SWIRL FLOW IN COMBUSTION CHAMBERS

AKMAL NIZAM BIN MOHAMMED

A thesis submitted in partial fulfillment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

NOVEMBER 2008

ABSTRACT

Swirling fluid motion in enclosed chambers was studied using two methods, simulations using Computational Fluid Dynamics, and experimentation using Particle Image Velocimetry. Using the tangential inlet configuration as the basic design, several different swirl generator models were created using Computer Aided Design software. The aim was to see whether a modified design from the original configuration could provide a reduction in the backflow effect that is constantly present in swirling flows. Simulations show that swirl generator inlets that are angled to 45 degrees from the original tangential position results in the backflow being slightly reduced. However, simulations in which the inlet angle was further increased yield inconclusive results. Later in the study, a model prototype of the 45 degrees inlet configuration was created for the purpose of PIV experimentation. From the experiments run, it was found that the results were comparable to that of the simulations.

ABSTRAK

Pergerakan bendalir secara berpusar dalam ruang tertutup telah dikaji menggunakan dua kaedah, iaitu secara Dinamik Bendalir Berkomputer (CFD) dan Penghalajuan Imej Partikel (PIV). Dengan menggunakan turus masuk secara tangen sebagai asas rekaan, beberapa model ‘penghasil pusaran’ telah dijanakan menggunakan perisian rekaan berkomputer (CAD). Tujuan penghasilan model-model ini adalah untuk menilai sama ada penngubahsuaian daripada konfigurasi asal dapat dimanfaatkan untuk mengurangkan kesan aliran bertentangan yang sering wujud dalam aliran berpusar. Hasil simulasi menunjukkan bahawa ‘penghasil pusaran’ yang mempunyai turus masuk pada 45 darjah daripada kedudukan tangen dapat mengurangkan sedikit kesan aliran bertentangan. Walau bagaimanapun, simulasi untuk konfigurasi yang mana sudut turus masuk telah ditambah tidak menunjukkan hasil yang konklusif. Kemudian, prototaip model untuk turus masuk 45 darjah telah dibina bagi tujuan menjalankan eksperimen. Daripada eksperimen yang dilakukan, telah didapati bahawa hasilnya menyerupai hasil simulasi.

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

This chapter provides an initial look to the study, outlining its objectives, listing its scopes and providing an historical background as reference and motivation.

1.1 Background

Swirl is a three-dimensional phenomenon in fluids flowing in a vessel with a circular cross-section at speed, where some of the particles follow a spiraling trajectory around the longitudinal axis of the vessel as they move along in the flow. Combustion systems that operate based on the principles of swirling fluid motion to facilitate the combustion process have increasingly been in extensive use. Swirl flows are now utilized in modern combustion machinery such as refinery or power station burners, gas turbine combustors, and internal combustion engines [1].

Swirling fluid motion has been introduced in the combustion engine of modern cars as part of an effort to reduce the emission of harmful gases, specifically Nitrogen Oxide (NO_x). Through the implementation of newly developed technologies in areas such as catalytic converters, fuel injection systems, and electronic engine management, the combustion process in these engines have become increasingly efficient. As a result, the emission of carbon monoxides, unburned hydrocarbons and other products of combustion in automotive vehicles have been greatly reduced. However, efficient combustion usually involves high combustion temperatures brought about by near stoichiometric fuel-air ratios in the chamber; such conditions have the negative impact of causing high thermal NO_x emissions [2]. This problem would need to be mitigated primarily through improvements in combustor design [3].

Thus, combustion systems that used alternating cycles of lean and rich burning mixtures were introduced. To achieve exact ratios of fuel-air mixture, these components of combustion would need to blend thoroughly before being ignited in the combustion chamber; hence the use of swirl generators. There have been many different ways to generate swirling flows being proposed according to their specific applications, for example using guide vanes, or rotating blades at the flow inlet. Regardless of the methods used, the generation of swirl flows enables the creation of a lean premixed reaction zone in the combustion chamber, which is required for lean mixture combustion. Furthermore, swirling flow helps in flame stabilization by creating flow reversal through vortex breakdown.

