Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

INVESTIGATION OF FLOW FIELD AROUND REUSABLE LAUNCH VEHICLE WITH DIFFERENT FUSELAGE CROSS SECTION AT SUPERSONIC SPEED

R Nesalingam, R. Mohankumar, K. P. Arulshri, R. Ramasamy, M. Venkatachalam, Banu T
nitradhakrishnan@nehrucolleges.com
Department of Aeronautical Engineering, Nehru Institute of Technology, Coimbatore

***Abstract*—In present study, an computational works has been carried out on different cross sections circular, hemispherical, triangular and rectangular models by taking the boundary conditions of Saturn V launch Vehicle at mach numbers 1.5, 2.7 and 4.4 considering flat angles to obtain the complex flow features and forces acting on the RLV models at supersonic speed. From the computation the losses in reusable launch vehicles at various sections has been determined by considering Three factors Strength of the shock, Aerodynamic heating and retardation of velocity in the base region by taking rate of change of entropy, temperature and velocity respectively. losses from these three factors are compared and a brief investigation is done on the models from which it was concluded that based on shock strength and aerodynamic heating circular cross-section had good aerodynamic characteristics and it was found that triangular model has a good aerodynamic characteristics as for as body of launch vehicle is concerned**

***Keywords*— RLV, Mach**

I. INTRODUCTION

Space exploration for space laboratories, space stations, space defense etc., is one of the strategic and prestigious projects of both developed and developing nations in the modern era. In the current scenario, satellites are launched for communication and educational purpose. These satellites can be launched into orbit by using any launch vehicles. In general launch vehicles are classified into two types, Expendable Launch Vehicles and Reusable Launch Vehicles. The main difference between these two vehicles is, we can reuse the parts of the Reusable Launch Vehicle, but expendable launch vehicles are designed for one time use. They usually separate from their payload and break up during atmospheric re entry. More over because of that reusability factor, the RLVs are cost effective when compared to the conventional launch vehicle like expendable launch vehicle. And RLVs are environmentally suitable, as the space debris can reduce by reusing the parts. In future space debris are going to be a big problem, because all countries in the world have started the space exploration program. Some of the applications of a Reusable Launch Vehicle are launching satellites, human transportation for space exploration, and transporting material to space stations, and retrieval of satellites from orbit, rapid passenger or package services to many points of the globe and tourism services that may carry civilians into space for entertainment.

The aerodynamic configuration of a RLV should be capable of providing required lift to drag ratio and controllability in all flight regimes. The basic aerodynamic requirements of a RLV are, The configuration should meet the mission objectives e.g. aero- thermodynamic flight environment, thermal protection system etc. The lift should be sufficient to support the overall mass of the RLV. RLV should have the capability to cruise at subsonic and hypersonic speeds. During landing the maximum angle of attack is usually limited to 20 degrees and speed to about 70 m/s to 80 m/s for safe landing. The vehicle should be statically stable ($CM\alpha < 0$) in subsonic speeds and neutrally stable ($CM\alpha = 0$) in hypersonic speeds.

Fuselages are generally the houses of payloads and navigational and control systems in launch vehicles. So the fuselage cross sectional shape will decide the amount of payload that can be launched at a time. At subsonic speed, cross sectional shapes of slender bodies (fuselages) have more effects on aerodynamic characteristics of the RLVs. The aerodynamic characteristics of a Wing- Body type RLV with small wings at the rear of fuselage could be improved by changing the cross sectional shape of fuselage. Some typical fuselage cross sections are mentioned below, Circular, Rectangular, Hemispherical and Triangular.

Apart from fuselage, wing design also plays a major role in the aerodynamic characteristics of the body. The complexity of wing plan form design increases for a winged Reusable Launch Vehicle (RLV), where as apart from aerodynamic and performance constraints, configuration has to be designed to reduce the aerodynamic heating during re- entry. The guidelines for the study of wing planform of a RLV as follows: Subsonic maximum landing speed is about 85 m/s, at an angle of attack of 14 degrees. Maximum subsonic untrimmed $L/D \geq 4.5$. Leading edge sweep angle of the wing must be ≥ 450 to minimize aerodynamic heating. Minimum pitching moment coefficient at typical subsonic, supersonic, hypersonic Mach numbers at corresponding angles of attack. The vehicle has to be stable/ near neutrally stable at complete trajectory Mach numbers and angles of attack. The vehicle must be trimmable/ controllable at complete trajectory Mach numbers and angles of attack.

