Study of Effect of Mach Number on Coefficient of drag of Sounding Rocket by Computational Method

Ashutosh Kumar1*, Rakesh Kumar Shukla2, Kushagra S. Gusain3, Aakash Sadar4
Department of Mechanical Engineering, Krishna Engineering College, Ghaziabad, India

Abstract— This research paper conducts a study of aerodynamic drag coefficient of a sounding rocket named Rohini300 MK-II. There are various factors which plays important role in rocket stability and aerodynamic drag coefficient is one of them. Zero-lift drag coefficient, C_D (i.e. drag coefficient at attack angle of zero degree) is evaluated and its behavior is observed with the different mach numbers. This paper defines the development of a CFD model of the RH 300 MK-II in the modeling software, then importing the 3D model into CFD software and then the creation of CFD domain, meshing of the domain of CFD, application of various boundary conditions and then obtaining plots and results for coefficient of drag. And Finally Result is concluded to get its Zero Drag coefficient.

Keywords – ANSYS Fluent 19, Drag Coefficient, Mach Number, RH 300 MK-II Sounding Rocket

1. INTRODUCTION
A. RH 300 MK-II Sounding Rocket
Indian Space Research Organisation (ISRO) developed Rohini – Sounding Rocket Series for meteorological and atmospheric study. RH 300 MK-II is a sounding rocket of Rohini sounding rocket series which is used for aerodynamic study. It is a single stage, fin stabilized sounding rocket which weighs nearly 500kg with a diameter of 305mm and overall length of 5.6m. A maximum thrust of 65kN is generated for about 20 seconds by the solid propellant boosters motor which powers the sounding rocket. The spin rockets function for 300 milliseconds and are then ejected out. The commands for all the activities after launch are provided by the vehicle’s centralized sequencer-encoder. Programmable S-band transmitter transmits the data collected by the various systems of the sounding rocket to the tracking station.

B. Computational Fluid Dynamics (CFD)
It is a branch of fluid mechanics that applies numerical methods and algorithms to solve problems that involve fluid flows is called CFD. Computer Software (like Ansys Fluent) are used to perform the calculations and analysis of liquid or gases. Initial validations of these softwares are performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

2. METHODOLOGY
A. Modeling of the RH-300 MK-II Sounding Rocket
The modeling of the RH 300 MK-II, is used in this paper. It is applied by using Solid Works model 2013. The External flow is solved and analyzed by using this solid work model. Some sketch commands are used in particular sounding rocket modeling are line, circle, ellipse, spline etc. The commands used are extrude, revolve, extrude cut,
mirror, etc. While solving and modeling of this sounding rocket, proper care should be taken because of any improper way of modeling leads overlapping of the geometry.

This enclosure around the sounding rocket is made in ANSYS Workbench 19.

B. Meshing
The meshing of the sounding rocket and atmosphere created around it, is done using ANSYS Fluent 19. The continuum for this as ANSYS Workbench is imported, then the continuum is subdivided into different categories like inlet, outlet, wall and rocket body. So the required meshing conditions are used and the continuum is meshed. We used Cut Cell meshing technique with inflation near the sounding rocket body.

![Image of meshing](image1)

The continuum is given a mesh size of 15mm and the rocket volume is given a fine mesh size of 5mm since it is a complex geometry. The difference between the element sizes can be seen in fig 5.

After the meshing of the continuum it is then exported to ANSYS Fluent Setup 19, where in the flow analysis over the sounding rocket is done.

C. Setup of the Continuum
The simulation on the domain is done in ANSYS Fluent Setup 19. Initially the meshing of the domain is checked and once the software approves it, the models, materials and boundary conditions are set. The meshing of the
sounding rocket and atmosphere created around it, is done using ANSYS Fluent 19. The continuum for this as ANSYS Workbench is imported, then the continuum is sub divided into different categories like inlet, outlet, wall and rocket body.

The set up conditions are 1. Model 2. Working Material 3. Boundary Conditions 4. Solutions.

Generally SST model is applied in simulation and SST is shear-stress transport which is used for the specific purpose.

The working Material is Air which is used for this purpose of analysis.

