http://ansinet.com/itj

ITJ

ISSN 1812-5638

INFORMATION TECHNOLOGY JOURNAL

ANSInet
Asian Network for Scientific Information
308 Lasani Town, Sargodha Road, Faisalabad - Pakistan

Information Technology Journal 12 (11): 2174-2180, 2013
ISSN 1812-5638 / DOI: 10.3923/itj.2013.2174.2180

Pressure Loss Computation on Inner-flow Field of a Muffler

[1,2]Bing Wang, [3]Yongjuan Wang and [3]Cheng Xu
[1]School of Mechanical Engineering, Nanjing University of Science and Technology, China
[2]School of Mechanical Engineering, Huaihai Institute of Technology, China
[3]School of Mechanical Engineering, Nanjing University of Science and Technology, China

Abstract: The muzzle resistant muffler is a common silencer in fire arms which prevent from sound transmission and allow flow pass. It is the main technical measure. The article mainly build the mechanics model of the muffler and numerical simulate the flow field using 3D CFD and then analyze variation process of muzzle interior flow including two cases about vent hole and non-vent hole. The results disclose the interior flow's structure and alteration and find the reason of the pressure loss. The numerical simulation results take on reference signification to evaluate the characteristic of the muffler and know the muzzle interior flow mechanism and offer technical support to same type mufflers.

Key words: Resistant muffler, interior flow, numerical simulation, pressure loss

INTRODUCTION

Various technical measures of elimination muzzle airflow noise will be from the internal ballistic scheme to consider, from view of gas dynamics and acoustic principle, these factors are interrelated and influence each other, all kinds of measures have some effects on noise elimination but actual bore hole silencer is relatively complex, when weapon launch, as the projectile leaves the chamber after muzzle, incomplete oxidation of high temperature and high pressure chamber formed by the gunpowder gas quickly out of shock wave, secondary combustion as well as the effect on weapon recoil and so on a variety of negative effects, will seriously affect the working performance of weapon system, the muzzle through the use of the device, change the chamber air emptying of mode and the flow equation, from the export momentum to change the air flow, the characteristics of muzzle shock wave and suppressing certain adverse effect. As a muzzle device design theoretical basis, people chamber of muzzle flow mechanism to explore continuously. Since the 80's, the experiment and the transient measurement technique in study of transient variable intensity of the complex flow plays a huge role, chamber flow field especially muzzle system device back flow field, a further study is made for the main structure and characteristics of the muzzle flow field (Huang *et al*., 1998; Zhang *et al*., 2008).

The development of Computational Fluid Dynamics (CFD) provides another fluid mechanics calculation and the corresponding analysis method. Muzzle flow field with TVD was computed (Jiang *et al*., 1998), considered in the calculation of the cylindrical projectile flow field, the influence of well bore flow field distribution. Using MUSCL format of 3D without a tiny muzzle of near field and far field are calculated (Sakamoto *et al*., 2001). Not considering the effects of the projectile are calculated to 3D muzzle peacekeeping chamber with chemical reaction flow field (Zibarov *et al*., 2002). So far, the study of flow field inside the chamber muzzle resistance muffler has yet to see literature reports. Resistant muffler using computational fluid dynamics method, the author of this study, through the numerical simulation of flow field structure, study the characteristics of silencer for internal flow field, trying to reveal the resistance muffler noise restraining mechanism provides a theoretical basis.

BASIC EQUATIONS AND CALCULATION METHOD

Basic equations: Research problem is the muzzle inner unsteady flow chamber with multi-cavity resistant muffler, after high temperature and high pressure gas after the device muzzle spray into the atmosphere, don't consider the effect of the projectile, resistant muffler is shown in Fig. 1 (Wang *et al*., 2010).

Adopting the ax symmetric compressible Navier-Strokes equation to describe the internal flow and complex flow of outflow, in initial time chamber inlet parameters of air chamber diameter as a dimensionless equation and characteristic parameters, get the following dimensionless form of the Navier-stokes equations (Siano, 2011).

