Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, 2016 **ICCFD9-242**

The effect of combination of counterflow jet and spike on drag reduction for a blunt body at high Mach number flow

Ziba Eghlima [1], Kamyar Mansour[2]

[1, 2] Department of Aerospace Engineering, AmirKabir University of Technology, Tehran, Iran

Corresponding author: mansour@aut.ac.ir

Abstract: The present study is a numerical analysis of flow field around two spiked hemispherical nose cylinder model in a free stream Mach number of 6 with and without a counterflow jet at the nose of their spikes. The gas is injected through the nozzle at the nose of the spikes at sonic speed in opposite direction of the free stream flow. The difference between two models is the lengths of the spikes. In this numerical analysis, axisymmetric Reynolds-averaged Navier-Stokes equations are solved by the standard k-ε turbulence model. The results for the without jet conditions are validated with an experimental study. The purpose of this paper is to study the effect of combination of counterflow jet and spike on drag coefficient over spiked-blunt bodies. The results for with jet condition show a significant reduction of the total drag coefficient.

Keywords: Counterflow jet, blunt body, spike, Hypersonic

1 Introduction

The wave drag and heat transfer reduction in aerodynamic applications by a spike or a jet spike on a blunt-nosed body is well known ([1]-[6]). The wave drag reduction is derived from both the splitting of a single strong shock into multiple shock waves and effectively replacing the blunt body by a slender displacement. Even if the accumulative pressure rise across the multiple and sequential shock wave is identical to that of a single shock, the entropy jump across the multiple wave system is much less. This difference is because the entropy increment across each shock wave is proportional to the cubic power of the pressure jump. The blunt body with injection will, thus, produce a lower wave drag.

Kalimuthu in ref.[6] has experimentally studied the pressure variation on the blunt nose body and the aerodynamic coefficients such as drag, lift and pitching moment over the forward facing hemisphere aero spike at Mach number 6. His studies show high drag reduction in spiked-body in comparison to the no spike body. This study was our reference for validating the present study for without jet condition.

With the development of the aeronautics and astronautics, the advantage of opposing jet is more and more outstanding. In this century, some scholars kept doing research on this method ([7]-[10]. Hayashi [8] did the numerical and experiment study of thermal protection system by opposing jet and obtained some valuable conclusions. The high precise simulation of Navier-Stokes equations was used by Tian [9] to study the detailed influences of the free Mach number, jet Mach number, attack angle on the heat flux reduction and the mechanism was discussed.

Then, an innovative method was proposed by Huang et al for Drag reduction mechanism induced by a combinational opposing jet and spike at a free stream Mach number of 2.5 in 2015 [10].

The present work is a numerical analysis of flow field around two spiked hemispherical nose cylinder model in the free stream of Mach number of 6 with and without a counterflow jet at sonic speed. This study adjusted the jet on the nose of the spike of a sphere cylinder and numerically calculated and showed drag reduction for different length to diameter ratios (L/D=1.3 and 3.5) compared with no jet condition. In the present study, we used the geometry and conditions of experimental study of ref. [6]. The results of no jet conditions are validated with experimental study in ref. [6]. The purpose of this paper is to study the effect of counterflow jet on drag coefficient of flow over a spiked- blunt body at Mach 6. The results for with jet condition show a significant reduction of the total drag coefficient.

2 Geometry and grid generation

Two hemispherical nose cylinders with spike are modeled. All geometric parameters are such as the parameters in reference [6], but the calculations have done for the length to base diameter ratios of 1.5 and 3.5. The structured grids are generated over the models and grid study has done by refining the grids. Figure 1 and figure 2 show respectively the geometry of the models with and without jet. The structured grids generated over the models with L/D=1.5 respectively for without and with jet conditions are illustrated in figure 3 and figure 4.

