6th ANKARA INTERNATIONAL AEROSPACE CONFERENCE
14-16 September 2011 - METU, Ankara TURKEY

AIAC-2011-112

THE UNSTEADY COMPUTATIONAL INVESTIGATION OF TANDEM BODIES

Mahdiyeh Kanani[1] and Mahmood Pasandideh Fard[2]
Ferdowsi University
Mashhad, Iran

Amirreza Faghihi[3]
Ferdowsi University
Mashhad, Iran

ABSTRACT

In this work, two dimensional numerical modeling of two projectiles with various geometries has been carried out in which one projectile with Mach number 2 was pursued by projectile with Mach number 3. A blend of second and fourth differences of the flow variables are used and the utilized grid scheme was a zonal method which separates the flow field into smaller components. Additionally, unsteady fluid stream has been considered turbulent, viscous and compressible. First projectile has been hold constant in the environment with Mach number 2 while the second one was moving towards it with Mach number (-1). The obtained results show that two mentioned projectiles do not leave any effect on each other under the condition that they are located far away from each other. However when the second projectile is located close to the wake of the first one, it is influenced by Mach number less than 2 and consequently formed shock in front of second projectile would become weak. As a result, the streamlines in this region would lose their regularity and the drag coefficient would decrease. A comparison has been performed between mathematical modeling data and the experimental results reported by Kayser L.D. et.al [2] for the first projectile.

Keywords:projectile, wake, tandem body, unsteady, two dimensional

NOMENCLATURE

L	Length of both projectile	x	Axial direction
u	Dimensionless Horizontal fluid velocity	y	Vertical direction
v	Dimensionless Vertical fluid velocity	e	Total energy
t	Dimensionless Time	R	Ideal gas constant
γ	Thermal coefficients ratio	C_P	pressure coefficient
P	Dimensionless Fluid Pressure	C_v	Volume coefficient
ρ	Fluid Density	T	Temperature
M	Mach number	T_∞	Far field fluid temperature
M_∞	Far field fluid Mach number	μ	Viscosity
u_r	Relative dimensionless Horizontal fluid velocity	u_m	Dimensionless Horizontal mesh velocity
v_r	Relative dimensionless Vertical fluid velocity	v_m	Dimensionless Vertical mesh velocity

1 M.Sc student in Ferdowsi University of Mashhad,Email:Mahdiyeh_Kanani@yahoo.com
2 Associated prof in Ferdowsi University of Mashhad, Email: fard_m@ferdowsi.um.ac.ir
3 M.Sc student in Ferdowsi University of Mashhad,Email: amir_jpl@yahoo.com

INTRODUCTION

External flow between two projectiles and their wake region are very sophisticated. The analysis of the stream between two projectiles are very unpredictable due to the fact that the wake of the first projectile influences flow field of the second one. Experimental studies of multibody configurations, with one body in the supersonic wake of another, are very limited. However, some information is obtainable from investigations carried out during the 1960s [1, 2]. A limited number of previous computational studies have been completed on tandem-body configurations. Sahu performed computations for cylindrical bodies in the wake of a parent projectile [8]. At *Ma = 4.4,* the small-diameter multiple segments were ejected by the large parent body into its own wake. The drag of downstream segments was as much *as* 40% less than upstream segments. Berner presented computational results for a related problem using a full Navier-Stokes code with $k-\varepsilon$ turbulence model [10]. Pressure distributions on the trailing body were compared for various separation distances and showed fair agreement with experiment. Ober *et.al* have also investigated the tandem-body configuration [11, 12]. However, the interest was primarily on flows with large separation distances. These results showed that a toroidal vortex formed at the nose of the trailing body. The size of the vortex was dependent on the relative strengths of the reattachment pressure and the stagnation pressure along the wake centerline. When the trailing body was displaced radially, the toroidal vortex was replaced with a horseshoe vortex which diminished in size with increasing displacement. After that in 2008 Michael.D.Johnson supersonic base store ejection simulated with using Beggar Code [14]. İn our investigation, two projectiles with different Mach number are investigated. First projectile moves with Mach number 2 and the other with Mach 3. The behavior of the two projectiles in various distances has been studied. Pressure and Mach contour have been analyzed and compared with each other.