Corresponding Author: Bing Wang, School of Mechanical Engineering, Nanjing University of Science and Technology, China

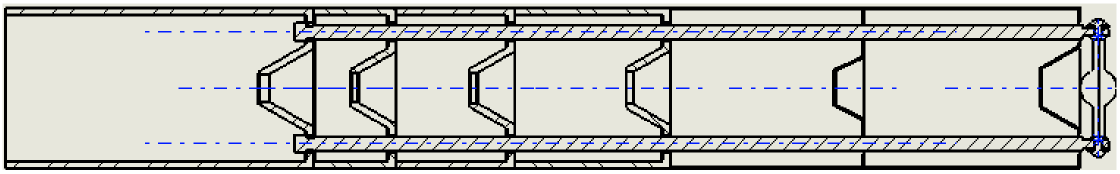

Fig. 1: Resistant muffler structure

$$\frac{\partial Uy}{\partial t}+\frac{\partial Fy}{\partial x}+\frac{\partial Gy}{\partial y}=\frac{\partial F_v y}{\partial x}+\frac{\partial G_v y}{\partial y}+H \quad (1)$$

While:

$$\begin{aligned}
&U=[\rho,\rho u,\rho v,\rho e]^T,\\
&F=[\rho u,\rho u^2+p,\rho uv,\rho ue+pu]^T,\\
&G=[\rho v,\rho uv,\rho v^2+p,\rho ve+pv]^T,\\
&F_v=Re^{-1}\left[0,\tau_{xx},\tau_{xy},\tau_{xx}u+\tau_{xy}v+\frac{\gamma}{\gamma-1}\frac{\mu}{Pr}\frac{\partial T}{\partial x}\right]^T,\\
&G_v=Re^{-1}[0,\tau_{xy},\tau_{yy},\tau_{xy}u+\tau_{yy}v+\frac{\gamma}{\gamma-1}\frac{\mu}{Pr}\frac{\partial T}{\partial y}]^T,\\
&H=Re^{-1}[0,0,-\tau_{\theta\theta}+pRe,0]^T,\\
&\tau_{xx}=2\mu\frac{\partial u}{\partial x}-\frac{2}{3}\mu\nabla\cdot V,\tau_{xy}=\mu(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial x}),\\
&\tau_{yy}=2\mu\frac{\partial v}{\partial y}-\frac{2}{3}\mu\nabla\cdot V,\\
&\tau_{\theta\theta}=2\mu\frac{v}{y}-\frac{2}{3}\mu\nabla\cdot V,\\
&\nabla\cdot V=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{v}{y},\\
&e=\frac{1}{\gamma-1}\frac{p}{\rho}+\frac{1}{2}(u^2+v^2)
\end{aligned} \quad (2)$$

While μ is specified by equation Sutherland, Prandtl Pr = 0.72.
Perfect gas equation of state:

$$p=\rho RT \quad (3)$$

While perfect gas unit volume total energy:

$$e=\frac{p}{\gamma-1}+\rho\frac{u^2+v^2}{2}$$

γ is specific heat ration of gas.

Finite volume method: Finite volume method is also known as control volume method. The basic idea is: calculated area was divided into grids and each grid point around a repetition of each control volume; will solve differential equations for each control volume integral, thus a set of discrete equations are obtained. The unknown is the dependent variable u on the grid. In order to calculate control volume integral, must assume that the value of the change rule between the grid points. From the selection method of integral area, the finite volume method belongs to the sub-domain method, the weighted residual method from the point of the approximate method of the unknown solution, finite volume method belongs to the local approximation of the discrete method. In short, sub-domain method with discrete, it is the basic method of finite volume method (Chaitanya and Munjal, 2011; Lee *et al*., 2006).

