International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 11, November-2015

Static Aeroelastic Analysis on Two Stage Rocket Body

Kothamasu Kiran (Author)
M.Tech in Aerospace
Institute of Aeronautical Engineering (IARE),
Hyderabad, India

Eshwar Reddy Cholleti (Author)
B.Tech in Aeronautical
MLR Institute of Technology,
Hyderabad, India

***Abstract*— In the design of a rocket, aeroelastic effects are rarely taken into account. But in real life, aeroelasticity plays an important role in deciding the performance of a rocket. This paper involves the aeroelastic analysis over a two stage rocket body with the simulation of fluid-structural interactions, initially starts with flow analysis over the geometry at different Mach numbers 1.5, 2.5 and 3 followed by structural analysis by linking the pressure distribution over the body obtained. Software's going to be used in this thesis is CATIA for modeling the object and then continued in ANSYS Workbench for Meshing and FLUENT for Flow analysis followed by Static Structural Analysis in workbench. After obtaining pressure distribution over the body, Normal force is going to be calculated using Numerical calculations. Thus the behaviour of Static Aeroelastic response over two stage rocket body is going to be predicted.**

Keywords*— Aeroelastic effects, Numerical calculations, Static Structural Analysis, Two-Stage Rocket body, Fluid-Structure Interactions.

I. INTRODUCTION

Rocket is a vehicle, missile or aircraft which obtains thrust by the reaction to the ejection of fast moving fluid from within a rocket engine. Rocket engines work by action and reaction. Rocket engines push rockets forward by expelling their exhaust in the opposite direction at high speed. Rockets are relatively lightweight and powerful, capable of generating large accelerations and of attaining extremely high speeds with reasonable efficiency. Rockets are not reliant on the atmosphere and work very well in space.

Looking back into the history of rockets and guided missiles, we find that rockets were used in China and India around 1000 AD for fireworks as well as for war purposes. During the 18th century, unguided rocket propelled missiles were used by Hyder Ali and his son Tipu Sultan against the British. The current phase in the history of missiles began during the World War II with the use of V1 and V2 missiles by Germany. Since then there has been a tremendous and rapid global advancement in this field.

Study of the movement of a body in the presence of air is called aerodynamics and this study is vitally important for the design of aircraft, missiles and rockets. The atmosphere as we know is densest close to earth's surface at sea level. As we go higher it becomes thinner (i.e. the pressure and density are lower). The aircraft and missiles are bodies that are heavier than air and so can support their weights only if they produce a force to counter it. This force can be either lift force generated by the flow of air over the wings and body or generated by means of an engine in the form of thrust.

The body of the rocket may be divided into three major sections as fore body or the nose, the mid-section and the aft or boat-tail section. The supersonic flow over a cone has characteristics which are similar in appearance as that of a conical one but are markedly different in nature from those corresponding to two-dimensional flow (i.e., flow over a wedge). The similarity in appearance is that an oblique shock wave formed at the tip of the wedge and apex of the cone as shown in the below fig 1.1.

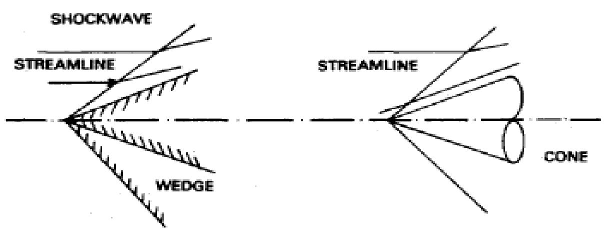

Fig 1.1. Flow past a wedge and a cone

Structural design considerations go hand in hand with the finalization of aerodynamic configuration in the preliminary design. The primary function of the structural design engineer is manifold: (i) To ensure structural adequacy of the missile airframe under its operating environment, (ii) To investigate the most suitable materials to meet the loadings and their associated operating environmental conditions in the missile weapon system and (iii) To analyze and select the optimum type of construction for the type of configuration from the standpoint of ease of manufacturing, cost per unit and interchangeability of parts.

Rockets are generally made from aluminum and its alloys, steel, magnesium and titanium. The major concern is the strength-to-weight ratio of the material. Higher ratio will be better for structure. On account of the high temperatures encountered flying at supersonic speeds and needs for lighter materials, newer materials are coming into usage. Fiber-reinforced plastics (FRP) like the carbon-carbon variety, graphite compounds, molybdenum, beryllium, etc. are some examples of this type.

