McNair Scholars Research Journal

Volume 2 Article 2

2015

Computational Analyses of Combustive Vortex Flows in Liquid Rocket Engines

Nadia M. Numa
numan@my.erau.edu

Computational Analyses of Combustive Vortex Flows in Liquid Rocket Engines

McNair Scholars Program, Embry-Riddle Aeronautical University
Nadia M. Numa

The objective of this research is to probe further into the propulsion applications of a bidirectional swirl chamber by simulating the combustive vortex flows inside multiple configurations of liquid rocket engines. The present work initially tested the combustion inside a two-dimensional, axisymmetric combustion chamber to validate the computational methods. Three-dimensional geometries were then modeled and tested based on the swirl chamber concept. ANSYS FLUENT was used to simulate the flow region numerically. A viscous, turbulent, non-premixed combustion model with radiation at steady state was adopted. The results were justified by making qualitative comparisons to previous research conducted by other investigators.

I. Introduction

For years, scholars have studied cyclonic flows which occur ordinarily in nature in the form of cyclones, tornadoes, hurricanes, whirlpools or cosmic jets [1]. They have sought to take advantage of the cyclonic flows' capabilities, which are normally harmful when they materialize as natural phenomena, by controlling the manner in which they occur, i.e., limiting or confining their occurrence to a fixed space. Today, their capabilities have been harnessed in air pollution control technology such as in cyclone separators (Figure 1) and cyclonic spray scrubbers where particulates from an air, gas or liquid stream are removed without the use of filters, through vortex separation. Cyclonic flows are characterized by a bidirectional coaxial motion that is not triggered by vortex breakdown or instability. The flow reversal in the conical section of a conventional cyclone may be attributed to the presence of strong centrifugal forces which produce a low pressure region near the chamber axis. At the head end, the suction-induced pressure effect draws the primary fluid inwardly and causes it to turn around as the core is approached. This results in a bipolar motion which is characterized by the presence of a non-translating layer separating the upward and downward drafts, also known as the mantle [1].

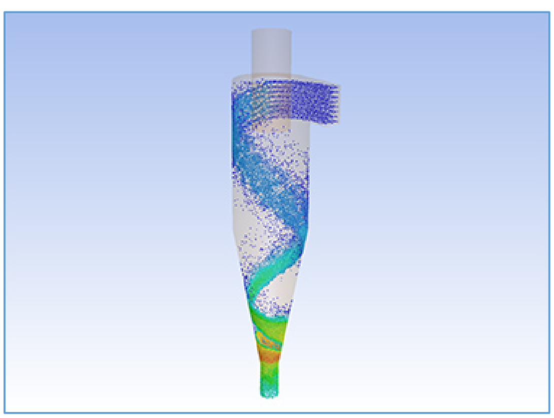

Figure 1: Cyclonic Separator [2].

A considerable amount of research has and still continues to be conducted to study the swirling motion of cyclonic flows. The structure of cyclonic flows makes them a good candidate for several applications, particularly, in the generation of swirls in the combustion chamber of transport systems since the performance of these systems is correlated with heat transfer, chemical dispensing and mixing of a variety of gases.

The principal investigators in the applications of cyclonic flows are Chiaverini et al. [3], who have examined the development of a self-cooled combustion chamber that would offer technological improvements such as lower temperatures at the walls of combustion chambers. This in turn translates into less cooling requirements which offer benefits in the areas of material selection, durability and weight. In their research, they report about an unorthodox combustion chamber where the oxidizer is injected tangentially, upstream of the nozzle (Figure 2). Once the oxidizer enters the chamber it spirals along the walls towards the entrance of the fuel for mixing to initiate. The fuel-oxidizer mixture then reverses direction and spirals towards the nozzle. The vortex-like motion allows for improved fuel mixture and increased turbulence which improves overall efficiency.

