

Simulation of Turbulent Swirling Flame in 3-D Can Combustion Chamber using 2D and 3D URANS RSM/SST $k-\omega$

A. GUESSAB¹, M. TAHA JANAN², M. MOKHTAR³ and A. ARIS⁴

1. Laboratoire de Recherche en Technologie de Fabrication Mécanique, Oran (Algérie), med_guessab@yahoo.fr
2. Laboratoire de Mécanique, Procédés et Processus Industriels, ENSET de Rabat, (Maroc), tahajanan@yahoo.fr
3. Laboratoire de Recherche en Technologie de Fabrication Mécanique, Oran (Algérie), mokhtarimohamed44@yahoo.fr
4. Laboratoire de Recherche en Technologie de Fabrication Mécanique, Oran (Algérie), arisaek@yahoo.fr

Abstract

The paper describes reacting flow analysis of a gas turbine combustion system. The method is based on the solution of Navier-Stokes unsteady equations using unstructured grid due to the complexity of geometry of the combustor. Turbulent swirling flow in can-type model of a gas turbine combustor is computationally investigated. The turbulence effect are modelled through the basic modeling strategy, a two and three-dimensional Unsteady Reynolds Averaged Navier-Stokes (URANS) approach is applied, employing a differential Reynolds Stress Turbulence model (RSM and SST $k-\omega$). The combustion system includes swirler vane passages, fuel nozzles, swirl cone, and all holes in primary, dilution. The total geometry has been created using the pre-processing GAMBIT and SolidWorks, and the meshing has been done using GAMBIT, and the analysis carried out in ANSYS FLUENT 14.0 solver. Numerical predictions of the mean flow field showed a large inner recirculation zone and an outer recirculation zone which represents a typical result of confined swirl flames. This comparison shows the importance of three-dimensional effects combined with flow unsteadiness.

Mots clefs: URANS, RSM, Gas turbine, SST $k-\omega$

1. Introduction

Numerical simulations of the flows in gas turbine combustors had become an unavoidable way to accelerate the design of this type of modern engines and optimize their performances: reducing fuel consumption, limiting noise and air pollution, avoiding combustion instabilities...etc. The simulations also facilitate the understanding and the visualization of physical phenomena often inaccessible by the experimental measurements. The use of numerical tools for simulating unsteady combustion phenomena still presents some issues; the Reynolds Averaged Navier-Stokes (RANS) approach, also in its time-resolved form, URANS, has been proven of not being capable of resolving all time and space scales, which play crucial roles in highly turbulent unsteady combustion. Swirling lean premixed flames are frequently used in modern gas turbine combustors since they offer a possibility of controlled flame temperature and thus favorable thermal NO_x emissions and avoid intrusive methods disturbing flow

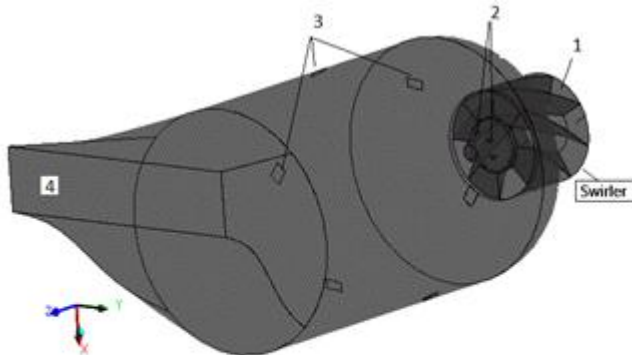
field. Combustion requires the effective mixing of fuel and oxidizer. For premixed flames, mixing takes place in a separated mixing chamber. This type of burner imposes the danger of a flashback of the flame into the mixing chamber. Another possibility is the mixing of fuel and air within the burner chamber. This prevents the danger of flashbacks, but homogeneous mixing is more difficult to achieve. A method often used in practical burners is the application of the swirling flows to improve mixing. In the swirl burner natural gas freely propagates into the burner chamber in the axial direction and is surrounded by the combustion air flow which has radial and tangential velocity components (swirl) in addition to the axial flow. The resulting flow field is strongly turbulent. Hot burned gases are transported back to the nozzle by internal recirculation in the flame and thus ensure effective mixing and stable ignition conditions. Swirling flows have been the subject of intensive experimental, analytical and numerical investigation over many years [1-2-3-4 and 5]. The application of swirling flows in industrial gas turbine combustors is of particular interest to the current work.

