Drag Reduction for Bluff bodies by controlling the Flow Field

Amit*

School of Mechanical Engineering

Lovely Professional University, Phagwara, Punjab, India

Abstract

This paper presents the drag reduction technique for blunt bodies at hypersonic flow. In hypersonic flow filed, drag is very high which increases the cost of space travel. There are different methods to reduce the drag for blunt bodies in hypersonic flow field. In this paper I selected a typical design of blunt bodies and then helium injection technique was used to reduce the drag by controlling the flow filed. Initially a model was developed of blunt body. And then analyses was done with the help of ANSYS-Fluent. And it was observed that drag can be reduced substantially by controlling the flow filed.

Introduction

Generally we can classify the bodies into two types with respect to the flow filed.

- Streamline body
- Bluff body

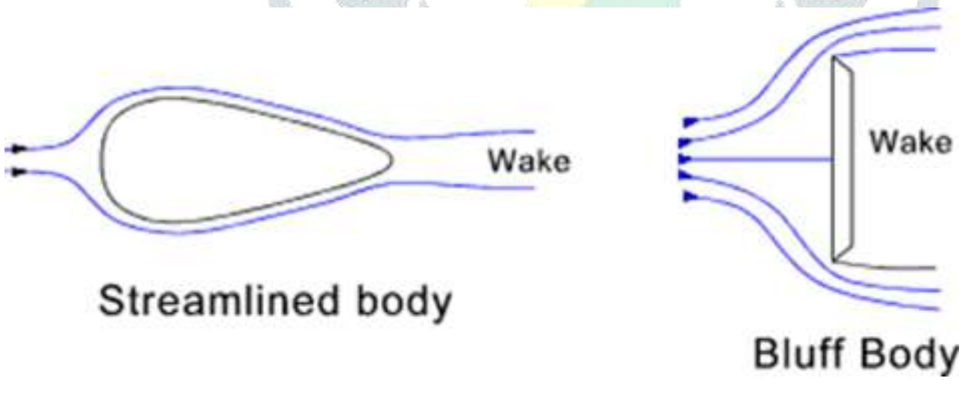

Fig 1. FLOW OVER AERODYNAMIC BODY AND BLUFF BODY

When we consider the bluff body moving through fluid at hypersonic speed the drag plays major role on it.

METHODS OF DRAG REDUCTION

Generally, there are two methods available to reduce the drag, which are passive method and active method. The active method also classified into active open loop method and active closed loop method.

PASSIVE METHOD

In this method the reduction of drag is achieved by modifying its shape without using any additional materials.

ACTIVE METHOD

In this active method some additional materials are require to reduce the drag. One active feedback is available in active open loop method. No feedback is consisting with active closed loop method

LITERAURE REVIEW

On 17th December 1903, Wright brothers initiated the development of aeronautics and in march 16th,1926 Robert h. Goggard had the first launch of first fuelled rocket had been driven with the thought to fly higher and faster. Exponential growth from 35 mph of Wright brothers in 1903 to 400 mph of fighters in world war II followed by transition to supersonic aircraft of 1200 mph at 30000 ft in 1960's- 1970's. Finally they achieved with re-entry vehicles at 25 mach. The aerodynamic flow at this Mach no level is known as the hypersonic aerodynamic flow.

The hypersonic regime is subset of the supersonic regime. At this hypersonic speed, some phenomena such as increasing temperature and the formation of a shock layer, low density effects begin to impact vehicle design.

GEOMETRICAL DETAILS AND MODELLING OF THE BLUFF BODY

And we, here had taken the configuration of the Scramjet Engine normally used by many of the research institutes. The Geometrical details and the Configuration of the bluff body currently used by the research institutes (NASA, etc) is collected and the same is shown below.

