Combustion instabilities of swirl combustors, with radial and axial air injection schemes; computational studies using LES and IDDES

Marcel Ilie[1], Jackson Asiatico[2] and Matthew Chan[3]
Georgia Southern University, Statesboro, GA, 30458

Combustion instabilities of swirl combustion, with axial and radial air injection schemes, are subject of investigation in the present research. The computational studies are performed using the large-eddy simulation approach. The computational studies are performed for a Reynolds number $R_e = 5.7 \times 10^5$. The analysis reveals that the radial air injection scheme enhance the turbulent mixing and thus, the efficiency of the combustion process. Thus, higher temperature values were observed for the case of radial air injection versus axial air injection scheme. The study also revealed that the radial air injection scheme reduces the CO_2

I. Introduction

In gas turbine engines, the swirl combustors are commonly used. In swirl combustors, the flame stabilization is achieved with the central recirculation region generated by the flow interaction between swirler and fuel injection. The central recirculation region of the combustor has significant influence on the thermal efficiency because the flow characteristics of the central recirculation region would affect the flame propagation and air/fuel mixing rate. Enhancing the mixing rate is the major objective to provide a robust combustion and various factors, such as swirl intensity, swirler geometry, injection scheme, etc., could have dominant effects on the combustion efficiency. Different injection schemes, like radial injection and axial injection, are extensively studied in the past years. However, the combustion instability is now well understood. Moreover, the assessment of the temperature field, experimentally, inside the combustor is a challenging task.
Therefore, numerical simulations are an alternative way to provide insight into the flow and combustion physics and thermodynamics. In the past decades, many experimental and computational studies, of backward facing step, have been performed. However, the combustion studies of backward facing step are still limited. Majority of the fluid dynamics studies employed the Reynolds averaged Navier-Strokes (RANS) simulation with various turbulence models. However, for highly turbulent and transient flows, the use of RANS poses significant challenges. RANS simulations are generally dissipative to capture weak pressure fluctuations and high-frequency tones. Accurate simulation of the radiated acoustic waves together with the unsteady turbulent shear-layer is needed to capture small amplitude of the acoustic field and their propagations over a long distance with little attenuation of numerical dissipation. Recently, large-eddy simulation (LES), detached-eddy simulation (DES), and RANS/LES hybrid methods were performed for supersonic cavities simulation with low-dispersive and low-dissipative numerical schemes. [12] Other studies employed the LES with a fourth-order compact difference scheme to investigate the efficiency and mechanism of the active noise suppression with pulsed mass injections.

---

[1] Assistant Professor, Department of Mechanical Engineering
[2] Undergraduate Student, Department of Mechanical Engineering
[3] Undergraduate Student, Department of Mechanical Engineering

Relatively recent work employed the DES for the computation of unsteady three-dimensional turbulent supersonic open cavity flow and results revealed the flow features of supersonic cavity flow, including the vortex shedding, shock waves, and coupling of the acoustic and vorticity fields.
Previous studies used the hybrid RANS/LES approach to study the control mechanism and dynamic loading of cavity tones using a rod spoiler as a device of passive actuation. Other studies employed RANS/LES with a fifth-order weighed essentially non-oscillation (WENO) scheme to investigate the two-dimensional supersonic cavity flows and the effects of inflow conditions and geometry structures on self-sustained oscillation characteristics.
In spite of these extensive studies, the generation mechanism of cavity tones in supersonic cavity flow is still not well understood. A detailed discussion of each component of compression wave is important to clarify the mechanism of cavity tones and to suppress cavity tones.
The aim of the present study is to provide a better understanding of the combustion instabilities, in backward facing step cavities.

II. Governing equations and numerical method

The main idea of LES is to separate the flow variable in two components, namely the mean $\overline{f}(x)$ or large scales and fluctuating component $f^{'}(x)$ or small scales. In LES, the large scales of the flow are completely resolved while the small scales are completely modeled using a sub-grid scale model. The governing equations of LES are the so-called filtered Navier-Stokes equations, which are a result of spatial averaging. The filtered Navier-Stokes equations are:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \left(\overline{u_i}\,\overline{u_j}\right)}{\partial x_j} = -\frac{1}{\overline{\rho}} \frac{\partial \overline{p}}{\partial x_i} - \nu \frac{\partial^2 \overline{u_i}}{\partial x_i \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + S_i \tag{2}$$

where t is the time, p is the pressure, ρ the density, ν kinematic viscosity, S_i a source term and τ_{ij} the subgrid scale (SGS) tensor expressed as:

$$\tau_{ij} = \overline{u_i}\,\overline{u_j} - \overline{u_i u_j} \tag{3}$$

An eddy viscosity model is used to model the SGS tensor which is then expressed as:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = 2\nu \overline{S_{ij}} \tag{4}$$

where $\overline{S_{ij}}$ is the strain rate based on the filtered velocity $\overline{u_i}$ and ν the eddy viscosity.