In the design of a missile or a rocket, aeroelastic effects are rarely taken into account. But in real life, due to aeroelastic effects following situations may occur:

- High aerodynamic forces may result in the mechanical failure of control surface(s) or stability fin(s), thus aerodynamic control of the missile may be lost.
- Deformations in the control surfaces or stability fins may reduce the lift force produced by these fin sets. Also reduced lift force in the control surfaces may result in reduced control effectiveness and manoeuvrability. Moreover stability characteristics of the missile may also change due to deformations.
- Deformations in the control surfaces may also change the centre of pressure location at the fin which may cause high hinge moments around hinge lines. Thus enough power may not be produced to control the missile aerodynamically.
- Vibrations that occur due to aeroelastic effects may affect the avionics of the missile. Sensors used in inertial measurement unit may be affected by these vibrations which would result in increased error in the flight computer calculations and trajectory. Also guidance systems of the missile may be affected from these vibrations that may reduce the hit probability and accuracy of the missile.

To overcome the problems described above, aeroelastic analysis should take place in the design of missiles and rockets. For different problems and situations, static and/or dynamic aeroelastic calculations, simulations and tests may be conducted. Aeroelastic analysis of rockets is an essential part of their design procedure. In most cases, the analysis is limited to calculation of the divergence velocity, sometimes leading to unrealistic prediction of the rocket response.

Much less interest has been given to study the dynamic behaviour of combined conical cylindrical shells coupled with fluid. This is mainly due to difficulties coupling the solid and fluid equations associated with two different geometries, conical and cylindrical.

In general two distinct types of Aeroelastic problems occur in nature. One involves the interaction of aero dynamic and elastic forces. Such interactions may exhibit divergent tendencies in a too flexible structure, leading to failure or in an adequately stiff structure, converge until a condition of stable equilibrium is reached. In this type of problem static or steady state systems of aerodynamic and elastic forces produce such aeroelastic phenomena as divergence and control reversal. The second class of problem involves the inertia of structure as well as aerodynamic and elastic forces.

Dynamic loading systems, of which gusts are of primary importance, include oscillations of structural components. If the natural or resonant frequency of the component is in region of the frequency of the applied loads then the amplitude of the oscillations may diverge, causing failure, and the presence of fluctuating loads is a fatigue hazard. For obvious reason we refer to these problems as dynamic. Included in this group are flutter, buffeting and dynamic response.

In this present work only static aeroelastic analysis is focused due to difficulties involved in the linking of loads and convergence of results during the dynamic analysis.

II. METHODOLOGY

Finite element methods are now widely used in the analysis of solids and structures, and they provide great benefits in product design. In fact, with today's highly competitive design and manufacturing markets, it is nearly impossible to ignore the advances that have been made in the computer analysis of structures without losing an edge in innovation and productivity. Various commercial finite-element programs are widely used and have proven to be indispensable in designing safer, more-economical products. In the analysis of fluids, significant advances also have been made during recent decades. A number of commercial programs have been developed, and applications in product design are increasing rapidly.

Numerical simulations have been used for quite some time in aeronautics, but their application in aerospace engineering is more recent. The total annual expenditure for flow simulations in aerospace engineering is still much smaller than for structural analysis, but the number of applications in fluid flow analysis is growing. This is largely due to valuable analysis capabilities that are now available for many practical cases of fluid flow in engineering applications. This new field of analysis is the fully coupled solution of fluid flows with structural interactions, commonly referred to as fluid-structure interaction (FSI).

The mechanical principles governing fluids and solids are the same, different characteristics in different difficulties in numerical simulations of fluids and structures. Now that these difficulties, for structures and fluids, have been largely overcome, the complex response of various combined fluid and solid media can be analyzed effectively. Exciting developments and applications are now on the horizon for FSI.

A. *Aim and Objectives*

Aim: To study the fluid structural interaction on Aeroelastic response over a two stage rocket body at different mach numbers.

Objectives: The main objectives of this project are the following

- Obtaining pressure distribution over rocket body at Mach numbers 1.5, 2.5 and 3.
- Calculation of Normal force coefficients with respect to mach numbers.
- Prediction of static aeroelastic behaviour in terms of structural deformations.

B. Project layout

This study involves the aeroelastic analysis, initially starts with flow analysis over the geometry at Mach numbers 1.5, 2.5 and 3 followed by dynamic analysis by linking the pressure distribution over the body obtained. Software's going to be used in this thesis is CATIA for modeling the object and then continued in ANSYS Workbench for Meshing and for Flow analysis followed by dynamic analysis. After obtaining pressure distribution over the body Normal force is going to be calculated using Numerical calculations.

C. Inference from literature survey

After going through a lot of published papers regarding the flow and structural interactions of launch vehicles, missiles and rockets, for accurate performance, and to reduce the failure of rockets, a study of fluid structural interactions based on aeroelastic response would be necessary. This can only be met by using a relatively conventional analysis employing a numerical analysis. Many experimental results from the previously published thesis have been referred and understood that Coupled Field Analysis is one of the kind of analysis which is responsible for countless useful effects in engineering. This project involves the Coupled Field Analysis performing both aerodynamic, structural analysis and taking different mach numbers in to consideration.