COMPUTATIONAL RESULTS AND ANALYSIS

Computational simulation to inner-flow field on automatic rifle resistance muffler: A numerical simulation of the flow field with muffler mainly includes the initial flow field and gunpowder gas flow field. The initial flow field is the result of the projectile move before air, gunpowder gas flow field in the muzzle is the gunpowder gas of high temperature and high pressure spray chamber outlet. On the muffler the flow field is used to numerically simulate, all did not consider the effect of initial flow field. Projectiles from guns and gunpowder gas begins to flow at moment as the initial moment, this moment interior gunpowder gas high temperature, high pressure and atmospheric pressure outside the chamber as the calculation of the initial state. This article is based on a study by 5.8 mm automatic rifles as application background, through the medicine chamber after rifling, barrel length is 468 mm and diameter 5.8 mm, not considering the influence of the rifling. Considering the symmetry model, only computes for about a quarter of the area, the calculation area is divided into interior ballastic and muffler internal two parts. Innersole center at the origin of coordinates, muzzle direction to x axis direction, the mesh density, the far field region near the muzzle grid is relatively sparse (Bogey and Bailly, 2004).

Classic interior ballistic assumes that the powder gas and not burning powder solid after the play space is uniform, which can be pushed out Speed is linear distribution in the chamber, the pressure within the chamber is a parabola distribution, that is:

$$v_x = \frac{x}{L} v \quad (4)$$

While, v_x is x coordinates the gunpowder gas velocity, L is space length after playing, v is gunpowder gas velocity after playing

$$p_x = p_d \left[1 + \frac{\omega}{2\varphi_1 q} \left(1 - \frac{x^2}{L^2} \right) \right] \quad (5)$$

Where:

p_x = x coordinates gunpowder gas pressure
p_d = Pressure at the bottom of bullet
ω = Explosive load
q = Pill quality
φ = Internal trajectory second power coefficients
φ_1 = 1.02

Boundary condition in numerical simulation of the work included symmetry boundary conditions including the solid wall surface and the conditions of pressure outlet, solid wall and the symmetry plane uses a default value, export the pressure condition is set to the standard atmosphere conditions (Bin *et al*., 2008).

Mechanism analysis

Velocity analysis: Figure 2 is multi-cavity resistance without vent muffler at T = 0.004~0.0076 sXOY the velocity contour map, Red stands for high-speed, blue stands for low speed, when T = 0.004 s, gunpowder gas exhausts from the barrel in body tube, because it flows from small to large diameter of the cavity, but due to the pressure, the gas is not just into the first cavity to expand, but along the diameter of the barrel into the cavity in the body, in the first half of the first cavity expansion, then in the middle part after two speed the larger vortex along the exhaust passage into the second chamber, another part of the first cavity of the cavity forming the backflow.

Gunpowder gas through the second, three, four cavity, in the second chamber to form two smaller eddies, entered the fifth cavity and no discharge the fifth cavity and at T = 0.0044 s, gunpowder gas decreases to the first chamber of the speed, but still form two symmetrical vortices in the middle of the cavity and is bigger than last time, then in the third and the second cavity through the exhaust passage of gas flow rate increases and when T = 0.0048 s, originally swirling gradually disappear in the middle of the first cavity, due to the third and the second cavity of the cavity length is very small, no formation of vortex in the third and the second cavity, at this time the largest gas speed still in the third and the second cavity exhaust passage, gas in the fourth form smaller reflux cavity, at the same time the fourth chamber exhaust passage near gas velocity increasing, the gas still to fifth cavity discharge at T = 0.052 s moment, the first chamber of the eddy current has disappeared, the exhaust passage near the gas velocity increasing, the gas already exhausts into the sixth cavity. To T = 0.056 s, at this point in the center of the first cavity gas velocity increases and area increase gradually, at the back of the first cavity gradually formed vortex, the third chamber to form two smaller vortexes, gas have already been to the sixth cavity, the fourth cavity of the vortex gradually become weak, in the fifth cavity gradually formed vortex. Gas exhaust muffler when T = 0.0076 s.