In the present work we employ the dynamic Smagorinsky sub-grid scale (SGS) model. The model is presented briefly in the following. In LES, the SGS model represents the effect of small scale (smaller than the grid size Δ) flow structures on the large ones (which are resolved). The large scale flow structures are obtained through a filtering process of the velocity and scalar fields at the grid scale such that

$$\widetilde{u}(x) = \int u(x^{'}) F(x - x^{'}) dx^{'} \tag{5}$$

where $\widetilde{u}$ is the filtered velocity and F is the filter function at scale Δ.

In 1991 Germano et al., [9], proposed the so-called dynamic Smagorinsky model. In this model the selected features of the resolved scales of the flow field are dynamically analyzed during the simulation, to determine the unknown model coefficient instead of using some predefined values. One fundamental characteristic of the dynamic Smagorinsky SGS model is that the resolved scales can represent much better the flow dynamics phenomena such as stratification, coherent structures and complex flow interactions compared with other turbulence models. The dynamic Smagorinsky SGS model is based on the Germano identity given by:

$$L_{ij} = \overline{\widetilde{u}_i \widetilde{u}_j} - \overline{\widetilde{u}_i}\,\overline{\widetilde{u}_j} = T_{ij} - \overline{\tau}_{ij} \tag{6}$$

where L_{ij} is the resolved stress tensor and T_{ij} is the subgrid stress tensor, at the test filter scale. For more details in the dynamic Smagorinsky SGS the reader is referred to Germano et. al [9].

The improved delayed detached-eddy simulation (IDDES), as the name suggests, it is an improvement of the delayed detached eddy simulation (DDES). The improvements of the IDDES arise from the fact that addresses favorably some of the drawbacks of the original DDES model. This is achieved by introducing a log-layer mismatch model as well a model that minimizes the effect of the separation induced by the grid. For more details on the development of DES models, the reader may refer to [5]. The model employed the present work is based on the SST-IDDES model proposed by Shur et. al [23]. This model offers the advantage that it is straightforward to implement, and it is explained in the following. The equation of the turbulence kinetic energy (TKE), for RANS, is given by:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho \mu_j k)}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\partial_k}\right)\frac{\partial k}{\partial x_j}\right]+\tau_{ij}S_{ij}-\frac{\rho k^{1.5}}{L_{RANS}} \tag{7}$$

where k is the TKE that is modeled. The main idea of IDDES is that the length scale of RANS (L_{RANS}) is replaced by the length scale of IDDES (L_{IDDES}). Thus,

$$L_{IDDES}=\tilde{f}_d(1+f_e)L_{RANS}+(1-\tilde{f}_d)L_{LES} \tag{8}$$

where

$$L_{RANS}=\sqrt{k}/(\beta^*\omega) \tag{9}$$

In equation (9), β^* is a constant in SST model $\beta^*=0.09$ and ω is the specific turbulence dissipation rate. The length scale for LES (L_{LES}) is given by equation (10)

$$L_{LES}=C_{DES}\times\Delta \tag{10}$$

In SST-IDDES C_{DES} is the blending of the $k-\varepsilon$ and $k-\omega$. C_{DES} is a constant calibrated by the decay of homogeneous isotropic turbulence, and thus $C_{DES,k-\varepsilon}=0.61$ $C_{DES,k-\omega}=0.78$. In equation (10) Δ represents the grid scale, and it is defined as

$$\Delta=\min\left[\max(C_w\Delta_{\max},C_w d,\Delta_{\min})\Delta_{\max}\right] \tag{11}$$

where $C_w=0.15$ and d is the distance from the cell center to the nearest wall. The minimum and maximum grid scales are defined as

$$\Delta_{\min}=\min(\Delta x,\Delta y,\Delta z) \tag{12}$$

$$\Delta_{\max}=\max(\Delta x,\Delta y,\Delta z) \tag{13}$$

The function $\tilde{f}_d$ in equation (8) is defined as

$$\tilde{f}_d=\max[(1-f_{dt}),f_B] \tag{14}$$

where f_B is a function of geometry, while $(1-f_{dt})$ is a function of flow physics. From equation (8) it can be seen that when $f_e=0$

