iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** IX **Month of publication:** September 2022

DOI: https://doi.org/10.22214/ijraset.2022.46550

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IX Sep 2022- Available at www.ijraset.com

Dynamic Design CFD Simulation of Geometric Canister Gasifier Stream

Nnadikwe Johnson[1], Samuel H. Kwelle[2], Ewelike Asterius Dozie[3], Onyewudiala Ibeawuchi Julius[4]
[1]*H.O.D in Department of Petroleum and Gas Engineering, Imo State University, Nigeria*
[2]*First Independent Power Limited, Rivers State, Nigeria*
[3] *H.O.D Department of Agriculture and Environmental Engineering, Imo State University, Nigeria*
[4] *Department of Mechanical Engineering, Imo State University, Nigeria*

Abstract: External streams in annuli are critical in regulating pressure loss, air stream distribution around the combustor liner, and the resulting impacts on performance, durability, and stability of the gas turbine combustion system. This article provides a computational fluid dynamics (CFD) simulation of the stream within a canister combustor's outer annulus. Validation of this simulation was accomplished by analysis of the stream within a canister combustor annulus utilized in a Afam/Nigeria gas turbine power plant facility. In 10 sites across the annular area, pitot static tubes were utilized to measure the velocity. By comparing the velocity profile, it was discovered that the CFD simulation and experimental work had a high degree of agreement.
Keywords: Geometric Stream, Canister Combustor, CFD Simulation, Pitot Static Tube, Velocity Profile.

I. INTRODUCTION

A gas turbine's canister combustor is an essential component. When compressed air is delivered to the turbine, it must be heated to an increased temperature before combustion can take place. This is ideal for improved overall efficiency and smoke-free combustion. Diffuser, liner, and casing-liner annulus are the three primary components of the combustor. The canister combustor is seen in Figure 1. There was a lot of research done in this work on the stream characteristics within a canister annular combustor computationally and physically, and a simulation constructed to analyze more effective instances on combustor cooling and air penetration into the liner. Gas turbine combustors stream and combustion numerical modeling has grown in popularity over the last few decades, thanks to advances in numerical computation methodologies and fast, highly competent computers. Analytical and numerical simulations based on CFD have shown that the stream within the canister combustor may be recognized in terms of its properties and gain significant insight into critical physical events.

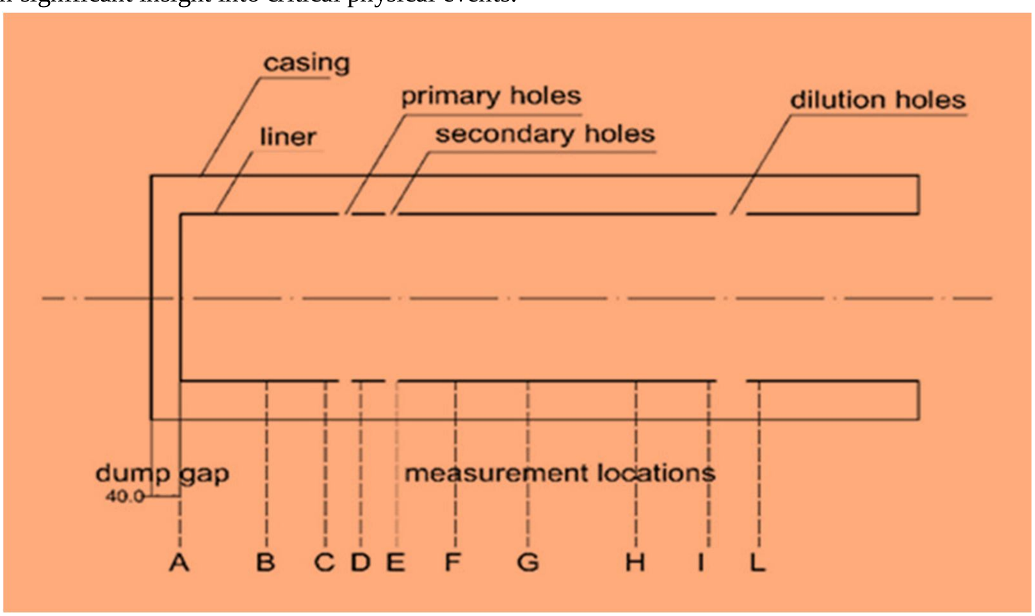


Fig. 1: Canister Combustor Layout

Fishenden and Stevens [2] examined Koutmos and McGuirk [1]'s stream model in the axisymmetric cone position and found their predictions to be accurate enough for engineering needs. Standard, RNG, realizable, Durbin modified, and the nonlinear k- model were among the models examined. The conventional and Durbin k- models had the greatest agreement with the experimental data, the findings revealed. According to Wennerberg and Obi [3,] excellent agreement between the k- model predictions and experimental data was found, and this backed up those findings.