Prediction of RLV aerodynamics with this separated flow over the wing depends on many parameters such as Reynolds number, compressibility, wing leading edge bluntness, shock interaction and many other factors. The objective of the present study is to investigate the flow field around the RLV with different fuselage cross sections. It has been planned to investigate overall flow features adopting computational work.

Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

Fig1.1: Body type RLV

Fig 1.2 : Lifting body type

INTRODUCTION OF COMPUTATIONAL FLUID DYNAMICS

To understand the complex flow field, over the RLV, complete three dimensional computations over the model is necessary. This can be done either by developing a code or by available CFD software. In the present study Computation has been attempted on four different cross-sections of fuselage of RLV model with the commercially available software ANSYS. The present Chapter discusses in detail about the solution methodology, problem setup and solver settings.

STEPS INVOLVED IN CFD PROCESS

Physical Phenomenon, Solid Model, Fluid Model, Meshing and physical Equations governing the flow.

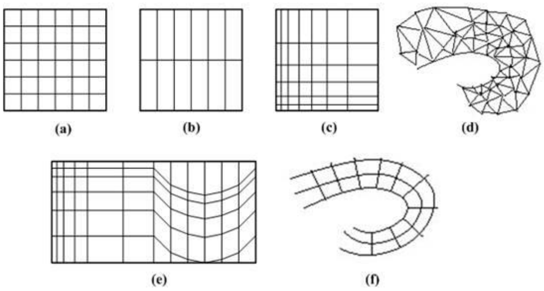

Fig. 1.3 Grid types. (a) Cartesian grid (b) Regular grid (c) Rectilinear grid (d)curvilinear grid (e) Block structured grid (f) Unstructured grid.

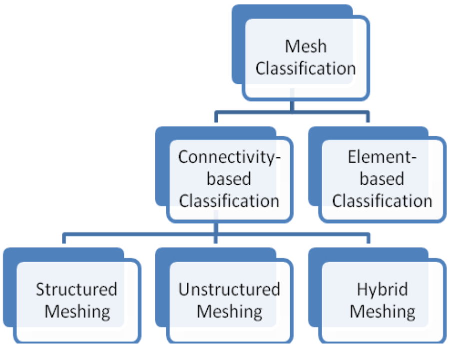

Fig. 1.4: Mesh classifications

II. LITERATURE REVIEW

Currently there is considerable interest in developing technology for design of Reusable Launch Vehicle specifically a reusable two stage to orbit. The development of extremely efficient and manoeuvrable RLV's has resulted in flight regimes that involve high angle of attack, raising interest in the study of three-dimensional separated flows. This Chapter briefly presents the theoretical, experimental and computational studies that have published concerning the flow physics.

Sundeep[1] conducted the experimental and computational studies to obtain the complex flow field features on reusable launch vehicle. The qualitative and quantitative results obtained from the experiments were been analyzed. The qualitative experiments performed were oil flow and tuft flow visualization and quantitative experiments were measurement of aerodynamic forces, moments and surface pressure. The experimental results were compared with computations made over the double delta wing and RLV using FLUENT software.

Karthikeyan.S[2] conducted an experimental and computational works to obtain the complex flow features and forces acting on the RLV fuselage and full RLV model at low subsonic speed. From the computation it could be concluded that a triangular fuselage has good aerodynamic characteristics when compare to the other cross section of fuselages. Hence the experiments were carried out only on triangular fuselage and RLV with triangular fuselage. Force measurement and Oil flow visualization were made on fuselage model and full RLV at 16 m/s, 20 m/s and 25 m/s. All experiments were conducted in low speed subsonic tunnel. Qualitative and quantitative analysis has been used to observe the effect of angle of attack on aerodynamic characteristics and Reynolds number was obtained on triangular fuselage and full RLV.

Takuro, et al.[3] conducted an experimental and numerical study for the configuration of reusable launch vehicle (RLV). Two wing- body models, whose cross sections is square and triangle, were used in the experiments to accumulate data for change of side slip angles. Oil- flow technique was applied to visualize the flow field on model surfaces. As for numerical

Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

study, three different cross section models were chosen for application to CFD method. In these experiments, not only longitudinal but also lateral characteristics were evaluated. Numerical results showed that cross sectional shapes of fuselages has significant importance to form the separated vortices in subsonic region.