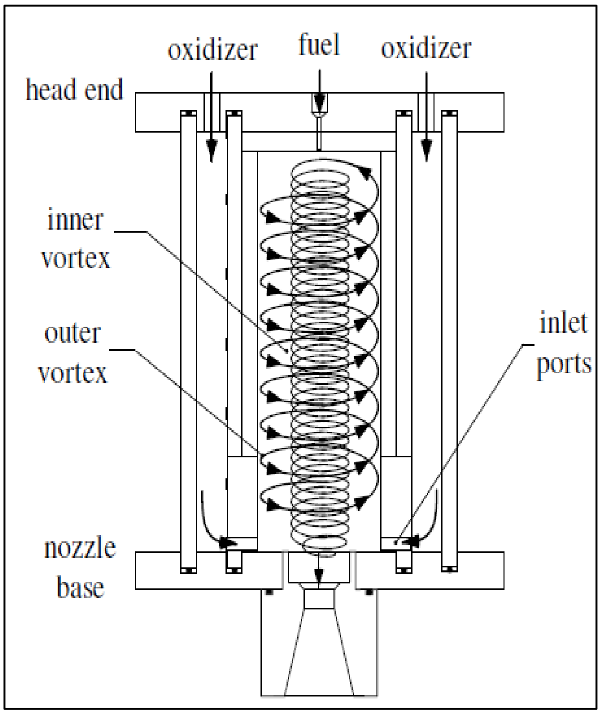

Figure 2: Sketch of a bidirectional swirl chamber (Chiaverini et al. [3])

The objective of this research study is to test the validity of the prototype (Figure 2) by simulating the occurrence of bidirectional combustive vortex flows numerically and to ascertain whether vortex flows do indeed improve the mixing and hence the efficiency of a combustion chamber. The results of this initial inquiry will then be succeeded by numerically experimenting to determine whether mixing is improved when the chamber geometry is altered to enhance vortex flow formation.

II. Methodology and Results

The limitations accompanied with analyzing complex flows necessitate the reliance on commercial computational fluid dynamics. While there are many reliable computational fluid dynamic solvers, ANSYS FLUENT was chosen to model the flow-field of the various engine configurations. The distinct combustion chamber geometries used in the flow analysis were designed in 3D using the CAD Software, CATIA V5 (Computer Aided Three-dimensional Interactive Application). The 3D geometries were then meshed using POINTWISE V17.0R2 prior to their analysis in ANSYS FLUENT. The baseline geometry was a two-dimensional axisymmetric model which was used to initially test the FLUENT results for conventional chambers. The second and third models were modified from the baseline by incorporating the oxidizer ports for the oxidizer to be injected tangentially. The third model featured an inner wall to enhance the vortex strength.

A. Methodology

Figure 3 represents the two-dimensional baseline geometry that was used to simulate the flow-field. It was designed with a diameter of 0.04 m for the fuel and a hydraulic diameter of 0.016 m for the oxidizer. As shown in Figure 3, the nozzle was not included in the design of the baseline geometry since the scope of this research was limited to the study and characterization of the behavior of the flow within the combustion chamber. Due to the complexity of the fuel injection systems in liquid rocket engines, the fuel injection ports in the computer models were designed using an example of a configuration shown by Sutton (Figure 4). Since the size of the chamber is small, the fuel injection system was simplified by the use of one port as was done by NASA researchers [4]. Therefore, this approach proved valid when compared to previous work done by Arshad et al. [5].

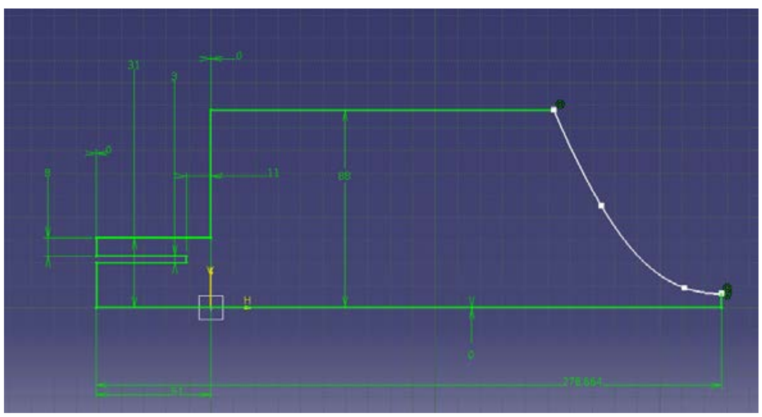

Figure 3: Baseline Geometry [6].