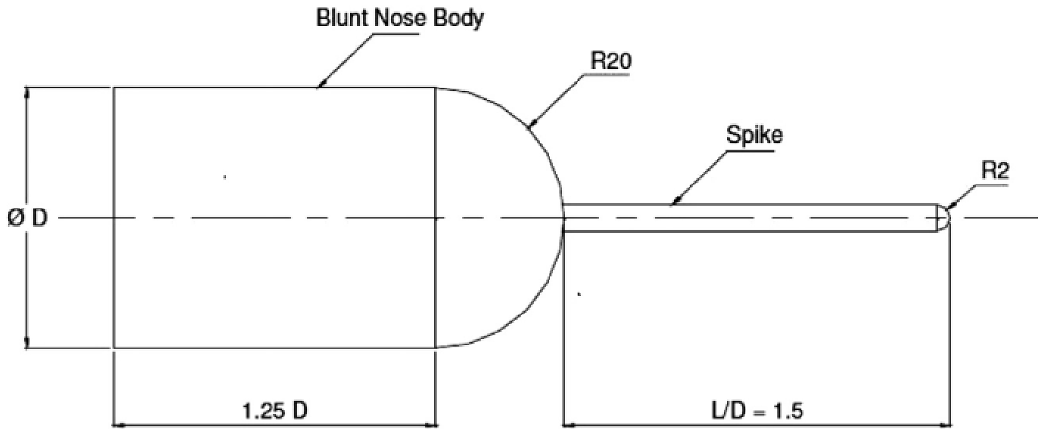


Fig.1. Geometry of model without jet (All dimensions in mm) [6]

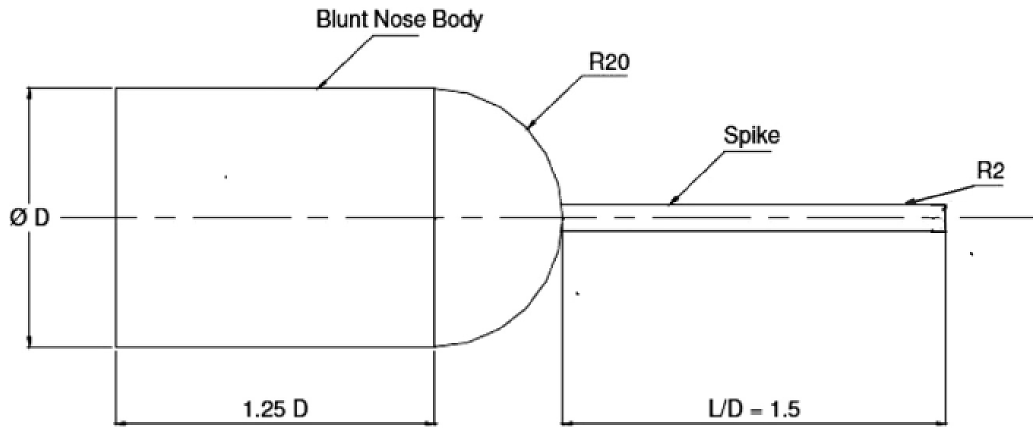
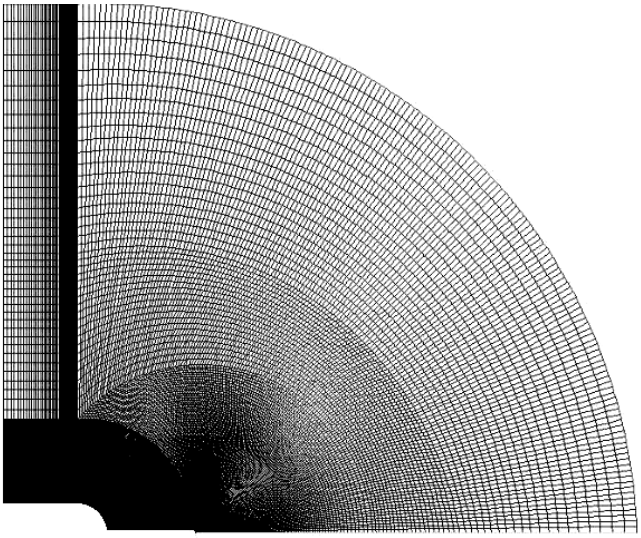


Fig.2. Geometry of model with jet (All dimensions in mm)



Figur 3.Grid generation over the model without jet (L/D=1.5)

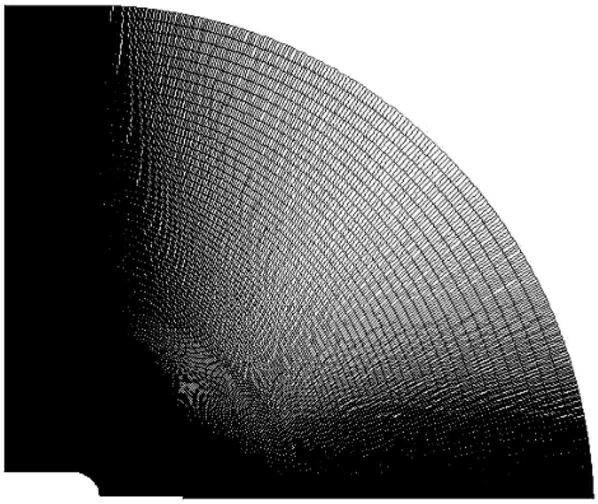


Figure 4.Grid generation over the model with jet (L/D=1.5)