The computational technique

The system of equations used in this investigation was the two dimensional, thin-layer, Reynolds-averaged, full Navier-Stokes equations for a perfect gas. All of the parameters are used dimensionless.

$$\frac{\partial W}{\partial t}+\frac{\partial E_i}{\partial x}+\frac{\partial F_i}{\partial y}+H_i=\frac{\partial E_v}{\partial x}+\frac{\partial F_v}{\partial y}+H_v \tag{1}$$

Where W is conservative parameter, the vectors E_i and F_i are the inviscid terms and H_i is inviscid Source term. These parameters are given as:

$$W=\begin{bmatrix}\rho\\ \rho u_r\\ \rho v_r\\ \rho e\end{bmatrix}\quad H_i=\begin{bmatrix}0\\ 0\\ \frac{P}{y}\\ 0\end{bmatrix}\quad F_i=\begin{bmatrix}\rho v_r\\ \rho u v_r\\ \rho v_r v+P\\ v_r\left(\rho e+P\right)\end{bmatrix}\quad E_i=\begin{bmatrix}\rho u_r\\ \rho u_r u+P\\ \rho u_r v\\ u_r\left(\rho e+P\right)\end{bmatrix} \tag{2}$$

Where u_r and v_r are given as:

$$u_r=u-u_m \qquad v_r=v-v_m \tag{3}$$

In these equations, u_m and v_m are mesh velocity and $v_m=0$ because there are no motion in y direction. For the first projectile $u_m=0$ because the invironment has mach number 2 but for the second projectile $u_m=0.5$.(in this code $u_m=0.5$ is equal to $M=1$)

Where the vectors E_v and F_v represent the viscous shear stress and heat flux terms and H_v is viscous Source term. Flux vectors are defined as follows:

$$E_v=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{yx}\\ u\tau_{xx}+v\tau_{xy}-q_x\end{bmatrix}\quad F_v=\begin{bmatrix}0\\ \tau_{xy}\\ \tau_{yy}\\ u\tau_{xy}+v\tau_{yy}-q_y\end{bmatrix}\quad H_v=\begin{bmatrix}0\\ 0\\ \frac{\tau_{yy}-\tau_{\theta\theta}}{y}\\ 0\end{bmatrix} \tag{4}$$

In energy equation, e consists of internal energy ($u=C_vT$) and kinetic energy. Air assumed a perfect gas so:

$$Pv=RT \tag{5}$$

$$R = C_p - C_v \tag{6}$$

And e is defined as:

$$e = \frac{P}{\rho(\gamma - 1)} + \frac{1}{2}\left(u^2 + v^2\right) \tag{7}$$

The vectors of Shear stress are given as:

$$\begin{aligned}
\tau_{xx} &= \frac{2}{3}\frac{\mu}{\mathrm{Re}_\infty}\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{v}{y}\right) \\
\tau_{yy} &= \frac{2}{3}\frac{\mu}{\mathrm{Re}_\infty}\left(2\frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} + \frac{v}{y}\right) \\
\tau_{\theta\theta} &= \frac{2}{3}\frac{\mu}{\mathrm{Re}_\infty}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} - 2\frac{v}{y}\right) \\
\tau_{xy} &= \tau_{yx} = \frac{\mu}{\mathrm{Re}_\infty}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)
\end{aligned} \tag{8}$$

μ is computed with Sutherland equation:

$$\mu = T^{\frac{3}{2}}\left[\frac{1 + \dfrac{110.4}{\bar{T}_\infty}}{T + \dfrac{110.4}{\bar{T}_\infty}}\right] \tag{9}$$

Dimenssionless heat fluxes in energy equation are calculated with:

$$\begin{aligned}
q_x &= -\frac{\mu}{\mathrm{Re}_\infty(\gamma - 1)M_\infty^2 \mathrm{Pr}}\frac{\partial T}{\partial x} \\
q_y &= -\frac{\mu}{\mathrm{Re}_\infty(\gamma - 1)M_\infty^2 \mathrm{Pr}}\frac{\partial T}{\partial y}
\end{aligned} \tag{10}$$

Where Pr is prandtl number and T is dimensionless temperature that is calculated as the following equations:

$$T = \gamma M_\infty^2 \frac{P}{\rho} \tag{11}$$

Numerical Method

As the discretization scheme leads to central difference approximation for the governing equations, additional artificial dissipative terms are necessary to damp out high-frequency oscillations. Here a blend of second and fourth differences of the flow variables are used as described in Ref. [4]. The second difference terms are used to prevent oscillations at shock waves, while the fourth difference terms are important for stability and convergence to a steady state[9].