Analysis of pressure loss: Muffler pressure loss mainly include pressure loss generated by the interior channel cross section and the airflow friction (referred as frictional pressure loss) and pressure loss by channel inflexion (inside the muffler), the structure of the cross section changes such as local changes in air pressure loss resulting from flow state changes (referred as local pressure loss). Both are caused by fluid movement overcome viscous shearing stress, including local losses. This muffler is six chambers in series, each cavity local decline. The pressure loss is mainly caused by partial loss. In local structure occurs in the redistribution of vortex region and velocity.

It can be seen from Fig. 2, in the vortex area, the rotation of the fluid is irregular, collision, reflux, often cause huge obstacles to the mainstream movement, energy consumption mainstream movement, resulting in a loss of pressure, energy, the vortex region is a partial loss of the common phenomenon. The redistribution of velocity not only intensifies internal friction in the mainstream, mainstream and micro fluid before and after the collision caused by increase of turbulent.

Fluid from the small diameter of the pipe flow to the large diameter pipe, due to the fluid inertia, it can't enlarge according to the shape of the pipe suddenly but expand gradually after leaving the small tube, therefore, in the form vortex and flow between the beam, wall corner vortex beam by mainstream drives spin. Main beam put energy to whirlpool. Vortex put the energy consumption in the rotation movement (into heat dissipation). Small diameter pipe in the flow of fluid has a high speed, is bound to impact large diameter low flow velocity of fluid in pipe to occur a collision. Fluid from the large diameter pipe flow to the small diameter pipelines, streamline must bend, beam must shrink.