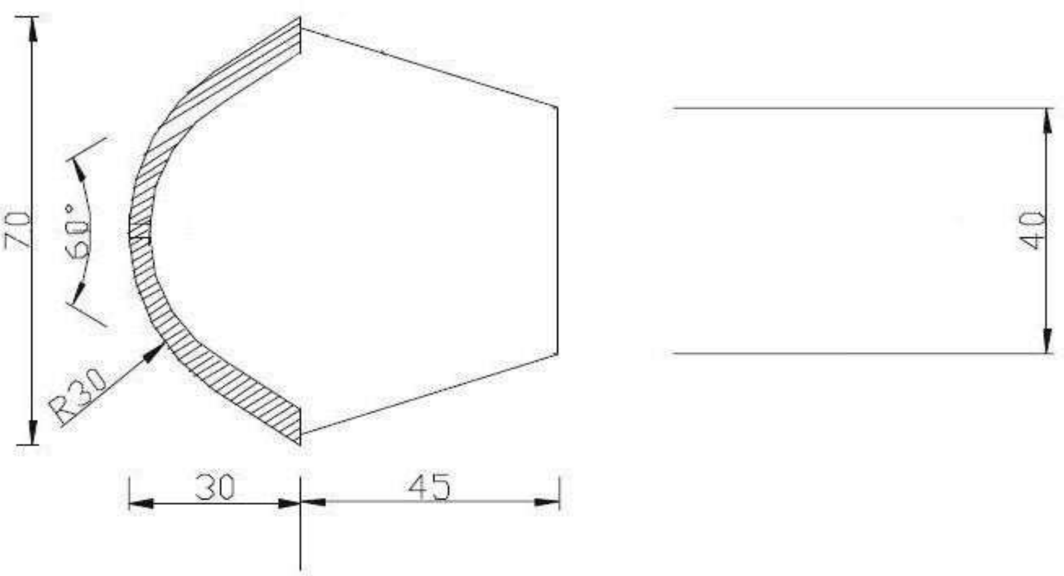

Fig 2. BLUFF BODY DIMENSIONS WITHOUT FUEL INJECTION

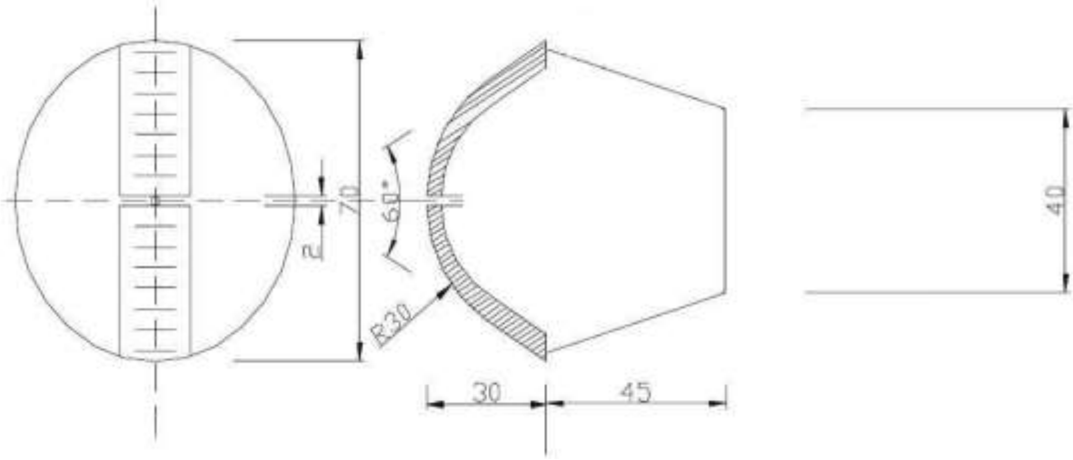

Fig 3. BLUFF BODY DIMENSIONS WITH FUEL INJECTION

CO-ORDINATES FOR THE BLUFF BODY CONFIGURATION

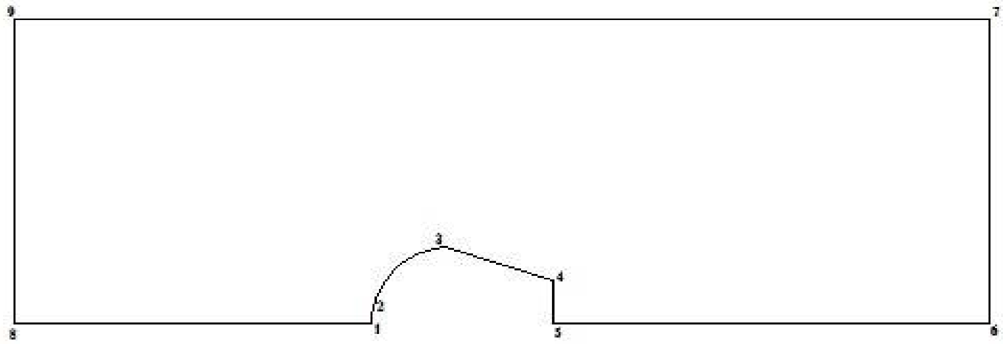

Fig 4 CO-ORDINATES FOR THE DESIGN CONFIGURATION

Points	X	Y
1	0	0
2	0	1
3	30	35
4	75	20
5	75	0
6	255	0
7	255	140
8	-150	0
9	-150	140

TABLE 1: CO-ORDINATES FOR ORIGINAL CONFIGURATION

MODELING OF THE BLUFF BODY

With the Geometrical details obtained, the bluff body have been modelled with the help of the Gambit software. The Modelled bluff body is shown below.

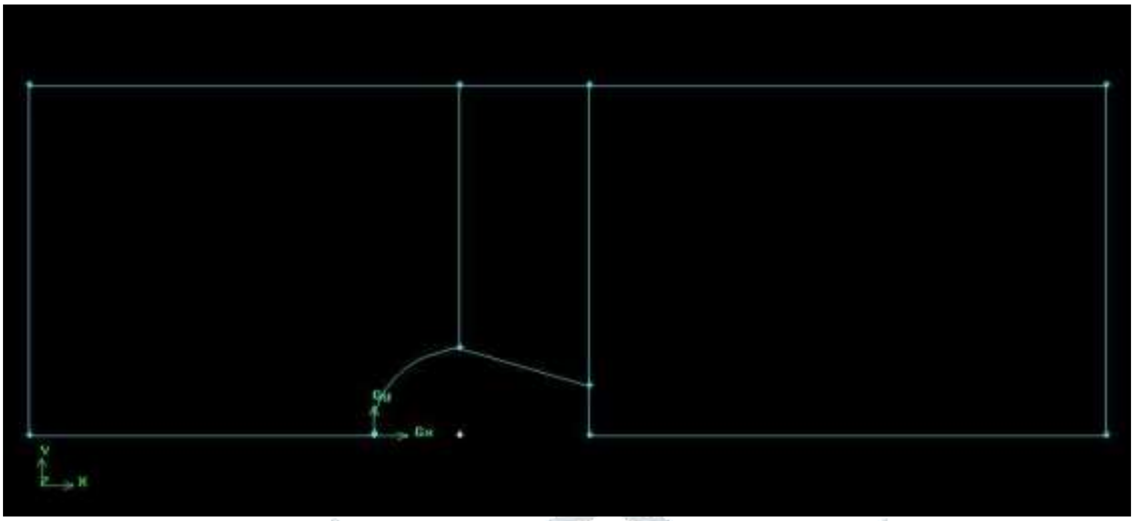

Fig 4 MODELED BLUFF BODY

CFD ANALYSIS OF THE BLUFF BODY

In today's world, CFD is used in a vast number of fields which include the examination of fluid motion. The industries range from the biomedical and pharmaceutical industries to the oil and natural gas extraction industries to the aeronautical and automotive industry. In the pharmaceutical industry, the CFD packages are used to model the flow of certain enzymes in different parts of the human body.

The oil extraction industry uses CFD to model the flow in the extracting station, and study different parameters in an isolated manner. For example, the flow may be modelled with or without consideration of the heat transfer characteristics. This may be particularly useful while determining only certain required behavioral characteristics.