The Scalar Dissipation Scheme has proved very effective in practice in numerous calculations of complex steady flows, and conditions under which it could be a total variation diminishing (TVD) scheme have been examined by Swanson and Turkel[13].

The spatial discretization results in a system of ordinary differential equations in time, which is solved by an explicit fourth-order Runge-Kutta time stepping scheme.

In the present calculations, for solving turbulence equations, Baldwin-Lomax model is used [3].

Computational Grid and Boundary Condition

İn this study two different geometries are used (fig.1&2). The utilized grid scheme is a zonal method which separates the flow field into smaller components. Communication between these zones insured the solution was continuous across zone boundaries. Further details are available in the literature [7, 6]. In Fig.3, a representative grid is shown. For the axisymmetric cases, a three-plane solution is used to reduce the computational costs. The C-type grid and H-type grid are used for the first projectile and the second one, respectively and the distance between two projectiles utilizes H-type grid as well. The freestream conditions are set to Ma = 2, U=1 and V=0. The applied boundary conditions are as follows: symmetry across the centerline, nonreflecting condition at upstream, adiabatic and no-slip conditions at the wall of the first projectile, Mach number (-1) for wall of the second projectile, extrapolation along the outflow boundary.

Verification of the present computational approach to supersonic wakes was completed for single-body wakes [12]. Comparison between the predictions, Sahu's results and experiments in Ma=0.96 was shown then regions of acceptable and unacceptable performance could then be assessed. (Fig.4)

RESULTS

The procedure is started by assuming uniform free-stream conditions for all grid points in the computational domain. With implementing boundary conditions the computation marches in time until a steady-state solution is reached. After that the zone of two projectiles is compacted and procedure repeats. Fig.5 shows the recirculation region of behind of the first projectile. In the Mach 2, freestream impings on the shoulder, while creating a high-pressure region, interacts with the low-momentum fluid along the centerline and causes the flow to separate and recirculate.

When 2 projectiles are far from each other, the second one does not leave effect on the first one. In fig.6, the formed shock in front of the second projectile is presented. According to this figure, contours in illustrated region are representative of the Mach number 3. Fig.7 portrays pressure contour. As it can be seen, weak shocks are formed under the condition that the second projectile stands close to wake of the first one. In Figs 6 and 7, continuity of Mach contour in different zones is shown, as well.

Figs. 8 and 9 show streamlines in front of the second projectile. When two projectiles are located far from each other, the streamlines are regular (Fig.8). However, when the second projectile stands close to wake of the first one, the streamlines present irregular pattern (Fig.9).

The decrease of the distance between two projectiles results in the reduction of the drag coefficient. A sharp decline in the region close to the wake of the first projectile can be observed obviously, as shown in Fig. 10.

CONCLUSION

A numerical investigation of the flowfield around a body located close to a supersonic wake was undertaken so as to determine aerodynamic characteristics. For coaxial alignments, when two projectiles are far from each other, shock of Mach number 2 and 3 occur in front of the first and second projectile, respectively. But under the condition that the second projectile stands close to the wake of the first one, the shock of second projectile becomes weak. In this case, the drag coefficient decreases and its gradient in mentioned region is sharper.

