Yinli Liu, Hao Tang, Zhaofeng Tian, Haifei Zheng

**CFD simulations of turbulent flows in a twin swirl combustor by RANS and hybrid RANS/LES methods**

Energy Procedia, 2015 / Jiang, X., Joyce, M., Xia, D.H. (ed./s), vol.66, pp.329-332



Originally Published at:
http://dx.doi.org/10.1016/j.egypro.2015.02.078

Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 66 (2015) 329 – 332
Energy Procedia

The 12th International Conference on Combustion & Energy Utilisation – 12ICCEU

# CFD Simulations of turbulent flows in a Twin Swirl Combustor by RANS and Hybrid RANS/LES Methods

Yinli Liu*,[1], Hao Tang[1], Zhaofeng Tian[2] , Haifei Zheng[1]

[1]College of Energy and Power Engineering,
Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China
[2]School of Mechanical Engineering,
The University of Adelaide, Adelaide, SA 5005, Australia

**Abstract**

A confined isothermal flow in a Twin Swirl Combustor (TSC) was studied under both steady and transient conditions. The Reynolds Averaged Navier-Stokes (RANS) simulation was carried out to investigate the time-averaged flow features in TSC under different ratios of the primary air flow rate to the secondary air flow rate. The steady-state velocity profiles inside the combustion chamber were analyzed using both Renormalized Group (RNG) *k-ε* model and Shear Stress Transport (SST) turbulence model. For the transient conditions, the Scale Adaptive Simulation (SAS) method based on the SST model was used to probe the instantaneous three dimensional (3D) vortex structures to better understand the formation of the internal recirculation zone (IRZ). The existence of a pressing vortex cores (PVC) and a hurricane-shaped vortical structure in the TSC were captured by the SAS model. The SAS model can yield vortex-level results similar to large eddy simulation (LES) prediction while saves much more computational resource. This paper briefly reports some preliminary results of this study.


Peer-review under responsibility of the Engineering Department, Lancaster University

Keywords: Twin swirl combustor; Computational Fluid Dynamics (CFD), Reynolds Averaged Navier-Stokes (RANS) models; Scale Adaptive Simulation (SAS); Vortex Structure

## 1. Introduction

Confined swirling flows are widely used in industrial combustion instruments. In these instruments, the recirculation zone generated by the negative pressure gradient and vortex breakdown of the swirling flows can intensify the mixing of air and fuel, while anchor the flame within the combustion zone. The utilization of Computational Fluid Dynamics (CFD) is in the cutting edge in optimized designing of the industrial applications. Two-equation Reynolds Averaged Navier-Stokes (RANS) models, e.g. standard *k-ε* model, Renormalized Group (RNG) *k-ε* model, and Shear Stress Transport (SST) model are widely used

* Corresponding author. Tel.: +86 158 5005 8778
*E-mail address: yinli.liu@nuaa.edu.cn*


Peer-review under responsibility of the Engineering Department, Lancaster University
doi:10.1016/j.egypro.2015.02.078

for the industrial calculations when the mean flow characteristics are of interest [1]. However, two-equation RANS models generally fail to predict the strong anisotropic turbulence like flow separation and instantaneous characteristics like vortex breakdown. Large Eddy Simulation (LES) is a turbulence resolving method that provides more accurate predictions than two-equation RANS models in many applications, though requires much more computational resource. A hybrid RANS/LES method has been established [2] and used in the present investigation to study the instantaneous vortex characteristics of the internal swirling flows inside the chamber of a Twin Swirl Combustor (TSC) that combusts pulverized coal [3]. The Scale Adaptive Simulation (SAS) method based on the SST model was used to probe the instantaneous three dimensional (3D) vortex structures to better understand the formation of the internal recirculation zone (IRZ). Furthermore, the steady-state velocity profiles inside the combustion chamber were analyzed using both Renormalized Group (RNG) $k$-$\varepsilon$ model and Shear Stress Transport (SST) turbulence model. This paper briefly reports some preliminary results of this study.

## 2. Computational models and numerical method

The commercial CFD code ANYSY FLUENT 14 was used for all the simulations. The geometry of the TSC has been reported in a previous paper [3]. Boundary conditions except the swirl intensity can be found in the previous paper and not reported herein due to the limited space. Mesh independence test had been implemented before all the simulations started. A total number of about 100,000 quadrilateral mesh cells were chosen for the 2D simulations, and the mesh with 2 million hexahedral cells was used for the 3D cases.

The RNG $k$-$\varepsilon$ model is based on the Navier-Stokes equations using a mathematical technique called the "renormalization group" (RNG) methods. It contains modifications for the swirling flows by modifying the turbulent viscosity appropriately according to different swirling intensity, though the performance of RNG $k$-$\varepsilon$ model in predicting swirling flows is controversial. The SST model combines the advantages of both $k$-$\omega$ and $k$-$\varepsilon$ models and the drawbacks for $k$-$\omega$ model solving the free flow areas and $k$-$\varepsilon$ model solving the near wall regions have been eliminated. Two blending factors determine the transition between these two models, based on the local solutions and the distance from the wall. The SAS model is one form of the Hybrid RANS/LES model and is evolved from SST model. One of the drawbacks for instantaneous SST model is that it produces too large length-scales and therefore the turbulent viscosities are too high [2]. In SAS model, this is eliminated by introducing the von Karman length-scale into the transport equations of the original SST model. More detailed information on the SAS model can be found in [2].

