Drag Reduction of an After Burner Flame Stabilizer in Passive Condition

Rakesh.C[1], Jafar M Sayed[2], M S Ganesha Prasad[3]

[1]*Dept. of Mechanical Engineering, New Horizon College of Engineering,*
[2] *Dept. of Mechanical Engineering, New Horizon College of Engineering,*
[3]*Dept. of Mechanical Engineering, New Horizon College of Engineering,*
Corresponding Author: Rakesh.C

Abstract: The technique of placing bluff bodies in downstream will nearly reduce the drag generated by single bodies. numerous numerical and test results have proved this phenomenon of drag reduction due to flow interference and placing the bluff bodies upstream and downstream of the flame stabilizer. A v-shaped gutter flame stabilizer is mostly used in afterburner of an aircraft turbine engine, which is used to hold the flame when the afterburner is in active condition. But when the burner is in passive condition the presence of gutter produce excess drag and pressure loss in the after burner section of the engine. Therefore it is important to reduce the pressure loss and drag generated by the flame stabilizer in afterburner. In this work an attempt has been made to understand the flow physics involved in keeping bodies in tandem and the effect of change in drag coefficient comparing to the single bodies. Three types of configuration of bodies are investigated considering the factors affecting the drag and the drag coefficient is calculated. The models are designed in CATIA V5 and the computational fluid dynamic analysis is done using ANSYS fluent software.

I. INTRODUCTION

In aircraft application, larger thrust for small duration is required sometimes and can be achieved by two methods. First method is by increasing the mas flow rate and the other is by thrust augmentation. Flame stabilizer is employed to hold and stabilize the flames, which are placed after the turbine section in an afterburner. Bluff bodies are characterized by large region of separated flow, a direct consequence of which is that they suffer from large values of C_d. Various research and development are carried out improve the performance of afterburner, both experimentally and theoretically. Computational methods have become highly useful tool to design, develop, and analyze the performance of an afterburner before implementing. The amount of drag generated by the presence of flame stabilizer and the recirculation zones in the wake of the stabilizer are very important factor to be considered for better performance of an afterburner in any gas turbine engines. In the past, various research activities have been carried out to reduce the drag of bluff bodies, by varying the orientation and arrangement of the bluff body such as, tandem, staggered, normal, and perpendicular to the flow. The above researches were carried out through CFD using ANSYS fluent software.A brief review of literature related to the above studies are presented.

Nakanishi et al... [1] Conducted experimental investigations on the effect of flame holder gutter shape on afterburner performance. Conventional v-gutter has minimum drag and minimum pressure loss both in burning and without burning condition and the flame holder shape had less effect on stability limits.

Kareem et al...[2] On cylinders in tandem, the flow in tandem arrangement in two regimes such as, for spacing up to the critical spacing, the vortex street is suppressed behind the front cylinder and beyond this critical spacing, both cylinders form vortex streets. Large values of drag co-efficient occur at ratio of cylinder spacing to diameter of cylinder between 3 and 4. When the spacing is kept below the above mentioned value the drag value is found to be less than the value obtained for the above mentioned spacing to diameter ratio. Similar works has been carried out by various researchers to show the effect of flow interference and drag characteristics' when two bodies are in tandem.

T. Morel et al…[3] Experiments with two disks of different diameters placed in tandem and connected together showed that very significant drag reductions from that of a single disk may be achieved by proper sizing of the disk diameters and of the gap between them. By Placing a second disk of diameter D1 ahead of a

single disk of interest with diameter D2 , and choosing D1 / D2 = 0.8 and the distance between them 0.54 D2. This represents an 81% drag reduction.

A. Prasad et al… [4] Has shown that it is possible to reduce bluff body drag dramatically with the use of small flat plates placed upstream. Not only can the drag of the bluff body be diminished to a value close to zero but, surprisingly, the drag of the system (body plus plate) is found to be a fraction of that of the bluff body alone.

S Srinivasan et al… [5] Has concluded from the CFD analysis that the drag of the v-gutter flame holder of an afterburner can be reduced by providing a buff body in tandem either upstream or downstream. The coefficient of drag is comparatively less for the bodies having rounded corners than the bodies with flat edges as expected. Further, the coefficient of drag varied with respect to the spacing between the bodies for all the cases. Further, from the analysis it is evident that the drag reduction is more when the body is kept downstream of the gutter, where as it is comparatively higher when it is kept upstream of the gutter.