FIGURES

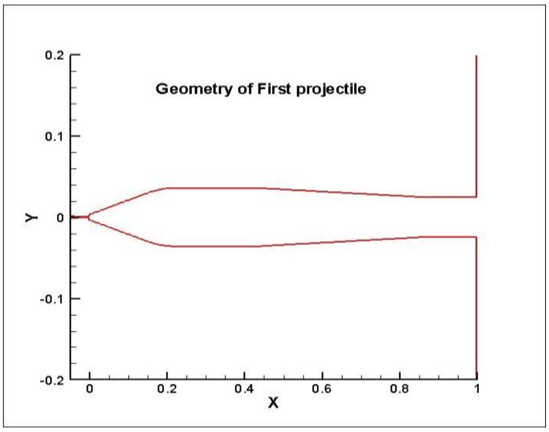

Figure 1: *Geometry of the first projectile*

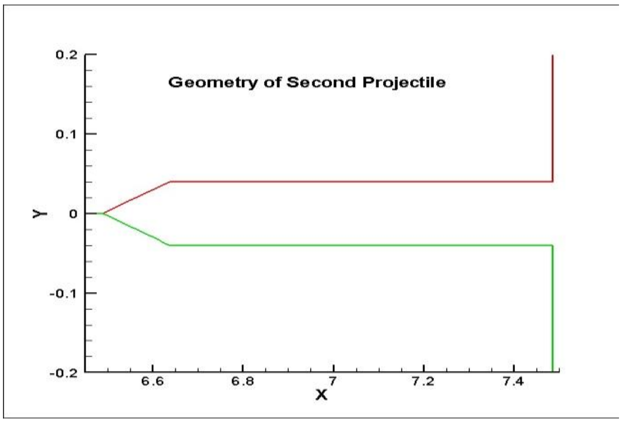

Figure 2: *Geometry of the second projectile*

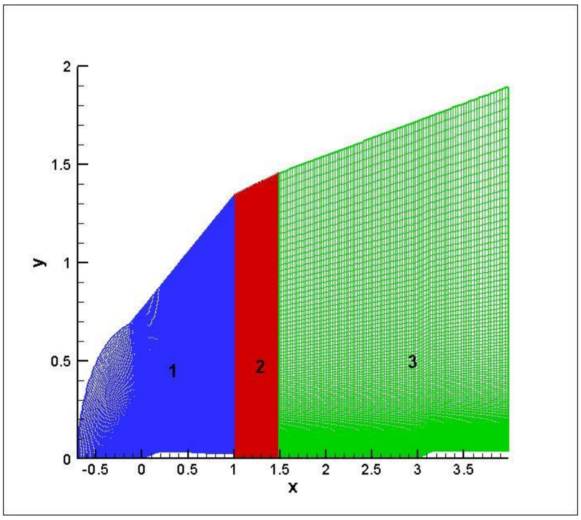

Figure 3: *Illustration of the zonal grid*

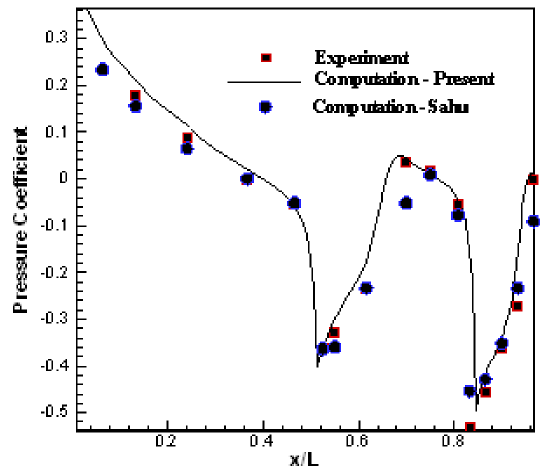

Figure 4: Longitudinal Surface Pressure in the Windward Plane ($M_\infty = 0.96$, $\alpha = 4^\circ$)