CONSTRUCTION OF GEOMETRY

Sweeping an vortex along a specific direction vector. Then the noteworthy feature in geometry construction is domain creation. Appropriate rectangular domain is created outside of the model. GAMBIT can generate both the quad and the hexahedral types of structured grids for 2D and 3D respectively. A typical structured grid is as shown in Figure.

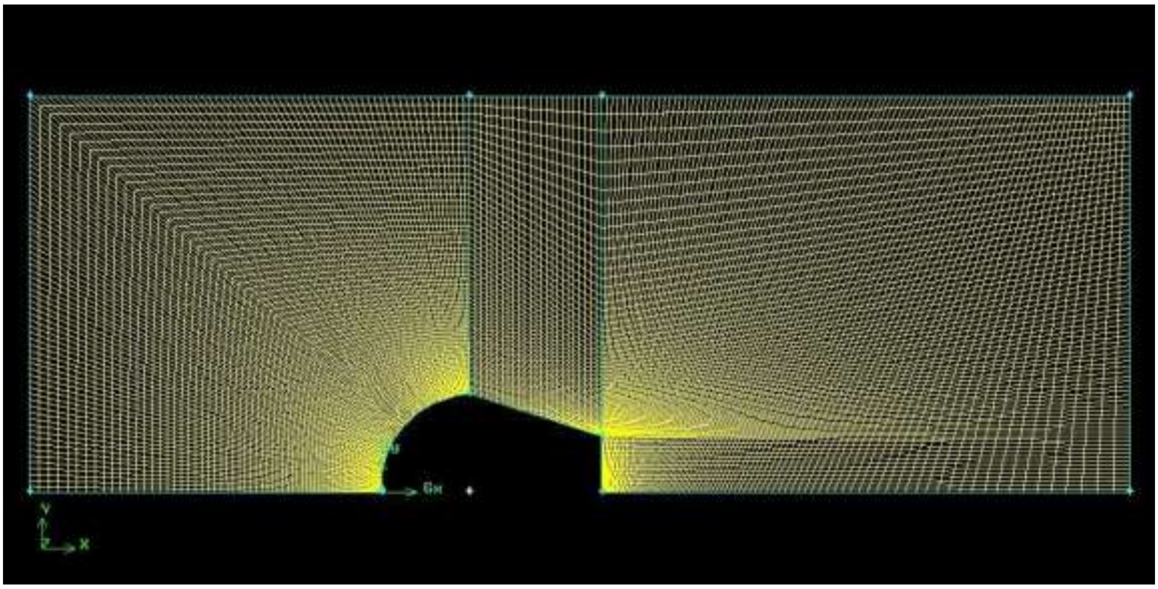

Fig 5. STRUCTURED GRID

in the sense that they can be scaled using those "numbers" to any units. The default units used by FLUENT are S.I Units and it is strongly recommended to stick to them. Hence the co-ordinates used in GAMBIT are scaled to a certain factor in order to keep the conditions close to the real world.

For the purpose of post processing the data obtained from FLUENT simulations, FIELDVIEW offers a more powerful platform when compared to the FLUENT GUI. This fact is valid, more for the 3D simulation results as opposed to the 2D ones. The data is exported from FLUENT in the FIELDVIEW Unstructured format in a file with an extension of .fvuns. This data contains the velocity values by default. The other required parameters may be manually selected from the list in the FLUENT Export menu. Once the data is read into the FIELDVIEW software via the "Structured Data Input" command, the required parameters may be selected for analysis from the list that was exported from FLUENT. The domain will be shown as a rectangular cube (for this case), or as it is for any other case. The boundary types selected can be displayed as Boundary Surfaces, upon which analysis can be done. Numerous display options are available.