II. METHODOLOGY

- Firstly, the case study on the different types of flame stabilizer geometries are done.
- The design parameters and the boundary conditions are determined.
- Validation of the Cd value from a research paper is compared with the CFD analysis value done in our software by taking the same boundary condition.
- The modeling of different geometries are done considering the factors affecting the drag using the CATIA V5 software.
- Then the meshing is done using ANSYS V.16 software.
- The fluid flow analysis is done using Ansys FLUENT to obtain forces acting on the geometries and Drag coefficient is calculated using Drag Equation.
- The results are compared and optimum results for better performance of afterburner is chosen.

ANSYS fluent software is used for the computational fluid dynamics analysis of flow over a V-gutter. The boundary conditions for above analysis are given in the table 1. A steady, viscous flow is considered and k-ε turbulence model is taken for the analysis. Standard wall function is chosen with v-gutter as no slip wall. A computational fluid domain is created surrounding the v-gutter model which is taken from the literature survey which corresponds to the section of an afterburner as shown in the figure 2.1. The analysis is carried out for the three different configuration of the bluff bodies each having three different cases. Drag coefficient is calculated using the drag force obtained by the software in the drag equation.

Table 1-Boundary Conditions

Parameters	Values
Inlet velocity	220 m/s
Inlet gauge pressure	124075 N/m^2
Turbulence kinetic energy (k)	187.8 m^2/s^2
Turbulence dissipation rate (ε)	655.57 m^2/s^3
Outlet gauge pressure	122675 N/m^2
Turbulence level (%)	5
Ratio of specific heat (ɤ)	1.35
Temperature	1009.1 K

Drag Equation is used to find the drag coefficient.

$$C_d = \frac{F_d}{0.5 * \rho * v^2 * A}$$

Where,

C$_d$ - Coefficient of Drag.
F$_d$ - Drag force in Newton.
A - Projected area on a plane perpendicular to the direction of motion in m^2.
ρ - Density of the fluid in Kg/m^3.
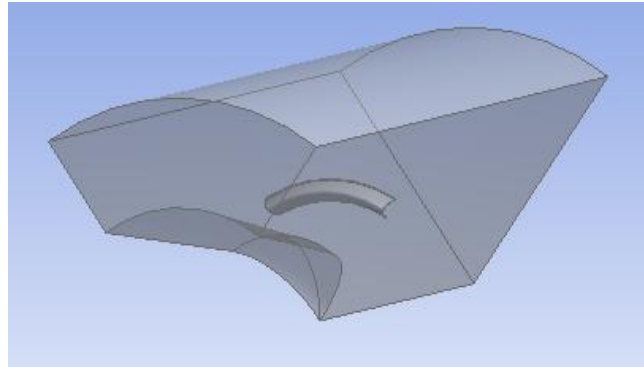v - Velocity of the fluid in m/s.

Fig-2.1 Computational fluid domain

The density and viscosity of the fluid is calculated and is constant for all the cases which are carried out and their values are 0.778 Kg/m^3 and 4.367*10^{-5} Kg/m-s respectively. Velocity ahead of the v-gutter is taken as 130 m/s and is also constant for the entire configuration.

III. RESULTS AND DISCUSSION

Coefficient of drag is calculated and flow over the bare v-gutter and when the bluff bodies are placed downstream are observed. Three different configurations were studied considering the factors affecting the drag and compared. The results show reduction of drag over the v-gutter with the bluff bodies placed downstream when compared with the bare v-gutter.

3.1 Case 1- Increasing the Apex Angle of the V-Gutter

One of the factors affecting the drag is inclination angle to the flow. According to the literature survey for 30° apex angle the drag coefficient is found to be 1.08, taking this as a reference the study is carried out. In this case the v-gutter apex angle is increased from 30° to 35° whereas the depth of the v-gutter is kept same. The drag force obtained from the CFD fluent software is used in the drag equation and coefficient of drag is calculated to be as 1.07.
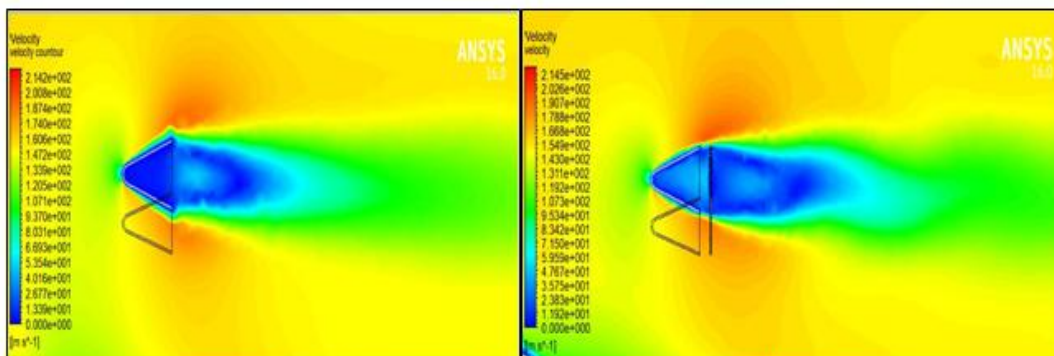

