Numerical study of non-premixed air-methane swirl combustor flows using RANS method

A F Sudarma[1,2] **and M H Morsy**[2,3]

[1] Mechanical Engineering Departement, Faculty of Engineering, Universitas Mercu Buana, Jakarta 11650, Indonesia
[2] Mechanical Engineering Departement, College of Engineering, King Saud University, Riyadh 11421, Saudi Arabia
[3] Mechanical Power Engineering Department, Faculty of Engineering, Port-Said University, Port-Said, Egypt

Email: andi.firdaus@mercubuana.ac.id

Abstract. The simulation is based on the experimental results obtained from German Aerospace Center (DLR) using a non-premixed swirling flow gas turbine combustor. The simulation case allows for the understanding of CFD's capabilities in modeling combustion for pre-mixed flows and adds the complexity of modeling mixing and combustion for un-mixed streams. The case is simulated in steady-state condition, with typical boundary conditions being applied. For turbulence modeling, the Reynolds Stress Model (RSM) approaches are being utilized here to study the flow and combustion processes occurring inside a combustion chamber where lean methane-air streams are burnt. For chemical kinetics calculations, the Probability Density Function (PDF) transport model and Eddy Dissipation Model (EDM) using a single-step reaction are considered. The simulation results are compared and evaluated qualitatively with the experimental data, including velocity field and temperature profile. The results of swirl flow combustor show that the adopted numerical model reproduced the trends of axial, radial and tangential velocity profiles, especially near the centerline. However, the model failed to predict the temperature and species mass fraction profiles.

1. Introduction

Study of combustion processes in all combustion systems is one of the critical and complex problems. Combustors play an important role in determining the operational characteristics of turbine engines, such as flame stability, fuel efficiency and engine performance. The combustion process involves complex interactions between chemical reactions and the turbulent flow field. Experimental investigations on industrial gas burners are very costly compared to CFD modeling [1], and the information presented is also limited. Computational Fluid Dynamics (CFD) can be very useful to provide important information about the flow field, temperature distribution and chemical reactions inside the combustion chamber.

Flames in high velocity flows are hardly stable, so some mechanisms are needed to enhance mixing and improve flame stability of the combustor. The recirculation zone formed behind the flame stabilizer helps in igniting the incoming fresh reactants through reprocessing hot products and radicals

from the burned mixture. The mixing process depends on the residence time of gases in the recirculation zone, and the residence time is affected by the size of stabilizer and velocity[2].

One kind of method to stabilize the flame is by introducing a swirl component to the incoming gases. Swirling flows can be produced by tangential jet injections or by vane swirlers. The swirler is a number of curved swirl vanes or tangential openings with different angles which promote the formation of recirculation zone, and this is the essential mechanism for flame stabilization. The swirl angle determines the size and the strength of the recirculation zone and most of the flame properties.

In combustion processes, swirling flows are widely used as a mechanism to enhance mixing of the reactants, the combustion performances and stabilization processes in the high intensity combustion chamber [3, 4]. Internal combustion engine cylinders, gas turbine combustors, power station boilers, and food dryers are typical examples where swirling flows are applicable. Recently, several significant results regarding the issues of the effect of swirl on the combustion and flame stability were available in the literature [5, 6].

The main objective of the present work is to examine the simulated results, in terms of velocity field, temperature profile and species distribution, and evaluated qualitatively through validation with available experimental data.