Creating animations is another useful feature of FIELDVIEW. Animations may be created using the Flipbook option. Animations may include the sweeping of a plane, showing the growth of streamlines, or simply showing the movement of vectors. Animations may be either saved as an MPEG file or in the MIFF format. An important fact while creating animations is that the MPEG movie will not function normally in a Media Player unless the correct resolution of the graphics window is not selected. If a resolution of 640 X 480 is selected, then the MPEG files function correctly. Streamlines are very useful to display the flow in case of the formation of vortices, or just to analyse the flow over a certain object. They may be created using the option of Streamlines from the Visualisations Panel. The originating points of a rake may be manually selected on a suitable co ordinate surface or may be arbitrarily selected by the program itself. Also, a certain number of points may be added between two selected points. Images may be saved in various formats ranging from the high resolution Bitmap(BMP) to the smaller sized GIF. The default black background may be changed to any colour of the user's choice. Also, a legend may be added onto the graphics window. The image is an exact snapshot of the current graphics window. Hence, it may be zoomed, rotated and panned as per the user's requirements.

CONVERGENCE CRITERIA

Convergence is absolutely essential in order to achieve the correct solution. A solution that is not converged is nothing but an incomplete set of equations with intermediate values. Convergence in FLUENT is open to adjustments by the user. The residuals method of checking convergence was adopted for this study whose default values are 0.001. The convergence can also be monitored during the solution of the equations by means of graphs that indicate the change of the required values with each iteration. This can be particularly helpful to see how and when each entity is converging to a low value during runtime. Hence monitors were setup for various parameters such as velocity and the residuals.

BOUNDARY CONDITION

CFD Analysis were carried out for the following configurations:

1. Original bluff body with no modification.
2. Bluff body with counter flowing supersonic jet of Helium (He) injection at stagnation point.
3. Bluff body with counter flowing supersonic jet of Nitrogen (Ni) injection at stagnation point.