The goal of actual study is flame behavior according to transit natural gas turbulent flame evolution is analyzed using commercial code ANSYS-FLUENT 14.0 [6] with URANS Simulation model to treat turbulence coupling to partially premixed model to treat combustion. Models are applied to a three-dimensional geometry; they gave suitable results and were able to describe a detailed flow field. Results of the mean as well as the time-dependent numerical predictions of the turbulent vortex structures were presented.

2. Combustor configuration

The can combustor is a feature of the gas turbine engine. Arranged around a central annulus, can combustors are designed to minimize emissions, burn very efficiently and keep wall temperatures as low as possible. The combustor configuration has been illustrated in Fig. 1. The flame is stabilized by a low swirl number ($S = 0.6$). The size of the combustor is 590 mm in the Z direction, 250 mm in the Y direction, and 230 mm in the X direction. The primary inlet air is guided by vanes to give the air a swirling velocity component. The total surface area of primary main air inlet is 57 cm². The fuel is injected through six fuel inlets in the swirling primary air flow.

There are six small fuel inlets, each with a surface area of 0.14 cm². The secondary air is injected in the combustion chamber through six side air inlets each with an area of 2cm². The secondary air or dilution air is injected at 0.1 m from the fuel injector to control the flame temperature and NO_x emissions. The can-type combustor outlet has a rectangular shape with an area of 0.0150 m².



- 1: Main air inlet. The inlet is guided by vanes to give the air a swirling velocity component. Total surface area is 57 cm².
 2: Six small fuel inlets, each with a surface area of 0.14 cm².
 3: Six side air inlets, each with a surface area of 2 cm².
 4: Outlet, has a surface area of 150 cm²

Fig. 1: Combustor configuration

3. Modelling, boundary conditions

The three-dimensional geometry was created using GAMBIT-FLUENT pre-processor [7]. A view of the grid system is shown in Fig.2. The meshed geometry contained 31567 Nodes, 106651 Elements (Tetrahedra: 74189, Pyramids: 1989 and Wedges: 30473). The boundary conditions are summarized in Table 1.

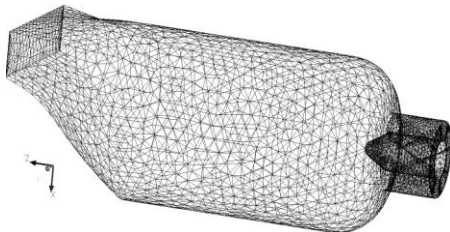
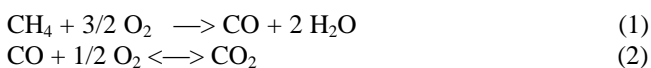


Fig. 2: 3D grid system

ANSYS-Fluent 14 offers a variety of different models for turbulence and combustion. A 3D URANS-RSM procedure is applied [6]. For comparison, the Shear Stress Transport (SST) model is also applied within the 3D URANS formulation for assessing the performance of RSM vs. SST. The RSM and SST models are also applied within a 2D-axisymmetric URANS formulation, and the results are compared with 3D URANS ones. The near-wall turbulence is modeled by the wall-functions approach [6]. The turbulence-chemistry interaction model called the eddy-dissipation model is based on the work of Magnussen and Hjertager [8]. In this work, a two-step mechanism to model the combustion of methane in air was employed:



The unsteady simulations were performed using the commercial CFD code ANSYS-FLUENT 14.0 [6], which is based on finite volume technique, is used to solve the governing equations. For the time discretization an implicit second order time differencing scheme was used. For the spatial discretization a bounded central difference scheme was used for the momentum equation, a second order upwind scheme for turbulence equations and a first order upwind scheme for the energy and species equations. The pressure-velocity coupling is numerically implemented using the SIMPLE algorithm.