For achieving the objectives, the brief description of research strategies is described below. Some important information has been gathered to model the experimental case accurately, including combustor geometry, boundary condition and experimental results.The combustor geometry is transferred to Solidworks for modeling the computational domain in three-dimension.
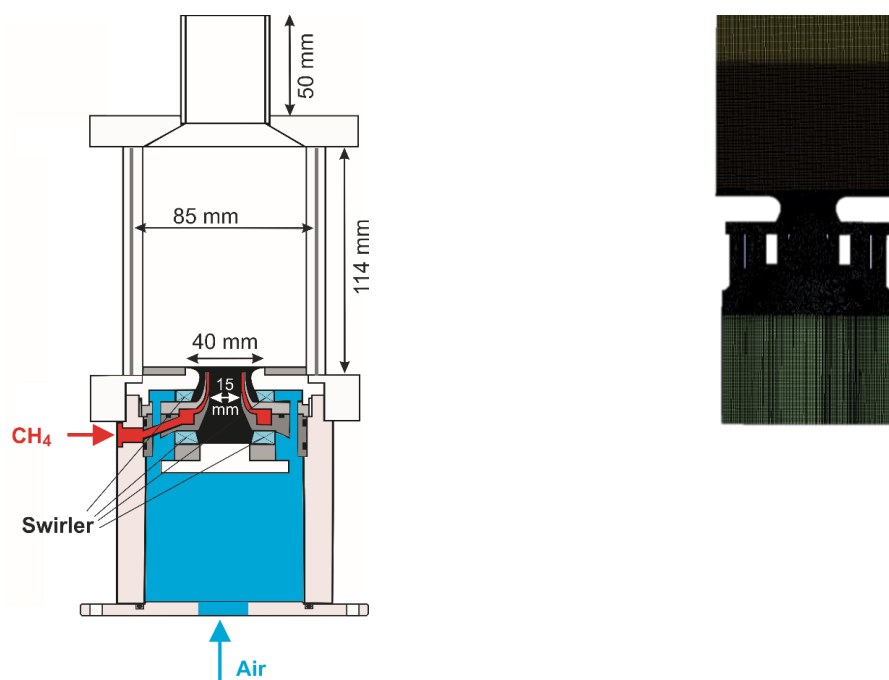
Figure 1**.** Schematic drawing of swirl flow combustor (left) [7] and Cells domain of swirl flow combustor case (right)

2. Procedure and boundary conditions

A brief description of the experimental method are explained below. The case is based on swirl flow combustion performed experimentally by German Aerospace Center (DLR) [7]. A schematic of the burner and the computational domain is illustrated in Figure 1 (left). The injector assembly consists of a central air nozzle, an annular fuel nozzle, and a co-annular outer air nozzle. Dry air at room temperature and atmospheric pressure is supplied from a common plenum and admitted through the inner and annular nozzle. The inner air nozzle has a diameter of 15 mm; the annular nozzle has an

inner and outer diameter of 17 and 25 mm, respectively. Nonswirling fuel is provided through 72 interior ports. The fuel nozzle is recessed by 4.5 mm below the burner face. The combustion chamber has a square cross section of 85 mm in width and 114 mm in height. The exit of the combustion chamber is connected to an exhaust tube with a diameter of 40 mm and a height of 50 mm. In the following, H denotes the axial distance from the bottom of the combustion chamber. The stable CH_4/air flame was operated at a global equivalence ratio of $\phi = 0.65$ with the corresponding mass flow rates of air and fuel were set for 1095 and 41.8 g/min, respectively.

2.1. Boundary conditions
The input flow for air and methane was modeled as "mass flow rate inlet" and the mass flow rates for air and fuel were set for 1095 and 41.8 g/min, respectively. The mixture flow into the combustion chamber at standard temperature and pressure. The combustor outlet is considered as "pressure outlet" at atmospheric pressure. The burner and combustion chamber walls were considered as no-slip and set to isothermal walls at temperature of 300 K.

2.2. Model application
The combustion case has been simulated numerically in steady state condition. It was decided to apply RSM approach for turbulent modeling, since the previous study shows that RSM is better than k–ε for complex flow modeling [8, 9]. The near wall flow was treated using a non-equilibrium wall function [10]. The first order Upwind Differencing Scheme (UDS) was used for the solution of the discretized momentum equations which were solved iteratively for the velocity components and pressure values using the SIMPLE algorithm. Radiation heat transfer was modeled as non-grey Discrete Ordinate (DO).

The Eddy Dissipation Model (EDM) was adopted in the present steady-state 3D calculation to model the mixing and reaction of the oxidant (air) and fuel (methane). The volumetric combustion reaction is represented by a one-step methane oxidation mechanism. The simulation was run on a Dell workstation with octa-core 64-bit processor and 24 GB RAM with ANSYS Fluent single-precision solver.

3. Results and discussions
The results and discussion of numerical study are presented below. The numerical results obtained from the swirling flow combustion simulation were compared against the experimental results provided by German Aerospace Center (DLR) [7, 11]. Experimental flow field measurements were obtained using laser Doppler velocimetry, temperature, temperature and the flame structures were visualized by Planar Laser-Induced Fluorescence (PLIF) of OH and CH radicals. The experimental measured data at several axial locations of H = 5, 10, 15, 20 and 45 are used to evaluate the model, where H represents the axial distance from the bottom surface of the combustion chamber.