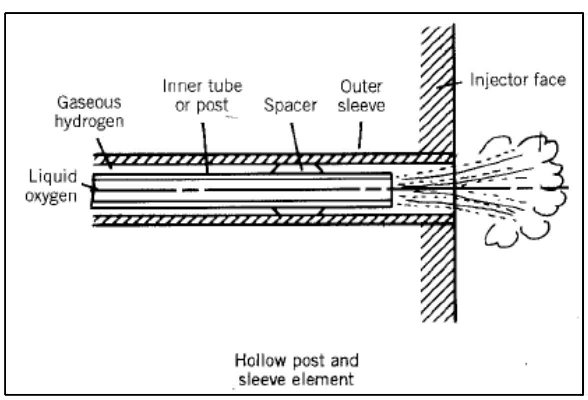

Figure 4: Simplified Injection System [7].

For each 3-D simulation, a successful mesh (Figure 5) was created using a combination of two different mesh styles. The first mesh was a structured mesh (quadrilaterals) applied to the boundary layer regions of the geometries to make sure that the boundary layer characteristics are captured with greater accuracy. The second mesh style is an unstructured mesh scheme (prisms) that populates the rest of the volume. A mesh-independence test was performed to obtain the mesh sizing that would produce the accurate results with minimal computation time. To ensure the quality of the mesh used, the equi-angle skewness (Q_{EAS}) was examined. A perfect mesh has a Q_{EAS} of zero. For the purpose of this study, the geometry had a maximum Q_{EAS} of 0.652 which suggested a favorable mesh quality.

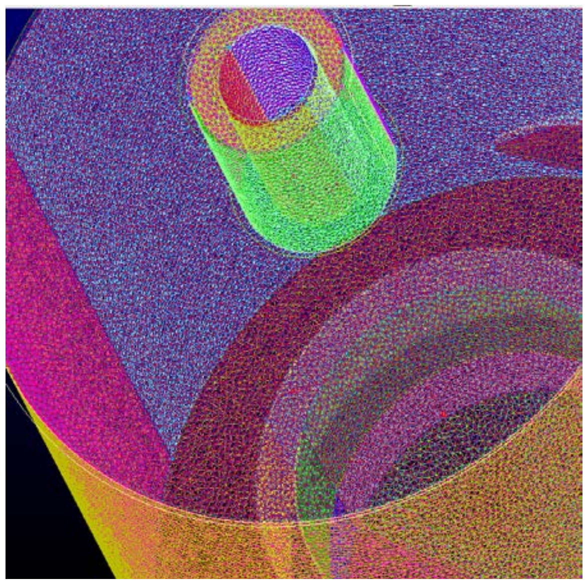

Figure 5: Mesh Creation Using Pointwise V17.0R2 [8].

ANSYS FLUENT was used to solve the steady-state, three-dimensional Navier-Stokes equations and to acquire the pressures, temperatures, velocities, etc. The standard κ-ε turbulence model was implemented. The reaction mechanisms of the reactants were simulated using the non-premixed combustion model with the Probability Density Function (PDF) tables. The discretization scheme used was PRESTO! (Pressure Staggering Option) to model pressure due to the pressure gradients. The popular SIMPLE pressure-velocity-coupling scheme was used and the discretization scheme for the other parameters was initially set to first-order upwind, and once convergence was achieved, it was changed to second-order upwind to increase accuracy. The residuals were monitored during the solution process and the simulations were continued until the residuals were decreased to sufficiently small values.