3 Numerical procedure and boundary conditions

Present study considered a supersonic flow with a free-stream Mach number 6. The gas is injected through the nozzle at the nose of the spikes at sonic speed. An ideal gas is assumed for the free stream and the jet The values of the total pressure or stagnation pressure ratio of the jet to the free stream, Po_j/Po_f are chosen as 0.8.The total temperature or stagnation temperature ratio Toj/Tof is 1.0 with To_f=449 K. The selected parameters are the same as those in the experiment in ref. [6]. In the numerical analysis, axisymmetric Reynolds-averaged Navier-Stokes equations are solved with k-ε turbulence model. The simulations have been done using an implicit CFD solver. The solver employs a conservative, cell-centered, control volume formulation. A second-order upwind scheme was used to discretize both momentum and continuity equations with a coupled solver. The convective fluxes were treated using the Roe Flux-Difference Splitting Scheme [11], which has proven beneficial for improvement in the treatment and accuracy of shock simulations.

4 Results and Conclusions

Figure 5 shows the results of grid study and validation for without jet condition by pressure coefficient distribution over hemispherical nose surface. This parameter is converged for maximum numbers of cells are shown in these figures (798900 cells). The grid study has done by refining the grids. As can be seen in figure 1, the present results are almost compatible with the experiment results in ref. [6].

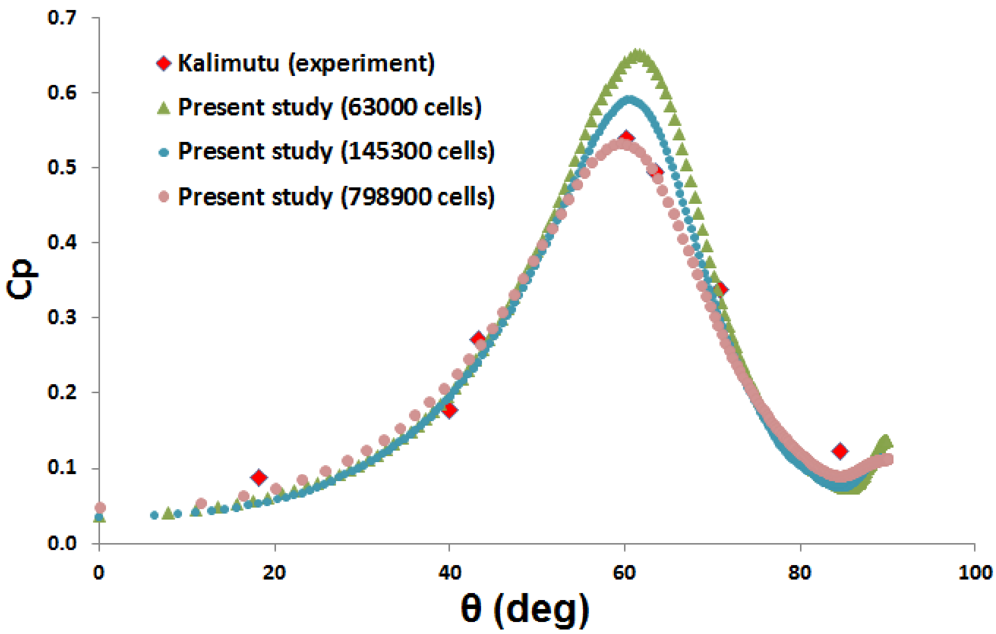


Figure 5.Grid study and validation for without jet condition (L/D=1.5)

For better description of the results, Mach contours over the models are illustrated. Figures 6 and 7 show respectively Mach contours for without jet conditions for L/D=1.5 and 3.5. The shocks near the tip of the spikes and along them over the main spheres are clear due to the sudden drop of Mach number. Figures 8 and 9 show respectively Mach contours for with jet conditions for L/D=1.5 and 3.5. The contractions between the conterflow jet and the freestream flow are seen. The structures of the new shock contours of these figures show the replacement of the slender shock by a thicker one for both models with jet.