3.1. Grid independence study
The numerical calculations were performed with four different grid sizes as indicated in Table 1 and the compared results were shown in Figure 2. The simulations were performed using the same condition of the experimental reacting flow in order to examine the solutions' quality in connection with the grid size (number of cells). Simulation results of velocity and temperature profiles of the four mesh sizes were compared. The figures show the results of mesh C and mesh D were almost similar and found to be grid independent. Therefore, mesh C, as shown in Figure 1 (right), was considered to be suitable and was used for the remainder of this work.

**Table 1.** Specification of different grid sizes.

Mesh A	Mesh B	Mesh C	Mesh D
2 million	3.5 million	10 million	20 million
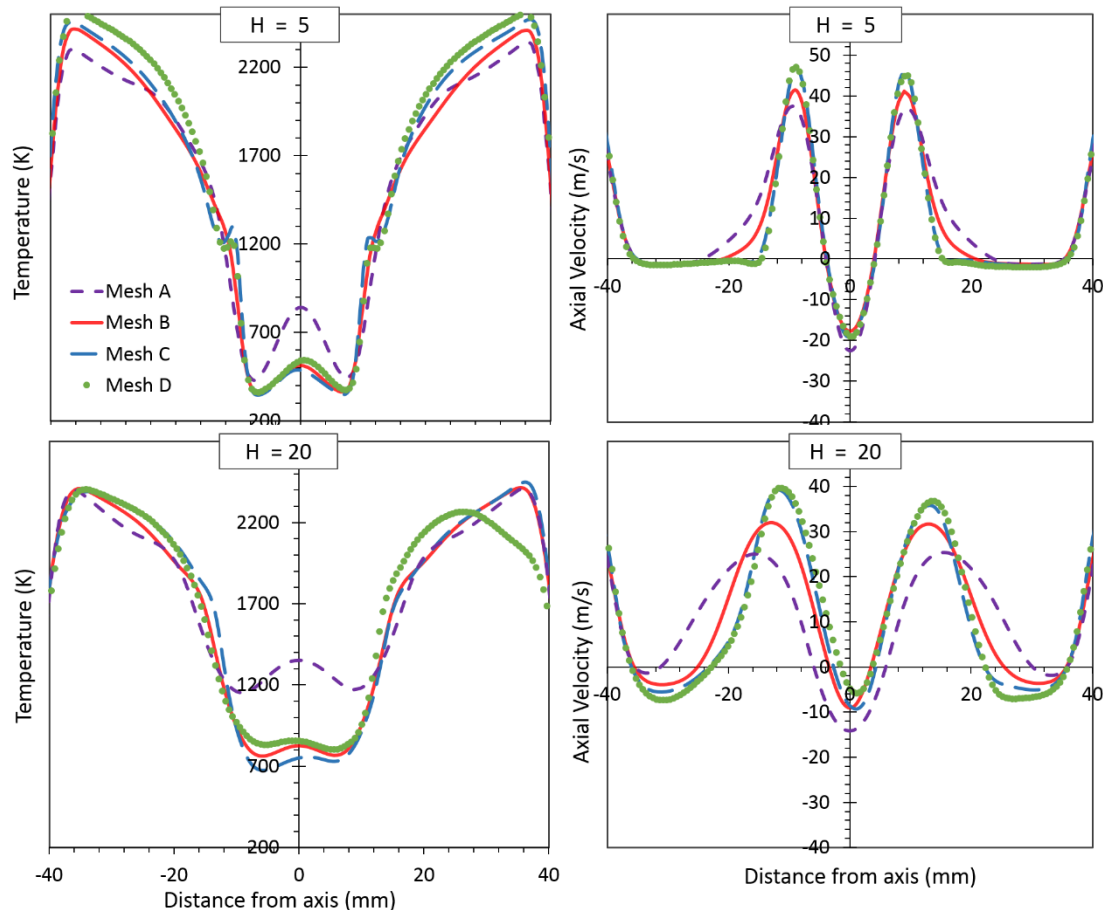
Figure 2. Grid independence test results of swirl flow combustor case.

3.2. Velocity field predictions

Figure 3 shows a comparison of measured and predicted mean axial velocity at various axial locations, e.g. H = 5, 10, 20, 45. The results show that axial velocity field was symmetrical for both positive and negative direction from centerline. It shows Outer Recirculation Zone (ORZ) does exist for both sides. Both ORZ were intersected at the centerline, and a reverse flow occurred at this region.

