ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Study of Flow Behavior over Different Bluff Body

Mr. Bhupal Rakham
Assistant Professor
Aerospace Engineering,
MIRIT, Hyderabad.

Mohd Nehal Aman
M.Tech Student,
Aerospace Engineering,
MIRIT, Hyderabad.

ABSTRACT

The main objective of the project is to study the performance of air flow around the bluff bodies. The bluff bodies are considered about the re-entry capsules and the flow behavior and its effect around the different bluff bodies is studied based upon the environment conditions at the certain altitudes and the Mach no which they enter into earth atmosphere from the outer space. A bluff body of certain length and various sizes are taken perpendicular to the flow direction. It has been flow analyzed for aerodynamics forces which leads to pressure distribution, a wake and also vortex shedding. This results in flow separation and velocity variation. The objective of this paper is to study and analysis this flow pattern on various bluff bodies.

Key words: ***bluff bodies, mach no, temperature, velocity, pressure distribution.***

Introduction

The topic of this thesis is the simulation of flow around bluff bodies and bridge deck sections using computational fluid dynamics (CFD). CFD calculates numerical solutions to the equations governing fluid flow. Bluff bodies are structures with shapes that significantly disturb the flow around them, as opposed to flow around a streamlined body. Examples of bluff bodies include circular cylinders, square cylinders and rectangular cylinders. Deck sections of long span bridges are another type of such a body that show similar flow characteristics as they are usually elongated with sharp corners which make flow around them very complicated. Computational flow involves the study of an object under moving condition. In this paper computational flow of bluff body was analyzed at different Reynolds's number to determine the pressure drag and also the drag coefficients. This study would help scientists and researchers to know about the importance of the Para-rec shape objects and its usefulness in designing a new type of re-entry vehicles. Most structures on land and in the ocean are in multiple forms and are confronted by a fluid flow. Vibrations of these structures due to fluid flows reduce the life of the respective installations and must therefore be taken into account in the design of the structure. For assessment of this vibration, it is important to understand the interaction of multiple structures in a flow. An elementary shape of a structure or a component of a structure is a circular cross-section, and a tandem arrangement of two circular cylinders is a basic example of an array of multiple structures. Flow past a bluff body such as a sphere and cylinder from engineering application point of view is a generic flow–structure interaction with important implications for flow-induced vibration and noise generation. Periodic vortex shedding patterns and fluctuating velocity fields downstream of the bluff bodies can cause structural Damage because of periodic surface loading, acoustic noise and drag forces. Most Work has been done on the flow passing a circular cylinder rather than a sphere. Coefficient of drag (Cd) and coefficient of lift (Cl) are the parameters used or developed here to obtained the results since these are the basic parameters that can give an idea on the vortex instabilities of the structures.

Bluff Body

- Bluff bodies which have blunt rear faces, or for streamline bodies at large angles of attack, viscosity always plays an important role for all the forces and moments.
- Such flows exhibit flow separation which is the sudden thickening or breakaway of the

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

boundary layer from the surface, resulting in a thick trailing wake.

- Flows with large wakes (comparable to the typical dimension of the body) and significant changes when compared to the ideal fluid model originating an high pressure drag force.
- Large wake is related to the body shape (cylinder, sphere) and/or with the orientation of the incoming flow (flat plate, foil).
- Near wake has small velocities and an approximately constant pressure smaller than the undisturbed pressure.
- Far wake exhibits vortices of symmetric strength aligned along two parallel lines and shifted half wave length van karman street. Vortex shedding leads to an unsteady flow and problem of induced vibration.