## 3. Results and discussion

The flow patterns of the confined co-axial swirling airflows in a twin-swirl combustor with a construction near the exit have been investigated in this study [3]. The axial and tangential mean velocity are discussed and compared for several different cases, which differs only in the swirl intensity adjusted by the secondary airflow. The first simulation case, named C1, has an inlet swirl intensity of 0.476. In Case C2, the inlet swirl intensity is 0.518. The inlet swirl intensity of the third case, C3, is 0.567. Figure 1 and Figure 2 illustrate, respectively, the mean axial and tangential velocity profiles along the radial direction at different axial positions, normalized by the bulk velocity of each case. To give a global structure of mean flow field, the profiles are plotted in the $x$ (axial direction)-$r$ (radial direction) plane, in which the $x$ axis also gives the value, when compared with the scaling in the upper-right corner of each figure. The dashed lines indicate the axial position of calculated velocity group, and they also stand for the zero value level of corresponding quantities. The red curves are the results obtained from RNG $k$-$\varepsilon$ model and the blue ones are from SST model.

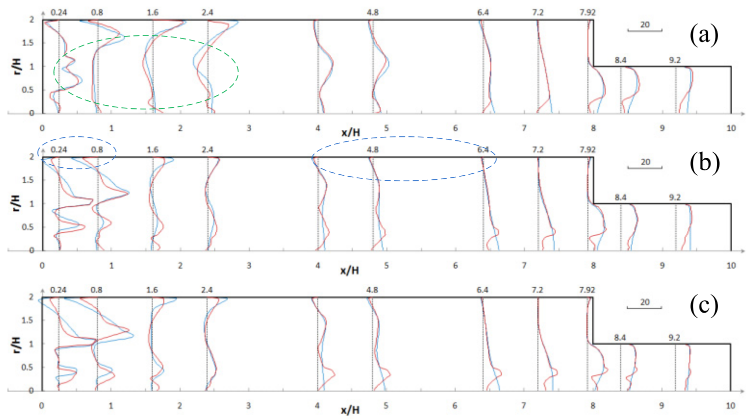

**Figure 1**. The axial velocity components yielded from RNG and SST models under different swirling intensities from C1 (top) to C3 (bottom). Blue curves: RNG results; red curves: SST results. Dashed lines denote both the calculating positions and zero level of shown quantities. The scaling of each shown quantities is given by the number and short line in the upper right of each figure. $H$=0.125 m.

As is shown in Figure 1, the trends of the calculated mean axial velocity profiles from both turbulence models consist well with each other in the front region ($x/H$<2.4), while deviations are found in the center region ($r/H$<0.5) after $x/H$ >4 for all cases. The inner recirculation zone (IRZ) is generated under intensive swirling conditions in the vortex breakdown zone in 0.24<$x/H$<2.4 (the green dished ellipse in Figure 1a). Besides, there are near-wall recirculation zones in the front part where 0.24<$x/H$<0.8 and in the middle of 4<$x/H$<6.4 (the blue dashed ellipses in Figure 1b). As the swirl intensity increases from C1 (0.476) to C3 (0.567), the location of IRZ moves towards upstream.

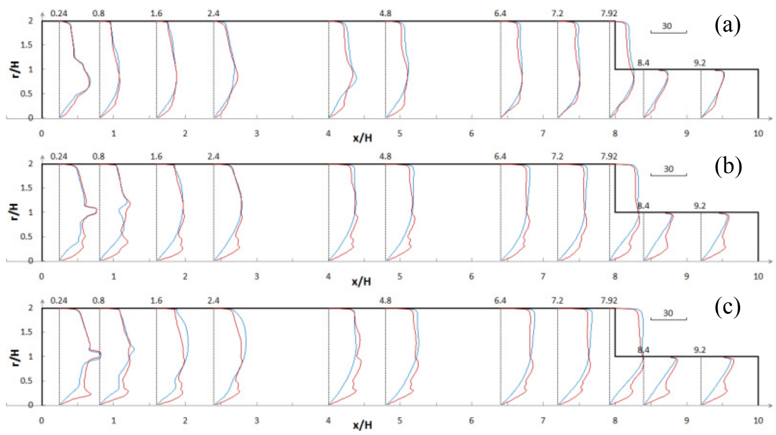

**Figure 2**. The tangential velocity compnents yielded from RNG $k$-$\varepsilon$ model and SST model of cases (a) C1, (b) C2, (c) C3. Blue curves: RNG results; red curves: SST results. It is worth noticing that the scaling for the tangenial velocity is 30, which is different from that of axial velocity with the value of 20.
See Figure 1 for further information.

