Numerical simulation of tangential inlet configuration for plenum chambers

A N Mohammed\(^1\), J M Sheriff\(^2\), M F Mohideen Batcha\(^1\), A Sapit\(^1\) and M A Razali\(^1\)

\(^1\) Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia,
\(^2\) Faculty of Mechanical Engineering, Universiti Teknologi Malaysia

E-mail: akmaln@uthm.edu.my

Abstract. Swirling fluid motion in enclosed chambers was studied using Computational Fluid Dynamics. Using the tangential inlet configuration as the basic design, 3 swirl generator models was 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 at different angles from the original tangential position results in a change in velocity profiles across the flow cross section. From the simulations performed, it was found that the swirl generator model with inlets set to 45 degrees produced the least backflow compared to other models.

1. Introduction

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 spiralling 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 [1]. Swirl flows are now utilized in modern combustion machinery such as refinery or power station burners, gas turbine combustors, and internal combustion engines.

An interesting phenomenon of swirling flows in enclosed boundaries is the existence of a precessing vortex core along the longitudinal axis of the flow [2]. 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. The behaviour of this vortex core is a significant occurrence in high velocity flows, as it affects the aerodynamic and thermal performance of the downstream turbine [3]. 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 [4].

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) [2]. Consistent with the title of their presentation, the authors’ work centres 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 modelling procedures for turbulent swirling flows, especially near the sub/supercritical vortex core transition. They then analysed the effectiveness of these predictions as compared to experimental results done on two
different laboratory test rigs with appropriately arranged parameters. It has been found that the tangential inlet configuration that is used to produce swirl flow can lead to undesirable backflow. Thus, for the current study, a newer design is proposed in order to seek possible improvements that can be made compared to the conventional design [5]. For the project, the number of inlets is increased from three to six, whilst the inlet orientation is varied from the original tangential position. The difference of results from three different inlet orientations is then examined and compared.

2. Methodology
For the model case considered, simulations were run for at least three different speeds. Other simulation parameters were kept constant for all simulation runs.

| Parameter            | Value   | Units          |
|----------------------|---------|----------------|
| Density              | 1.225   | kg/m³          |
| Viscosity            | 1.789 x 10⁻⁵ | kg/ms⁻¹  |
| Specific Heat, Cp    | 1006    | J/kg-K         |
| Molecular Weight     | 28.97   | kg/kg-mol      |
| Reynolds Number a     | 2300 ~ 18000 |               |
| Strouhal Number       | ~ 0.86  |                |
| Swirl Number a        | 0.07 ~ 1.0 |               |

* a The range given are based on the minimum and maximum inlet velocities set in the simulation.

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 modelling approaches.

The model geometry used for simulation work has inlets that are positioned on the sides of the primary cylinder (main body) as shown in Figures 1, 2, and 3 depicting the three configurations named Model 1, Model 2, and Model 3 respectively. An example of the computer generated image for the model is shown in Figure 4, whilst in Figures 5 and 6, sample mesh illustrations of the simulation model is shown. The length of the inlet tubes for this geometry is 124.88 mm at maximum and 20 mm minimum, whilst the main body has a total length of 500 mm as shown in Figure 7. Each of the inlet diameter is set to 30 mm, with the main body being 305 mm in diameter. This geometry was chosen since tangential inlet is a traditional design commonly found for this type of swirl generators.
Figure 1. In Model 1, the six inlets are offset by 10° from perpendicular to the plenum.

Figure 2. In Model 2, the inlets are tangential to the main plenum chamber.

Figure 3. In Model 3, the inlets are slanted by 45° from perpendicular to the plenum.

Figure 4. Computer generated model of the swirl generator.

Figure 5. Sample model mesh of the flow domain viewed from the front.

Figure 6. Sample model mesh of the flow domain viewed from the side.

Figure 7. Dimensions of the main plenum and the position of the inlets.
3. Results and discussion

Figure 8 shows the velocity vector plot for the finished simulation run at an inlet velocity of 10 m/s. Following that is Figure 9, which shows a velocity contour plot at the outlet for the same case.

Figure 8. Velocity vector plot for the case of Model 2 at inlet velocity of 10 m/s.

Figure 9. Contour plot for Model 2 at 10 m/s inlet velocity.

Figure 10 shows the velocity profile along the centre axis of the cross section at the exit of Model 1. The high velocity contours occur around the outer regions of the cross-section, which eventually drops off around the centre. This indicates backflow occurs along the longitudinal axis of the primary cylinder, an expected result for this particular geometry model. However, significant differences in profiles are depicted for all three velocities, which point toward the different phases that the flows of the three cases are experiencing near the exit. The combination of these patterns indicates the possibility of a secondary flow inside the chamber. This recirculation zone induces a backflow along the length of the cylinder.