An annular reverse stream combustor model's stream split via the liner holes was numerically examined by Mohan, Singh, and Agrawal [4]. According to Garg et al., cold stream modeling using CFD may be improved by adjusting the height of the inner and outer annuli. On flat, three-dimensional film cooling geometries, Miao and Wu [6] studied the main stream, injection tubes, impingement chambers, and supply plenum areas using numerical methods. For a low-Reynolds k- model, they observed that the projected data matched the actual data.

[7] A non-reacting combustor simulator with film cooling holes and two rows of in-line jets was also tested and simulated by Barringer et al. [7] to achieve comparable levels of turbulence around 18 percent. The RNG k- turbulence model was utilized in the computational simulations, and the findings were consistent with those discovered by Holdeman [8].

Canister combustors with non-swirling and swirling streams at the intake were studied by Alkhafagiy and Rahim [9] in an isothermal environment for CFD analysis. They demonstrate that the numerical findings that are verified against the experimental data are reasonable. For both non-swirling and swirling streams, Rahim, Singh, and Veeravalli [10] investigated the annulus stream characteristics of a canister combustor model for various liner dome shapes under isothermal stream conditions, and they discovered that swirling streams with a hemispherical dome liner give better annulus stream characteristics in swirling streams.

II. EXPERIMENTAL FACILITY

An actual canister combustor from the Afam gas turbine power plant facility in Nigeria was analyzed. The square-sectioned supersonic wind tunnel was linked to this component in the laboratory. Due to its circular shape, the wind tunnel's outflow has been shielded by a piece of wood. The circular opening in this wood shell enables the square and circular pieces to be matched. Experimentation in the lab is shown in Figure 1. The combustor was fed air at 314°K temperature and 31 m/s velocity from the wind tunnel. The experimental setup is shown in Figure 2. The canister-annulus combustor's interior liner has a diameter of 280mm and a length of 1020mm, while the air casing has a diameter of 410mm.

An annular gap of 65 mm and a 40 mm dump gap are created when the Liner is put into the Casing in an orthogonal fashion. Primary, secondary, and dilution holes are scattered throughout the liner's perimeter. The primary holes have a diameter of 19.5mm and have a total of eight in them. A total of eight 19.5mm-diameter secondary holes complete the set. The four 40mm-diameter Dilution holes. Table 1 explains that 10 locations were selected for examination. It was necessary to use a pitot static tube to get the velocity profile, and this tube was placed horizontally into the combustor from the opposite direction of air delivery in order to measure the velocity at ten different stations while being wired into a water manometer.

Table 1: Measurment Location in the Outer Annular of Canister Combustor

Position	A	B	C	D	E	F	G	H	I	L
X/Dc	0	0.29	0.48	0.61	0.73	0.92	1.17	1.53	1.78	1.95

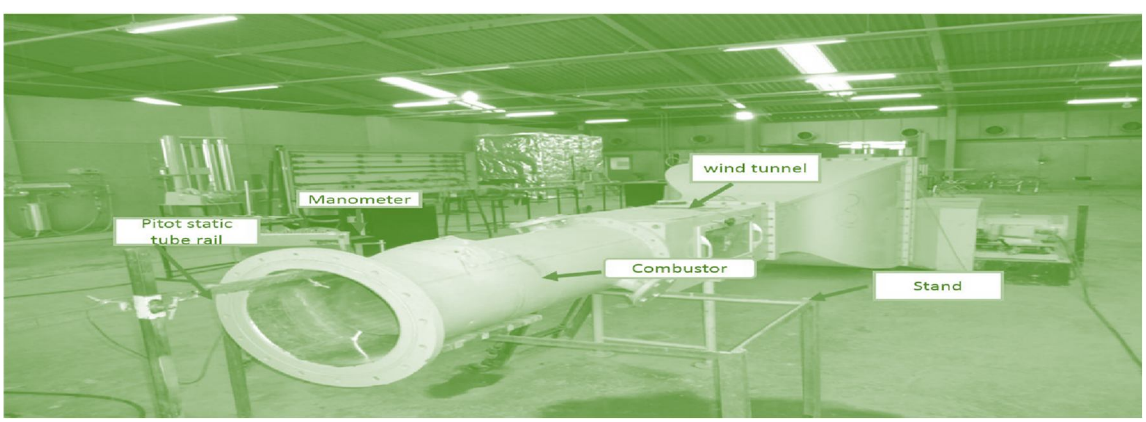


Fig. 2: Experimental Setup

III. COMPUTATIONAL MODEL

Commercial CFD software Fluent was used in the research.. The canister combustor was created using the GAMBIT package of the FLUENT code. Sensor data from the combustors was used to calculate the velocity stream at the combustor intake. By taking readings at the combustor's end, we were able to determine the overall combustor output pressure. It was made clear that this was the situation. Figure (3) shows the CFD simulation's working fluid, air, boundary conditions.