The tangential velocity profiles predicted by RNG $k$-$\varepsilon$ model and SST model in Figure 2 show great similarities against each other for C1 case (Figure 2a), while the deviations existing in center regions for C2 and C3 cases (Figure 2b and 2c). SST model yields larger tangential velocity in $r/H$<1, and decays faster in the regions where 1<$r/H$<2 for C2 and C3 cases. The Rankine vortex structures can be clearly observed from the curves of tangential velocity profiles. The turbulent kinetic energy is transported along the radial direction due to the high centrifugal force brought in by the strong swirling airflows and dissipated by the viscous layer near the wall.

The Precessing Vortex Core (PVC) is a rotating flow structure, which precesses around the central axis. The PVC locates in the IRZ and is determined by swirl number, equivalence ratio and geometry of the combustor [4]. It is a coherent structure which can lead to combustion instability and oscillations. The instantaneous structure of PVC in the TSC chamber of C2 was visualized by instantaneous 3D SAS simulation. Figure 3 demonstrates the vortex core regions using different $Q$-criteria, colored by the eddy viscosity ratio. When the actual value of $Q$ increases, more detailed cortex structures around the axis have been revealed. The PVC in the TSC was seen from the primary air inlet and propagated to the end near the constriction part . A hurricane shaped vortical structure was observed inside the PVC in the rear, propagating to the exit of the combustor. This structure is generated by the increasing angular velocity, which is caused by the contraction part in the end of the chamber [4]. It can interact with the PVC. Furthermore, the location and shape of both IRZ and PVC can be influenced by this hurricane-shaped vortical structure.

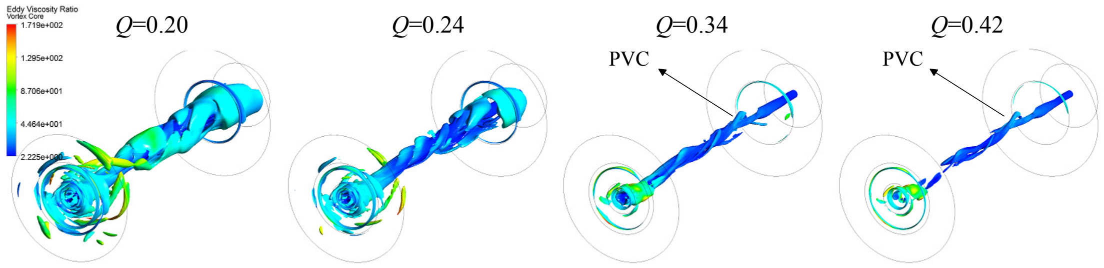


**Figure 3** Precessing vortex core (PVC) visualized by different Q-criteria, colored by the eddy viscosity ratio, at simulation time 7.473s

## 4. Conclusion

The turbulence features in the TSC chamber were numerically studied using two equation RANS and Hybrid RANS/LES methods. The time averaged axial and tangential velocity components were simulated with RNG $k$-$\varepsilon$ and SST models. Both model predict mean axial and tangential velocities that have similar trends, but with some deviations in the near-axis regions and center region. The IRZ moves upstream when the swirl intensity increases. The Rankine vortex structures were also observed under the coaxial swirling flow conditions. A 3D SAS method was applied to depict the instantaneous vortex structure. The PVC was visualized and propagates downstream around a hurricane-shaped vortex structure to the constriction part. The SAS model is able to capture smaller and more detailed vortex structures than the RANS methods with much less computational effort compared with LES method.

A detailed measurement of the flow field in the TSC is essential to further compare and evaluate the performance of different turbulence models and is expected to be conducted soon.

## References

[1] Paik, J., Sotiropoulos, F., **Numerical simulation of strongly swirling turbulent flows through an abrupt expansion.** *International Journal of Heat and Fluid Flow,* 2010. **31**(3): p. 390-400.

[2] Menter, F. R., Egorov, V., **A scale-adaptive simulation model using two-equation models.** In *43rd AIAA Aerospace Sciences Meeting and Exhibit* , Reno, Nevada, 10 - 13 January 2005, AIAA 2005-1095.

[3] Liu, Y. L., Tang, H., **Numerical Simulation on the Flow Characteristics of Twin Swirl Combustor.** In *2011 International Conference on Materials for Renewable Energy & Environment*, 20 - 22 May 2011, Vol.2 p:1559-1564.

[4] Widenhorn, A., Noll, B., Aigner, M., **Numerical study of a non-reacting turbulent flow in a gas turbine model combustor.** In 47th AIAA Aerospace Science Meeting Including the New Horizons Forum and Aerospace Exhibition, Orlando, Florida, 5 - 8 January, 2009, AIAA 2009-647.

## Acknowledgement

This research is supported by the Funding of University-industry Cooperation and Innovation of Jiangsu (BY2012020) and Funding of Jiangsu Innovation Program for Graduate Education (CXLX11_0193).