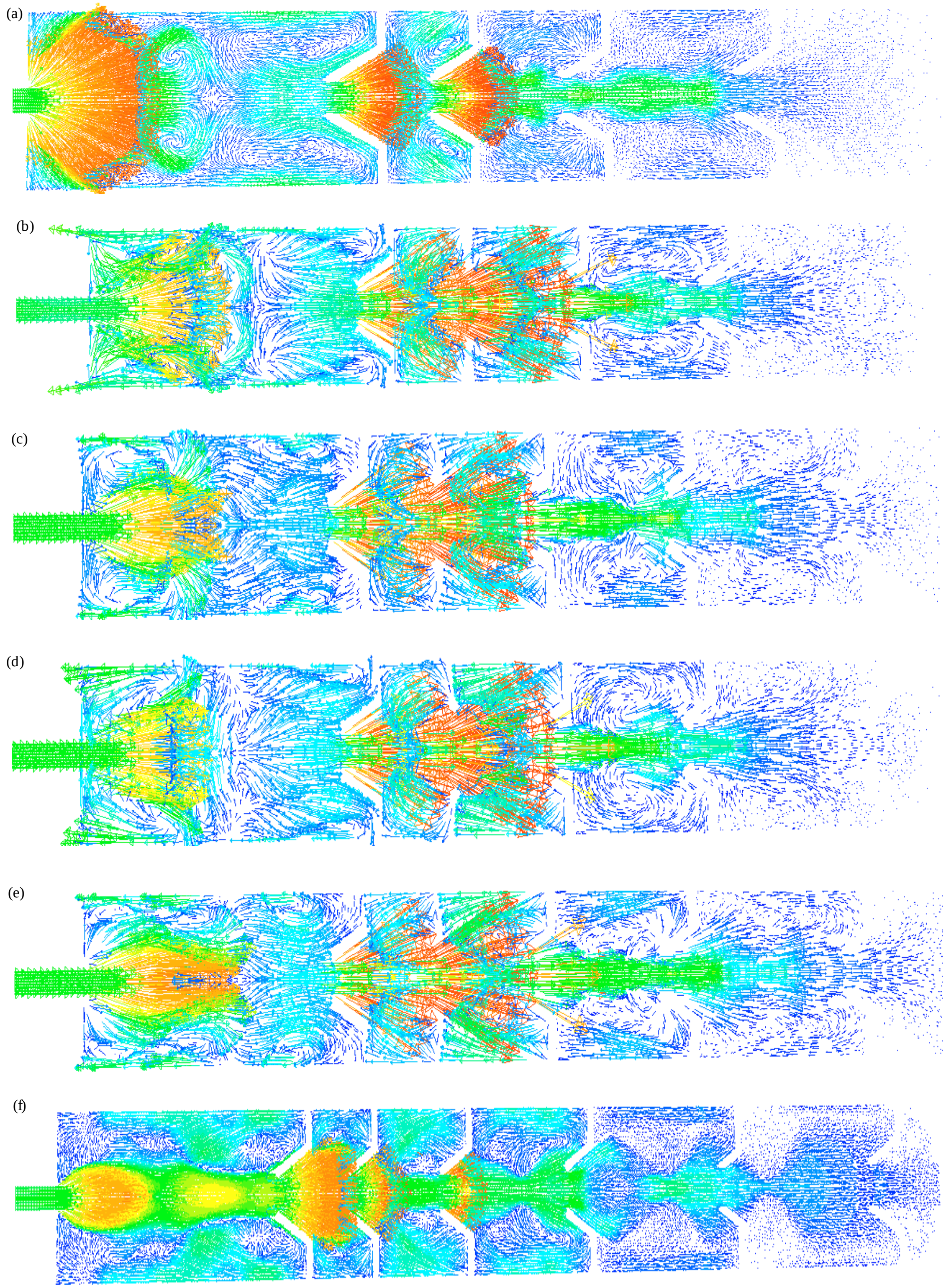

Fig. 2(a-f): Different time velocity contour graph, (a) T = 0.004 s, (b) T = 0.0044 s, (c) T = 0.0048 s, (d) T = 0.0052 s, (e) T = 0.0056 s and (f) T = 0.0076 s

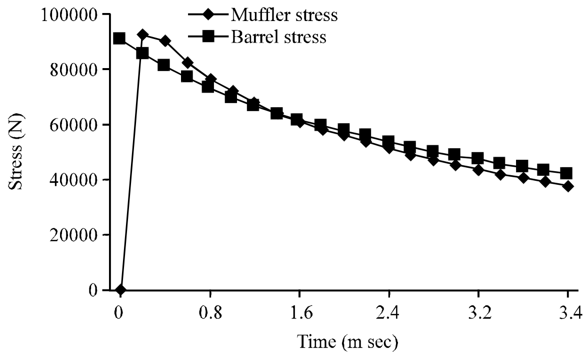

Fig. 3: Barrel and muffler stress variation with time

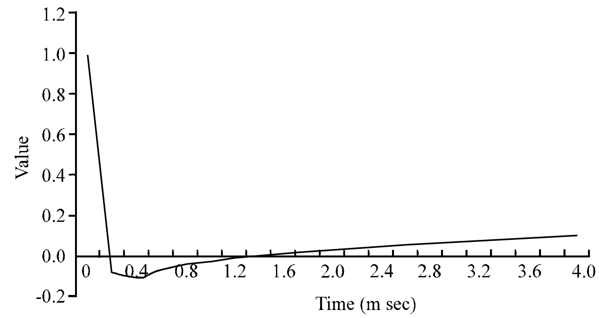

Fig. 4: Impulse characteristic value variation with time

After the fluid enters into the small diameter pipe, due to the fluid inertia, a fluid continues to shrink until the minimum cross section (called the neck), then gradually expand, until it is full of cross section of the small diameter. Near the neck of a beam between the wall and a low pressure area is full of small vortex. In large diameter section connected to the small diameter of the cross section of convex shoulder, also often form spiral. All of the vortex movement consumes energy; In streamline curvature, fluid acceleration and deceleration process, the fluid particle collisions, the velocity distribution change also will cause energy loss, resulting in the muffler pressure loss.