D. Motivation to the work

Fluid Structural Interaction has a critical impact on the design and performance of any structure in these days. The basic idea started with the basic idea of problems faced by aerospace vehicles within the atmosphere. If the aerodynamic loads are making any difference in structures, then the rocket vehicles which will get deformed because of aeroelastic loading. These vehicles which are acted upon by gravitational force and thermal forces may show some additional effects on the performance of the vehicle.

Many of the previous works were performed on single and simple structures, only few studies were carried on multistage rockets. The class dealing with problems where more than one physical effect is involved comprises multi-physics problems, among the most important of which is Coupled Field Analysis, challenging with respect to both modeling and computational issues. Coupling here is a very tough task to be accomplished i.e. coupling of flow analysis results with structural solver and assigning the obtained pressure loads. A two stage rocket body from the references is considered and some modifications are done to the design and additionally various flow velocities are applied to see the effects that are levied on the Rocket body.

III. MODELING AND NUMERICAL CALCULATIONS

A. Geometry Modeling

CATIA is chosen for modeling the two stage rocket body. CATIA (Computer Aided Three dimensional Interactive Application) consists of modules each Module specialized in specific design field. There are many types of nosecone shapes followed by cylindrical structured rockets are using for different purposes. The selection of shape is purely depends on the type of application for which it is being developed. Rocket having conical nose followed by cylinder and frustum cross section is modeled and analyzed in this project due to its wide range of applications. The model chosen for static aeroelastic analysis is shown in below fig 3.1.

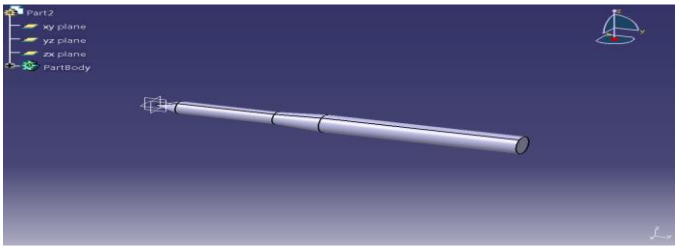

Fig 3.1: Three dimensional model of two stage rocket body

B. Flow Analysis

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.

FLUENT is a CFD software package to simulate flow problems. It uses the finite volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. The mesh models of domain and rocket body are inscribed in the below figures 3.2.1 and 3.2.2.

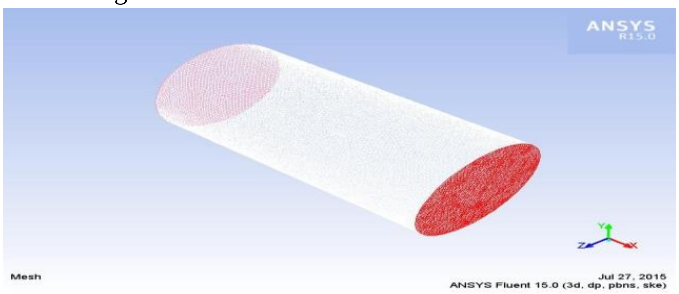

Fig 3.2.1: Display of domain with mesh in FLUENT solver

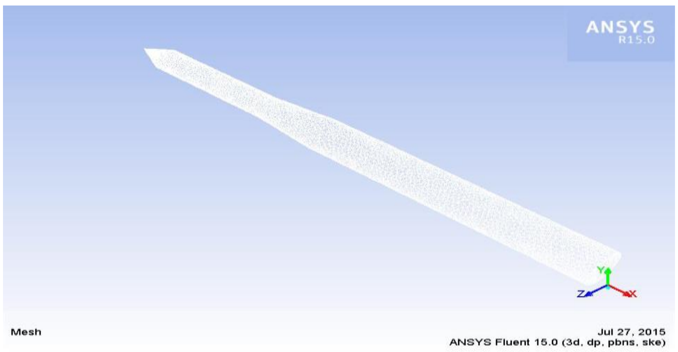

Fig 3.2.2: Display of rocket body with mesh in FLUENT solver

The domain including body shown above consists of 15,70,993 tetrahedral cells, 10,716 triangular wall faces, 31,21,888 triangular interior faces, 2,573 triangular pressure-far-field faces, 2,583 triangular pressure-outlet faces, 24,324 triangular wall faces. Advanced solver technology provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability.

The integration of ANSYS Fluent into ANSYS Workbench provides superior bi-directional connections to all major CAD systems, powerful geometry modification and creation with ANSYS Design Modeler, and advanced meshing technologies in ANSYS Meshing. The platform also allows data and results to be shared between applications using an easy drag-and-drop transfer, for example, to use a fluid flow solution in the definition of a boundary load of a subsequent structural mechanics simulation.