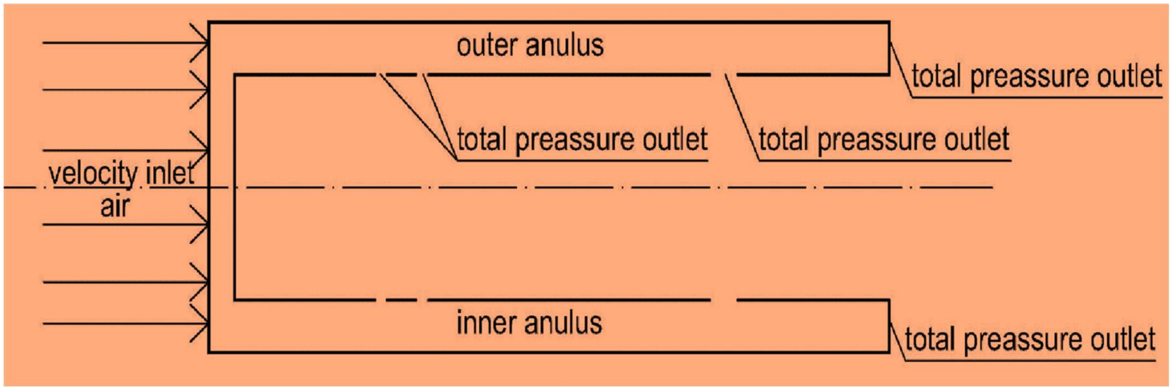


Fig. 3: Flipchart Shows the Boundary conditions in CFD

There are 85,915 meshes in this geometry, which was meshed using the Quad-Pave type. This was determined by comparing the percentage change in total pressure in the outstream to the grid independence (4). It was decided to use boundary layer meshing along the wall, however in other areas, it was decided to use a different meshing method. For turbulent viscous streams, the Prise layer approach is utilized because it is beneficial for calculating the viscosity-dominated near-wall areas. The K- model is a well-known simulation model.