Figure 3 is 1/4 barrel and 1/4 muffler stress variation with time. As the gunpowder gas chamber to drain out of the insole pressure decreases, the barrel axial force has been smaller. And the stress of the silencer size is a process of first big and after small, this is because at the beginning of the after-effect period, the gunpowder gas is a gradual process, before the gunpowder gas filled with muffler, muffler grows in the axial force. Since the gunpowder gas is full of muffler, with the muzzle pressure drop, muffler force will gradually become smaller.

Figure 4 is impulse characteristic value χ variation with time. Because in the after-effect period muffler stress has a larger size after first process, so the impulse characteristic size has an obvious descent, muffler reached the maximum force, its value and the body tube stress are constantly decrescent, the magnitude of the impulse characteristics also is just a small change. After the stress peak and traditional theory thinks that muffler will keep geometric with rob cannon force drops, meaning that impulse characteristics will remain unchanged. Caused the main reason for this difference may be due to the traditional theory ignores the practical factors such as viscosity, is established based on many assumptions. Because in order to get the entire aftereffect period requires a lot of machine, this paper take when t = 0.76 m sec instead of on the value of engineering application. When t = 0.76 m sec, χ = 0.04 and efficiency η_r = 31.8% as the muzzle system can be calculated out device.

SIMULATION RESULTS OF THE COMPARISON WITH VENT HOLE AND WITHOUT VENT HOLE

According to the muffler in above, in the case of the other conditions unchanged, compared the two kinds of mufflers, one is no vent, another adds two symmetrical vent in each silencing the side of the bowl, as can be seen from the Fig. 5, two kinds of mufflers gas velocity distribution is similar, but the time in the vent gas from barrel to the outlet is longer than there is no vent need long time, when T = 0.3932 s with the vent hole muffler has reached the maximum value at the exit. For the non-vent hole muffler, maximum velocity when T = 0.488 s only reached the fifth cavity, this is due to the absence of air breather to hinder the role of gas is reduced, makes the flow more smoothly, the pressure loss is reduced.

Figure 5a is velocity contour without vent hole. Most of the air flows out through the perforated pipe, tube and expansion of gas cavity in the gas exchange is less, so expansion velocity inside the cavity is low, the impact of the cavity wall rather small, so the energy loss is small.

Figure 5b is velocity contour with vent hole. From radial perforated tube muffler expansion chamber, gas flow inside the muffler to reduce gradually but near the perforated tube muffler is greatly improved, some is small but relatively strong turbulence. Expansion chamber internal gas flow velocity is more uniform and the impact on the expansion chamber is small. But because gas needs to pass through expansion chamber into the tube, near the holes in the process of the turbulent kinetic energy than the pass-through hole muffler parts produce turbulent kinetic energy, so this type of pressure loss of muffler is far bigger than muffler without vent hole.