Figure 10. Velocity profile along the centre axis of the cross section at the exit of Model 1.

Figure 11. Velocity profile along the centre axis of the cross section at the exit of Model 2.
In Figure 11 representing Model 2, the profiles for inlet speeds of 10 m/s and 3.87 m/s are closer in shape to each other, whilst the profile for the inlet speed of 1 m/s seems skewed, having lower speeds close to the bottom of the cylinder. The ‘double-valley’ pattern that occurred for only the high-speed inlet flow in the previous geometry now occurs also for the medium inlet speed case, indicating the existence of the central recirculation zone. However, the recirculation zone for the high-speed inlet flow seems to be somewhat wider than the medium speed case, but a bit weaker.

In Figure 12, there exist similarities in the profile pattern between this model geometry and the previous two models. Again, the profile at low speed is somewhat misleading due to its skew, but profiles at medium and high inlet speeds exhibit similar characteristics to what is expected of a swirling flow. Comparing all three models, it has been calculated that Model 2 consistently provides the highest average velocity magnitude around the central area of the cylinder at all speeds considered, which means that the backflow for Model 2 is the lowest of the three configurations tested. On the other hand, Model 1 exhibits significant peaks and valleys whilst Model 3 has average velocity magnitudes that are simply lower than Model 2.

4. Conclusions and recommendations
Swirl flow behaviour in enclosed chambers has been studied using both Computational Fluid Dynamics and Particle Image Velocimetry. Geometry models of swirl generators were created, meshed, and simulated. It has been found from simulations that using the semi tangential inlet geometry instead of the traditional tangential inlet provides a discernible reduction in the backflow. Further reductions of backflow could possibly be achieved using the near perpendicular inlet geometry, but the flow pattern in this model deviates from the conventional swirling flow pattern, and is less predictable.

Acknowledgments
This research was made possible through funding from Universiti Tun Hussein Onn Malaysia (UTHM) IGSP Grant U413 and UTHM STG Grant U126. The authors would also like to acknowledge the contributions made by members of Flow Analysis, Simulation and Turbulence Research Group (FAST), the Centre for Energy and Industrial Environment Studies (CEIES) of UTHM, the Faculty of Mechanical and Manufacturing Engineering at UTHM, and the Faculty of Mechanical Engineering at Universiti Teknologi Malaysia (UTM).
References

[1] Syred N 2006 Progress in Energy and Combustion Sci. 32 (2006) 93-161
[2] Benim A, Escudier M P, Stopford P J and Syed K J 2007 Computational analysis of turbulent swirling flows for gas turbine combustor applications 9th Asian Int. Conf. Fluid Machinery AICFM9-IL06
[3] Batcha M F M, Hafiz M A, Sulaiman S A and Raghavan V R 2013 Asian. J. Sci. Research. 6(2) 157-66
[4] Hafiz M A, Batcha M F M and Asmuin N 2013 Material Sci. Eng. 50 012021
[5] Mohammed A N 2008 Swirl Flow in Combustion Chambers (Universiti Teknologi Malaysia)
[6] Binu T S and Mathew E M 2015 Int. J. Sci. Eng. Tech. Research 4(10) 3601-10
[7] Sabudin S, Wan Salim W S I and Batcha M F M 2016 ARPN J. Eng. Applied Sci. 11(18) 11129-34
[8] Naz M Y and Sulaiman S A 2016 J. Inst. 11 05019
[9] Cloos F J, Zimmermann A L and Pelz P F 2016 A second turbulent regime when a fully developed turbulent flow enters a rotating pipe Proc. ASME Turbo Expo 2016: Turbomachinery Tech. Conf. and Expo. (Seoul, South Korea, 13–17 June 2016) (ASME) GT2016-57499
[10] Othman S, Wahab A A and Raghavan V R 2008 Numerical study of the plenum chamber of a swirling fluidized bed Proc. Int. Conf. Mech. Manuf. Eng. 2008 (Johor Bahru, Malaysia, 21–23 May 2008) 97–98 –2963–59–2
[11] Herrmann-Priesnitz B, Calderón-Muñoz W R, Valencia A and Soto R 2016 Appl. Thermal. Eng. 109 22-34
[12] Mufeed V E and Goutham R 2016 Int. J. Sci. Research and Management 4(6) 4333-37