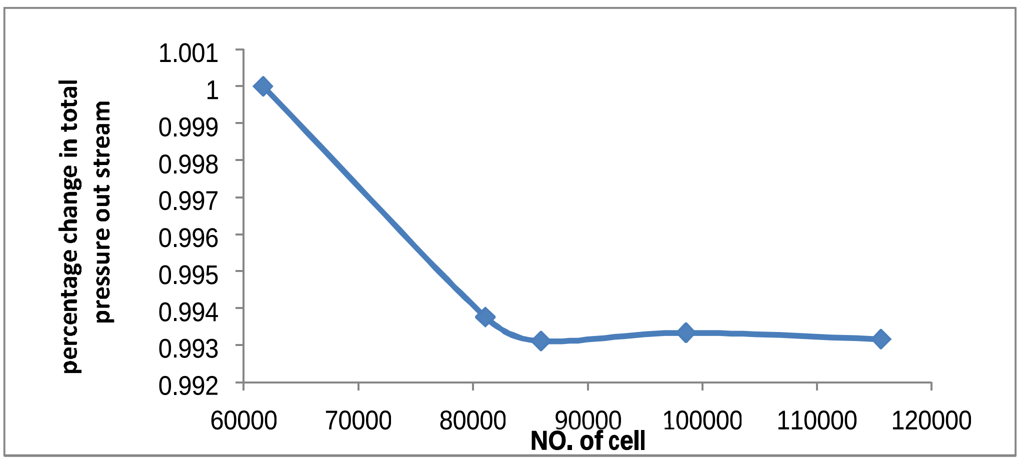


Fig. 4: Mesh Dependent Study

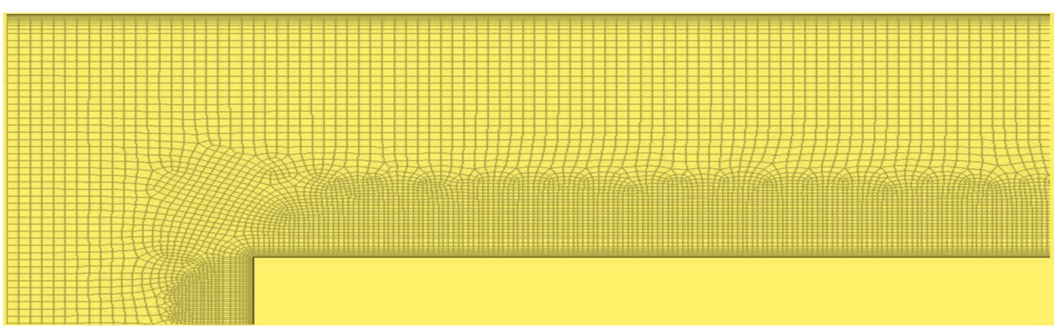
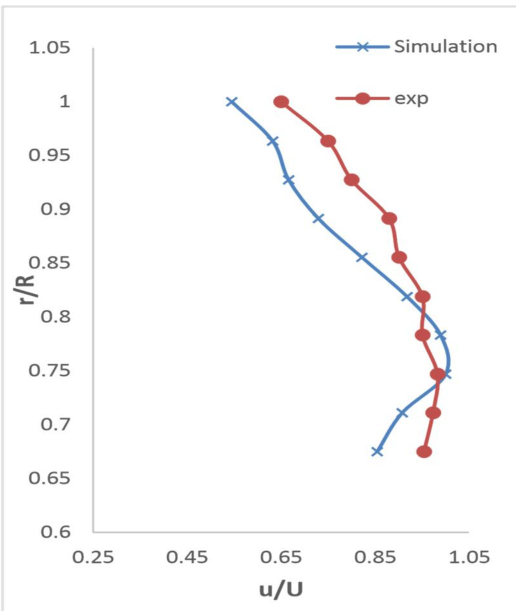


Fig. 5: Shows the Grid Mesh

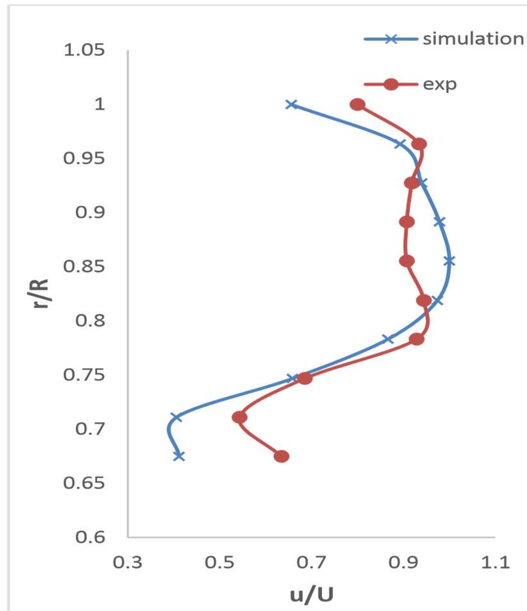
IV. RESULTS AND DISCUSSION

Experimental and theoretical investigations have been conducted into the annular canister combustor's stream characteristics. As may be seen in the figures (6A to 6L), the outer annular's axial velocity profile was studied experimentally and theoretically.

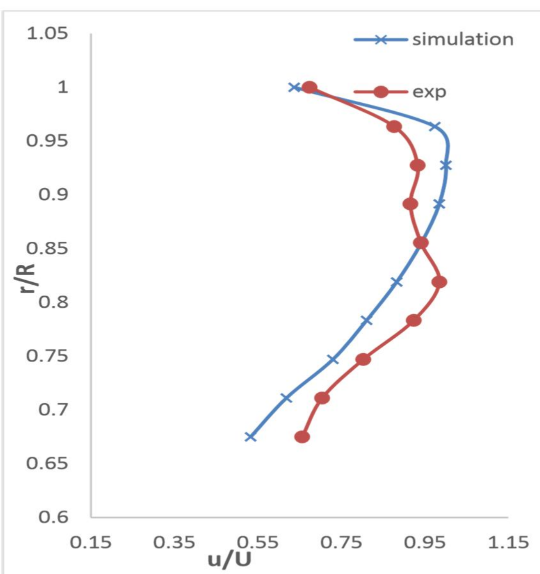
The practical and theoretical findings are found to be in excellent agreement, despite some recognized deviations in the velocity profile of the object. If one notices a skewed velocity profile at point A, it's because of a disruption that occurred at that point in the combustor's annular route (the beginning of the stream's journey). Fig. 6B shows the reversing stream near the liner wall as a result of a sharp edge on the liner head (dome) interfering with the stream homogeneity (6D). However, as seen in figures 6C to 6L, the stream becomes more and more uniform, better approximating the expected profile of velocity as it nears the exhaust of the combustor.