Kenji, et al.[4] performed an experimental study using lateral blowing to improve aerodynamic characteristics of RLVs (Reusable Launch Vehicles). Square and triangular cross sections of fuselage were considered to investigate effects of shape of cross section of fuselage on aerodynamic characteristics during lateral blowing. Results showed that lateral blowing increases the lift off over a wide range of angles of attack. Especially a significant increase in lift is achieved in the case of a "square" model at high angle of attack. The lift of the model with small sweepback angle and a rectangular wing was found to be larger than that of the other wings in higher angles of attack. As a result of calculation of landing process, lateral blowing makes possible a shorter landing distance or a smaller landing velocity of a RLV.

III. RESEARCH METHODOLOGY

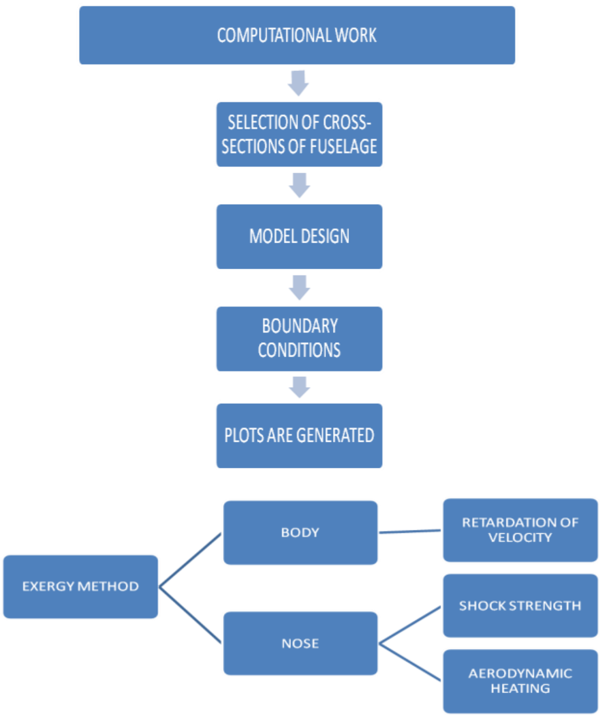

Fig 2.1: Research methodology chart

As in every CFD package solution can be divided into three parts. They are Pre-processing, Solution and Post processing.

GRID GENERATION ON DIFFERENT FUSELAGE MODELS

Computational work has been carried out on four different fuselage cross section, Circular, Rectangular, Hemispherical and Triangular

Structured grid has been made for all the fuselage model. The first cell distance has been maintained as 0.1 for all the cases from the body surface. Inlet and Outflow has been kept at distance of 5L from the body. Unsteady, laminar solutions were obtained in order to observe the force acting on the body.

GOVERNING EQUATIONS

Fluent uses a control volume based technique to convert the following governing equation as conservation of mass, conservation of momentum equation and conservation of energy to algebraic equation that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equation that conserves each quantity on a control volume basis.

Equation of Conservation of Mass

$$\frac{\partial \rho}{\partial t} + \Delta(\rho v) = 0$$

Equation of conservation of momentum

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial x} + \rho f_x$$

Equation of conservation of energy

$$\rho \frac{D}{Dt}\left(e + \frac{V^2}{2}\right) = \rho \dot{q} + \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k \frac{\partial T}{\partial z}\right) - \frac{\partial (up)}{\partial x} - \frac{\partial (vp)}{\partial y} - \frac{\partial (wp)}{\partial z} + \frac{\partial (u\tau_{xx})}{\partial x}$$

$$+ \frac{\partial (u\tau_{yx})}{\partial y} + \frac{\partial (u\tau_{zx})}{\partial z} + \frac{\partial (v\tau_{xy})}{\partial x} + \frac{\partial (v\tau_{yy})}{\partial y} + \frac{\partial (v\tau_{zy})}{\partial z} + \frac{\partial (w\tau_{xz})}{\partial x} + \frac{\partial (w\tau_{yz})}{\partial y} + \frac{\partial (w_{zz})}{\partial z} + \rho f \bullet V$$

CONDITIONS	MACH NO	FREE STREAM PRESSURE (Pa)	FREE STREAM TEMPERATURE(K)
1	1.5	12111	217
2	2.7	2250	221
3	4.4	151	264

IV. INVESTIGATION OF FLOW FIELD

Prediction of best design of different fuselage cross-sections by changing the angle of attack can be done by estimating the Cl and Cd but for fuselage without changing angle of attack it is hard to predict the best design, and this

Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

analysis is purely done by taking flat angle so it is necessary to identify the various sources of efficiency loss and the quantity of efficiency loss associated with each source within the system. Although energy cannot be created or destroyed, it can be transformed into unavailable forms.