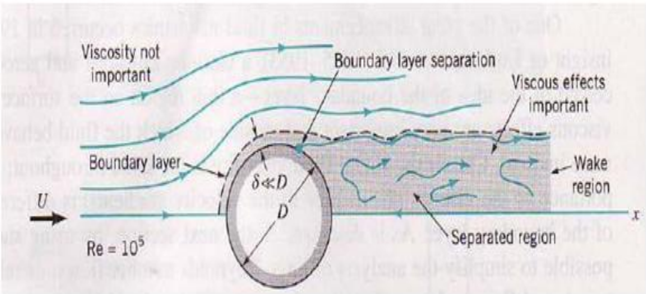

Figure 1.1: Flow around bluff bodies with flow separation

Types of Bluff Bodies

Circular Bluff Body

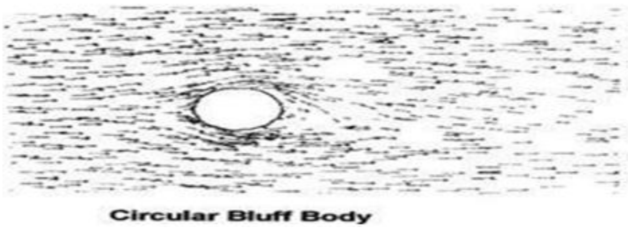

Figure 1.2: Circular Bluff Body

The flow around circular cylinders has been extensively studied due to its practical importance in engineering and scientific relevance in fluid mechanics. On the engineering side, there are a number of devices in mechanical, civil and naval engineering where circular-cylindrical structures are used. Examples of such devices are heat exchangers, chimneys and offshore platforms. In scientific terms, the flow around circular cylinders exhibits various important physical phenomena, such as separation, vortex shedding and turbulence in the wake, at relatively low flow speed. When circular cylinders are grouped in close proximity, the flow field and the forces experienced by the cylinders are entirely different from those observed when the bodies are isolated in the fluid stream. The effect of the presence of other bodies in the flow is called flow interference, and it has crucial importance in aerodynamics and hydrodynamics. For example, in all the devices mentioned at the beginning of this chapter it is common to have circular-cylindrical structures grouped together. This thesis examines the effect of a specific type of flow interference on two different aspects of the flow: the early stages of the transition to turbulence in the wake of the flow around fixed cylinders and the flow-induced vibrations of an elastically-mounted rigid cylinder. We start by presenting a synopsis of some pertinent features of the flow around a single cylinder.

Rectangular Bluff Body

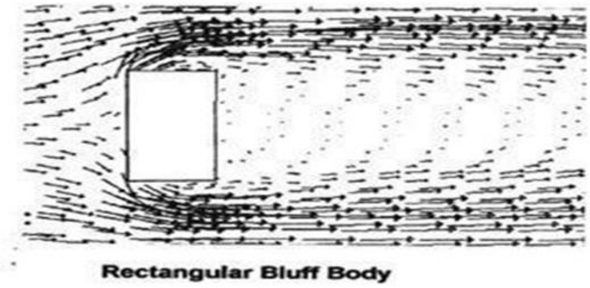

Figure 1.3 Rectangular Bluff Body

Rectangular presents an interesting summary of the sources of unsteadiness in the flow-field around a static bluff body. It was proposed in the review work by ahsan on bluff body aerodynamics and aero elasticity. This a qualitative comparison of the frequency content of the different sources of unsteadiness in the flow-field. The three subplots in the bottom part of the correspond to the oncoming

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

turbulence, the flow separation (and re-attachment) and the vortex shedding process respectively. From these subplots, it is clear that three distinct bands of frequency characterize the aerodynamic system. As shown in the next section, when the dynamics of the flexible system are added, important interactions between these different sources of unsteadiness can occur.

Triangular Bluff Body

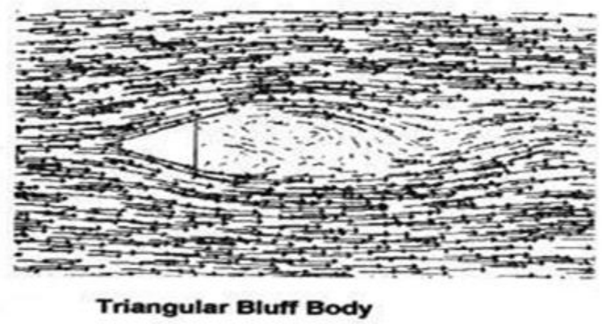

Figure 1.4: Triangular Bluff Body

A method widely used in combustors to anchor a turbulent flame consists in establishing a recirculation region do hot gases which continuously ignites the re-action stream. A backward facing step or a bluff body is typically used for this purpose .Developing future combustors and active or passive combustion control system requires better understanding of flame holding of turbulence kinetics interaction vortex dynamics combustion instability ignition and quenching. Analysis of combustion instabilities is particularly important in the design of reduced emission.