$$L_{IDDES}=\tilde{f}_d L_{RANS}+(1-\tilde{f}_d)L_{LES} \tag{15}$$

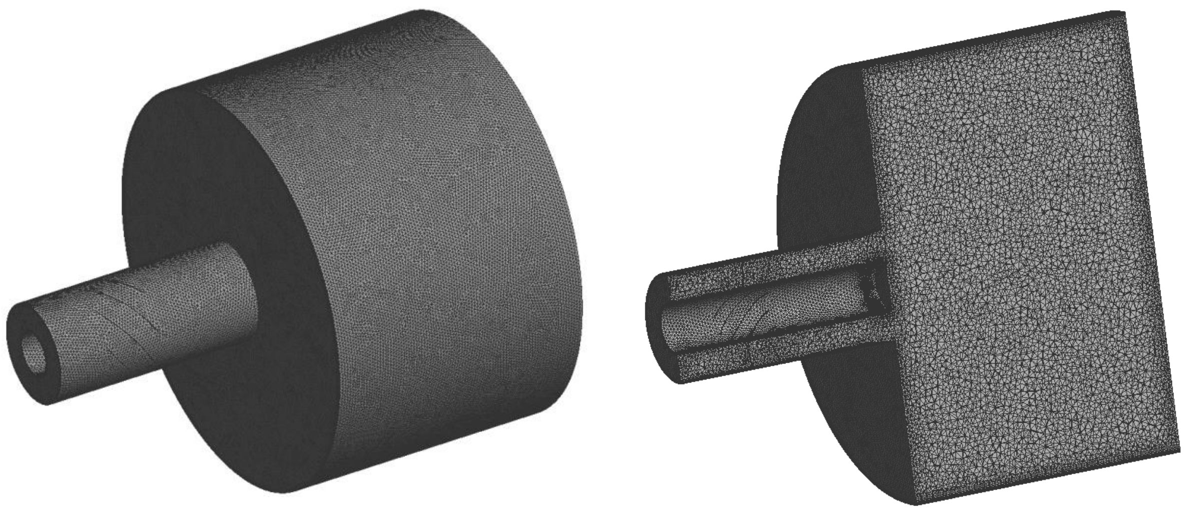
while, when $f_e>0$ and $\tilde{f}_d=f_B$

$$L_{IDDES}=f_B(1+f_e)L_{RANS}+(1-f_B)L_{LES} \tag{16}$$

For more details on the reader is referred to [23].

III. Computational model

Computational studies of combustion phenomena, in swirl combustor, are performed to study the effect air mass injection scheme on the combustion instabilities. The computational domain consists of 4.8 million grid points. For all of the computations in this paper, a dimensionless time step $\overline{\Delta t} = \Delta t U_\infty / c$ = 5x10^{-5} is chosen, where U_∞ is the free- stream velocity. The time-step is determined with respect to the explicit time-marching scheme and temporal resolution requirement of LES (CFL $\leq$ 1). In the present investigations a value of Courant–Friedrichs–Lewy (CFL) number of 0.9 was chosen.