BOUNDARY LAYER

The theory that the flow surrounding an immersed body could be divided into two parts was first proposed by L.Pranadtl he stated that the viscosity in the major part of the flow field has a negligible effect on its motion and consequently in most cases could be ignored. The other part of the flow field consisted of a narrow region next to the body where the viscous forces are not negligible. This narrow region near the body is known as the boundary layer. In the boundary layer the velocity of the fluid increases rapidly from zero at the surface to a value which then changes slowly with increasing distance from the body. At an infinite distance from the body, the boundary layer and the free stream merge smoothly together. For practical applications, the boundary layer conditions approach the free stream condition very rapidly within a relatively small distance from the surface.

RETARDATION OF VELOCITY

The boundary is one of the key aspect where some amount of work is done because of shear force so it causes the retardation of velocity along the longitudinal axis within the boundary layer this difference in velocity can be found by taking two specific point in the flow field within the boundary layer such that both are separated by a distance along the longitudinal axis and this sort of energy loss is mainly because of the surface roughness of the boundary layer and the exposed area and by finding the retardation of velocity the efficiency of the fuselage wall can be estimated

WAKE BOUNDARY DETERMINATION

If the assumption is made that the launch vehicle is a projectile with no exhaust plume, the pressure behind the body in the wake is assumed to be the free stream static pressure. The assumption of no exhaust plume allows calculation of the velocity change as a result of the boundary layer on the body, but ignores the effects of boundary layer/plume interaction. With the pressure in the wake being approximately equal to static pressure and the stagnation pressure behind the shock, the Mach number in the wake can be determined The difference between the Mach number in the wake and in the flowfield can be determined and the point in the longitudinal axis has to be cheked for variation of velocities with free stream velocities and the wake boundary layer has to be estimated from which the fuselage having efficient wake boundary layer can be analyzed.

V. RESULT AND DISCUSSIONS

EFFICIENCY OF NOSE

STRENGTH OF THE SHOCK

The strength of the shock depends upon mach number and the shape of nose of launch vehicle so by determining the strength of the shock the best nose section can be determined

Table 5.1 : Hemispherical model

Mach	Before Shock	After Shock	Rate of change
1.5	-351.502	-323.273	28.229
2.7	-303.942	-81.7933	222.1487
4.4	-121.12	322.185	443.305

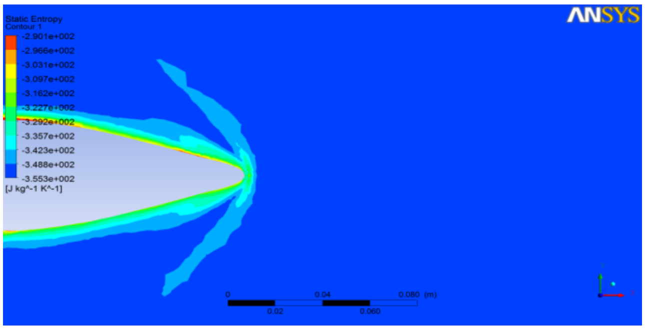

Fig.5.1 condition:1(Hemispherical model entropy plot)

Table 5.2 Circular model

Mach	Before Shock	After Shock	Rate of change
1.5	-351.507	-326.734	24.773
2.7	-303.546	-127.211	176.335
4.4	-121.412	303.933	425.345

Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

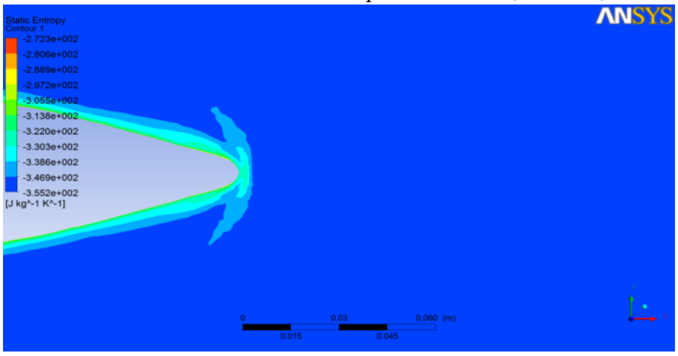

Fig.5.2 condition:1(Circular model entropy plot)