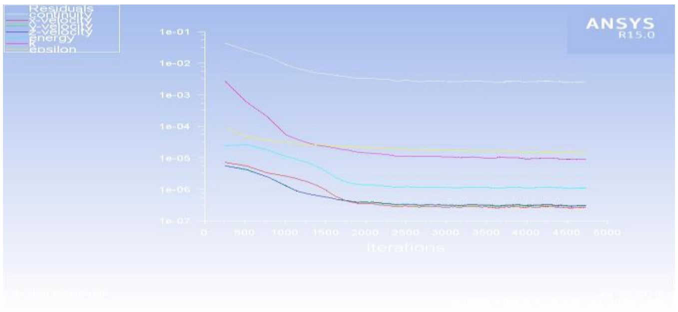

Fig 3.3: Residuals Plot at Mach number 2.5

After initializing the solution Calculation was run up to 5,000 iterations to get better and accurate visualization of flow field patterns at different mach numbers 1.5, 2.5 and 3. The residuals plot which shows the resulting iteration and convergence phenomena at 2.5 mach number is presented in figure 3.3.

In the same manner with the flow conditions described above flow analysis is carried at other mach numbers 1.5 and 3. After getting the results, pressure loads generated over rocket body at various mach numbers is imported into the structural analysis module and then static structural analysis is carried by linking the output file of fluid dynamics as input to the structural analysis solver.

C. Static Structural Analysis

Steps required for solving a problem: In performing any finite element analysis we must complete certain tasks which can be thought of as the steps required for completing the analysis. Regardless of what FEA tool is being used, these same tasks must be performed in order to complete the analysis. These tasks are listed below.

1. Generation of the mesh
2. Define/Assign material properties
3. Define the analysis type
4. Set loading and boundary conditions
5. Solve
6. Review the results

As the steps mentioned above, after linking the output of fluid analysis file to static structural analysis again the body is imported and then meshed. The block pictures of these are inserted below indicating figure 3.4 and it consists of 38389 Nodes, 25293 Elements of Tetrahedral shape. Material properties considered for this structure selected are the following since these are the materials with properties widely using in various applications. Young's Modulus is 70 GPa, with Poisson's ratio 0.35 having the Density 2700 Kg/m^3.

Finally Pressure loads are imported over the body obtained in the flow analysis by using the option called "Import Load" in the left side tree of present in the Static structural Analysis module. After importing the pressure it is indicated over the body as displayed in figure 3.4.

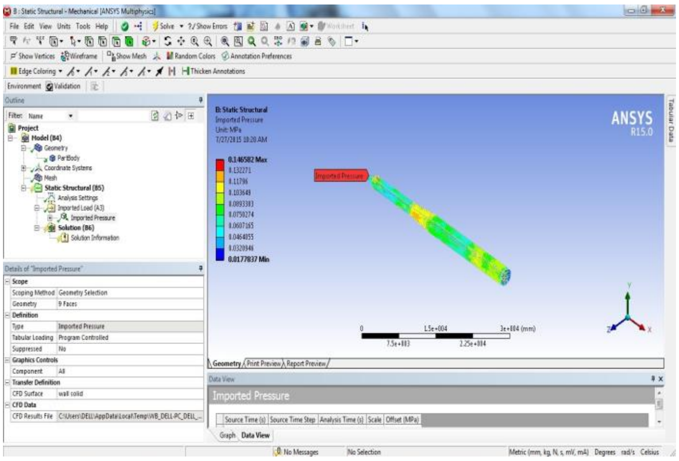

Fig 3.4: Imported Pressure over the Rocket body for Mach 3

While importing loads if the window shows any errors, they have to be rectified before proceeding to the solution. As the successful application of loads and all other boundary conditions it can be solved easily to obtain deformation and stress results. The same steps have to be repeated for other two mach numbers.

D. Numerical Calculations

In reality, the rigid geometry assumption holds for many engineering problems. In many cases, where the structures are flexible, fluid-structure interactions become important. In this work, a two stage rocket body is studied. Static aeroelasticity considers the non-oscillatory effects of aerodynamic forces acting on the elastic structure. Because of the elastic nature of the rocket body, aerodynamic forces acting on the rocket contribute to structural deformation. This deflection of the structure tends to redistribute the aerodynamic forces acting on the rocket, and this interaction continues by leading to each other. As a result, a coupling approach is used to solve the static aeroelastic problem as a significant part of rocket design workflow.

Normal force can be calculated from the above pressure coefficient distribution using the relation described. Trapezoidal method has been chosen for calculating Normal force, in numerical analysis, the trapezoidal rule also known as trapezium rule, it is a technique used for approximating the definite integral mentioned below. This rule has an advantage of faster convergence.

$$\int_{x_0}^{x_n} f(x)dx = \frac{h}{2}\{(y_0 + y_n) + 2(y_1 + y_2 + ------- + y_{n-1})\}$$

Where, $x_n = x_0 + nh$, 'n' is the number of intervals, y terms are the corresponding function values with respect to x values.