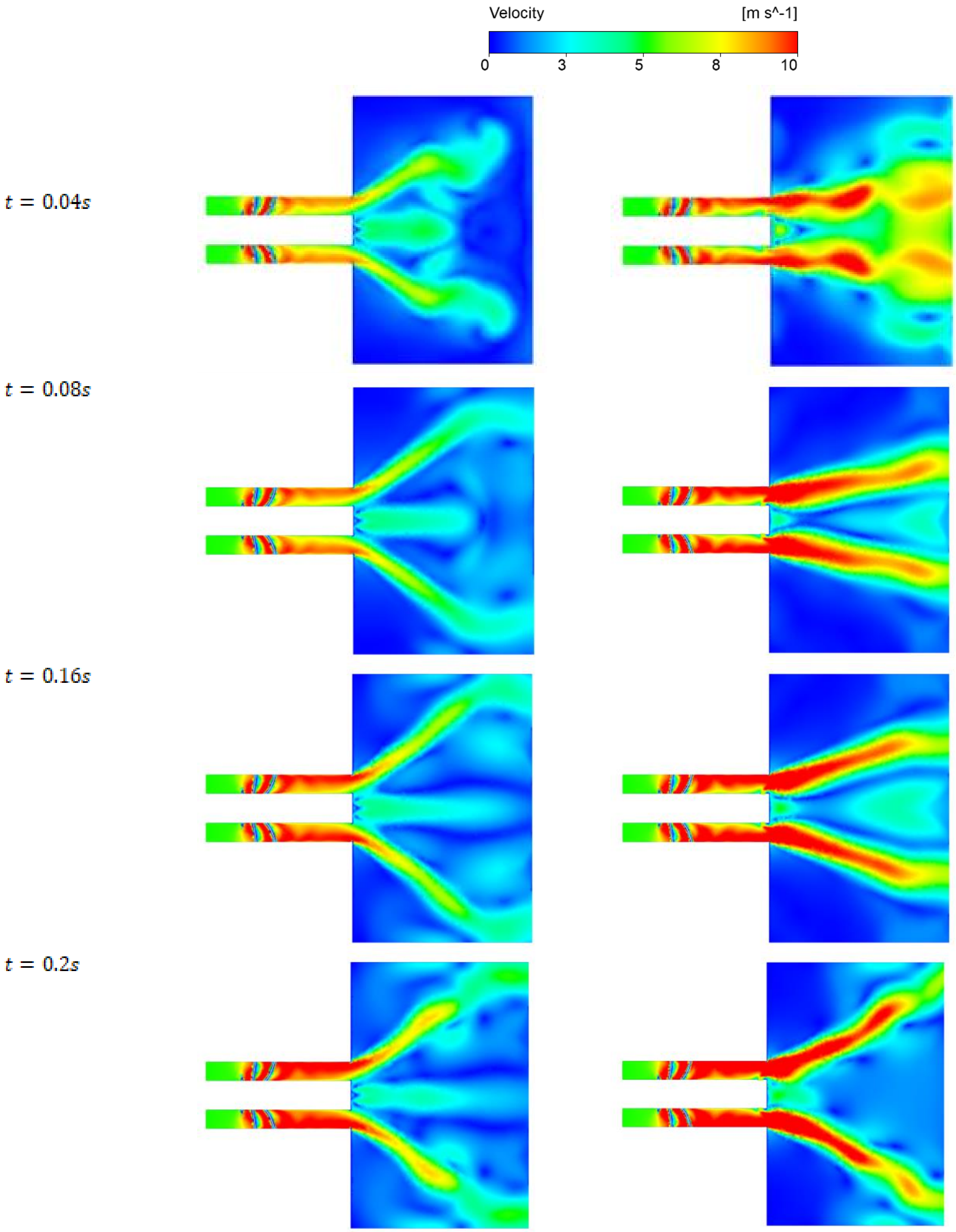
a. Iso-view   b. Cross-section view

Figure 1. Computational domain of swirl combustor

IV. Results and Discussion

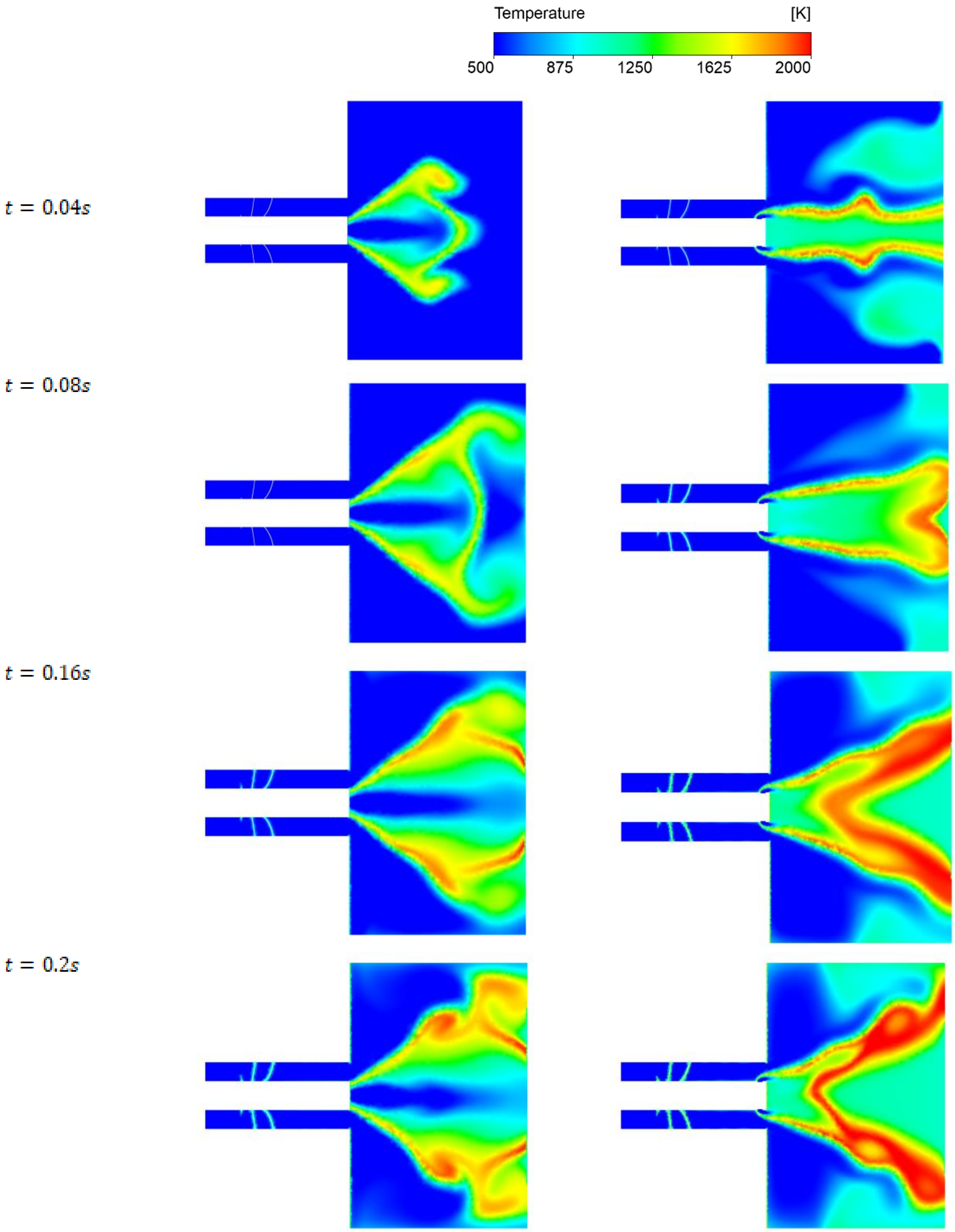
Figure 2 presents the comparison of the time-dependent velocity field from LES computations, for the two types of swirl combustors. Thus, Figure 2a presents the velocity field for swirl combustor with axial flow injection. Figure 2 presents the velocity field for swirl combustor with radial flow injection. The analysis of the velocity field for the two swirl combustors reveals that the radial air mass injection improves the turbulent mixing which is desired for increased efficiency of the combustion process. An important observation is the axial air mass flow injection causes a more confined velocity distribution. This is well-illustrated by the comparison of the velocity fields, at instant t=0.08s. The analysis that, in the case of radial air mass flow injection, there is a larger recirculation region inside the combustor.
It is expected that the turbulent mixing, inside the combustor chamber, would enhance the combustion efficiency which may be assessed by the temperature values. Thus, Figure 3 presents the time-varying temperature variation inside the combustor. The analysis of the temperature shows that the temperature is higher in the case of radial air mass injection, reaching a maximum value of 2,000[K].
. It is worth to notice that the radial air mass flow injection reduces the combustion instabilities, and this is illustrated by the less fluctuations of the temperature field. The analysis of the temperature field reveals that there are high fluctuations of the temperature, due to the interaction between and turbulence and combustion.