Table 5.3Triangular model

Mach	Before Shock	After Shock	Rate of change
1.5	-349.891	-325.164	24.727
2.7	-306.604	-72.3717	234.232
4.4	-120.986	421.781	542.767

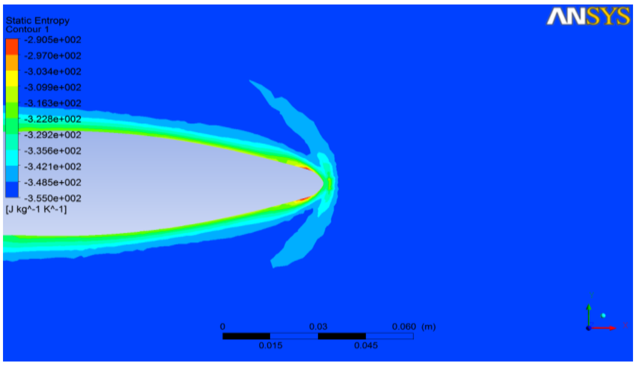

Fig.5.3 condition:1(Triangular model entropy plot)

Table 5.4 Rectangular model

Mach	Before Shock	After Shock	Rate of change
1.5	-348.884	-320.36	28.534
2.7	-306.543	-53.547	252.996
4.4	-121.677	614.324	736

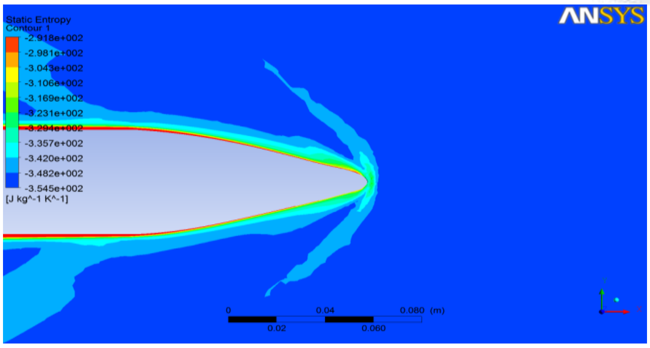

Fig 5.4 condition:1(Rectangular model entropy plot)

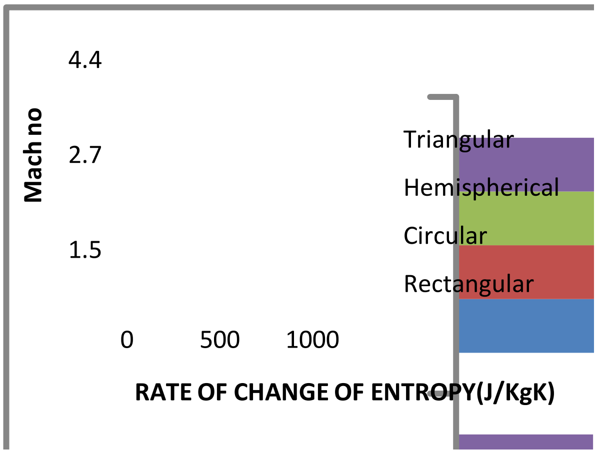

Fig 5.5 Strength of the shock (MACH NO VS ENTROPY PLOT)

Based on the plot obtained from the computational report it is found that the strength of the shock is lower for circular section when compared with the other sections and then comes the hemispherical and triangular cross-sections when the strength of shock is less then the energy required to overcome the shock gets reduced thereby reducing the amount of fuel consumed.