According to Trapezoidal rule Normal force values are found to be 2.617KN, 1.3713KN and 1.3242KN at Mach numbers 1.5, 2.5 and 3 respectively. And this variation is shown in the below figure 3.5. Normal force value decreases due to rocket deformations. Accordingly, the flight trajectory may be affected by the change of these aerodynamic force variations. In this work, the far-field free stream condition is standard temperature and pressure (288K, 101.325 KPa) and the far-field boundary reflecting boundary conditions. The air is assumed as an ideal gas. The solution method is implicit formulation and Second order Upwind.

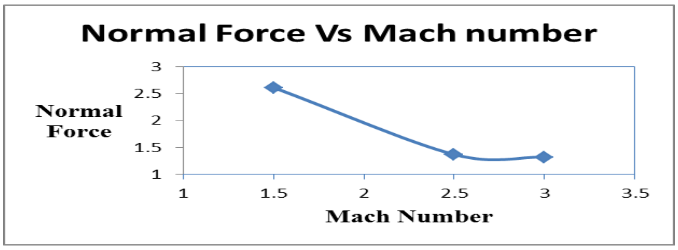

Fig 3.5: Variation of Normal Force with Mach Number

The changes of Normal force values with Mach number of two stage rocket body compared are shown in Figure 3.5. It reveals that the drag and lift force coefficients also decrease with mach number due to elastic deformations.

The changes of Normal force values with Mach number of two stage rocket body compared are shown in Figure 3.5. It reveals that the drag and lift force coefficients also decrease with mach number due to elastic deformations.

IV. RESULTS AND DISCUSSIONS

Initially the Normal force values are calculated and compared with Mach number. In this present section Results obtained in Flow analysis continued to the Static Structural analysis by means of fluid-structural interaction, thus Aeroelastic effects over two stages Rocket body with mach number in terms of deformations are going to be represent schematically. For better understanding present concept is divided into sub-sections described below.

A. Computational Flow Simulation Results

Computations were performed to understand the flow field around a model of two stage rocket body. Computations using the commercially available software FLUENT were carried out for three dimensional fully developed flows at three different Mach numbers. The results of computations performed are discussed in detail in the following sections. For all the cases, maximum of 5000 iterations were performed until desired convergence is obtained. Compressible effects were considered for comparing the results obtained for viscous flows. The standard k-epsilon model was adopted for the viscous computation after checking with the other models like inviscid and k-ω.

Static Pressure distribution over the body at three different Mach numbers 1.5, 2.5 and 3 obtained in the flow simulation are shown in the below figures 4.1(A), 4.1(B) and 4.1(C) respectively. From these contours it has been observed that as the Mach number increases, Shock wave attached moves closely to the body. So it can estimate that as the mach number increases it cause more deformation to the structure due to Fluid-Structural Interference. And the figures 4.2(A), 4.2(B) and 4.2(C) give the clear view of distribution of Static Pressure across its longitudinal axis.

As the Mach number varies, Aerodynamic Pressure distribution i.e. aerodynamic loading changes thus the behaviour of Rocket body is need to be studied, how the Aeroelastic response in terms of structural deformations is carried in the following section.