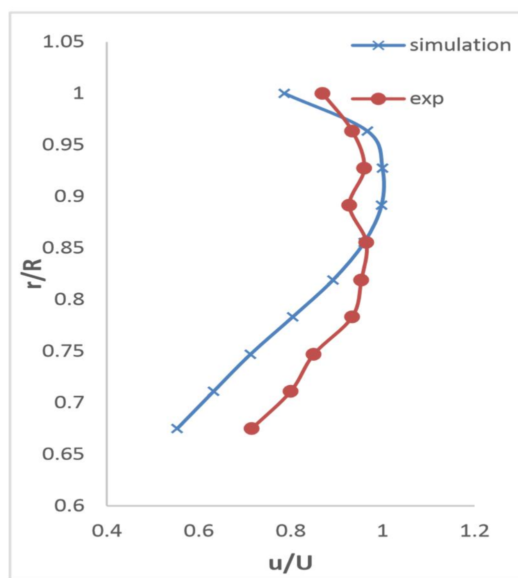
Axial Velocity profile at (A)



Axial velocity profile at (B)

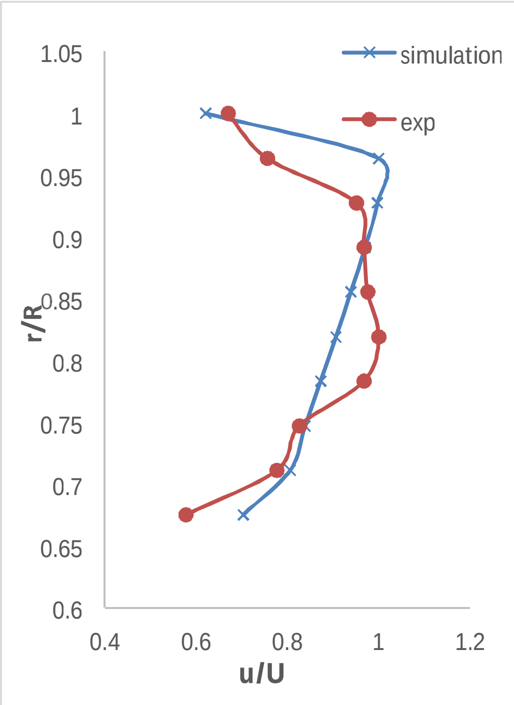


Axial velocity profile at (C)

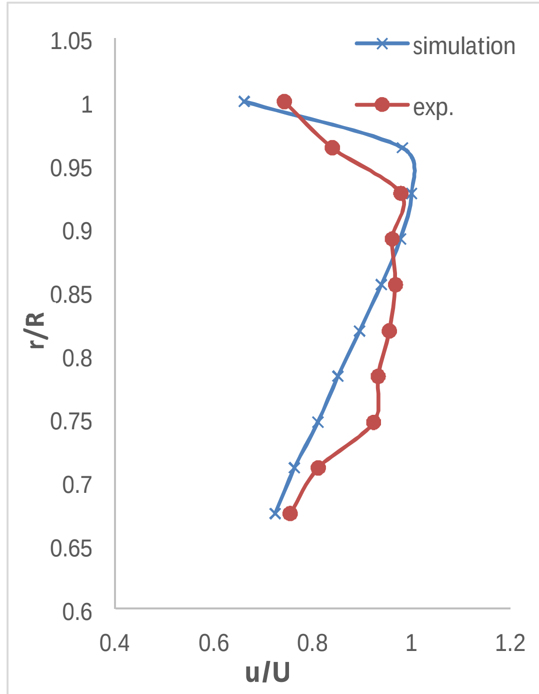


Axial velocity profile at (D)