The results show that the predicted axial velocity at r = 10 mm (r represents the distance from axis/centerline) were slightly overpredicted and the maximum axial was velocity occurred at this point. However, the reverse flow trends at centerline are almost similar to experimental data, especially at H = 5 and 10 mm. The zero velocity at the edge (r > 20 mm for H = 5 and r > 24 mm for H = 10) reflecting the size of ORZ. At the axial location H = 20 and 45 mm, the reverse flows are underpredicted. The figures show predicted data produces lower reverse velocity compared to the experimental measurements.
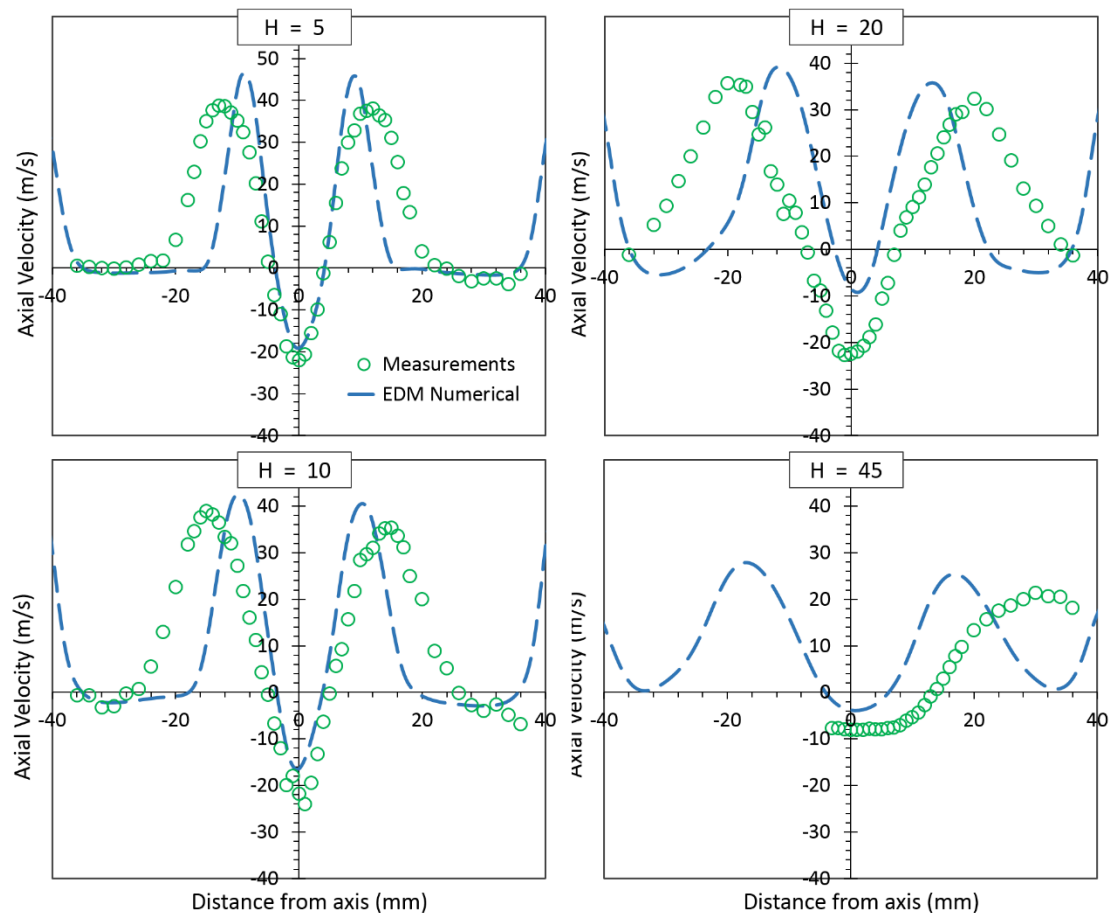
Figure 3. Measured and predicted mean axial velocity profiles.

The radial velocity profiles are presented in Figure 4. Overall, the predicted data given by EDM simulation gives similar trend compared to the experimental data only at the region r < 10 mm (near the centerline). Beyond this region, the numerical results show the underprediction and fail to replicate the negative flow behavior near the combustion chamber walls. The results show the existence of Outer Recirculation Zone (ORZ). The negative flow direction indicates the ORZ. At H = 5, the zero velocity at r = 20 indicates the ORZ edge while at H = 10 and 15, the ORZ size is decreasing.