Table 1 : Boundary conditions

	Type	values
Primary air inlet	Mass flow inlet	U = 10 m/s; Q= 0.0688875kg/s D _h = 85 mm. T = 300K It = 10%
Fuel inlet	Mass flow inlet	U = 40 m/s; Q = 0.00218033kg/s D _h = 4.2 mm. T = 300K It = 10%
Secondary air inlet	Mass flow inlet	U = 6 m/s; Q =0.00845225kg/s D _h = 16 mm. T = 300K It = 10%
Combustion chamber exit	Outflow Flux	
Walls of combustion chamber	Wall	T = 320 [k]

4. Results and discussion

In this section, 2D and 3D URANS modeling the Reynolds Stress Model (RSM) and the Shear Stress Transport model (SST) are compared for ability to predict flow field. Figure 3 compares the predicted distributions of the circumferential (W) velocity, along a traversal line z = 0.1m, for different turbulence modeling approaches. A similar comparison for Z= 0.2m is provided in Figure 4. Comparing the time-averaged circumferential velocity profiles, one can observe that 3D URANS RSM predicts a more confined vortex core with substantially higher velocities compared to 3D URANS SST. The radial profiles of the time-averaged circumferential velocity components extracted 0.1m above the injector inlet, Fig. 3, reveal that case 3D-URANS RSM, overestimates the negative circumferential velocity on the axis in comparison to case 3D-URANS SST k- ω by approximately 5%. Then again, case 3D-URANS RSM captures the intensity of the flow between the inner and outer shears better, Figure 3, and also the shape of the outer recirculation zone is better reproduced from the profiles extracted for cas 3D-URANS RSM. All testcases show the right penetration of the outer recirculation zone in the chamber: at z = 0.1m, the outer recirculation zone is not present anymore, Figure 4. Due to the contraction of the chamber into the exhaust canal rectangular a peak of velocity was registered next to the outlet. A similar comparison for z = 0.2m is provided in Fig. 4. Based on the profiles of time-averaged circumferential velocity at

$z = 0.1\text{m}$, one can see that 3D URANS SST k- ω over-predicts the size and intensity of the recirculation zone compared to 3D URANS RSM (Fig. 3). Comparing the time-averaged circumferential velocity profiles, one can observe that 3D URANS RSM predicts a more confined vortex core with substantially higher velocities compared to 3D URANS SST k- ω . For the comparison of time-averaged circumferential velocities by 3D URANS RSM and 3D URANS SST k- ω , similar trends are observed, also for $z = 0.2\text{m}$. The 2D URANS SST k- ω results predict an even larger recirculation zone and a broader vortex core compared to 3D URANS SST k- ω . It is already mentioned above that no convergence could be obtained by 2D URANS RSM, which can be seen as the manifestation of ability of RSM to capture low frequency flow unsteadiness. The 2D URANS RSM results displayed in the figures predict a qualitatively complete different W velocity field, implying a region of forward flow (central jet) enveloped by a recirculation zone. The circumferential velocity profiles of 2D URANS RSM also differ considerably from those of 3D URANS RSM. The all cases 2D-URANS, reveals better profiles for time averaged circumferential velocity. This comparison shows the importance of three-dimensional effects combined with flow unsteadiness. Generally, the time averaged characteristics of the flow field were very well captured by all numerical simulations.

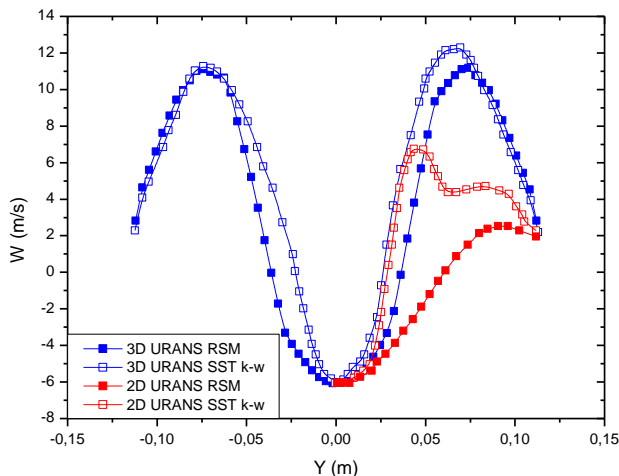


Fig. 4: Time-averaged circumferential velocity profiles for different turbulence models along a line at $z = 0.1\text{m}$