The analysis of the turbulence kinetic energy (TKE) shows that there is a significant increase of the turbulence kinetic energy for the case when the air injected radially. The turbulence kinetic energy is a god indicator of the turbulent mixing, as well the interaction between the turbulence and combustion. The high turbulent mixing, in the combustor with radial air injection, ensures a better combustion efficiency and therefore, less CO_2. The comparison of the CO_2, for the two air injection schemes, shows that the swirl combustor with radial air mass injection performs better in term of CO_2 products.



a. Axial b. Radial

Figure 2. Time-dependent velocity



a. Axial b. Radial

Figure 3. Time-dependent temperature

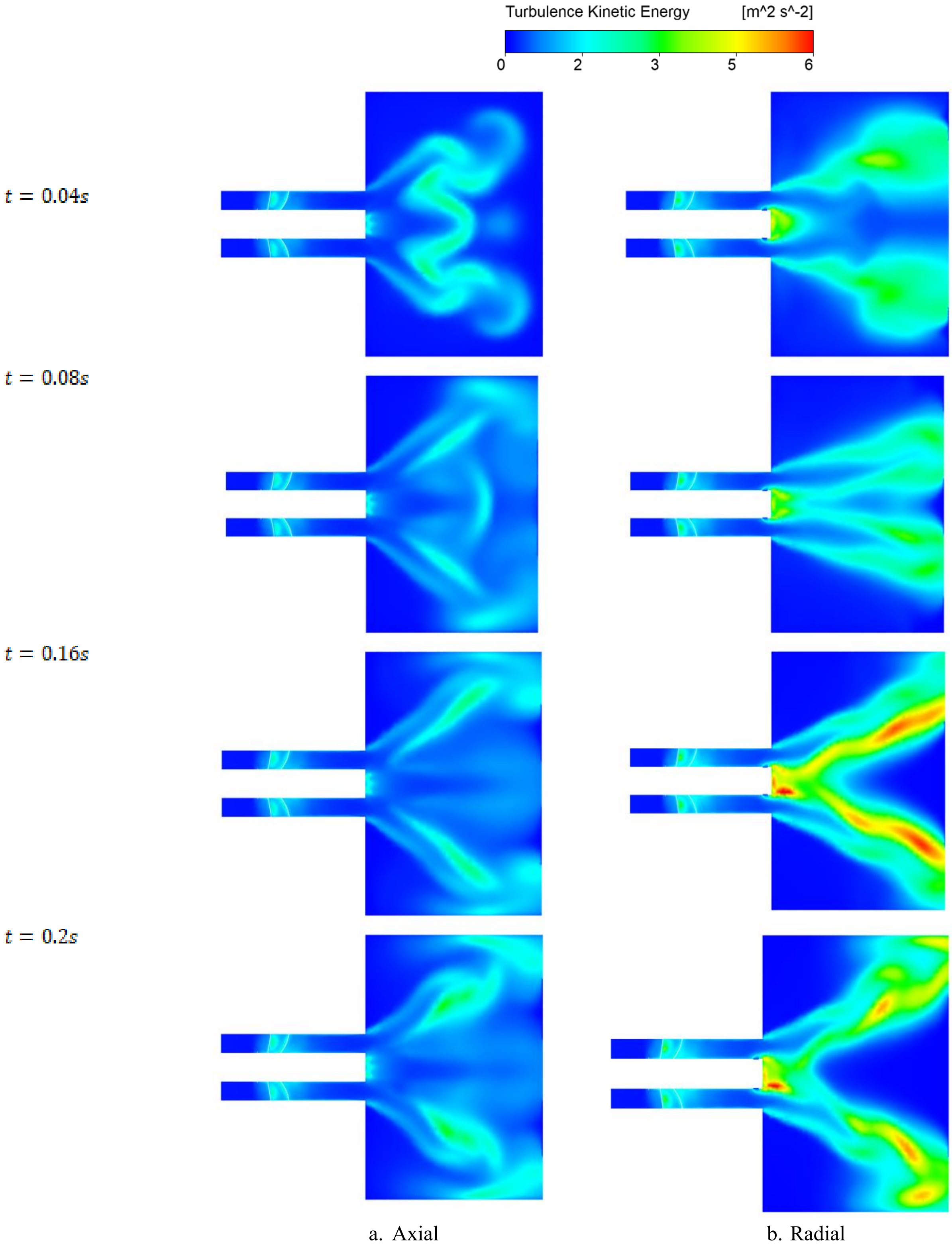
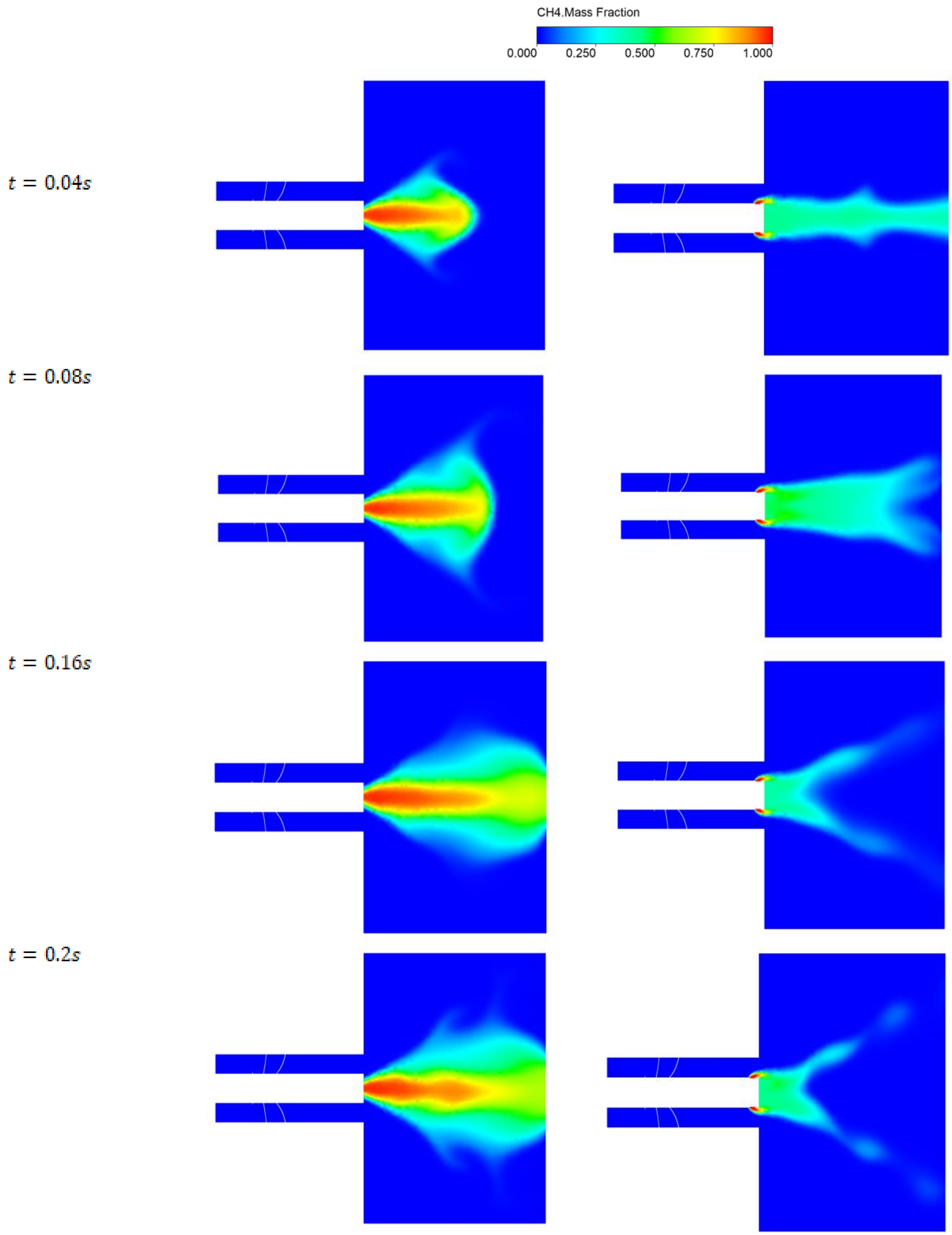