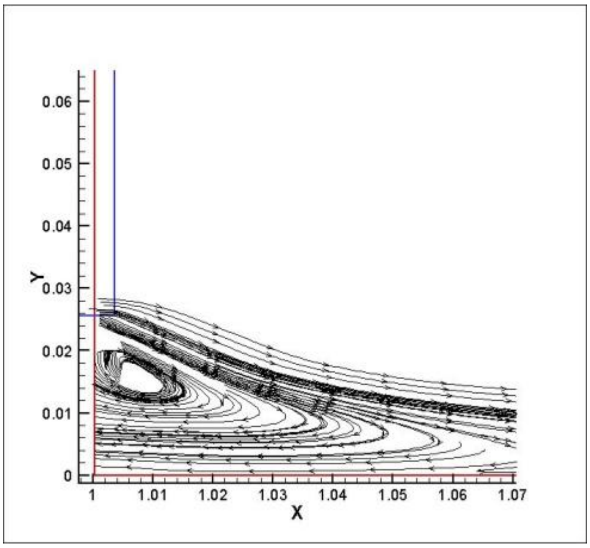

Figure 5: *Recirculation region in the behind of the first projectile*

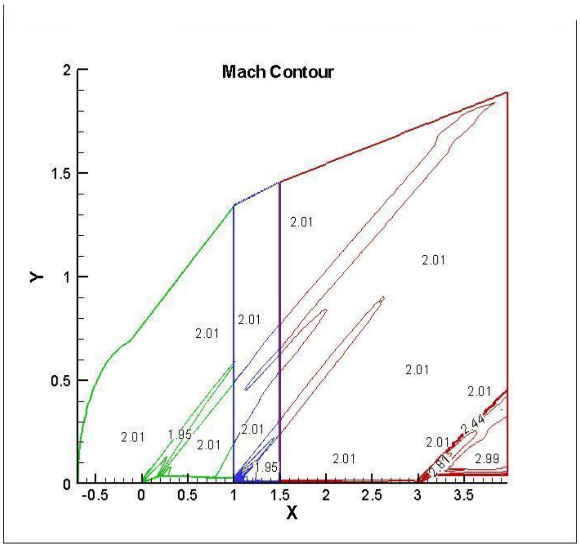

Figure 6: *Mach contour under the condition of being away from each other*

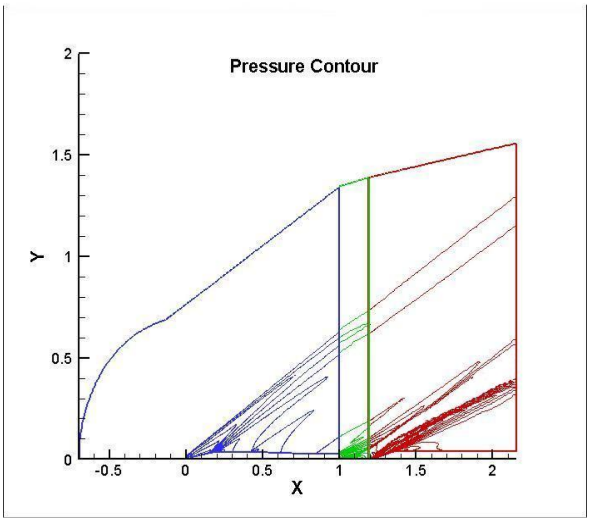

Figure 7: *Pressure contour under the condition of being located close to each other*

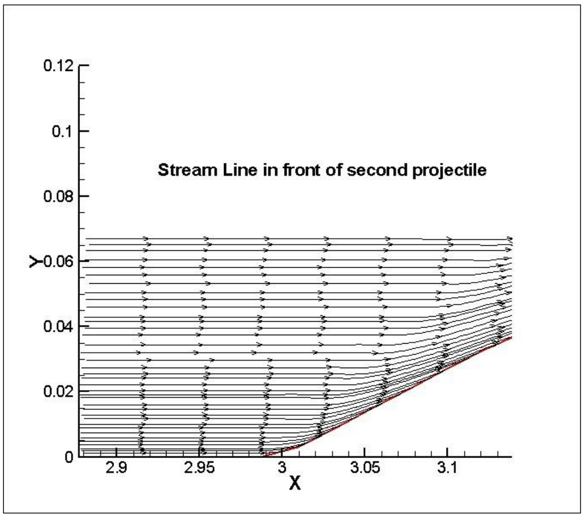

Figure 8: *Streamlines in front of the second projectile under the condition of being away*