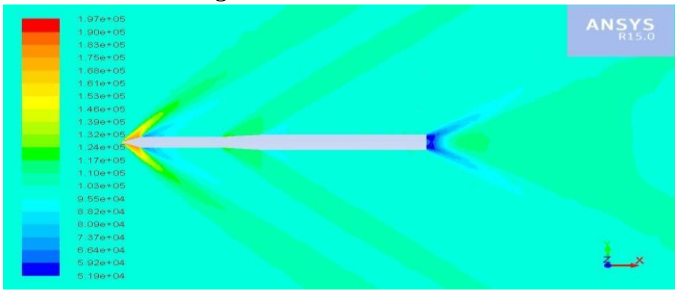

Fig 4.1 (A): Contours of Static Pressure at M=1.5

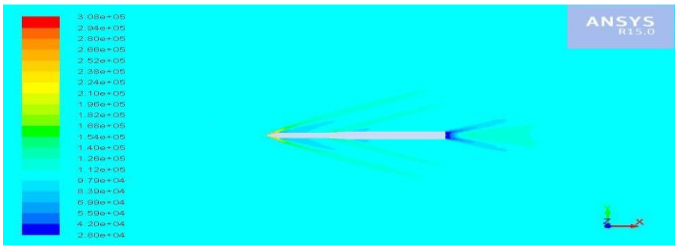

Fig 4.1 (B): Contours of Static Pressure at M=2.5

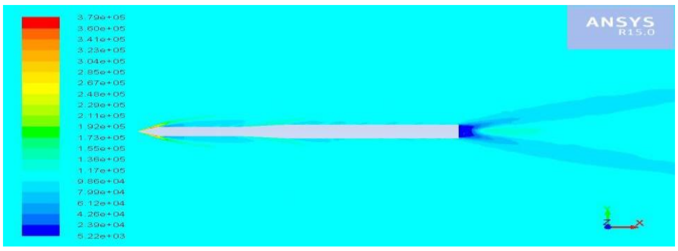

Fig 4.1 (C): Contours of Static Pressure at M=3

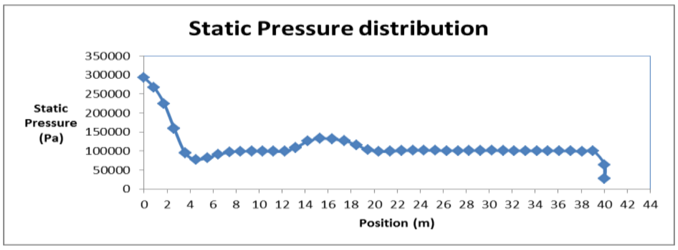

Fig 4.2 (A): Static Pressure distribution over the Rocket body at M=1.5

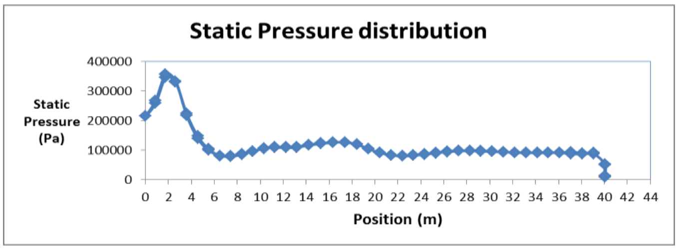

Fig 4.2 (B): Static Pressure distribution over the Rocket body at M=2.5

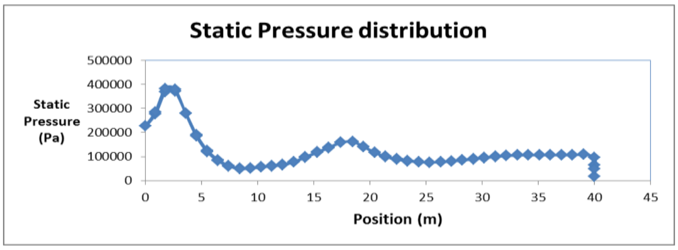

Fig 4.2 (C): Static Pressure distribution over the Rocket body at M=3

B. Static Aeroelastic Simulation Results

The structural grid of the flow field around the required model is constructed to compute the aerodynamic pressures distribution along the whole rocket body using CFD. Then, map the pressures at the CFD grid points to be replaced by forces on the CSD nodes. Consequently, stress and deformation distributions of the structure are obtained by solving it. To obtain accurately the deformation and aerodynamic load distributions of the rocket, a one-way fluid-structural interaction coupling method is applied and compared with mach numbers. The deformations of the two stage rocket body are shown in Figure 4.3. These deformations are increasing with increase of mach numbers. It is due to the normal force of the deformed rocket becoming smaller than the rigid rocket. Obviously, with increasing the Mach number, the deformations of the rocket become larger as presented in Figures 4.3(A), 4.3(B) and 4.3(C).

From these plots it reveals that the drag and lift force coefficients also decrease due to elastic deformations, which contributes to the stability reduction of the rocket. Comparison of the Total deformations with mach number for two stage rocket body considered is given in Table 4.1.

Table 4.1: Comparison of maximum Pressure and Total deformation

S.No	Mach number	Maximum Pressure (Pa)	Maximum Total deformation (mm)
1	1.5	1.97e5	2.0831
2	2.5	3.08e5	3.603
3	3	3.79e5	19.473

From these results, it has been observed that there is an unexpected and sudden increase of pressure as well as total deformations occurs with rise in mach number. This leads to unstable and wide variation in the performance of rocket.