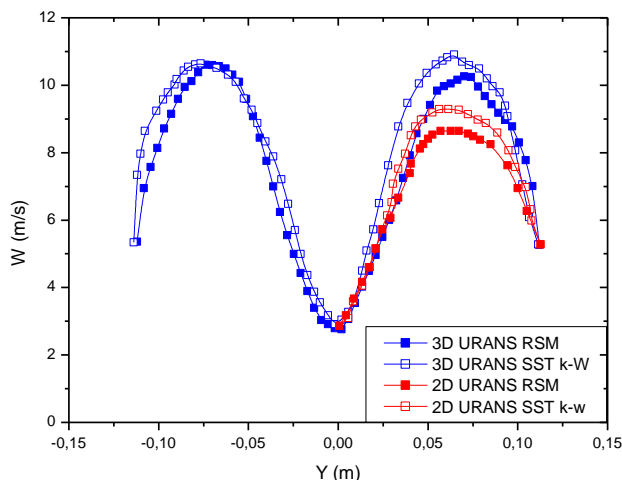


Fig. 5: Time-Averaged circumferential velocity profiles for different turbulence models along a line at $z = 0.2\text{m}$

5. Conclusion

In this work, numerical simulations of complex reactive turbulent swirled flow correspondent to a gas turbine model combustor were established by means of the ANSYS FLUENT 14.0. Turbulent swirling flow in can-type combustor model of a gas turbine combustor is computationally investigated. As the basic modeling strategy, a 3D URANS approach is applied, employing a differential Reynolds Stress Turbulence Model (RSM/ SST K- ω). A highly unsteady and 2D/3D flow structure, the vortex breakdown and a precessing vortex core are observed. Main findings can be summarized as:

- Differences between 3D URANS RSM and 3D URANS SST k- ω are rather substantial. The larger differences are observed for the time-averaged circumferential velocity profile, as the former predicts a more intense vortex core with higher maximum velocities.
- 2D URANS (RSM/SST K- ω) results may only be used for purely qualitative purposes.

Références

- [1] Benim A. C., Cagna M., Joos F. and Wiedermann A.: Computational analysis of turbulent swirling flow in water model of a gas turbine combustor (2009). Proceeding of 6th International conference on computational heat and mass transfer, May 18-21, 2009, Guangzhou, China 243.
- [2] Selvakuma K., M., Young Kim, 2014, "Combustion and emission characteristics in a can-type combustion chamber", International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, Vol. 8, No: 7, pp. 1300-1303.
- [3] Widenhorn A., B. Noll and M. Aigner (2010), Numerical Characterization of a Gas Turbine Model Combustor High Performance Computing in Science and Engineering '09, Part 4, 179-195.
- [4] Khelil, A., Naji, H., Loukarfi, L., Mompean, G., 2009, "Prediction of a high swirled natural gas diffusion flame using a PDF model", Fuel, Vol. 88, Issue 2, pp. 374-381.
- [5] Lysenko D. A. and Solomatnikov A. A. (2006), Numerical analysis of aerodynamics and hydraulic loss in the GTE'-150 combustion chamber of gas-turbine power plant with the use of the Fluent suite, Journal of Engineering Physics and Thermophysics, Vol. 79, No. 4.
- [6] ANSYS, Inc. ANSYS FLUENT v14.0: User's Guide; Theory Guide; UDF Manual, November 2010.
- [7] ANSYS Gambit 2.4, <http://www.ansys.com>.
- [8] B. F. Magnussen. "On the Structure of Turbulence and a Generalized Eddy Dissipation Concept for Chemical Reaction in Turbulent Flow". Nineteenth AIAA Meeting, St. Louis 1981.