B. Results and Discussion

The initial step of the experimental part of the study was to simulate the flow-field inside a small two-dimensional axisymmetric liquid rocket engine to validate the parameters and methods which would be used to run the three-dimensional simulations. As expected, the fuel and oxidizer were mixed to produce a rich flame with a maximum temperature of 2010 K (Figure 6).

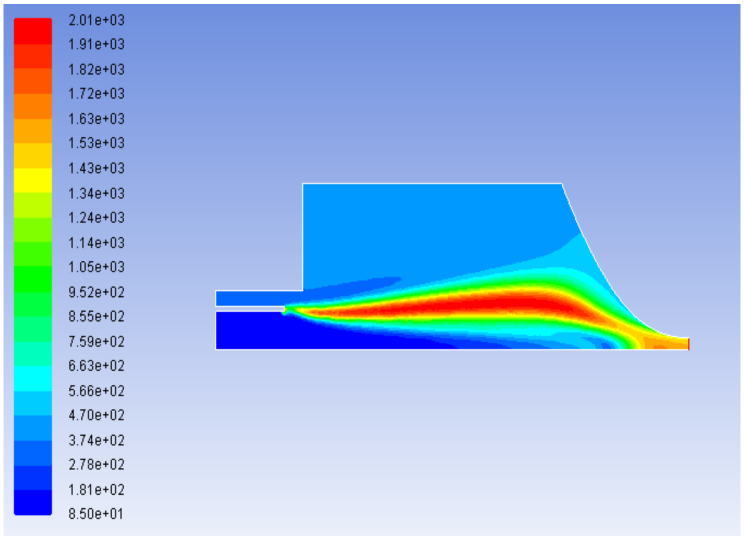

Figure 6: 2D Axisymmetric Flow-Field

The next step was to introduce the three-dimensional geometry model based on the bidirectional swirl chamber concept by Chiaverini et al [3]. The objective of using this geometry was to have the fuel enter axially, and the oxidizer enter tangentially at the base and spiral towards the head of the chamber (Figure 7). The cross-sectional areas of the two tangential injectors combined were equal to the total area of the oxidizer port in the baseline geometry (Figure 3). This was done to ensure consistency with Chiaverini's prototype to acquire accurate results.

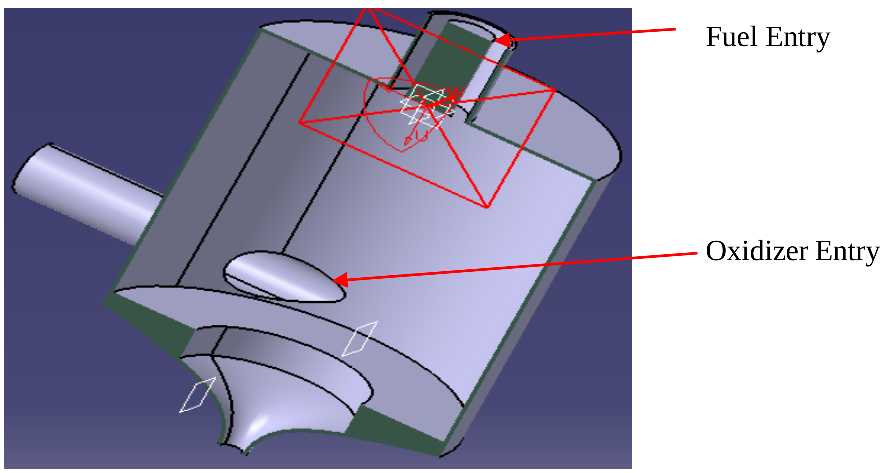

Figure 7: 3D Vortex Inducing Geometry Concept.

Figure 8 reveals the structure of the flame within the combustion chamber. It was observed that the flame was confined to the inner region of the combustion chamber confirming the creation of a mantle. The maximum temperature recorded was 2850K. To ascertain whether a vortex was created, velocity vectors were plotted along a plane perpendicular to the fuel injection ports. The maximum velocity registered was about 350m/s. This result affirmed the hypothesis that inducing a vortex inside the combustion chamber allows for better mixing, improving the overall combustion process.