RESULTS AND DISCUSSION

Pressure contour

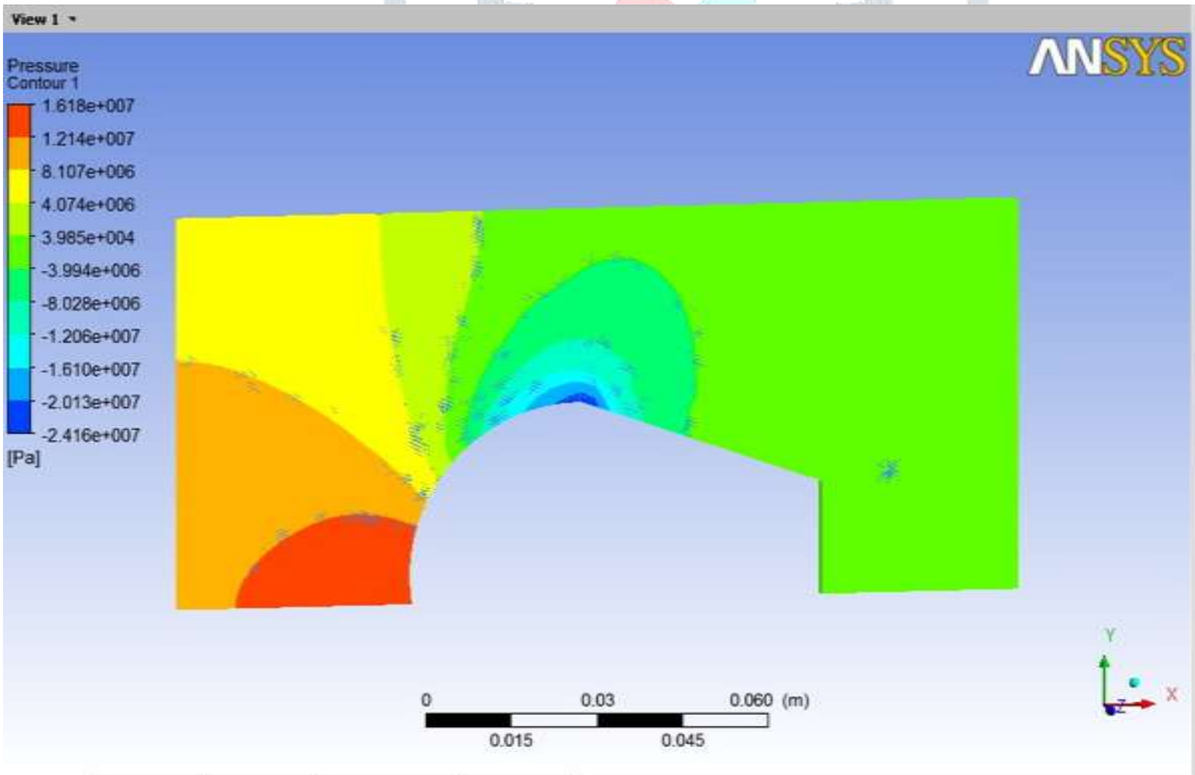

Fig. 6

PRESSURE CONTOUR

The above CFD result is the pressure contour for the original configuration without fuel injection. And it shows very clearly that the high pressure formed at the nose of the bluff body. This pressure cause higher pressure drag in this hypersonic speed.

Turbulence contour:

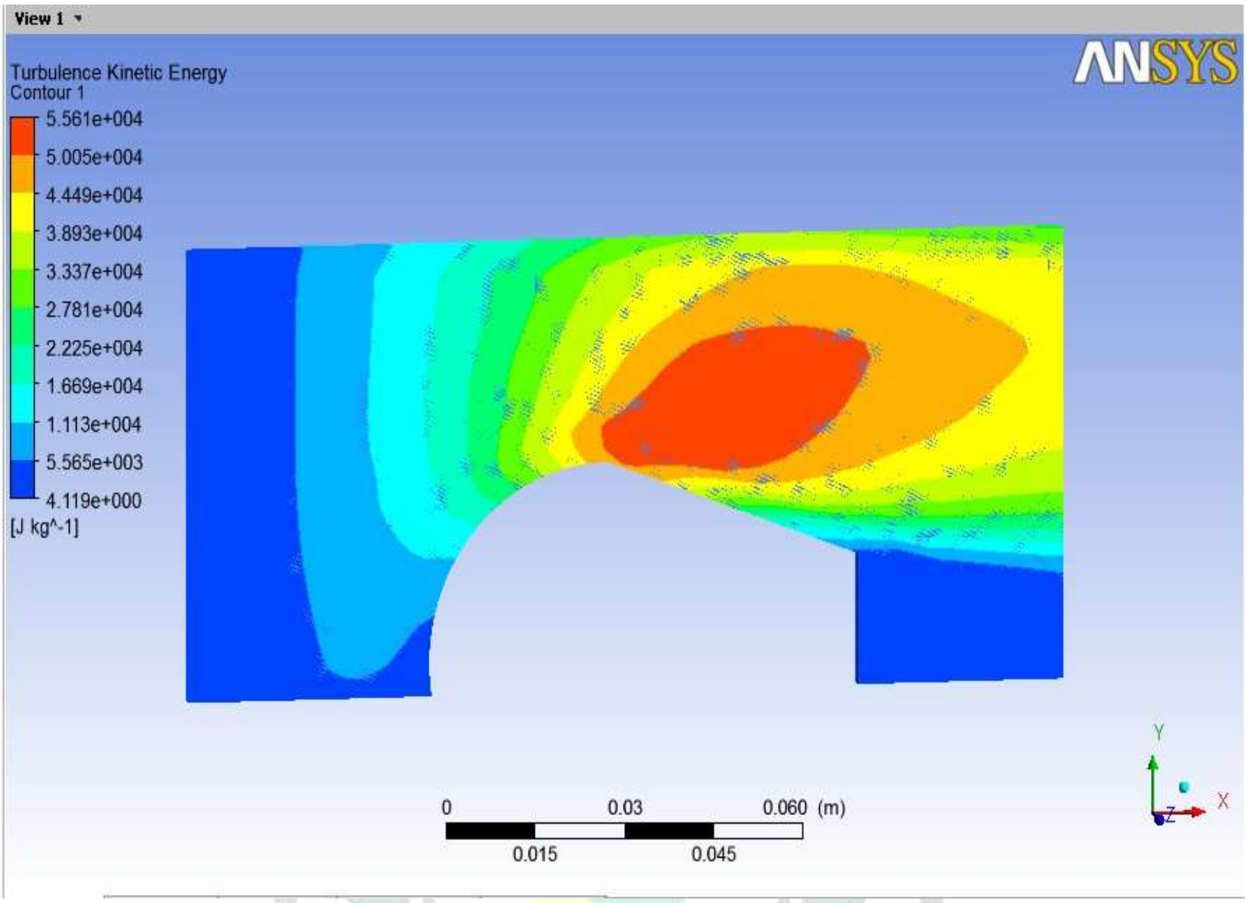

Fig 7. TURBULENCE CONTOUR

The above CFD result is the turbulence contour for the original configuration without fuel injection. This shows that there is more turbulence produced over the surface of the bluff body. This turbulence produces more drag.

CFD RESULTS FOR HELIUM INJECTION

Pressure Contour

The below contour is the pressure contour at 30 psi pressure. In this 30 psi the pressure value is reduced to some amount from the pressure of original configuration.