from each other

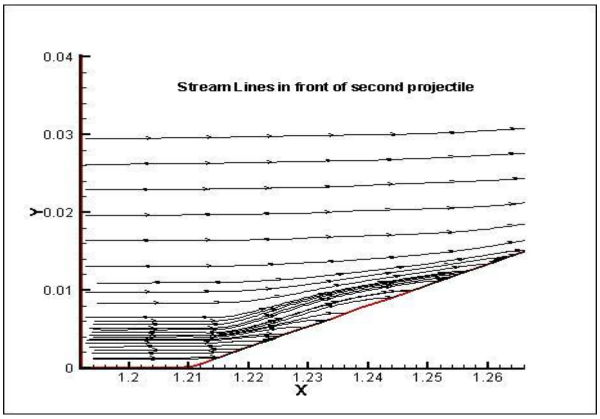

Figure 9: *Streamlines in front of the second projectile under the condition of being located close to each other*

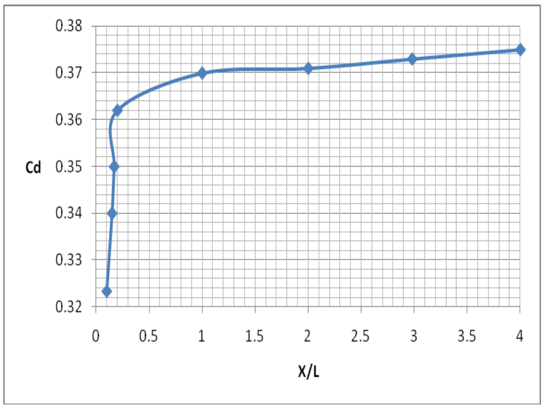

Figure 10: *Drag coefficient in the various distances*

References

[1] Beheim, M., "Flow in the Base Region of Axisymmetric and Two-Dimensional Configurations," NASA TR R-77, Sept. 1960.

[2] Roshko, A., and Thomke, G., "Observations of Turbulent Reattachment behind an Axisymmetric Downstream-Facing Step in Supersonic Flow,"AIM Journal, Vol. 4, No. 6, pp. 975-980, 1966.

[3] Baldwin B.S. and Lomax H., “Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows,” AIAA Paper, 78-257, 1978.

[4] Jameson A. and Schmidt W., “Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes,” AIAA Paper 81-1259, June 1981.

[5] Kayser L.D. et al, “Surface Pressure Measurement on a Boattailed Projectile Shape at Transonic Speeds,” ARBRL-MR-03161, March 1982.

[6] J.Sahu and J.Steger, "Numerical Simulation of Three Dimensional Transonic Flows", AIAA paper 87-2293, 1987.

[7] J.Flores, "Simulation of Transonic Viscous Wing and Wing-Fuselage Flows using Zonal Methods". Inn Proceedings of the First International Symposium on Domain Decomposition Methods for Partial Differential Equations, pp. 381-415, SIAM, 1988.

[8] J.Sahu and C. Nietubicz, "A Computational Study of Cylindrical Segments in the Wake of a projectile", Tech.Rep.BRL-TR-3254. Ballistic Research Laboratory, U.S.ArmyCommand, Aberdeen Proving Ground, August 1991.

[9] Swanson R.C. and Turkel E., “*On Central Difference and Upwind Scheme*,” J. Comp. Phys., 101:297-306, 1992.

[10] C.Berner, "Aerodynamic Interference Between Projectiles: Measurements and computation", in 14th *international symposium on Ballistics.* (Quebec, Canada), pp. 775-784, September 1993.

[11] C.C.Ober, J.P.Lamb, and T.M.Kiehne, "A computational study of the Aerodynamics of a Body Immersed in a Supersonic Wake". AIAA paper 94-3519, 1994.

[12] C.C.Ober. "A Computational study of the Aerodynamics of a Body Immersed in a Compressible Wake". PhD thesis, University of Texas at Austin, Austin, Texas 78731, May 1994.

[13] Jameson A., “*the Present Status, Challenges, and Future Developments in Computational Fluid Dynamics,*” Twelfth Australasian Fluid Mechanics Conference, The University of Sydney, Australia 1995.

[14] Michael D. Johnson."Dynamic Sup e r sonic Base Store Ejection Simulation Using Beggar". PhD thesis, Department Of The Air Force and Air University, September 2008.