An interesting phenomenon of swirling flows in enclosed boundaries is the existence of a precessing vortex core along the longitudinal axis of the flow. The core of the swirl flow is visually distinct from the rest of the fluid movement, and can be described as having the shape of a continuous corkscrew spinning in the opposite direction of the general flow, as shown in Figure 1.1.

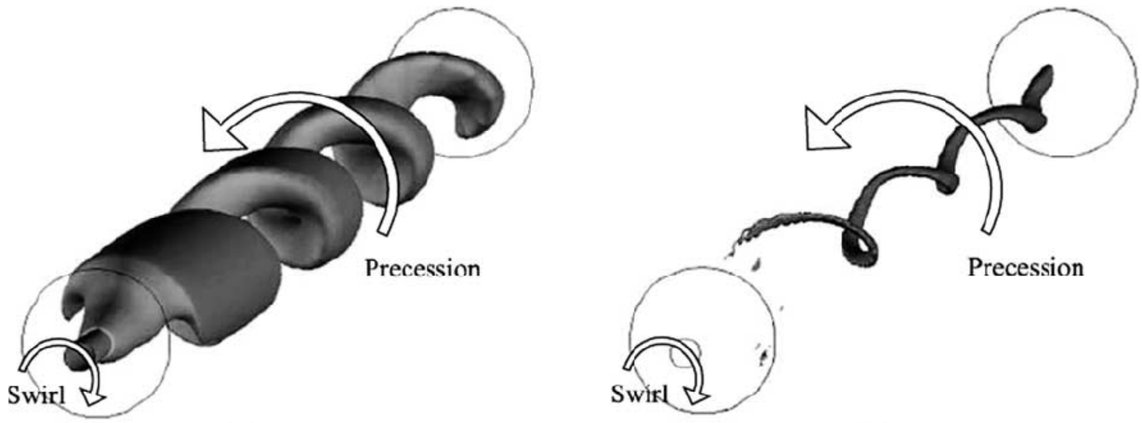

Figure 1.1 Precessing Vortex Core (PVC) [1]

The behavior of this vortex core is a significant occurrence in high velocity flows, as it affects the aerodynamic and thermal performance of the downstream turbine. In the combustion process, the existence of the vortex core in the combustion mixture flow will cause the flame itself to have the form of a vortex, as shown in Figure 1.2.

Figure 1.2 Precessing Vortex Core Flame [1]

1.2 Previous Study

The current study is principally based on a conference paper entitled 'Computational Analysis of Turbulent Swirling Flows for Gas Turbine Combustor Applications' by Benim et al. (2007) [4]. Consistent with the title of their presentation, the authors' work centers on investigating the existence of highly rotating vortex core in the flow within the combustion chamber of a modern gas turbine combustor and the impact of this phenomenon on the nozzle guide vanes at the combustor/turbine interface. They firstly sought out to validate the predictive capability of current modeling procedures for turbulent swirling flows, especially near the sub/supercritical vortex core transition. They then analyzed the effectiveness of these predictions as compared to experimental results done on two different laboratory test rigs with appropriately arranged parameters. All of the turbulent models used in the study were picked based on their capability of resolving three-dimensional transient motion of coherent flow structures without assuming a scalar turbulent viscosity at all scales; they are the URANS-RSM, LES and DES modeling approaches.

The result of the simulations is as shown in Figure 1.3. The figure is a temperature contour in a longitudinal plane of a combustion chamber; the core of the vortex is the hottest part of the flow. The vortex core extends from the inlet to the outlet of the chamber, interacting intensively with the nozzle guide vanes as a result. This interaction not only transfer massive amounts of heat to the guide vanes, but also disturbs the cooling aerodynamics of these vanes, causing them to overheat.