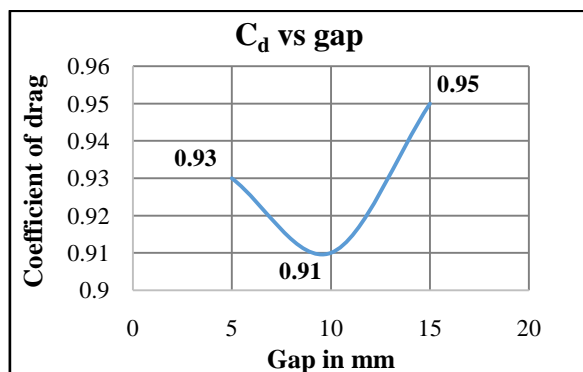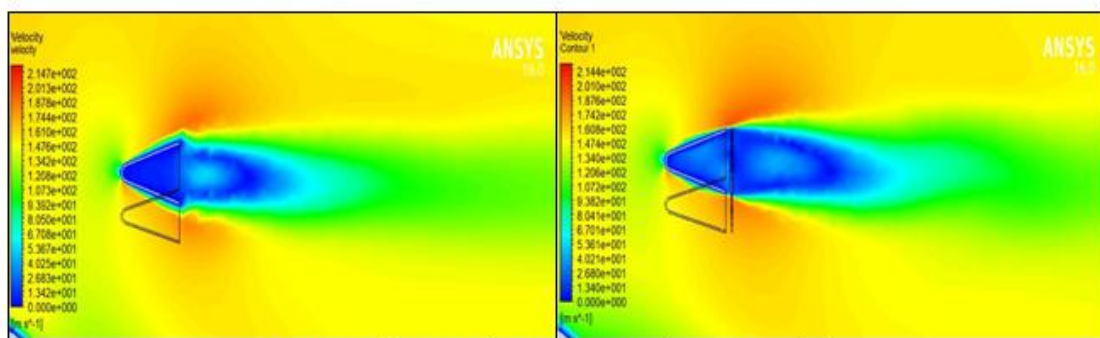*Fig 3.1- Flow over bare v-gutter and v-gutter with plate*

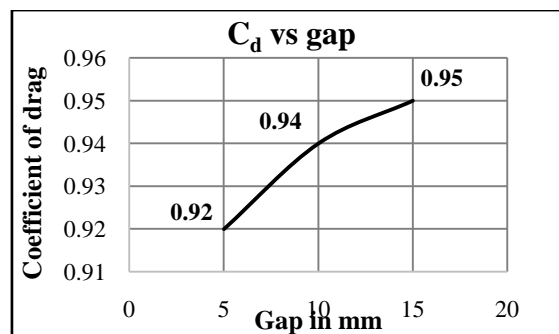*Fig 3.2- variation of C$_d$ with respect of spacing*

Keeping this value as the base, rounded edged disk is placed downstream of the gutter. The distance between the disk and the v-gutter was varied to 5mm, 10mm and 15mm and the drag coefficient were calculated to be as 0.93, 0.91 and 0.95 respectively. Hence it can be observed clearly that the reduction of drag obtained is maximum15.3% and minimum 11.2% for this geometry. Fig 3.1 shows the flow over bare v-gutter and v-gutter with plate. Fig 3.2 shows the variation of C_d with respect to spacing.