Figure 5 shows the prediction of tangential velocity compared to measurements. In general, the numerical simulation produces a reasonable agreement with experimental measurements. Compared to axial and radial velocity prediction, the predicted tangential profiles almost similar and typical to the measurements.Overall, the simulation model produces a reasonable velocity prediction near the center line and shows the similar trends of the flow.

3.3. Temperature field predictions

A comparison between measured and predicted temperature profiles at different axial positions from the bottom combustion chamber, e.g. H = 5, 10, 15 and 20, is presented on Figure 6. The combustion process was conducted with overall equivalence ration $\phi = 0.65$ and adiabatic temperature reached at $T_{ad} = 1750$ K [7].

In general, one-step reaction EDM species transport model failed to predict the temperature profiles correctly. The results show that the maximum temperature predicted by the numerical model were ~ 2500 K located near the combustor walls which is exceeding the adiabatic temperature. In this case,
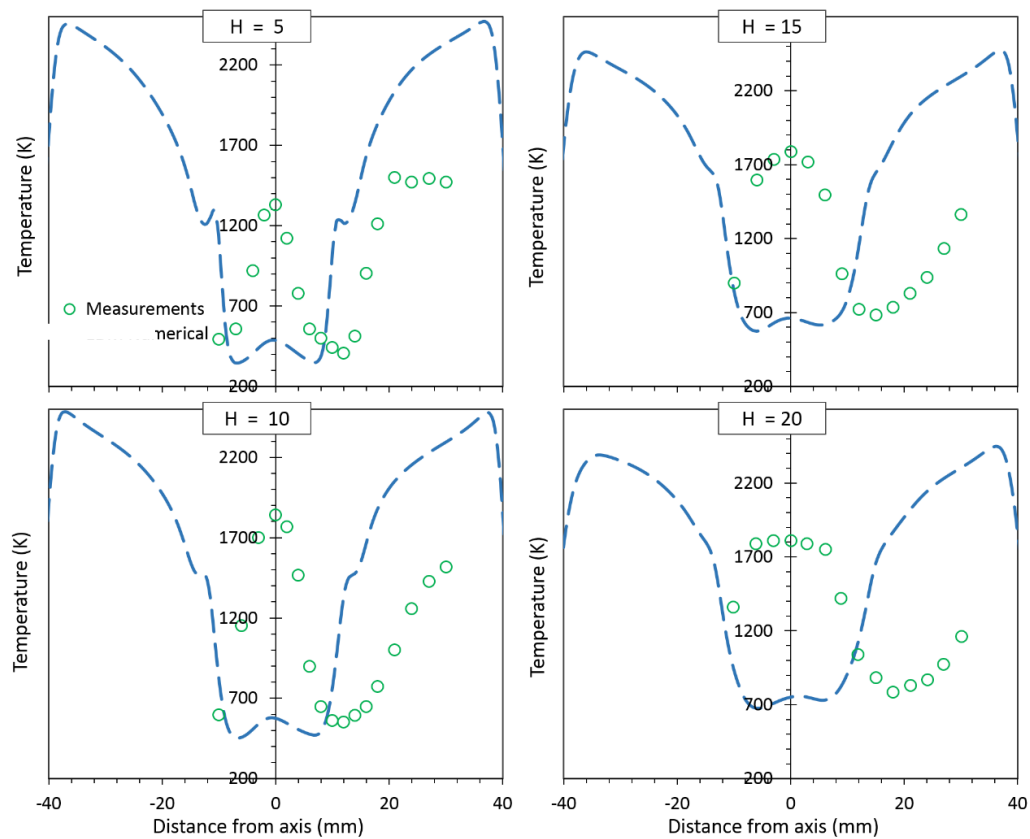
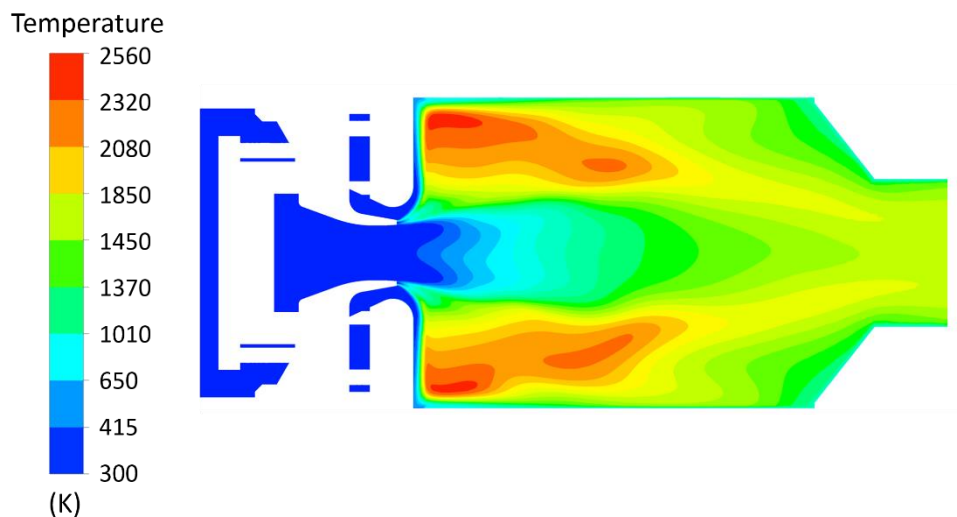the temperature inside the combustion chamber should lower than adiabatic temperature because of heat losses.
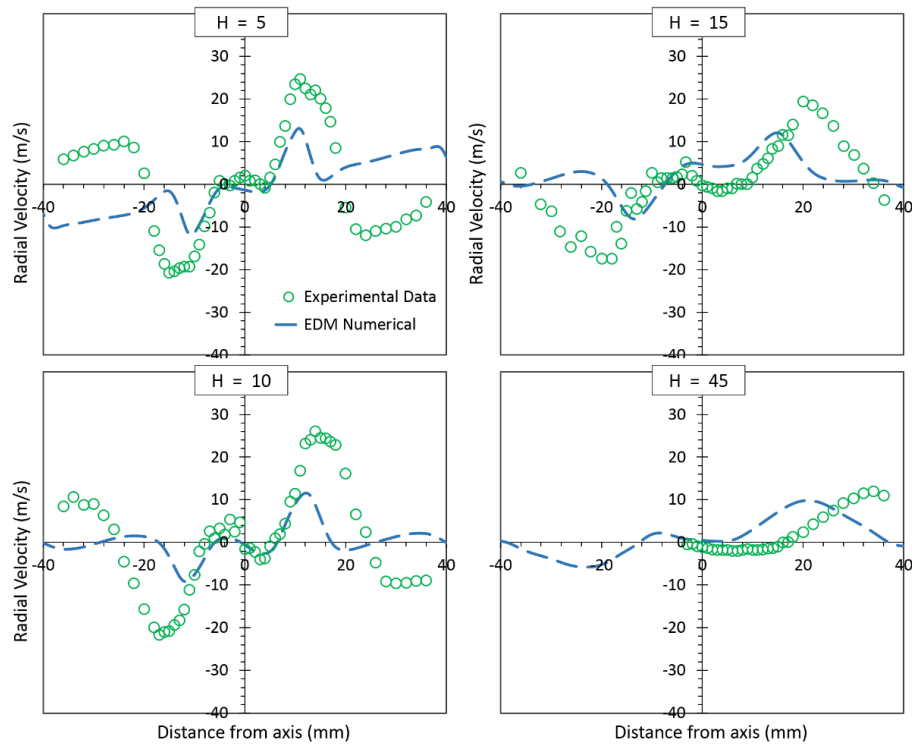
Figure4. Measured and predicted mean radial velocity profiles.
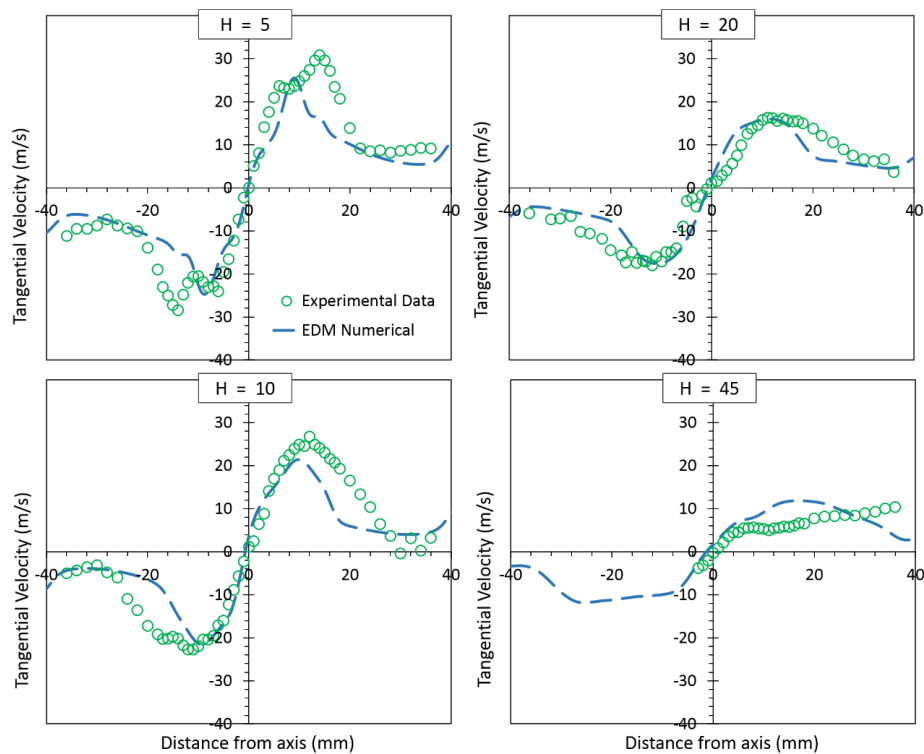
Figure 5. Measured and predicted mean tangential velocity profiles.