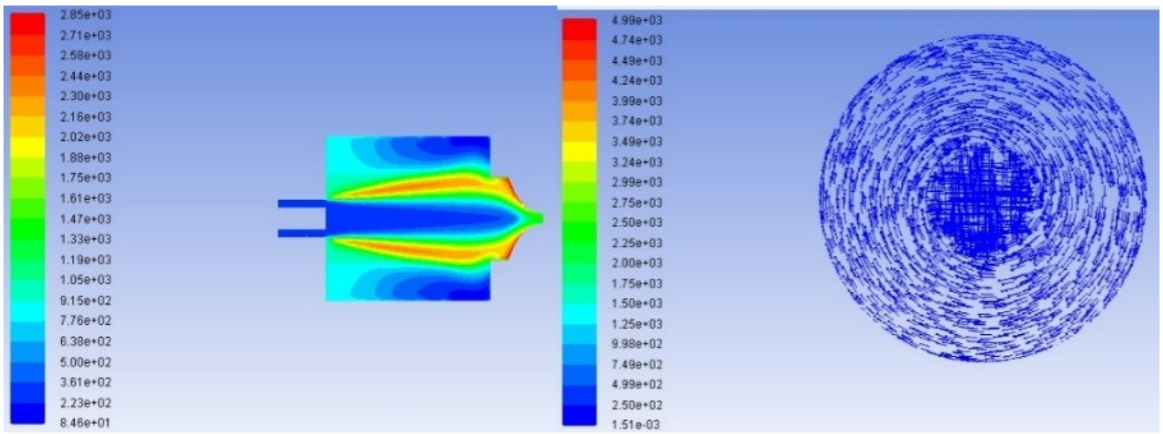

Figure 8: Flame structure and velocity vectors of configuration 2.

The third geometry featured a wall near the throat of the combustion chamber. This was done to ensure that the structure of the vortex could be created faster and stronger, as well as to confine the oxidizer from escaping radially into the nozzle. The third configuration can be seen in Figure 9. Again the baseline dimensions were retained.

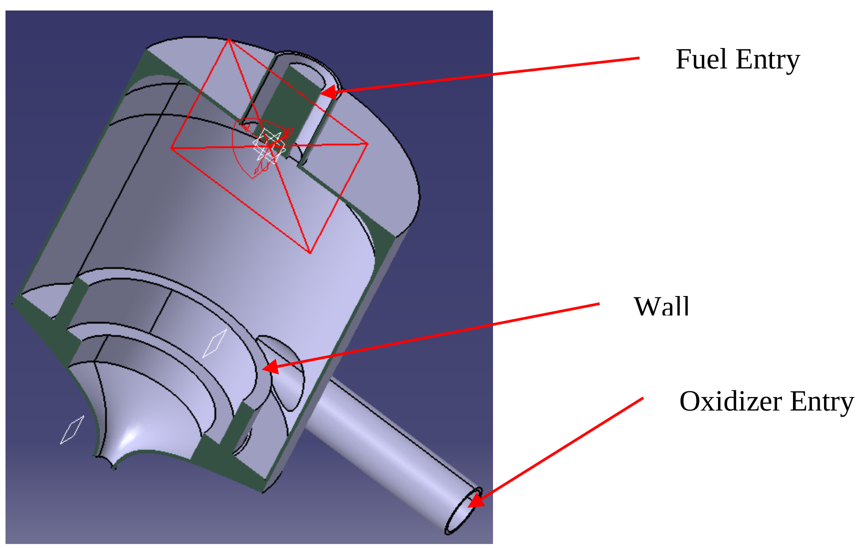

Figure 9: Configuration 3

Figure 10 characterizes the flow field within the combustion chamber of the third configuration. As shown in the figure, combustion was again confined to the inner region of the chamber as well as near the throat entrance. The maximum temperature recorded was 2873K. This is slightly higher than the temperature recorded for the second configuration which proves that the mixing was enhanced to some extent. Recurrently, it was observed that the temperature near the walls was lower than the flame temperature. This is shown in Figure 10, configuration 3, which featured a wall near the throat of the combustion chamber. The velocity registered was about 400 m/s. This was a consequence of the increased strength of the vortex.