3.2 Case2 -Increasing the Sizeof theV-Gutter

Another factor affecting the drag is the size of the object. In this case the v-gutter size is increase from 35mm to 40mm whereas the angle of the v-gutter is kept same. The drag force obtained from the CFD fluent software is used in the drag equation and coefficient of drag is calculated to be as 1.05. Keeping this value as the base, rounded edged disk is placed downstream of the gutter. The distance between the disk and the v-gutter was varied to 5mm, 10mm and 15mm and the drag coefficient were calculated to be as 0.92, 0.94 and 0.95 respectively. Hence it can be observed clearly that the reduction of drag obtained is maximum 11.5% and minimum 8.7% for this geometry. Fig 3.3 shows the flow over bare v-gutter and v-gutter with plate. Fig 3.4 shows the variation of C_d with respect to spacing.

Fig 3.3- Flow over bare v-gutter and v-gutter with plate

Fig 3.4- variation of C_d with respect of spacing

3.3 Case-3: Aerofoil Shape Gutter

Shape is one of the important factors affecting the drag. In this case the gutter is in the form of an aerofoil and the coordinate are of NACA 0012 Aerofoil and the depth is kept same. The drag force obtained from the CFD fluent software is used in the drag equation and coefficient of drag is calculated to be as 0.98. Keeping this value as the base, rounded edged disk is placed downstream of the gutter. The distance between the disk and the v-gutter was varied to 5mm, 10mm and 15mm and the drag coefficient were calculated to be as 0.851, 0.853 and 0.89 respectively. Hence it can be observed clearly that the reduction of drag obtained is maximum 13.5% and minimum 8.6% for this geometry. Fig 3.5 shows the flow over bare v-gutter and v-gutter with plate. Fig 3.6 shows the variation of C_d with respect to spacing.
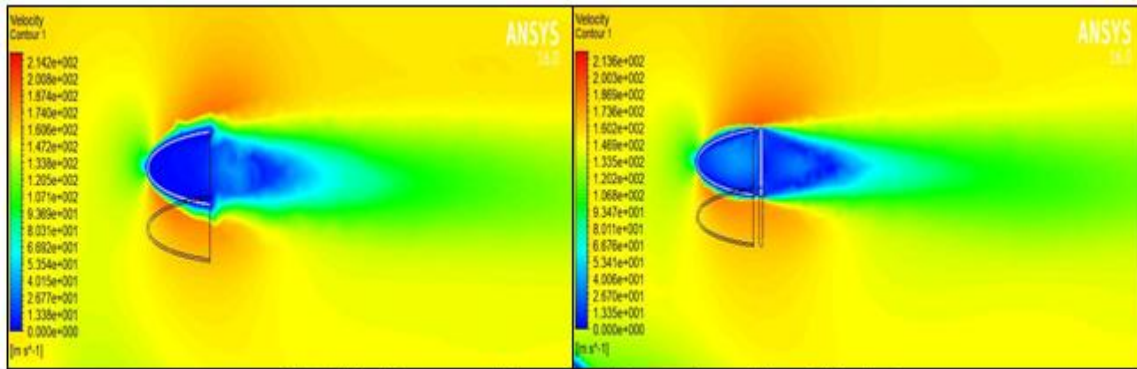
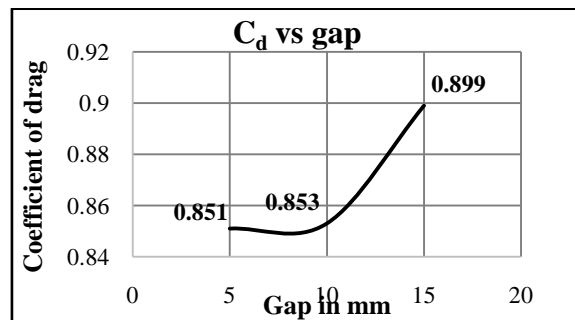*Fig 3.5- Flow over bare v-gutter and v-gutter with plate*