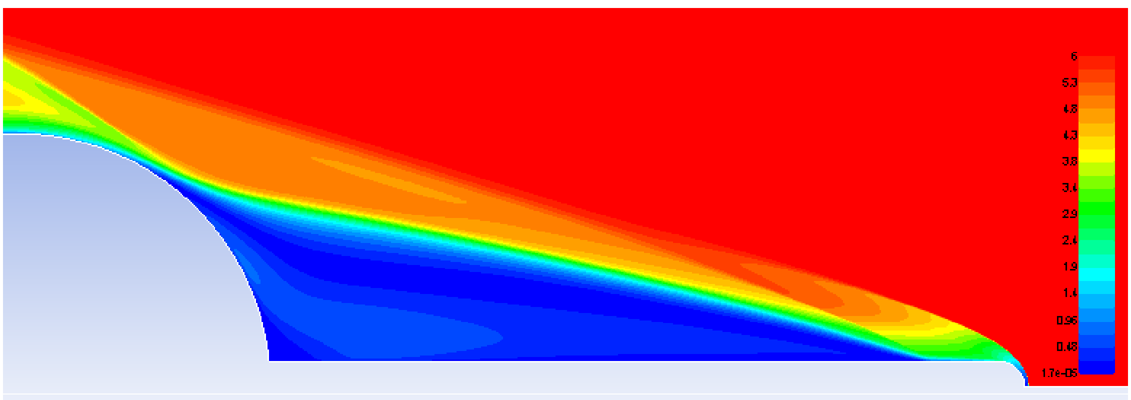


Figure 6. Mach contour for without jet condition (L/D=1.5)

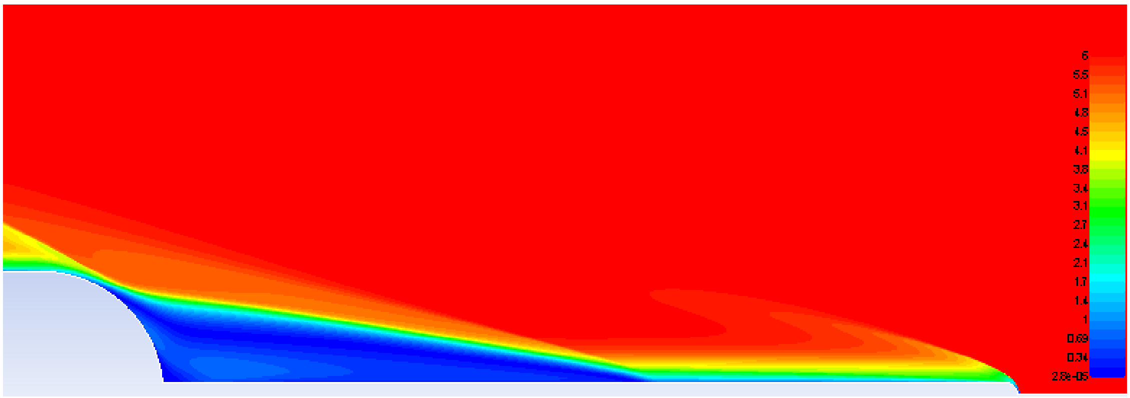


Figure 7. Mach contour for without jet condition (L/D=3.5)

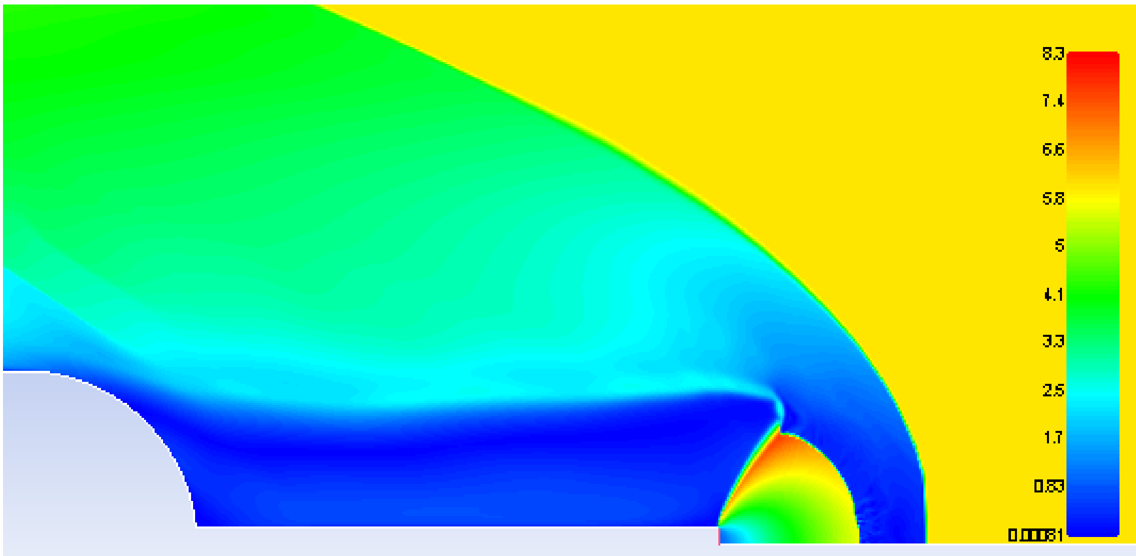


Figure 8. Mach contour for with jet condition (L/D=1.5)