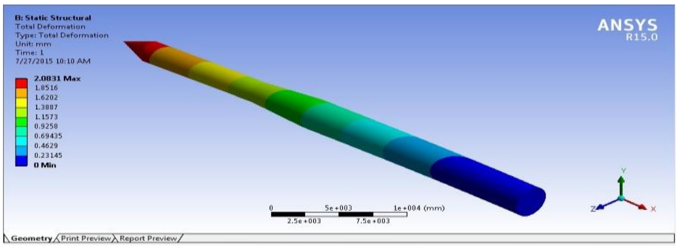

Fig 4.3 (A): Total deformation distribution at M=1.5

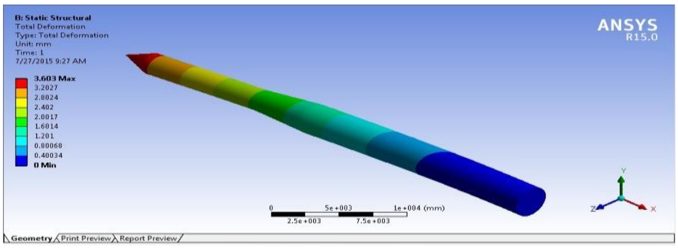

Fig 4.3 (B): Total deformation distribution at M=2.5

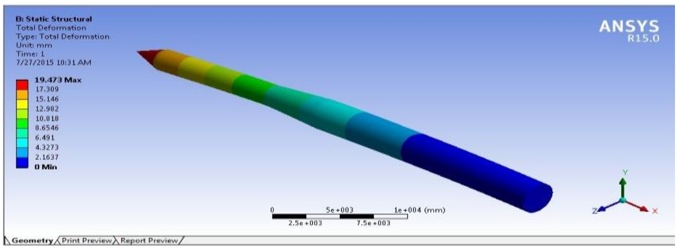

Fig 4.3 (C): Total deformation distribution at M=3

V. CONCLUSIONS

Computational studies were carried out to get an understanding of the flow field around two stage rocket body Mach 1.5, 2.5 and 3. Three dimensional simulations of the flow field using FLUENT were performed. k-epsilon model was adopted to capture the flow field. Computations were validated through a simulation of flow field around the similar bodies by earlier investigators. After a good agreement with reported results, simulation of the present case was carried out and compared with respect to mach number. Results obtained through the flow analysis linked to static structural analysis using ANSYS Workbench were performed. Thus the Aeroelastic responses on two stage rocket body through fluid-structural interaction tests are discussed in the previous chapter.

The following important observations were made from the results obtained through computations:

The basic flow structure around two stage rocket body was captured through 3D computations. Pressure contours showed the shock wave structure, near the nose cone of the main body. Shock and boundary layer interferences were observed near the frustum structure. Comparison of k-epsilon modeled viscous simulations around the body showed predicts well for the flow field features of fluid-structural interactions. A good comparison of Aeroelastic results was achieved. Numerical calculations were performed to estimate Normal force, as the mach is increased the Normal force value is decreased.

VI. SUGGESTIONS TO FUTURE WORK

3Dimensional Computations were carried out to get an understanding of the flow field around a two stage rocket body at a free stream Mach numbers 1.5, 2.5 and 3, at zero angle of attack. Fluid structural interaction on aeroelasticity response of rocket at different mach numbers was carried. After obtaining the results it was observed that further studies related to the topic can be carried out for getting a better understanding of the flow phenomenon. Some of the suggested work is outlined below:

Finer 3D grid can be generated for better capturing of the shock waves near the nose cone. Study of flow field by keeping the model at various angles of attack. Base flows were not considered in the present work, it can be studied further. In this present work one way coupling is used. Two way coupling can be carried further. For the present study only computations were performed. Experiments and computations can be carried out by considering the protrusions for a complete rocket body.

ACKNOWLEDGMENT

The authors would like to acknowledge and express sincere thanks to the technical help and guidance of Prof. Dr. Amalesh Barayi, Prof. Dr.P.Dass.

REFERENCES

[1] Laith K. Abbas, Dongyang Chen, and Xiaoting Rui, "Numerical Calculation of Effect of Elastic Deformation on Aerodynamic Characteristics of a Rocket", International Journal of Aerospace Engineering, Volume 2014, Article ID 478534.

[2] J. D. Baum,H.Luo, andE. L.Mestreau, "Recent developments of a coupled CFD/CSD methodology," in Computational Science, vol. 2073 of Lecture Notes in Computer Science, pp. 1087–1097, 2001.

[3] J.Gai and F. Liu, "Static aero-elastic computation with a coupled CFD and CSD method," Tech. Rep. AIAA-2000-0717, 2000.

[4] E. Bas¸kut and A. Akg¨ul, "Development of a coupling procedure for static aeroelastic analyses," Scientific Technical Review, vol. 61, no. 3-4, pp. 39–48, 2011.

[5] J. R. Wright and J. E. Cooper, Introduction to Aircraft Aeroelasticity and Loads, JohnWiley & Sons, 2007.

[6] N.Sreedhar, G. Ravindra Reddy, "Flow Simulation over a Multi-Stage Launch Vehicle with Strapons Using CFD Techniques", International Journal of Computer Trends and Technology (IJCTT) – volume 4 Issue 7–July 2013.

[7] "Computation of flow past aerospace vehicles"S.K.Chakrabartty,K. Dhanalakshmi and J.S. Mathur".

[8] Kamakoti, R., Shyy, W. "Fluid- Structure Interaction for Aeroelastic Applications", Progress in Aerospace Sciences 40 (2004) 535-558.

[9] Slone, A., Pericleous, K., Bailey, C., Cross, M. "Dynamic Fluid-structure Interaction Using Finite Volume Unstructured Mesh Procedures", Computers and Structures 80 (2002) 371-390.