Figure6. Measured and predicted mean temperature profiles.

Figure 7. Temperature contours.

The discrepancies may happened because the model predicts the mass concentration in the particular area higher than the experimental values. The numerical model also failed to predict the temperature at the centerline. The discrepancies occurred because the model failed to reproduce the diffusion process correctly.

Figure 7 shows the temperature contour inside the combustion chamber, the existence of recirculation zone outside the swirling flow is clearly shown, and this region gives the highest temperature.

4. Conclusion
The numerical cases were being analyzed through the combustion processes simulation inside a non-premixed swirl flow combustor. The numerical study performed using RSM combined with one-step reaction eddy dissipation model approaches and the results were compared against the experimental data in terms of velocity fields, temperature profiles and species distribution. The results show that the numerical model produced a reasonable behavior of flow velocity. However, the model failed to capture the combustion features for temperature.

Acknowledgments

References
[1]   Köten H 2018 Performance analysis of a diesel engine within a multi-dimensional framework *Journal of Thermal Engineering* **4** 2075-82
[2]   Longwell J P, Frost E E and Weiss M A 1953 Flame stability in bluff body recirculation zones *Industrial & Engineering Chemistry* **45** 1629-33
[3]   Li Y, Pei L, Li J, Zheng J, Wang Y, Yuan J and Tang Y 2015 Instantaneous microwave frequency measurement with improved resolution *Optics Communications* **354** 140-7
[4]   Musa O, Xiong C, Changsheng Z and Li W 2017 Effect of inlet conditions on swirling turbulent reacting flows in a solid fuel ramjet engine *Applied Thermal Engineering* **113** 186-207
[5]   Liu Y, Tang H, Tian Z and Zheng H 2015 CFD simulations of turbulent flows in a twin swirl combustor by RANS and hybrid RANS/LES methods *Energy Procedia* **66** 329-32
[6]   Al-Abdeli Y M and Masri A R 2015 Review of laboratory swirl burners and experiments for model validation *Experimental Thermal and Fluid Science* **69** 178-96
[7]   Weigand P, Meier W, Duan X R, Stricker W and Aigner M 2006 Investigations of swirl flames in a gas turbine model combustor: I. Flow field, structures, temperature, and species distributions *Combustion and Flame* **144** 205-24
[8]   German A E and Mahmud T 2005 Modelling of non-premixed swirl burner flows using a Reynolds-stress turbulence closure *Fuel* **84** 583-94
[9]   Sudarma A F, Al-Witry A and Morsy M H 2017 RANS numerical simulation of lean premixed bluff body stabilized combustor: Comparison of turbulence models *Journal of Thermal Engineering* **3** 1561-73
[10]   Kim S E and Choudhury D 1995 A near-wall treatment using wall functions sensitized to pressure gradient *Separated and Complex Flows* **1** 273-80
[11]   Meier W, Duan X R and Weigand P 2006 Investigations of swirl flames in a gas turbine model combustor: II. Turbulence–chemistry interactions *Combustion and Flame* **144** 225-36