Factors Affecting the Bluff Bodies

Thin Shock Layers

For a given flow deflection angle, the density increases across the shock wave and become progressively larger, as the Mach number increases. At high density, the mass flow behind the shock wave, easily "squeeze through" the smaller areas. And the distance between bow shock and the body can gets very small. So the flow field between the shock wave and the body is defined as the shock layer

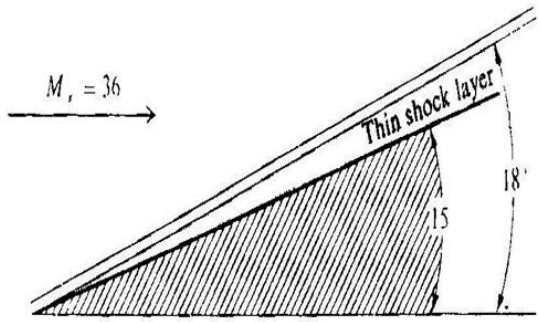

Figure 1.6: Thin Shock Layers

Entropy Layer

At hypersonic Mach numbers, the shock layer adjacent the body is very thin and this thin layer detach at a small distance from blunt nose. In the blunt nose region, the shape of the shock is highly curved. From the shock property, entropy increases across shock wave. And as stronger the shock, the larger the entropy increase. So flow passing near the normal portion will experience large entropy change than neighboring streamline that passes through a weaker portion of the shock wave. Therefore at the nose a large entropy gradient exists, that is called entropy layer.

Viscous Dissipation

A high velocity hypersonic velocity contain a large amount of kinetic energy .When the flow is slowed down in the boundary layer by the viscous effect, the lost kinetic energy changes to the internal energy of the gas molecule .This phenomena is called viscous dissipation.

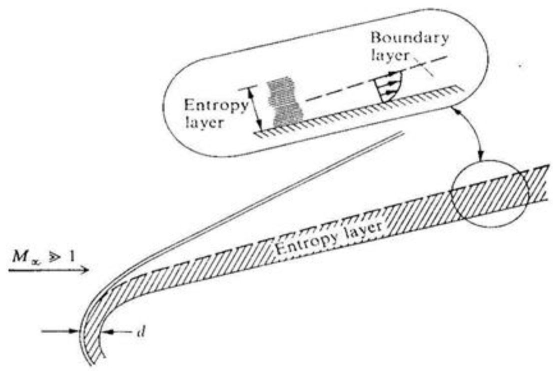

Figure 1.7: Entropy layer near the blunt nose

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Bow shock wave

These shocks are curved in geometry and detached from the body with a small distance in front of the body. In front of the body, these wave stand at 90 degrees to the oncoming flow, and then curve around the body. Detached shocks allow the same type of analytic calculations as for the attached shock for the flow near the shock. The detachment distance of shock to body depends upon the body's shape and deflection angle. Additionally, the shock standoff distance varies drastically with the temperature for a non-ideal gas, causing large differences in the heat transfer to the thermal protection system of the vehicle.

METHODOLOGY

To achieve the above-mentioned objectives, the work is split into various stages. Initially, modeling of the flow around a circular cylinder within range of Reynolds number is done by using basic steady state simulation methods. This is done as a pilot study for the further application of CFD on the computation of a more complex flow using advanced CFD techniques at a later stage of the work. The next stage is the simulation using the unsteady and more advanced LES model on the flow around a circular cylinder to study the vortex shedding phenomenon in the wake region of the flow.

This acts as a first step towards the investigation of the effect of vortices on bluff body flow. The simulation of flow around rectangular sections investigates the changes of Strophe number and the drag coefficient with the increase in aspect ratio. Apart from that, validation of the turbulence models (LES and DES) at higher Reynolds number of for the flow around a square cylinder is conducted. This ensures that the turbulence models are capable of capturing the flow characteristics accurately not only at low Reynolds number. The work on the simulation of flow around rectangular cylinders provides a general idea of the flow patterns and the expected outcomes on the flow around the bridge deck section of similar aspect ratio Based on the findings of the study of the flow around a circular cylinder and the rectangular sections, numerical modeling techniques of the simulation are then applied on the flow around a bridge deck section to investigate the wind effect on the bridge. A geometrically similar sectional model of the Kessock Bridge is simulated using DES to predict the flow parameters such as force and velocity distribution around the deck section. Evaluated drag, lift and moment coefficients are then compared to the measurement from the 1:40 scale sectional wind tunnel test of the bridge.