Figure 4. Time-variation of TKE



a. Axial

b. Radial

Figure 5. Time-variation of CH_4

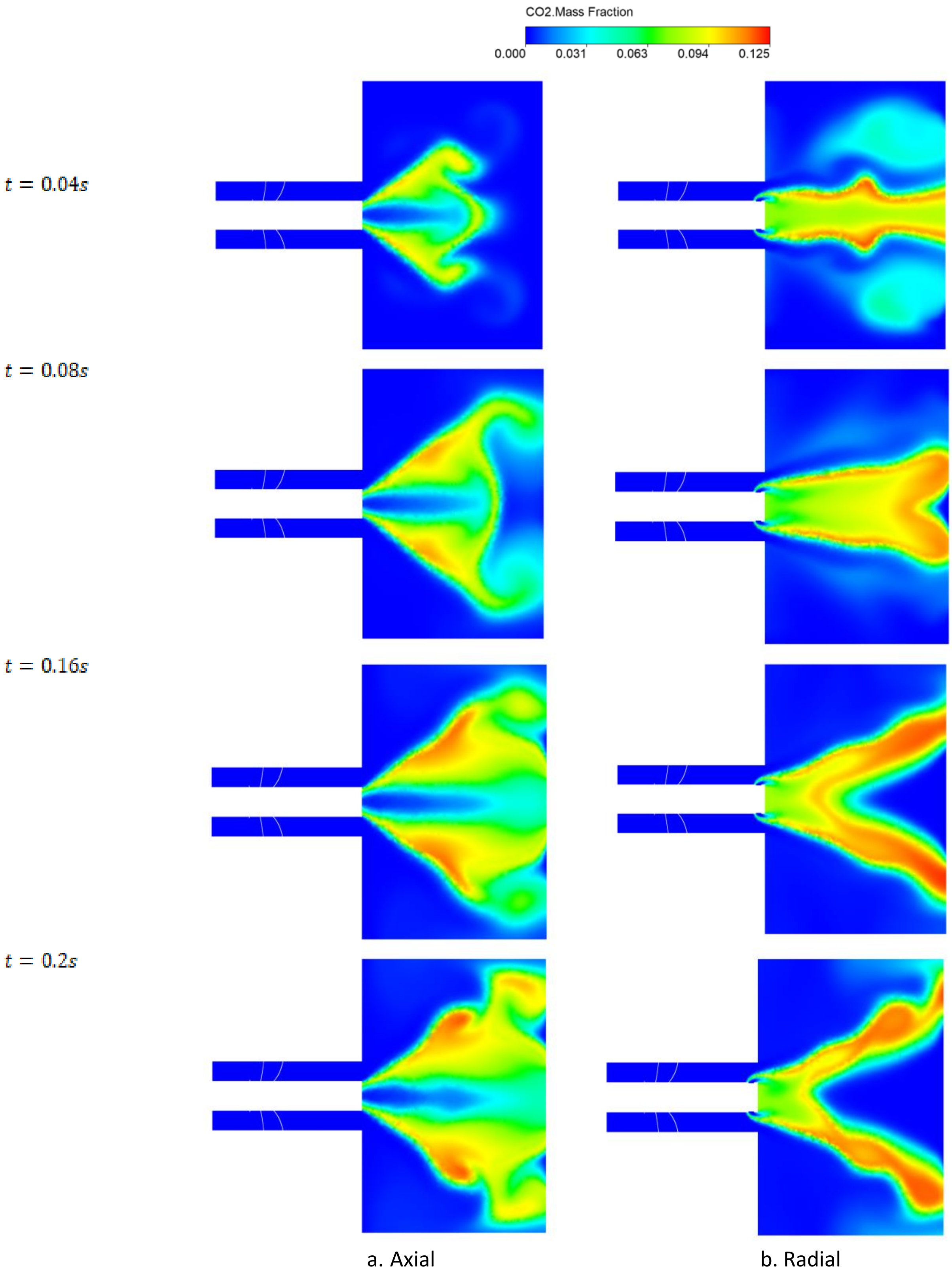


Figure 6. Time-variation of CO_2

V. Conclusions

Combustion efficiency and instabilities are studied for swirl combustor, with two different schemes of air injection, namely axial and radial schemes, respectively. The analysis shows that radial air injection schemes performs better than the axial air mass injection scheme, for the case of swirl combustor. The radial air injection scheme is associated with higher turbulent mixing and thus, higher combustion efficiency. Therefore, higher overall temperature and less CO_2 values are observed for the case of radial air injection scheme.

VI. References

[1]Sheryl M. Grace. An overview of computational aeroacoustic techniques applied to cavity noise prediction. AIAA 2001-0510. January, 2001

[2]Tim Colonius. An overview of simulation, modeling, and active control of flow/acoustic resonance in open cavities. AIAA Paper 2001-0076, 2001

[3]Robert C. Murrayand Gregory S. Elliott. Characteristics of the Compressible Shear-layer over a Cavity. AIAA Journal.Vol. 39, No. 5, May 2001

[4]Adela Ben-Yakar and Ronald K. Hanson. Cavity Flame-Holders for Ignition and Flame Stabilization in Scramjets: An Overview. Journal of Propulsion and Power. 17(4) 869-877. 2001

[5]Asbury, S. C., Gunther, C. L., and Hunter, C. A., Passive Cavity Concept for Improving the Off-Design Performance of Fixed-Geometry Exhaust Nozzles, AIAA Paper 96-2541, July 1996.

[6]Plentovich, E., "Three-dimensional Cavity Flow Fields at Subsonic and Transonic Speeds," Tech. Rep. TM 4209. NASA Langley Research Center, 1992.

[7]N. Zhuang, F. S. Alvi, M. B. Alkislar, and C. Shih. Supersonic Cavity Flows and Their Control. AIAA Journal. 44(9),2006: (2118-2128)

[8]N. Zhuang, F. S. Alvi and C. Shih. Another Look at Supersonic Cavity Flows and Their Control. AIAA Paper 2005-2803, 2001

[9]A. Powell. On the edge tone. Journal of the Acoustical Society of America. 33:395.1961.