Fig 3.6- variation of C_d with respect of spacing

3.4 Comparison of cases

After the overall results are achieved and analysis is completed the comparison is been made. Fig 3.7(a) shows the comparison for v gutter with plate downstream, it can be seen from the above fig that the case 3 which is aerofoil shape gutter is more efficient when compare to other two cases. When v gutter shape is required then case 1 is better than the case 2. Fig 3.7(b) shows the comparison for all the cases when no plate is placed downstream of the gutter. Comparing the above results it can be said that the aerofoil shape gutter is more efficient than the v-gutter shape flame stabilizer.
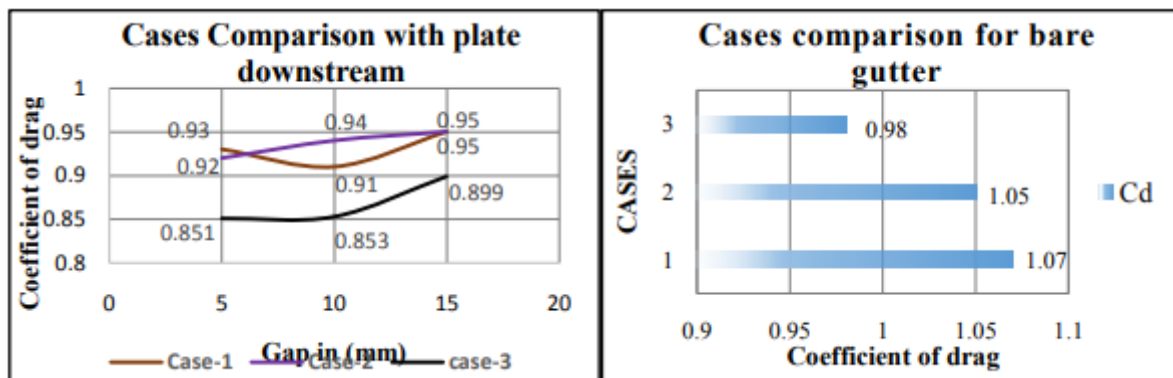

Fig-3.7(a) &3.7(b) Comparison of Cases

IV. CONCLUSION

It can be concluded from the CFD analysis carried out that the drag can be reduced by placing the bluff bodies downstream of the V-gutter. It can be observed from the above studies that coefficient of drag is less when compare to the bare gutter and further the drag varies with respect to the spacing between the gutter and the disk placed downstream of the v-gutter or aerofoil gutter. Considering the above cases it is good to use aerofoil shape gutter in place of v-gutter when only bare gutter are in used, whereas v-gutter shape is more efficient provided bluff bodies in downstream. Thus taking performance of the afterburner as consideration the suitable gutter has to be used as flame stabilizer in the afterburner section.

ACKNOWLEDGEMENTS

REFERENCES

[1]. S.Nakanishi, W.W.Velie, and L.Bryant ,*An investigation of effects of flame holder gutter shape on afterburner performance,* NACA Research memorandum, NACA RM E53J14, Feb-1954

[2]. Kareem.A, Tracy Kijewski, Po-chienlu, *Investigation of interference effects of group of finite cylinders*, Journal of wind engineering and Industrial aerodynamics, Vol-77-78, pp.503-520, Sep-1998.

[3]. T.Morel and M.Bohn*, Flow over two circular discs in tandem*, Journal fluids Engineering, Vol-102, pp.104-111, March-1980.

[4]. A.Prasad and C.H.K. Williamson, *A method for reduction of bluff body drag*, Journal of wind engineering and Industrial aerodynamics, Vol-69-71, pp.155-167, Jul-Oct-1997.

[5]. S Srinivasan, J J Isaac, C.Rajashekar, A.Arokkiaswamy, *Drag reduction of V gutter of an afterburner by tandem bluff bodies using CFD,* OSR Journal of Mechanical and Civil Engineering (IOSRJMCE), Aug-2012

[6]. P.F.Zhang, J.J.Wang, S.F.Lu.J.Mi, *Aerodynamic characteristics of square cylinder with a rod in staggered arrangement*, Experiments in Fluids, Volume 38, Issue 4, pp.494-502, March-2005.

[7]. Mehdi R. Khorrami, David P. Lockard, Meelan M. Choudhari, Luther N. Jenkins,Dan H. Neuhart, and Catherine B. McGinley "*Simulations Of Bluff Body Flow Interaction For Noise Source Modeling*" Report Number: AIAA Paper 2006-3203.

[8]. V.Ganesan, *Numerical computation of turbulent flow behind a conical baffle* Indian journal of Technology, Vol-18, pp.447-450, Nov-1980.

[9]. Shaik ShahabazAhamed, Dr. K.M Parammasivam "*Experimentation Of Flame Stabilization Using Strong Swirl For Afterburner Applications*" Proceedings of International Conference on Advanced Engineering and Technology-2015

[10]. Anand R, Lokesharun D, Rajkumar S, Kirubakaran R "*3D CFD Analysis In An Afterburner Using Numeca*". International Journal of Advanced Research in Engineering & Management (IJAREM) 2017.