Work done in this chapter then progresses to the investigation and development on the Fluid Structure Interaction (FSI) study of wind effect on long span bridges as discussed in Chapter Seven. Current work imposes significant impact on the development of the FSI capability on long bridges within the research group. Findings and observation from the work contributes to the computational aspects of the FSI investigation on various similar projects within the group.

Computer simulation software is used to study the flow characteristics of the Bluff bodies and this was done by using FLUENT software. The objective of the simulation is to focus on the study of flow in the PDE so that any important factor that contributes to total pressure, velocity distribution may be identified. The steps of procedure involved are presented in the following flow.

RESULTS AND DISCUSSION

It is assumed that we have the free stream Mach number M_∞ and the shape of the body. Now we want to calculate the shock wave size, shape and flow properties, which is called direct problem. In the time marching technique first we assume the shock wave shape at the time t=0. The abscissa of the body and the shock wave is defined by b and S. the flow field between the assumed shock wave and the given by is divided by into a number of grid points. After that we set the initial value of the grid points using the time marching technique new values of the flow field variable.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

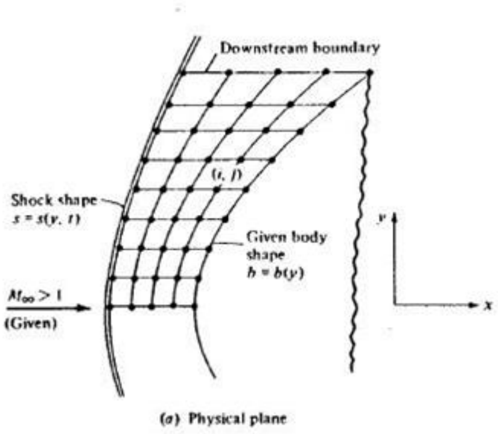

Figure 6.1: Physical plane

In the flow field around the body, gas molecules which impact on the body experience a change in momentum, and by the random molecular collision of the molecule this change transmitted to the neighboring molecules. In this fashion, information about the presence of bluff body transmitted to the surrounding flow via molecular collisions. If the upstream flow is supersonic the disturbance wave pile up and coalesce, form a standing wave in front of the body. That's why ahead of the bluff body a bow shock is generated because deflection angle is very large at the body nose. This shock wave is clearly seen in the figures. And ahead of this flow properties changes drastically.

Most frequently used properties in aerodynamic analysis of any object are given as:
1. Pressure
2. Flow velocity
3. Reynolds number

Pressure:
In the flow field around the blunt body, due to hypersonic flow ashock wave generate ahead of the body. This shock wave is called bow shock.

This shock is detached from the body due to the high deflection angle .Pressure changes drastically across the shock wave; at the stagnation point pressure is at peak value because at the stagnation pressure shock wave is normal to the body. Area of the contour shows the sonic region here velocity of the flow hypersonic.

Flow velocity:
Flow compress along the bow shock that decrease the Machnumber .At the stagnation point the Mach number is about .01 .Dividing line between subsonic and supersonic region in the blunt body is called sonic line. Behind the blunt body flow separates and create wake.

Reynolds number:
Flow around a circular cylinder varies with the Reynolds number. Small Reynolds number corresponds to slow viscous flow where frictional forces are dominant. When Reynolds number increases, flows are characterized by rapid regions of velocity variation and the occurrence of vortices and turbulence. Mathematically, Reynolds number of the flow around a circular cylinder is represented by, where D is the diameter of the cylinder; “u” is the inlet velocity of the flow, the kinematic viscosity of the flow. Experimental study of the flow around a circular cylinder has identified regions where significant patterns of flow occur as the Reynolds number changes, especially when the flow changes from laminar to turbulent stat.