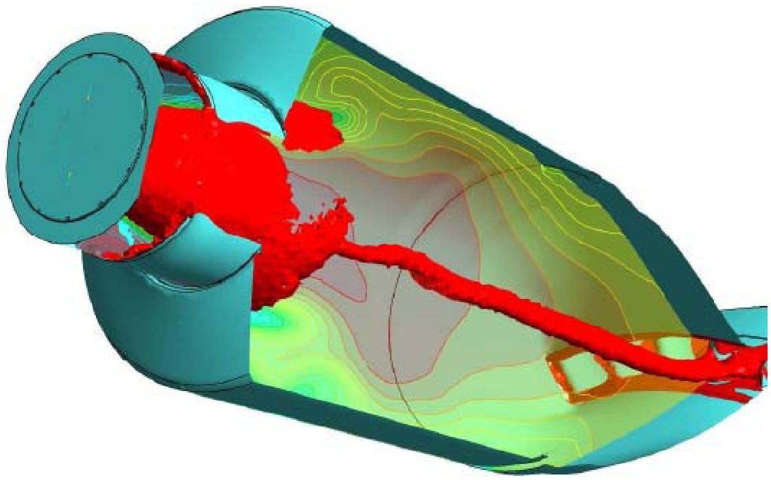

Figure 1.3 Result of CFD simulation on swirl flow in combustion chamber [4]

1.3 Problem Statement

As exhibited from the previous study, the formation of secondary flows that moves in the reverse direction from the primary swirling flow brings a negative effect to components in the combustion chamber. Therefore, this phenomenon, also known as backflow, needs to be eliminated, or at least reduced.

1.4 Objective

The objective of this study is to reduce the backflow that occurs in swirling flows, specifically swirls that are produced by tangential inlets that are commonly found in thermo fluid equipments such as combustion chambers, plenum chambers, and fluidized beds. As an attempt to improve on the current design, the inlets will be tilted from the tangent position up to certain angles, the result of which will be studied through simulations and experiments. To determine whether the backflow has reduced or increased, the velocity profile at several points in the flow region will be produced.

1.5 Scope

The scope of the study includes:

- Development of swirl generator model
- Determination of the swirl flow pattern using both Computational Fluid Dynamics (CFD) and Particle Image Velocimetry (PIV)
- Simulation and experimentation using air as the working fluid at atmospheric temperature (cold flow), thus without involving heat transfer
- Comparison of results between CFD and PIV

1.6 Past Researches on Swirl Flow

Some of the most important research papers concerning this topic are listed in Table 1.1. These researches are focused on the flow behavior in the chamber and not the actual combustion process.

Table 1.1 Past researches on swirl flow

Date	Occurrence
1983	Escudier and Keller propose the idea that a vortex breakdown along the swirl axis can be thought of as an abrupt transition between an upstream supercritical flow and a downstream subcritical flow. A supercritical condition occurs if the axial flow velocity exceeds the relative phase velocity of upstream-directed inertia waves, while a subcritical condition is the opposite.
1986	Weber et al. produces computation results of near-field aerodynamics of swirling expanding flows.
1988	Hogg et al. provide computation results of highly swirling confined flows with using the Reynolds Stress Turbulence Model.

1990	Benim publishes a finite element analysis of confined swirling flows
1997	Xia et al. publish a study on the effects of three-dimensionality on swirling flows with or without combustion.
2004	Turell et al. CFD simulation of the flow within and downstream of a high-swirl lean premixed gas turbine combustor.

1.7 History of Numerical Modeling in Turbulent Flow

The complexity of fluid flow in general makes it almost impossible to obtain complete analytical solutions for most real-world applications of fluid dynamics. However, without the ability to explain and predict the flow of fluids in the form of mathematical expressions, one cannot conceivably design any machine or apparatus that deals with fluids to work effectively under wide ranging circumstances of fluid movement. Therefore, scientist and researchers have long been involved in creating models that approximate the actual behavior of fluid flow to varying degrees of accuracy depending on the situation. In order to understand the correct method of simulating fluid flow structures, sufficient knowledge of fluid dynamics, computational methods, and numerical modeling is required. All these fields are encompassed in the study of turbulent flow modeling.