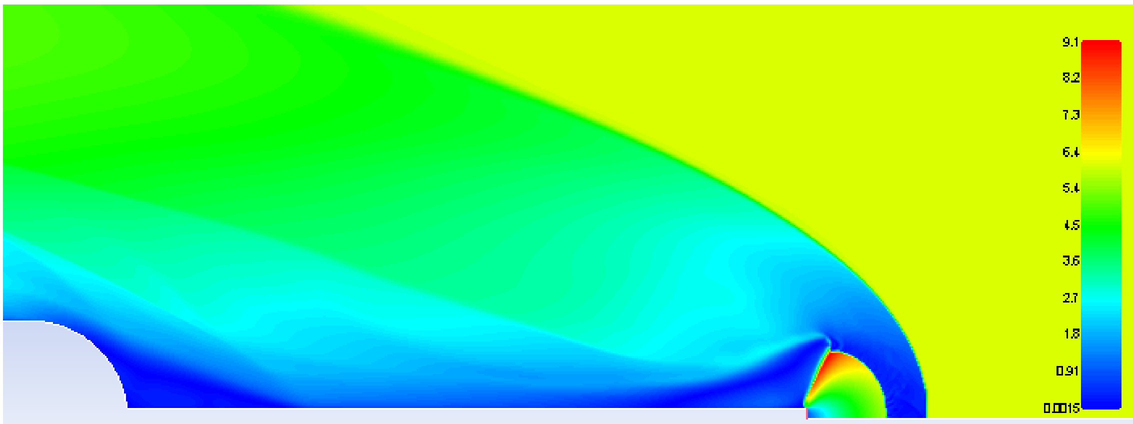


Figure 9. Mach contour for with jet condition (L/D=3.5)

Table.1 shows the Drag coefficients and drag reduction percentages for different conditions. This table shows a positive reduction for both viscous and pressure drag coefficients. The greatest contribution of the decline belongs to the pressure drag coefficient. Total drag reductions for spiked-body with L/D=1.5 and 3.5 with jets compared to the without jet model (L/D=1.5) in ref. [6] are about 62% and 69%. Also table.1 shows that the model with longer spike has a little more drag reduction at the same jet total pressure value.

Table 1. Drag coefficients and drag reduction percentage for different conditions

	Pressure drag	viscous drag	total drag	Total drag reduction (%)
L/D=1.5 (Without jet)	0.256	0.021	0.277	0.000
L/D=1.5	0.093	0.011	0.104	62.3
L/D=3.5	0.076	0.010	0.086	68.9

References

[1] Finley, P. J., The Flow of a Jet From a Body Opposing a Supersonic Free Stream, Journal of FluidMechanics, Vol. 26, Pt. 2, 1966, pp. 337–368.

[2] Bogdonoff, S. M., and Vas, I. E., Preliminary Investigations of Spiked Bodies at Hypersonic Speeds, Journal of the Aerospace Sciences, Vol. 26, No. 1, 1959, pp. 65–74.

[3] Barber, E. A., Jr., An Experimental Investigation of Stagnation-Point Injection, Journal of Spacecraft and Rockets, Vol. 2, No. 5, 1965, pp. 770–774.

[4] Warren, C. H. E., An experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body, J. Fluid Mech. 8, 1950, pp.400–417.

[5] Karashima, K., Sato K., An experimental study of an opposing jet, Bull., Inst. Space Aeronaut. Sci. Univ. Tokyo 11, 1975, pp.53–64.

[6] R.Kalimuthu, E.Rathakrishnan, Aerospike for drag reduction in hypersonicflow, AIAA4707, 2008, 21–23.

[7] Meyer B, Nelson H F, Riggins D., Hypersonic drag and heat-transfer reduction using a forward-facing jet, Journal of Aircraft, 2001, 38(4): 680-686.

[8] Hayashi K, Aso S, Tani Y., Experimental study on thermal protection system by opposing jet in supersonic flow, Journal of Spacecraft and Rockets, 2006, 43(1): 233-235.

[9] Tian T, Yan C., Numerical simulation on opposing jet in hypersonic flow, Journal of Beijing University of Aeronautics and Astronautics, 2008, 34(1): 9-12.

[10] Wei Huang, JunLiu,Zhi-xunXia ,Drag reduction mechanism induced by a combinational opposing jet and spike concept in supersonic flows, Acta Astronautica115(2015)24–31.

[11]Roe, P.L., Approximate Riemann solvers, parameter vectors, and difference schemes. J. Computational Physics, 1981, 43:357-372.