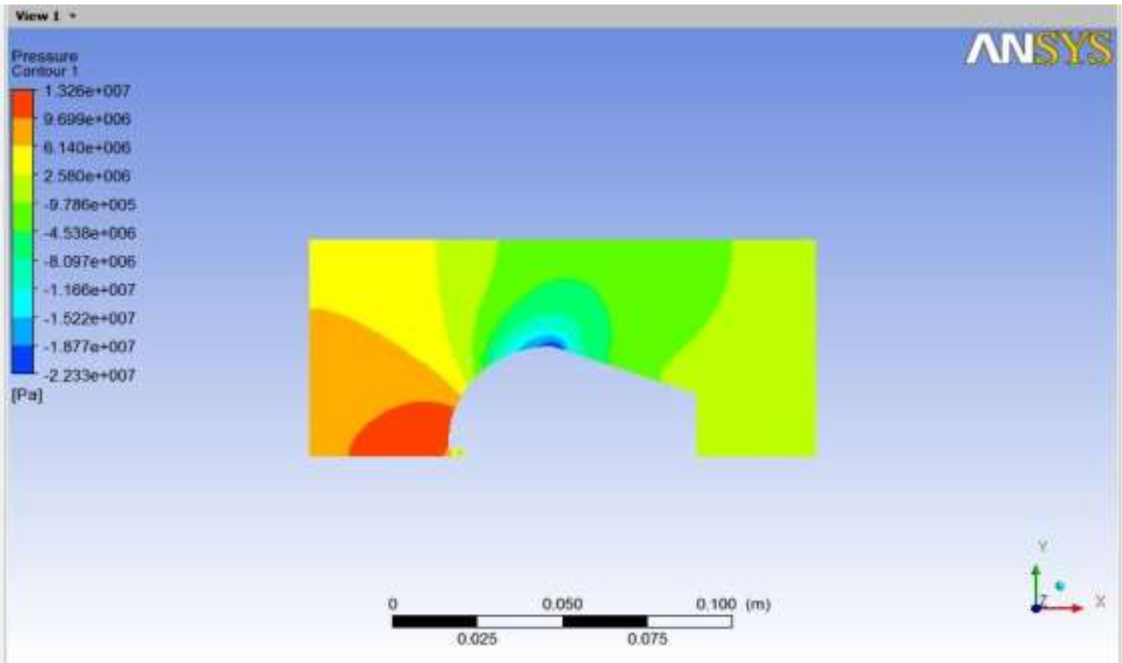

Fig 8. PRESSURE CONTOUR

Turbulence contour

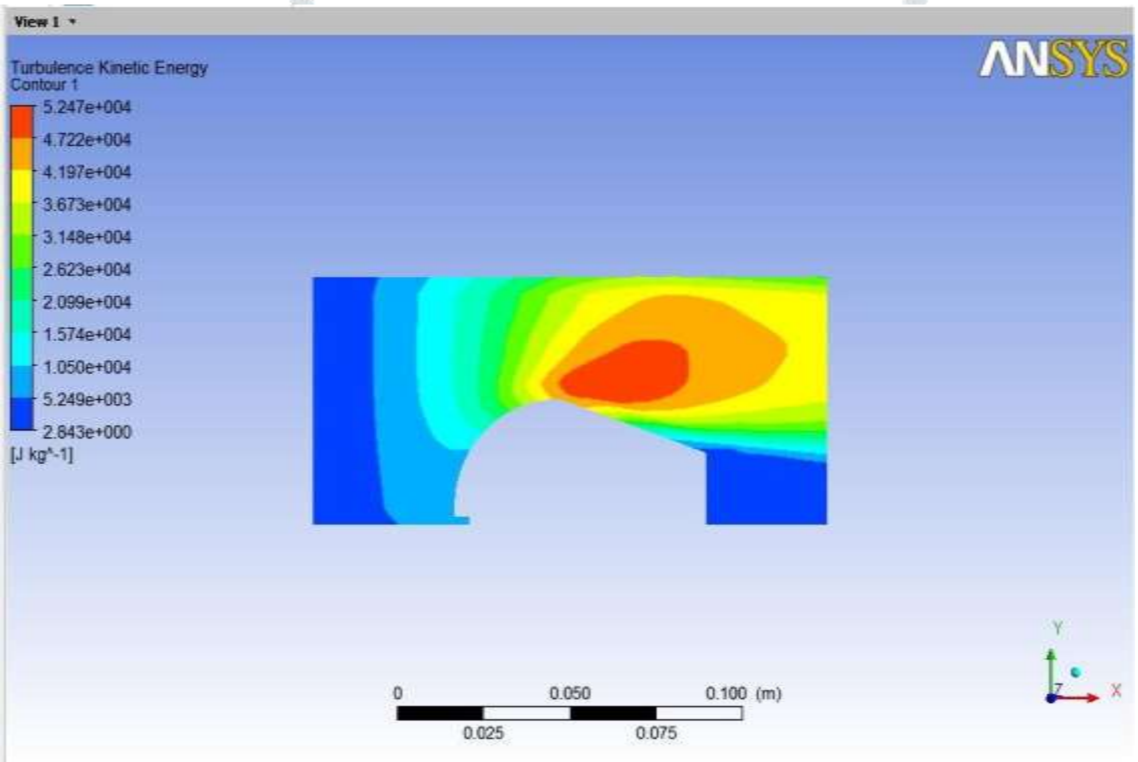

Fig 9. TURBULENCE CONTOUR

This is the turbulence contour for 30 psi pressure. Turbulence also reduced to some amount from the turbulence of original configuration.

Conclusion

This paper presents the drag reduction technique for blunt bodies at hypersonic flow. In hypersonic flow filed, drag is very high which increases the cost of space travel. There are different methods to reduce the drag for blunt bodies in hypersonic flow field. In this paper we selected a typical design of blunt bodies and then helium injection technique was used to reduce the drag by controlling the flow filed. Initially a model was developed of blunt body. And then analyses was done with the help of ANSYS-Fluent. And it was observed that drag can be reduced substantially by controlling the flow filed. Maximum drag reduction was found at the injection pressure of 40 psi.

REFERENCES

1. G. N. Abramovich, The Theory of Turbulent Jets, MIT press, 1963.
2. J.D Anderson, Introduction to Flight (2006), eighth edition, Tata Mcgraw-Hill Publications.
3. J.D Anderson, Fundamentals of Aerodynamics (2008), eighth edition, Tata Mcgraw-Hill Publications.
4. E. Radhakrishnan, Gas Dynamics (2008), second edition, PHI publications.
5. J.D. Anderson, Hypersonic and high temperature gas dynamics, Mcgraw-Hill Publications.
6. 7. M. Toyoshima, S. Okawa, An effect of a horizontal buoyant jet on the temperature distribution inside a hot water storage tank, *Int. J. of Heat & Fluid Flow 44* (2013) 403-413.
7. 8. N. E. Kotsovinous, A note on the spreading rate and virtual origin of a plane turbulent jets, *J. of Fluid Mechanics 77* (1976) 305-311.