[10] Cai J., Liu F., Tsai H.M., Wong A.S.F., "Static Aero-elastic Computation with a Coupled CFD and CSD Method", AIAA 2001-0717, Aerospace Sciences Meeting and Exhibit.

[11] Ralph J. Mnruca, An Empirical Method for determining Static distributed Aerodynamic loads on axisymmetric multistage launch vehicles", NASA TN D-3283.

[12] Anoop Thankachen, Santosh kumar, Design Optimization and Analysis of Rocket Structure for Aerospace Applications, International Journal of Engineering Trends and Technology (IJETT) – Volume 24 Number 6- June 2015.

[13] Subramanyam.B, S.Venkateswarlu, Prangyadeeptha Choudhury, Static and Dynamic Analysis of Rocket Structure, International Journal of Technology and Engineering Science [IJTES] Vol 1 (5), pp 405 – 414, August 2013.

[14] E. D. V. Bigarella, J. L. F. Azevedo O. A. F. Mello, Normal Force Calculations for Rocket like Configurations.

[15] R. van Loon et al. Comparison of various fluid-structure interaction methods for deformable bodies.

[16] Gene Hou, Jin Wang2 and Anita Layton, NumericalMethods for Fluid-Structure Interaction- A Review,Commun. Comput. Phys., 12 (2012), pp. 337-377.

[17] Wang, J. and Layton, A., Numerical simulations of fiber sedimentation in Navier-Stokes flows, Communications in Computational Physics, Vol. 5, No. 1, 2009, pp. 61-83.

[18] Ana Cristina Avelar et al., Investigation of the Flow Patterns on a Sounding Rocket Fore-body Section,29th Congress of the International Council of the Aeronautical Sciences, Russia, September 7-12, 2014.

[19] Gregory J. W., Asai K., Kameda M., Liu T. and Sullivan J. P. A Review of Pressure Sensitive Paint for High Speed and Unsteady Aerodynamics. Proc of the Institution of Mechanical Engineers, Part G, Journal of Aerospace Engineering, Vol. 222.

[20] Houghton E. L. and Carpenter P. W. Aerodynamics for Engineering Students, 5th ed. – Oxford, Boston: Butterworth/Heinemann, 2003.

[21] Brandt, S. A., Stiles, R. J., Bertin, J. J., and Whitford, R., Introduction to Aeronautics: A Design Perspective, 2nd ed., AIAA Education Series, Reston, VA, 2004.

[22] Edson Basso et al., Aerodynamic study of sounding rocket flows using Chimera and patched multiblock meshes, J. Aerosp.Technol. Manag., Vol.3, No.1, pp. 87-98, Jan. - Apr., 2011

[23] Papa, J.C., Azevedo, J.L.F., 2003, "Three dimensional fl ow simulations over a typical sounding rocket", Proceedings of the 21st AIAA Applied Aerodynamics Conference, AIAA Paper No. 2003-3421, Orlando, FL, USA.

[24] Fletcher, C.A.J., 1991, "Computational techniques for fl uid dynamics," Vol. 2, Spring-Verlag, NewYork, USA.

[25] Pulliam, T.H., Steger, J.L., 1980, "Implicit fi nite-difference simulations of three-dimensional compressible fl ow," AIAA Journal, Vol. 18, No. 2.

[26] "Static Aeroelastic analysis of a generic slender missile using a loosely coupled Fluid Structure Interaction Method", Thesis, Mechanical Engineering Department, Middle East Technical University.

[27] "Fluid-Structure Interactions" by Klaus Jurgen Bathe.

[28] "Space Mission Analysis and Design", third edition, by Larson and Wertz.

[29] J. D. Anderson Jr., Fundamentals of Aerodynamics, McGraw-Hill, 1984.

[30] White, F., M., "Viscous Fluid Flow", McGraw and Hill, 1991.

[31] http://sciencelearn.org.nz/Contexts/Rockets/Science-Ideas-and-Concepts/Rocket-aerodynamics.

[32] https://en.wikipedia.org/wiki/Rocket.

[33] http://exploration.grc.nasa.gov/education/rocket/rktaero.html.

[34] http://catiav5cad.blogspot.in/2013/03/catiav5cadcamintroduction.html.

[35] FLUENT 15.0 User's Guide, 2015.

[36] ANSYS web page: www.ansys.com.

BIOGRAPHIES

Mr. kothamasu Kiran did M.Tech in Aerospace Engineering with distinction from esteemed Institute of Aeronautical Engineering affiliated to JNTU-Hyderabad, India. After completing B.Tech in Aeronautical Engineering from esteemed MLR Institute of technology affiliated to JNTU-Hyderabad with distinction.

Mr. Cholleti Eshwar Reddy pursuing M.Tech in Thermal Branch from PRIT affiliated to JNTU-Hyderabad, India. After completing B.Tech in Aeronautical Engineering from esteemed MLR Institute of technology affiliated to JNTU-Hyderabad with distinction.