AERODYNAMIC HEATING

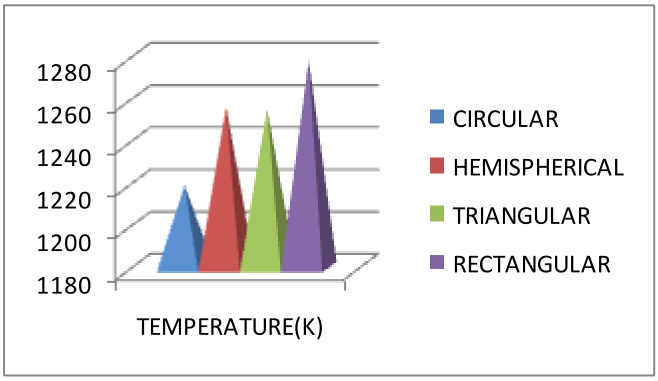

Fig 5.6 Aerodynamic heating (Temperature plot)

Special Issue on Proceedings of International Conference on Materials Manufacturing and Nanotechnology, June 2021. International Journal of Aquatic Science, Vol 12, Issue 03, 2021

Considering the nose of RLV from the results obtained from the strength of shock as well as aerodynamic heating it was concluded that the circular cross section has the effective reduction in both the strength of the shock and in the heating effect.

VI. CONCLUSION

In present days, There is a great rush towards space administration and research so design of an efficient launch vehicles for transportation and for other applications has a great importance. There has been an increased emphasis on improving the efficiency of vehicles used for space launch services. In order to improve efficiency of launch vehicles it is necessary to predict the losses in various sections. In the present study, the computational analysis is done and various plots are produced above from which efficient models are taken.

The model of launch vehicles excluding wings consists of nose and body this report has concentrated in these two regions, the shock waves plays a major role in the supersonic regime more energy is spent to overcome it, the shape and the strength of shock waves depends upon the the shape of the nose and the mach number at which the vehicle operates by designing an efficient nose the strength of shock waves can be minimized.

Aerodynamic heating, this has a major significance at high supersonic speeds were this factor is also a function of shape of the nose and mach number so considering these two factors it was found that from computational report that among the four models the circular cross-section is efficient compared to the other sections as for as nose of the launch vehicle is concerned.

While considering the body of the launch vehicles the boundary layer is the major contributor of losses in the vehicles based on the computational results it was found that the triangular cross-sections suffers less when compared to others models. So if an axis symmetric body is to be designed for launching purpose. The combination of circular and triangular model would serve with high aerodynamic characteristics as for the compared models are considered.

VII. REFERENCES

A. Anderson, J.D., Modern Compressible Flow With Historical Perspective, McGraw Hill Book Co., 1990.

B.Anderson, J.D., Fundamentals of Aerodynamics, McGraw Hill Book Co., 1984.

C. Benjamin F. Roper Exergy decrease in shock waves and boundary layers of space launch vehicles March, 1993

D. D.R. Chapman, National Advisory Committee for Aeronautics Technical Note 2137, An Analysis of Base Pressure at Supersonic Velocities and Comparison With Experiment, July 1950.

E. E.S. Love, National Advisory Committee for Aeronautics Technical Note 4170, A Reexamination of the Use of Simple Concepts for Predicting the Shape and Location of Detached Shock Waves, December 1957.
`

F. J.O.Reller and F.M. Hanaker, National Advisory Committee for Aeronautics Technical Note 3393, An Experimental Investigation of the Base Pressure Characteristics of Non Lifting Bodies of Revolution at Mach Numbers from 2.73 to 4.98,March 1955.

G. Kubota, H., “Some Aerodynamic and Aero thermodynamic Considerations for Reusable Launch Vehicles,” 34th AIAA Fluid Dynamics Conference and Exhibit, AIAA 2004- 2428.

H. Mortia, “Experimental Study on Effects of Fuselage Configuration for RLV on Aerodynamic Characteristics,” 34th AIAA Fluid Dynamics Conference and Exhibit, AIAA 2004- 2151, 8 p.

I. Schlichting, H., Boundary Layer Theory, 7th Ed., McGraw Hill Book Co., 1979.

J. Von Karman, T., and Moore, N.B., Resistance of Slender Bodies Moving With Supersonic Velocities With Special Reference to Projectiles, A.S.M.E., Transactions, pp. 303-310, 1932.

K. W.E. Moeckel, National Advisory Committee for Aeronautics Technical Note 1921, Approximate Method for Predicting Form and Location of Detached Shock Waves Ahead of Plane of Axially Symmetric Bodies,1949.

L. Yasuhiro, T., Takuro, I., Shigeru, A., Kenji, T., and Mitsumasa, N., “Experimental Study of Fuselage Effects on the Aerodynamic Performances of Reusable Launch Vehicle,” AIAA journal. IAC- 06- D. 2. 1. 03, pp. 1- 7.