[10]Rossiter, J. E.,“Wind-Tunnel Experiments on the Flowover Rectangular Cavities at Subsonic and Transonic Speeds,” Aeronautical Research Council, Repts. and Memoranda No. 3438, London, Oct. 1964.

[11]Heller, H., Holmes, G., and Covert, E., “Flow-induced Pressure Oscillations in Shallow Cavities”, AFFDL TR-70-104

[12]Ö. H. Ünalmis, N. T. Clemens, and D. S. Dolling. Experimental Study of Shear-Layer/Acoustics Coupling in Mach 5 Cavity Flow. AIAA Journal, 39(2), 2001: 242-252

[13]N. S. Vikramaditya and Job Kurian. Pressure Oscillations from Cavities with Ramp. AIAA Journal, 47(12), 2009: 2974-2984

[14]Hanno H. Heller and Donald B. Bliss. The Physical Mechanism of Flow-induced Pressure in Cavities and Concepts for Their Suppression. AIAA Paper 1975-491, 1975 American Institute of Aeronautics and Astronautics

[15]Rockwell, D., and Naudascher, E., “Review: Self-Sustaining Oscillations of Flow Past Cavities,” Journal of Fluids Engineering, 100(2), 1978, pp. 152–165.

[16]S. J. Lawson and G. N. Barakos. Assessment of Passive Flow Control for Transonic Cavity Flow Using Detached-Eddy Simulation. Journal of Aircraft, 46(3), 2009:1009-1029

[17]Chung-Jen Tam, Paul D. Orkwis and Peter J. Disimile. Supersonic open cavity flow physics ascertained from algebraic turbulence model simulations. AIAA Paper 96-0075. January 1996

[18]Shih, S. H., Hamed, A., and Yeuan, J. J., "Unsteady Supersonic Cavity Flow Simulations Using Coupled k-epsilon and Navier-Stokes Equations," AIAA Journal, Vol. 32, No. 10, October 1994, pp. 2015-2021.

[19]Zhang, X., "Compressible Cavity Flow Oscillation due to Shear-layer Instabilities and Pressure Feedback," AIAA Journal, Vol. 33, No. 8, August 1995, pp. 1404-1411.

[20]Selin Aradag, Doyle D. Knight. Simulation of Supersonic Flow over a Cavity. AIAA Paper 2005-848. January 2005

[21]Donald P. Rizzetta and Miguel R. Visbal Large-Eddy Simulation of Supersonic Cavity Flowfields Including Flow Control. AIAA Journal,41( )8, 2003:1452-1462

[22]A. Hamed, D. Basu, K. Das. Detached Eddy Simulations of Supersonic Flow over Cavity. AIAA 2003-549. January 2003

[23]Arunajatesan, S., Sinha, N., and Menon, S., "Towards Hybrid LES-RANS Computations of Cavity Flowfields," AIAA Paper, 2000-0401, 2000.

[24]Liang Jianhan, Sun Mingbo. LES of Cavity Oscillation Characteristics in High-Speed Flows. AIAA 2009-7231, 2009

[25]Taku Nonomura and Kozo Fujii. Effects of difference scheme type in high-order weighted compact nonlinear schemes. Journal of Computational Physics. 228 (2009) 3533-3539

[26]Taku Nonomura, Nobuyuki Iizuka, Kozo Fujii. Freestream and vortex preservation properties of high-order WENO and WCNS on curvilinear grids. Computers and Fluids 39 (2010) 197-214

[27]Taku Nonomura and Kozo Fujii. Computational Analysis of Characteristics of Mach Wave Sources in Supersonic Free-jets. AIAA 2009-16, 2009
[28]Shima, E. and Jounouchi, T., Role of CFD in Aeronautical Engineering (No. 14) -AUSM type Upwind Schemes, Proceedings of the 14th NAL Symposium on Aircraft Computational Aerodynamics, National Aeroautical Laboratory, Tokyo, 1997, pp. 7-12.
[29]Hiroyuki Nishida, Taku Nonomura. ADI-SGS scheme on ideal magnetohydrodynamics. Journal of Computational Physics. 228 (2009) 3182–3188
[30]D. A. Lyn and W. Rodi. The flapping shear layer formed by flow separation from the forward corner of a square cylinder. Journal of Fluid Mechanism, 267: 353-376, 1994
[31]Lionel Larchevêque, Pierre Sagaut, Ivan Mary and Odile Labbé. Large-eddy simulation of a compressible flow past a deep cavity. Physis of Fluids,15(1), 2003
[32]G. berkooz, P. Holmes, and J. L. Lumley. The Proper Orthogonal Decomposition in the Analysis of Turbulent Flows. Annual review of Fluid Mechanism, 25:539-575. 1993.
[33]Yang Qin and Fu Song. Analysis of flow structures in supersonic plane mixing layer using the POD method. Science in China series G: Physis, mechanics and Astronomy, 51(5): 541-558
[34]J. B. Freund and T. Colonius. Turbulence and sound-field POD analysis of a turbulent jet. Aeroacoustics, 8(4):337‑354, 2009.