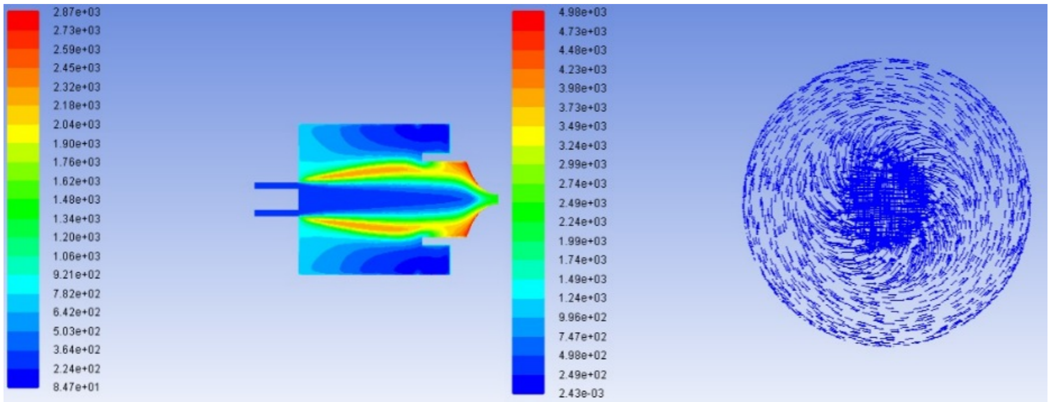

Figure 10: Flame structure and velocity vectors for configuration 3.

III. Conclusion

The results of this study affirm the reports of other scholars demonstrating that a bidirectional vortex enhances the efficiency of the combustion process of a liquid rocket engine. The simulation of different internal geometries, with the base line dimensions remaining unchanged, show that the temperatures at the chamber walls are lower due to the protective thermal blanket that the oxidizer creates as the combustion is confined to the inner region. This, therefore, eliminates the need for high-temperature withstanding materials along the walls, which reduces cost. The strength of the vortex is correlated with the degree of mixing and hence the maximum temperature attained in the combustion process. Future studies will explore the use of different geometries with the objective of further enhancing vortex strength, mixing, and maximum temperature in liquid rocket combustion chambers.

IV. Acknowledgements

I would like to express gratitude to the McNair Program for granting me the opportunity to perform this research. I would also like thank Professor Yongho Lee of the Mechanical Engineering Department at ERAU for his guidance in completing this research.

V. References

[1] Vyas A.B., M. J. *"Exact Solution of the Bidirectional Vortex"*. AIAA Vol. 44 No.10. October 2006.
[2] FLUENT ®. CFD software developed by FLUENT, Inc, (acquired by ANSYS, Inc). http://www.fluent.com
[3] Chiaverini, M. M. *"Vortex Combustion Chamber Development for Future Liquid Rocket Engine Applications"*. AIAA Paper 2002-2149. July 2002.
[4] Farmer R., C. G., C. YS. "*CFD Simulation of liquid rocket engine injectors. Part 3 simulation of the RCM-3 experiment."* 2nd International workshop on rocket combustion modeling, atomization, combustion, heat transfer. Lampoldshausen, Germany: Denfense Technical Information Center. 2001.
[5] Arshad A., K. M. *"Numerical Studies on Hydrogen Combustion in a Film-Cooled Cryogenic Rocket Engine"*. International Journal of Engineering Science and Technology (IJEST) Vol. 4 No. 07. July 2012.
[6] CATIA® (Version 5, Release 17). Developed by Dassault Systemes http://www.3ds.com/products-services/catia/
[7] George P. Sutton, O. B. *Rocket Propulsion Elements*. John Wiley and Sons. 2001.
[8] Pointwise ® (Version 17.0, Release 2). Developed by Pointwise. http://www.pointwise.com