The drag coefficient decreases with increase in the blockage ratio, and increases with decrease in the Reynolds number.

ACKNOWLEDGEMENTS

We would like to thank the SRM High Performance Computing Centre (HPCC) for providing access to the computational facility.

REFERENCES

[1] Selvi Rajan, Santoshkumar. M, Lakshmanan, N, Nadaraja Pillai, S and Paramasivam, M. 2009 CFD Analysis and Wind Tunnel Experiment on a Typical Launch Vehicle Model, Tamkang Journal of Science and Engineering, 12 223-9.

[2] J.Anthoine, D. Olivari, and D.Portugais, 2009 Drag Coefficient Determination of Bluff Bodies – Analysis of Blockage Effect, Wind and Structures An International Journal.

[3] Mihai-Victor PRICOP, Ionut BUNESCU, Adrian DINA, 2018 Estimation of Wind Tunnel Corrections using Potential Models, International Conference of Aerospace sciences “AEROSPATIAL”, DOI: 10.13111/2066-8201.2019.11.1.4.

[4] A. Maheshwari, R.P. Chitahra, and G.Biswas, 2006 Effect of Blockage on drag and heat transfer from a single sphere and in-line array of three spheres, Powder Technology 168 74–83.

[5] Suresh Krishman, and Kannan A, 2020 Effect of Blockage ratio on drag and heat transfer from a centrally located sphere in pipe flow, Engineering Applications of Computational Fluid Mechanics 4 396 – 414.

[6] Miroslav Mokry, 1995 Wall Interference Correction to drag measurements in Automotive Wind Tunnels, Journal of Wind Engineering and Industrial Aerodynamics 56 107-122.

[7] Xikun Wang, Jiaqi Chen, Bo Zhou and Yajie Li, 2021 Experimental investigation of flow past a confined bluff body: Effects of body shape, blockage ratio and Reynolds number, Journal of Ocean Engineering, 220, 108412.

[8] A.P Roychowdhury, and C. Unnikrishnan, 2014 Numerical Simulation of Wind tunnel Blockage and Wall Interference Effects at Supersonic Mach Number Flows, International Journal of Innovative Research in Technology & Science(IJIRTS), 2 (4), 63-72. ISSN:2321-1156.

[9] G. Lombardi, M.V Salvetti and M. Morelli, Correction of the wall interference effects in wind tunnel experiments, Transactions on Modelling and Simulation 30 ISSN 1743-355X.

[10] Taha Ahmed Abdullah, Zlatko Petrovic, Zoran Stefanovic, Ivan Kostic and Jovan Isakovic, 2015 Two Dimensional Wind Tunnel Measurement Corrections By The Singularity Method, Tehnicki Vjesnik, 22(3), 557-565. DOI: 10.17559/TV-20140214114718

[11] Pankaj Kumar and Santosh Singh 2020 “Flow past a bluff body subjected to lower subcritical Reynolds number” Journal of Ocean Engineering and Science 5 173-179.