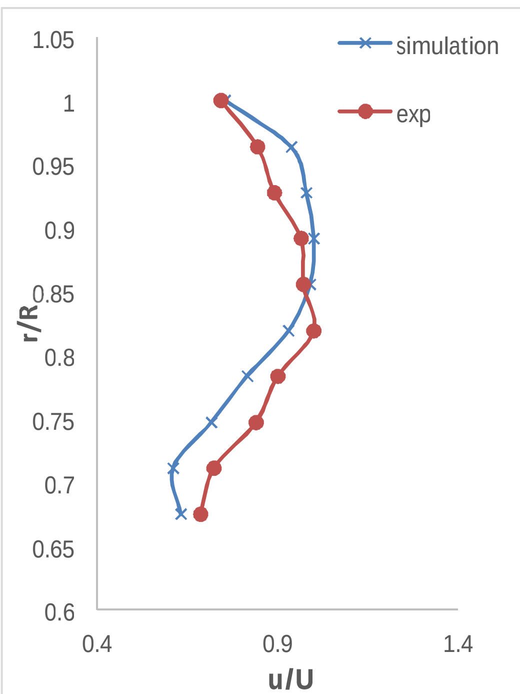
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IX Sep 2022- Available at www.ijraset.com



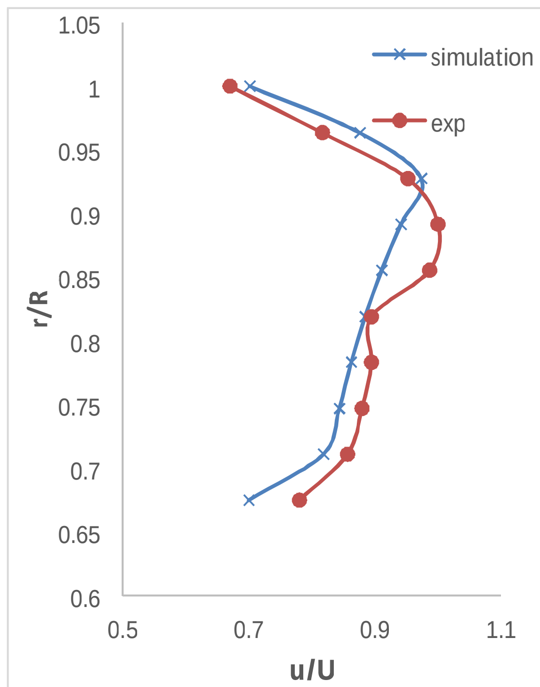
Axial velocity profile at (E)



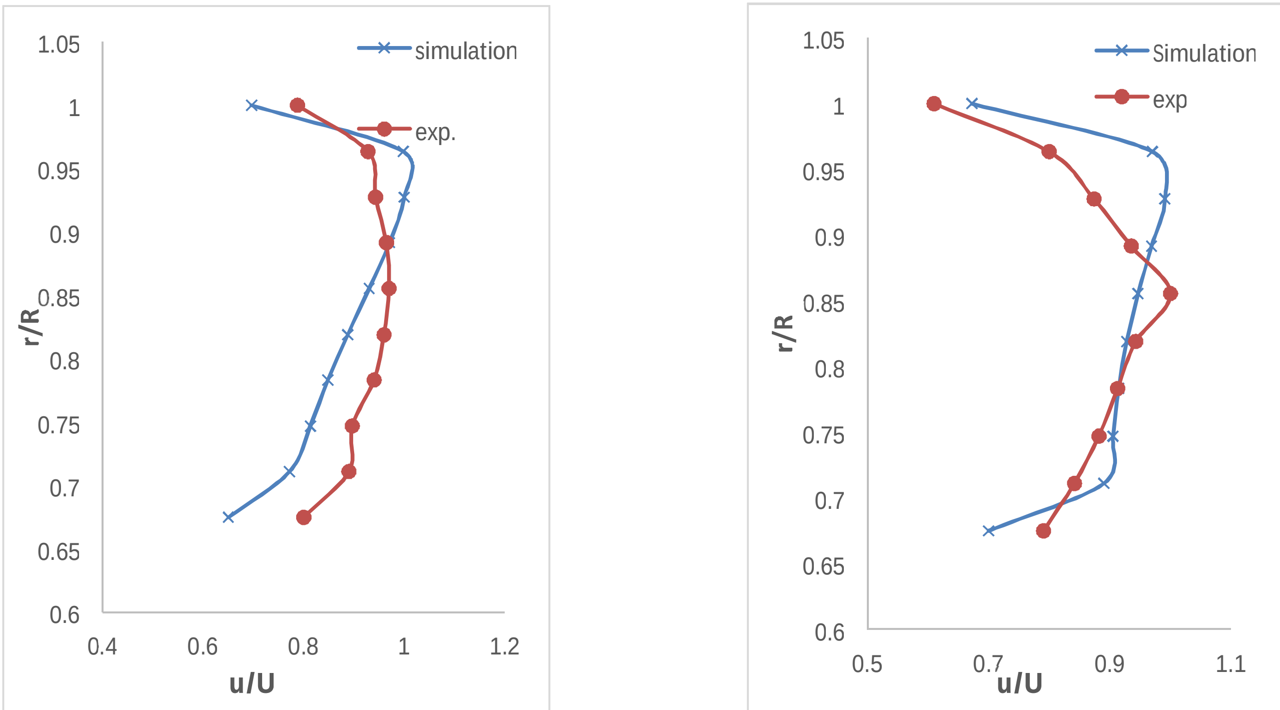
Axial velocity profile at (F)



Axial velocity profile at (H)



Axial velocity profile at (G)



Axial velocity profile at (I) Axial velocity profile at (L)

Fig. 6: Shows the Comparison Between the CFD Simulation and Experimental Work.