CONCLUSIONS
From the fluent simulation it is concluded that at high Mach number a detached bow shock at the front of the body generates, which highly influence the flow properties around the body. Mach number suddenly decrease drastically behind the wave and flow compressed to a high level at the stagnation point. Temperature rise at stagnation point is very high; due to this high heat transfer rate is set up between the flow and body. At the apex of the body a sonic region is generates where the flow is subsonic, and the flow properties changes drastically at this region due to a strong bow shock. Calculation of flow variables at a point just behind the bow shock wave confirm that at the apex the bow shock wave can be treated as a

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

normal shock . From the theoretical formulation we conclude that aerodynamic heating of the body initially depends on its kinetic energy and bluntness of the nose cone decrease the aerodynamic heating over the body by generating the strong bow shock. And aerodynamic heating varies inversely proportional to the radius of the nose cone. Another conclusion from the literature study is that the flow behind the shock wave is rotational, because each streamline pass through a different strength shock wave and entropy change behind the shock wave is different for every streamline.

The computational investigation of minimum-drag bodies at supersonic a moderate hypersonic speeds (Mach 3-12) confirms that, the bodies with the lowest wave drag have to be geometrically blunt. Aerodynamic heating or temperature rise at the stagnation point over the body depends over the ratio of coefficient of friction and coefficient of drag.

REFERENCES

1.Bruno Souza Carmo (2009), On Wake Interference in the Flow around Two Circular Cylinders: Direct Stability Analysis and Flow-Induced Vibrations.

2.D.A. Jones and D.B. Clarke (2008), Simulation of Flow Past a Sphere usingthe Fluent Code.

3.Bengt Fornberg (2014), Some Observations Regarding Steady Lamina Flows Past Bluff_ Bodies.

4.Kai Fan Liaw (2005), Simulation of Flow around Bluff Bodies and Bridge Deck Sections using CFD.

5.Jeffrey Newcamp (2002), Bluff Body Aerodynamics and Flow Separation.

6.Merle Potter, David.C.Wiggert (1986). Fluid Mechanics. 2nd ed. Newyork: Schaum's series. Pg. 156-176.

7.W.C.L.Shih, C.Wang, D.coles and A.Roshko (1993), Experiments on flow past rough circular cylinders at large Reynolds numbers. Journal of wind engineering and industrial aerodynamics.

8.John C K Cheung, William H Melbourne, Effects of surface roughness on a circular cylinder in supercritical turbulent flow, Department of Mechanical Engineering, Monash University.

9.Pijush.K.Kundu, Ira M.Cohen (2002). Fluid Mechanics. New York: Academic Press. Pg.256-300.

10.Fluent 14.5. Available: www.sharcnet.in. Last accessed 20th July 2013. 11.J.D.Anderson (1998).Fundamentals of Aerodynamics. New Delhi: Tata McGraw Hill Publications.Pg.348-352.

12.Anderson,john,D, Albacete M.Lorenzo and Winkelmann E Allen: “On Hypersonic Blunt body fields Obtained with A time dependent Technique:Naval Ordnance Lab Itr ,1971.

13.Anderson john,D.,jr: Computational Fluid Dynamics :The basic with applications, McGraw Hill ,New York ,1995.

14.Anderson J.D,Jr , 2005,McGraw Hill, Fundamental of aerodynamics, New York.

15.Anderson J.D, Jr, McGraw-Hill ,Hypersonic and high temperature gas dynamics, New York.

16.Anderson J.D,jr,1978 , McGraw Hill, Introduction to flight , New York.

17.Anderson J.D, Jr, McGraw-Hill, Modern compressible flow, New York.

18. FLUENT USER GUIDE 6.3.

19.Meir Gerick Bar, Fundamentals of Compressible Fluid Mechanics.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

20. Moretti Gino and AbbtitMichael : “A Time-Dependent Computational Method for Blunt Body Flows”AIAA Journal ,Vol 4 No 12 ,1970.

21.Santos W.F.N, 2005, Leading edge bluntness effect on aerodynamic heating, Combustion and Propulsion Laboratory National Institute for Space Research 12630-000 Cachoeira Paulista, SP. Brazil.