Applied Thermal Engineering 64 (2014) 19–31

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

PIV measurement and numerical simulation of fan-driven flow in a constant volume combustion vessel

Hai-Wen Ge [a], Michael Norconk [b], Seong-Young Lee [b,*], Jeffrey Naber [b], Steve Wooldridge [c], James Yi [c]

[a] ESI Inc., United States
[b] MEEM, Michigan Technological University, United States
[c] Research and Innovation Center, Ford Motor Company, United States

HIGHLIGHTS

- PIV measurement was conducted for model validation.
- Fan models, a body force and a sliding mesh models, were successfully built.
- A reasonably good agreement between both models and PIV measurement.
- Computational cost of sliding mesh model was much higher than body force model.
- Body force model is a more applicable model.

ARTICLE INFO

Article history:
Received 1 October 2013
Accepted 30 November 2013
Available online 12 December 2013

Keywords:
Fan
Wake
PIV
Unsteady RANS
Moving mesh
Body force model
Sliding mesh model
Combustion vessel

ABSTRACT

In the present work, flow motions in a constant volume combustion vessel driven by two fans were investigated using experimental measurement and numerical simulation. The flow field between two fans was measured using particle image velocimetry (PIV) technique. Unsteady RANS (Reynolds-averaged Navier–Stokes) method was used to simulate the fan-driven flow. Two different fan models, a body force model and a sliding mesh model, were used to model the effects of fan blades. Two rotating modes, co-rotating and counter-rotating, were considered. The numerical results of both models are in reasonably good agreement with the PIV measurement. The velocity field predictions of the sliding mesh model are close to the predictions of the body force model in a region just below the spark plug adaptor, and correlate better with the PIV measurements than the body force model further below the spark plug adaptor. However, the computational cost of the sliding mesh model is about 10 times more than the body force model. Thus, the body force model is a more applicable model for the current application in the constant volume combustion vessel.

1. Introduction

As the most widely used device in transferring energy between rotor and fluid, the fan is multi-functional and thus has been extensively used in ventilation, air condition, drying, mechanical draft, local exhaust, conveying, air-cooled heat exchange, air cleaning, etc. [1]. Rotors in turbomachinery [2], turbochargers [3], and wind turbines [4] have similar features as generic fans. The flows induced by fans are usually very complex, typically unsteady and three-dimensional, and strongly influenced by surrounding geometry.

Computational fluid dynamics (CFD) methods that numerically solve the Euler or Navier–Stokes equations have the potential to provide a consistent and physically realistic simulation of the flow field. Thanks to significant development of computer technology, CFD has been extensively used in many areas relating to fluid dynamics, including fan flows. CFD has become an important tool for design and optimization of fan design. Different fan models with various levels of simplifications have been developed and implemented into CFD codes. A Multiple Reference Frame (MRF) model [5] is the most common fan model. The MRF model separates the entire computational domain into a non-rotating region and a rotating region. The governing equations in the rotating region are

* Corresponding author. 815 R.L. Smith Bldg., 1400 Townsend Dr., Houghton, MI 49931, United States. Tel.: +1 906 487 2559.
E-mail address: sylee@mtu.edu (S.-Y. Lee).