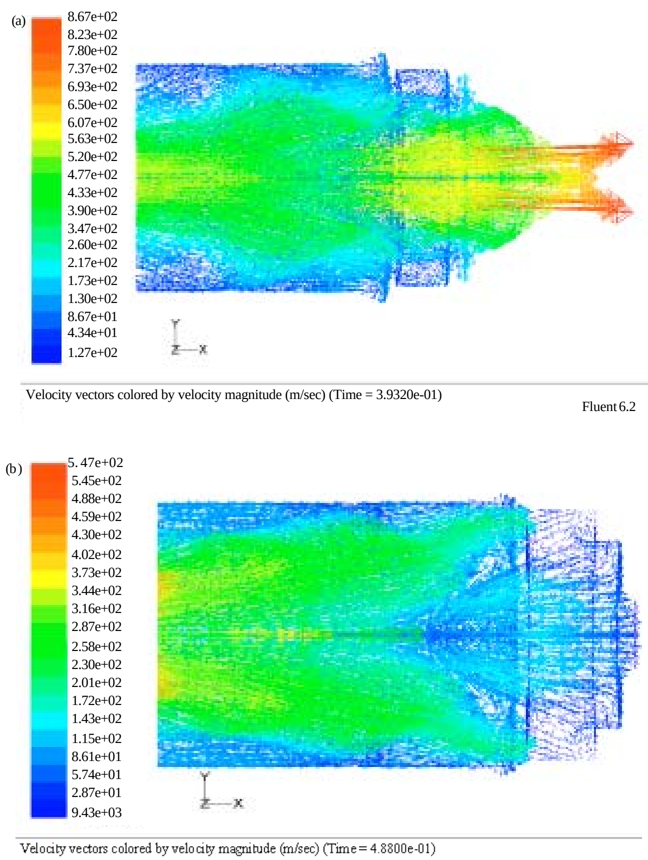

Fig. 5(a-b): Velocity contour map of muffler, (a) T = 0.3932 no vent hole and (b) T = 0.488 with vent hole

CONCLUSION

Based on muffler airflow speed in XOY distribution and pressure distribution on the surface of the detailed description, from figure can intuitive see all kinds of air flow in all kinds of sports phenomenon, according to the calculation results may draw the conclusion as follows:

- Gas from the pipe into the first cavity occurred at a time when inflation and form a vortex
- When big tangent direction of the whirlpool and middle air inlet exhaust passage in the opposite direction, will weaken the inlet pressure and inlet velocity, is not conducive to airflow, when the high-speed airflow impacts wall directly, backflow effects upstream fluid movement, the pressure loss is produced
- For muffler with vent hole and without vent hole, muffler decreases without vent hole due to the effect of blocking of the gas and so to export, reach a maximum speed of the time is short

REFERENCES

Bin, J., M. Kim and S. Lee, 2008. A numerical study on the generation of impulsive noise by complex flows discharging from a muzzle. Int. J. Numer. Meth. Eng., 75: 964-991.

Bogey, C. and C. Bailly, 2004. A family of low dispersive and low dissipative explicit schemes for flow and noise computations. J. Comput. Phys., 194: 194-214.

Chaitanya, P. and M.L. Munjal, 2011. Effect of wall thickness on the end corrections of the extended inlet and outlet of a double-tuned expansion chamber. Applied Acoustics, 72: 65-70.

Huang, S., H. Wang and Y. Ma, 1998. Numerical study of muzzle flow with inlet/outlet expansive suppressor. Explosion Shock Waves, 18: 155-161.

Jiang, Z., K. Takayama and B.W. Skews, 1998. Numerical study on blast flowfields induced by supersonic projectiles discharged from shock tubes. Phys. Fluids, 10: 277-288.

Lee, C.L., D.J. Lee, S.H. Ko, D.S. Lee and G.J. Kang, 2006. Numerical analysis of a blast wave using CFD-CAA hybrid method. Proceedings of the 12th AIAA/CEAS Aeroacoustics Conference, American Institute of Aeronautics and Astronautics, May 8-10, 2006, Cambridge, Massachusetts, pp: 3873-3881.

Sakamoto, K., K. Matsunnagak and J. Fukushima, 2001. Numerical analysis of the propagating blast wave in a firing range. Proceedings of the 19th International Symposium in Ballistic, May 7-11, 2001, Interlaken, Switzerland, pp: 289-296.

Siano, D., 2011. Three-dimensional/one-dimensional numerical correlation study of a three-pass perforated tube. Simulat. Modell. Pract. Theory, 19: 1143-1153.

Wang, B., Y.J. Wang and C. Xu, 2010. Experimental study on muzzle noise suppression of minor-caliber bullet. J. Nanjing Univ. Sci. Technol., 35: 160-163.

Zhang, H., Tan J. and D. Cui, 2008. Numerical simulation of 3D muzzle brake flow field. Mechanics Eng., 30: 34-37.

Zibarov, A.V., D.B. Babayev, A.A. Mironov, I.Y. Komarov, S.V. Malofeev and V.N. Okhitin, 2002. Numerical simulation of 3D muzzle brake and missile launcher flowfield in presence of movable objects. Proceedings of the 20th International Ballistics Symposium, September 23-27, 2002, Orlando, USA., pp: 225-232.