After it has ensured from the agreement between the theoretical and experimental results, it will be useful to study the other characteristics of the stream that have an effects on the performance of combustor work. Figure (7) shows contours of the velocity magnitude.it is observed big recirculation region at beginning of annuls stream and high level of velocity magnitude near the casing wall, this effect on penetration of air through the holes and cooling the liner wall performance. When arrives to the dilution holes the velocity decrease and will be uniform until the end.

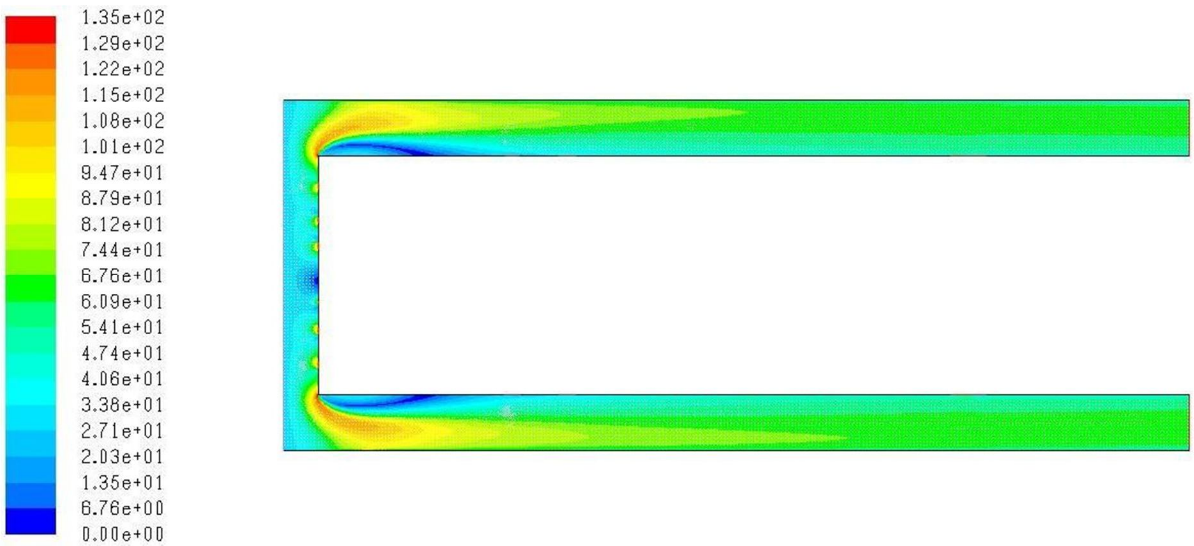


Fig.7: Contours of Velocity Magnitude

Turbulence intensity is seen in Figure 8. The turbulence around the corners of the annuli is particularly intense, and it progressively decreases until it reaches the end of the combustor. An increase in turbulence results in smoother surfaces, which decrease pressure loss and the risk of stream separation.