1359-4311/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2013.11.073

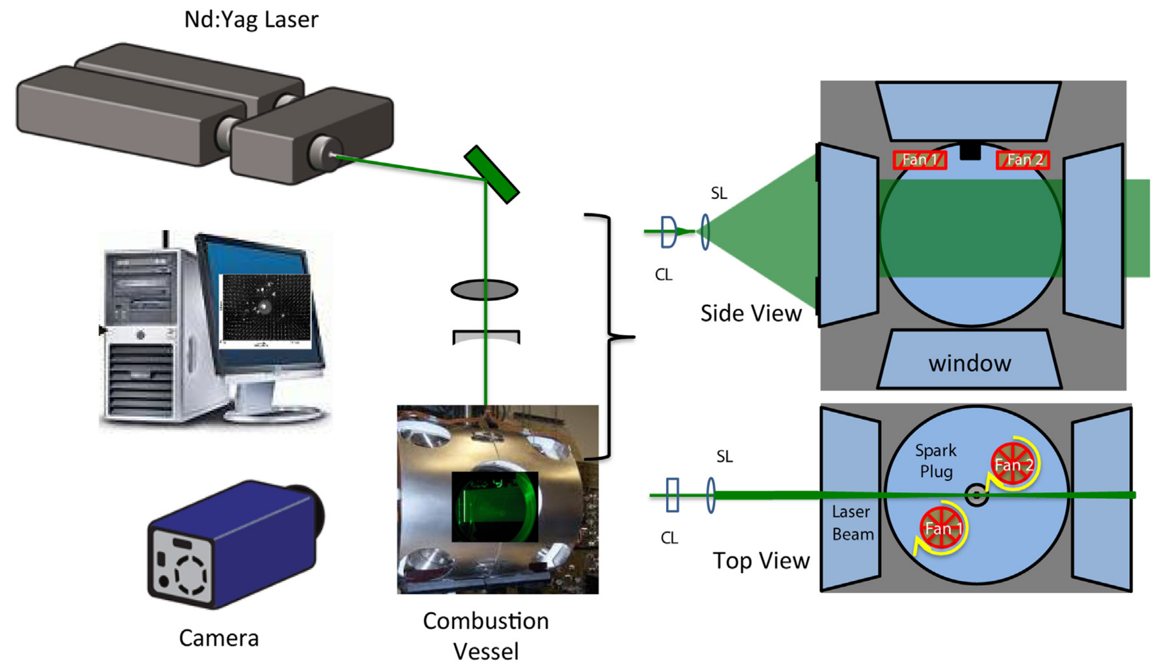

Fig. 1. Schematic of PIV setup (left) and the sheet beam orientation with respect to fans position (right).

transformed into a rotating frame of reference. A local reference frame transformation is performed to allow for flux calculations at the interface between these two regions. The model is valid only when the flow field at the MRF interface is in steady state. The results are sensitive to the size of MRF domain [6]. The body force model is another kind of fan model [7,8] in which the fan blades are not explicitly meshed. Instead, the fan region is meshed as a regular fluid domain. The fan effects are modeled by adding a body force to the momentum equation. In a simple approach, the fan effects are modeled by applying a pressure drop across the fan, which has been widely used in fan cooling [9]. The velocity field in the fan plane is estimated using a polynomial formula [10]. Duvenhage et al. [9] developed a body force model based on blade element theory and implemented the model into a general-purpose CFD code—PHOENICS. Meyer and Kröger [11] developed a body force model based on an actuator disk approach. The axial flow fan is modeled as an actuator disc, where the actuator disc forces are calculated using blade element theory. The calculated disc forces are expressed as a source term in the momentum equation. Tzanos and Chien [12] presented a similar body force model. Predicted velocity is compared with the results of a rotating reference frame model as well as experimental data. The results show that the accuracy of the body force model is comparable to the rotating reference frame model. Liu et al. [13] developed a downstream flow resistance method which employs an additional source term for the momentum transport equation to represent the resistance effects.

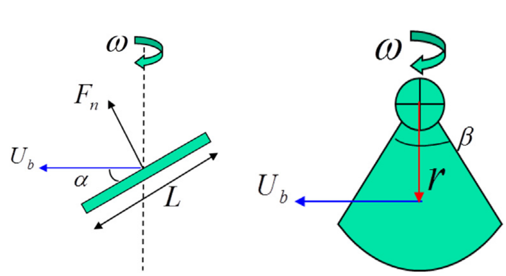

Fig. 2. Schematic of fan geometry. Left: side view; right: top view.

The third class of model is known as a sliding mesh model or rigid body rotation model. It considers the actual rotation of fan blades during simulation. Blades are explicitly meshed. It is the most accurate but the most expensive way to simulate the fan flow. Gullberg and Sengupta [14] compared numerical results of the MRF model and sliding mesh method with experimental data in a fan test rig. The results show that the sliding mesh method is an accurate methodology for fan flow simulation while the MRF model may have problems in transitional and radial flow regimes. Ota et al. [6] applied the PIV technique and CFD to develop radiator cooling fans for automotive applications.

Different measurement techniques have been applied to characterize the fan flows. Simple techniques, such as hot-wire anemometry, were used to measure the wake flow of ceiling fans [15]. Ota et al. [6] measured the flow velocity around the fan using a stereo PIV system. Zhu et al. [16] used a PDA system to measure the velocity on the blade-to-blade surface. Chen and Shu [17] studied near-wall flow structure using laser Doppler velocimetry (LDV). Estevadeordal et al. [18] used a two-color digital PIV system to study the flow field in the rotor passage of a low-speed axial fan. Yoon and Lee [19] used a stereoscopic PIV system to simultaneously measure three orthogonal velocity components of flow behind an axial fan. Morris et al. [20] used an X-array hot-wire probe to measure the velocity of wake flow behind an automobile cooling fan. Hurault et al. [21] used a two-dimensional hot fiber film probe to measure the components of the instantaneous velocity of the wake flow of an axial fan. Measurement of the three velocity components was achieved by measuring two different angular positions. Toffolo et al. [22] measured flow fields within the impeller of a cross-flow fan using a three-dimensional five-hole aerodynamic probe. Kohri et al. [23] visualized the flow around blades with tufts posted on the blade surface and simulated the fan flows using a MRF model.

Most of these studies focus on the wake of the fan flows [4,9,12,14,17,19–25] or the flow in the rotor passage [3,12,16,18,22,23,25,26]. The Alternative Fuels Combustion Laboratory at the Michigan Technological University developed a constant volume combustion vessel (CV). The CV is a fundamental research