Turbulent modeling has been around since the late 1800s, when the analytical approach to solve for turbulent fluid flow has been all but exhausted. Some of the prominent figures in fluid mechanics began to introduce various concepts to describe the interactions between fluid particles and the factors that come into play at various conditions. The following Table 1.2 gives an overview of turbulent modeling development in chronological order.

REFERENCES

[1] N. Syred (2006). A Review of Oscillation Mechanisms and the Role of the Precessing Vortex Core (PVC) in Swirl Combustion Systems. Progress in Energy and Combustion Science 32 (2006) 93-161.

[2] S. L. Yang, Y. K. Siow, B. D. Peschke and R. R. Tacina (2003). Numerical Study of Non-Reacting Gas Turbine Combustor Swirl Flow Using Reynolds Stress Model. ASME J. Eng. Gas Turbines Power Vol. 125, 804-811.

[3] B. L. Koff (1994). Aircraft Gas Turbine Emission Challenge. ASME J. Eng. Gas Turbines Power Vol. 116, 474-477.

[4] A. C. Benim, M. P. Escudier, P. J. Stopford, and K. J. Syed (2007). Computational Analysis of Turbulent Swirling Flows for Gas Turbine Combustor Applications. The 9th Asian International Conference on Fluid Machinery NO. AICFM9-IL06.

[5] B. E. Launder and D. B. Spalding. *Lectures in Mathematical Models of Turbulence*. Academic Press, London, England, 1972.

[6] V. Yakhot and S. A. Orszag. Renormalization Group Analysis of Turbulence: I. Basic Theory. *Journal of Scientific Computing*, 1(1):1-51, 1986.

[7] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu. A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. *Computers Fluids*, 24(3):227-238, 1995.

[8] M. D. Turrell, P. J. Stopford, K. J. Syed and E. Buchanan (2001). CFD Simulations of the Flow within and Downstream of High Swirl Lean Premixed Gas Turbine Combustor. ASME Turbo-Expo Vol. 1, Combustion and Fuels, Education; 2004 31-8.

[9] M. Escudier, and J. J. Keller (1983). Vortex Breakdown: A Two-Stage Transition. Aerodynamics of Vortical Type of Flows in Three Dimensions, AGARD CP No. 342, Paper No. 25.

[10] S. Hogg, and Leschziner (1988). Computation of Highly Swirling Confined Flow with a Reynolds Stress Turbulence Model. AIAA J., Vol.27, pp.57-63.

[11] B. E. Launder, G. J. Reece, and W. Rodi (1975). Progress in the Development of a Reynolds-Stress Turbulence Closure. Journal of Fluid Mechanics, Vol. 68, pp. 537-566

[12] F. R. Menter (1994). Two Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal, Vol. 32, pp. 1598-1695.

[13] J. Smagorinsky (1963). General Circulation Experiments with the Primitive Equations. I: The Basic Experiment. Month. Weath. Rev., Vol. 91, pp. 99-164.

[14] I. G. Currie (2003). Fundamentals Mechanics of Fluids, Third Edition, New York: Marcel Dekker Inc.

[15] B. Massey (2006). Mechanics of Fluids, Eight Edition, New York: Taylor and Francis Group.

[16] H. Schlichting (1970). Boundary Layer Theory, Seventh Edition, New York: McGraw-Hill Book Company.

[17] H. K. Versteeg, and W. Malalasekera (1995). An Introduction to Computational Fluid Dynamics: The Finite Volume Method, First Edition, Essex: Longman Scientific and Technical.

[18] T. J. Chung (2002). Computational Fluids Dynamics, First Edition, Cambridge: Cambridge University Press.

[19] J. C. Tannehill, D. A. Anderson, and R. H. Pletcher (1997). Computational Fluid Mechanics and Heat Transfer, Second Edition, Washington, D.C.: Taylor and Francis Group.

[20] Singiresu S. Rao (2002). Applied Numerical Methods for Engineers and Scientists, New York: Prentice Hall.