The boundary conditions are applied for calculating Mach number or velocity at inlet and outlet of the system by using the CFD analysis method. We analyzed RH 300MK-II in the speed range of 0 to 900 m/s. Therefore, the boundary condition at inlet for the continuum is given as a range of 0 m/s to 900 m/s.

Solution are selected for the analysis purpose and relaxation factor for momentum and pressure to get the desired kinetic energy turbulent dissipation rate and turbulent velocity which is at are set as 0.5 and for the turbulent kinetic energy, turbulent dissipation rate and turbulent viscosity is set to 0.75, 0.75 and 1 respectively.

3. RESULTS AND DISCUSSIONS

The expected graphical results are set to drag coefficient. The graphs shown below shows the value of coefficient of drag on the rocket at different Mach Numbers. Solution are selected for the analysis purpose and relaxation factor for momentum and pressure to get the desired kinetic energy turbulent dissipation rate and turbulent velocity.

A. Graphs
1. Mach 0.5

It can be seen that the value of Coefficient Drag for Mach No. 0.5 has converged on the value 0.4 at 220 iterations

2. Mach 1
It can be seen that the value of Coefficient Drag for Mach No. 1 has converged on the value 0.5 at 300 iterations.

3. Mach 1.5

It can be seen that the value of Coefficient Drag for Mach No. 1.5 has converged on the value 0.75 at 350 iterations.

4. Mach 2

It can be seen that the value of Coefficient Drag for Mach No. 2 has converged on the value 1.2 at 400 iterations.

5. Mach 2.62

It can be seen that the value of Coefficient Drag for Mach No. 2.62 has converged on the value 1.75 at 500 iterations.

4. CONCLUSION

After the simulation, the obtained results (i.e. coefficient of drags) are plotted against mach numbers. The plotted graph gives the clearer picture of the aerodynamic drag coefficient.
As we can see in the figure 8.1, the zero lift drag coefficient at Mach 0.5, Mach 1, Mach 1.5, Mach 2 and Mach 2.62 is 0.4, 0.5, 0.75, 1.2 and 1.75 respectively. With the increase of velocity the drag coefficient increases exponentially.

The zero lift drag coefficient obtained from the CFD analysis lie within satisfactory range. So, it can be concluded that the sounding rocket RH 300 Mk-II is aerodynamically (in terms of drag) sound. It is important for getting desired coefficient of performance to set the value. Mach Number is deciding parameter for this set up. The velocity is also important for its performance measurement and decision analysis. Hence, The zero drag is used to get for CFD analysis lie within satisfactory range. So, it can be concluded that the sounding rocket RH 300 Mk-II is aerodynamically (in terms of drag) sound.

5. FUTURE SCOPE

Scope of future work is very wide as there are numbers of project work possible. One of the possible future project works can be the comparison of CFD simulations with the current practical analysis. It can give us a clear picture how the CFD analysis is close to practical analysis.

Another potential future project work could be related different nose designs. As we know that coefficient of drag is majorly dependent on the design of the sounding rocket. So, with change in design of nose of rocket, we can simulate the results in Ansys Fluent and compare them with each other at different mach numbers. This could give us a more feasible conclusion on the design aspect in terms of aerodynamics.

REFERENCES

(1). Algimantas Fedaravičius, Sigitas Kilikevičius, Arvydas Survila, Investigation on the aerodynamic characteristics of a rocket-target for the system “Stinger, JVE International Ltd. Journal of Vibroengineering, Dec 2015, Vol. 17, Issue 8. ISSN1392-8716.
(2). Aerodynamics Modeling of Sounding Rockets by Kristoffer Hammargren, 2018.
(3). Olalekan, A.H. Attah, O.C. Bisola, O. Jimoh, H. Atinuke, Ojo. (2014). Flight Trajectory Simulation of a Single Stage Sounding Rocket. International Journal of Scientific & Engineering Research, Volume 5, Issue 11, November 2014. ISSN 2229-5518.
(4). Fedaravičius A., Kilikevičius S., Survila A. Optimization of the rocket’s nose and nozzle design parameters in respect to its aerodynamic characteristics. Journal of Vibroengineering, Vol. 14, Issue 3, 2012, p. 1390-1398.
(5). Menter F. R., Kuntz M., Langtry R. Ten years of industrial experience with the SST turbulence model. Turbulence, Heat and Mass Transfer 4, Begell House, 2003, p.625-632.