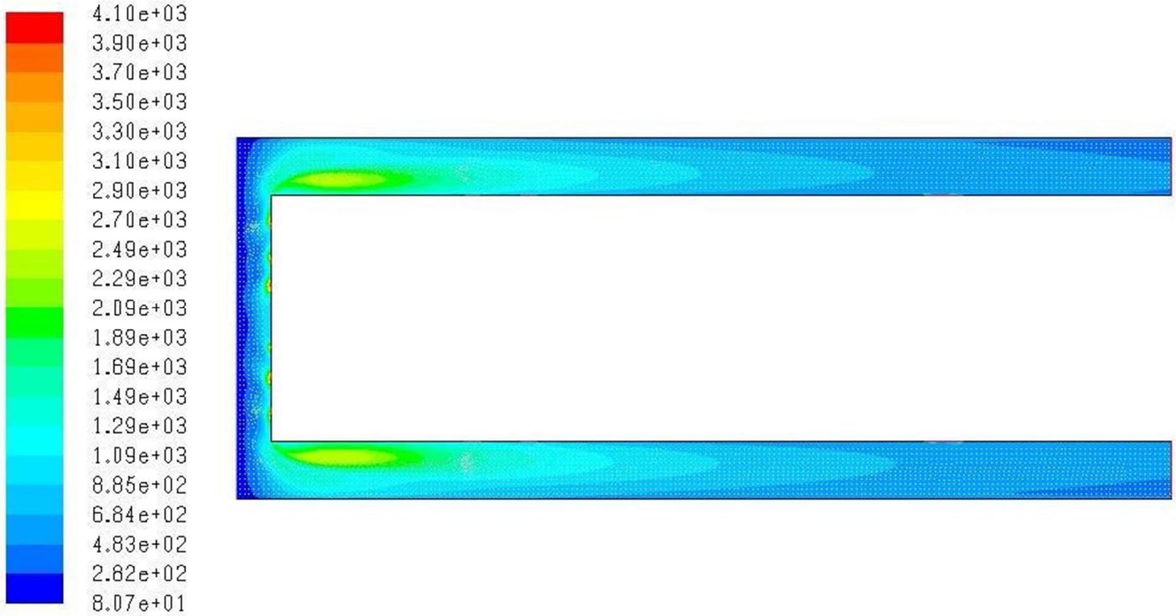


Fig. 8: Contours of Turbulent Intensity

V. CONCLUSION

An experimental and theoretical investigation of the canister combustor's stream characteristics has been conducted The combustor's performance and, ultimately, the plant's overall output power, are directly impacted by the velocity and pressure, respectively. Experimentation was carried out on an actual combustor from the Afam Electrical power plant. The findings of both the FLUENT simulation and the experiments were confirmed. An annular combustor combustor flow study using CFD seems to be useful, and the k- model provides a somewhat accurate prediction of the stream..

VI. ACKNOWLEDGEMENTS

The authors would like to acknowledge the Centre for Energy Resources and Refining at Imo State University, Owerri, Nigeria for their technical support, especially providing access to their own software application.

REFERENCES

[1] Koutmos, P. and McGuirk, J.J. (1989), Isothermal stream in a gas-turbine combustor—a benchmark experimental study. Experiments in Fluids 7(5), pp. 344-354. http://dx.doi.org/10.1007/BF00198453.
[2] Fishenden, C.R. and Stevens, S.J. (1977), Performance of annular combustor dump diffusers, Journal of Aircraft, 14(1), pp. 60-67. http://dx.doi.org/10.2514/3.58749.
[3] Wennerberg, D. and Obi, S. (1993), Prediction of strongly swirling streams in quarl expansions with a non-orthogonal finite-volume method and a second-moment turbulence closure. In: Rodi, W. and Martelli, F. (eds.) Engineering Turbulence Modelling and Experiments, vol. 2, pp. 197206. ISBN 0444898026.
[4] Mohan, R., Singh, S.N. and Agrawal, D.P. (1995), Stream split in a reverse stream combustor, Proceedings of the 22nd National Conference on Fluid Mechanics & Fluid power (IIT, Madras), 182-186.
[5] Garg, G., Bharani, S., Singh, S.N. and Seshadri, V. (2011), Stream characteristics around the liner of an annular gas turbine combustor model, Proceedings of the 28th National Conference on Fluid Mechanics & Fluid Power (PEC, Chandigarh) 13-15 Dec, pp. 3-11.
[6] Miao, Jr. M. and Wu, C-Y. (2006), Numerical approach to hole shape effect on film cooling effectiveness over flat plate including internal impingement cooling chamber. International Journal of Heat and Mass Transfer, 49, pp. 919-938. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2005.09.015.
[7] Barringer, M., Richard, O., Stitzel, S., Walter, J. and Thole, K. (2002), "Stream Field Simulations of a Gas Turbine Combustor," Journal of Turbo Machinery, 124, pp. 508-516. http://dx.doi.org/10.1115/1.1475742.
[8] Holdeman, J.D. (1993), Mixing of Multiple Jets with a Confined Subsonic Crossstream, Progress in Energy and Combusion Science, 19, pp. 3170, 60-67.
[9] Alkhafagiy, D. and Rahim, A. (2007), Application of CFD in combustor design technology, National Conference on State of Art Technologies in Mechanical Engineering, STEM, and pp. 311-316.
[10] Rahim, A., Singh, S.N. and Veeravalli, S.V. (2007), Liner dome shape effect on the annulus stream characteristics with and without swirl for a canister combustor, Journal of Power and Energy, IMech, 221, Part A.