Journal of Physics: Conference Series

OPEN ACCESS

CFD Modelling of a Quadrupole Vortex Inside a Cylindrical Channel for Research into Advanced Hybrid Rocket Designs

To cite this article: B Godfrey and J Majdalani 2014 *J. Phys.: Conf. Ser.* **548** 012067

Journal of Physics: Conference Series **548** (2014) 012067 doi:10.1088/1742-6596/548/1/012067

CFD Modelling of a Quadrupole Vortex Inside a Cylindrical Channel for Research into Advanced Hybrid Rocket Designs

B Godfrey[1] **and J Majdalani**[2]

[1] Department of Mechanical, Aerospace, and Biomedical Engineering, Ph.D. Candidate, University of Tennessee Space Institute, Tullahoma, TN 37388, USA
[2] Department of Aerospace Engineering, Auburn Alumni Engineering Council Endowed Professor and Chair, Auburn University, Auburn, AL 36849, USA

E-mail: bgodfrey@utsi.edu; joe.majdalani@auburn.edu

Abstract. This study relies on computational fluid dynamics (CFD) tools to analyse a possible method for creating a stable quadrupole vortex within a simulated, circular-port, cylindrical rocket chamber. A model of the vortex generator is created in a SolidWorks CAD program and then the grid is generated using the Pointwise mesh generation software. The non-reactive flowfield is simulated using an open source computational program, Stanford University Unstructured (SU2). Subsequent analysis and visualization are performed using ParaView. The vortex generation approach that we employ consists of four tangentially injected monopole vortex generators that are arranged symmetrically with respect to the center of the chamber in such a way to produce a quadrupole vortex with a common downwash. The present investigation focuses on characterizing the flow dynamics so that future investigations can be undertaken with increasing levels of complexity. Our CFD simulations help to elucidate the onset of vortex filaments within the monopole tubes, and the evolution of quadrupole vortices downstream of the injection faceplate. Our results indicate that the quadrupole vortices produced using the present injection pattern can become quickly unstable to the extent of dissipating soon after being introduced into simulated rocket chamber. We conclude that a change in the geometrical configuration will be necessary to produce more stable quadrupoles.

1. Introduction

Hybrid rocket motors are motors where the fuel and oxidizer are stored separately in two different states. The typical configuration for a hybrid rocket motor (Figure 1) has the oxidizer stored as a liquid or gas and the fuel stored as a solid. A hybrid rocket motor has many advantages over liquid and solid fuel rocket systems. Hybrids tend to exhibit a simpler design than liquid fuel systems, as they require only one pumping system for the oxidizer, whereas liquid rocket motors require two pumping systems, one for each of the fuel and the oxidizer. Hybrids also have a higher specific density than liquid rocket engines, which enables them to produce more thrust energy for a given rocket volume. Unlike solid rocket motors, however, hybrid rockets can be throttled to some degree, and they can be shut down before the fuel is used up, thus increasing its mission capability and overall safety. Since most fuels for hybrid motors are more stable than solid fuel grains, a hybrid motor can be stored for longer periods of time without worrying about the stability of the fuel grain. With the oxidizer being stored separately, the fuel grain is not capable of igniting under non-operating conditions as is the case with solid rocket motors.

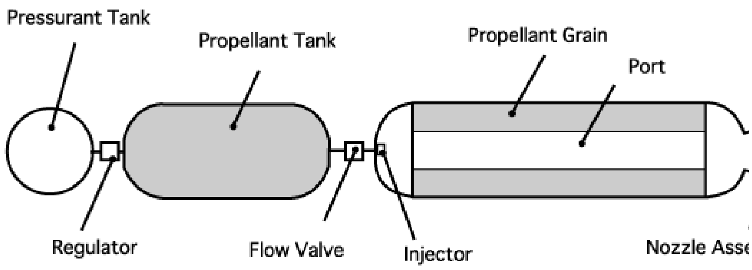

Figure 1. General hybrid rocket motor design.

1.1. Hybrid disadvantages

There are several disadvantages to hybrid rocket motors including the difficulty in ignition of the propellant and the low regression rate of the fuel grain. This paper is concerned with a flow configuration that seeks to enhance the regression rate of the fuel grain to the extent of improving the combustion efficiency and the thrust performance of the corresponding motor. To set the stage, several methods can be used to increase the fuel regression rate and thrust performance for hybrids. One such method consists of increasing the surface area of the fuel grain by incorporating multiple ports with a large exposed "wet" area at the beginning of the burn. However, this approach leads to complex fuel grains that are difficult to produce. Moreover, it leads to weaker grains as the grain can structurally fail during the firing, thus causing pieces of fuel to detach from the main grain, travel down the bore, and clog the nozzle entrance. Flow blockage can, in turn, result in motor detonation (e.g., the AMROC family of large scale motors). Increasing the fuel regression rate can also be accomplished by embedding energetic materials in the form of additives into the fuel grain. Most popular examples include ammonium perchlorate (oxidizer) and/or aluminum, which decrease the stability of the fuel grain and must be carefully formulated to mitigate the risks of self-ignition. Studies have shown that an addition of ammonium perchlorate and aluminum can lead to an increase of up to 170% in regression rate over unmodified fuel samples [1]. Another method for improving the regression rate consists of changing the internal ballistics of the gaseous motion within the motor by modulating the flow field. In this vein, a popular technique for enhancing the erosive capability of the mean flow while increasing the pressure at the fuel surface and prolonging the residence time of fuel particles within the chamber can be achieved by introducing a coaxially rotating, swirling oxidizer stream. At the outset, test results show that the introduction of unidirectional swirl can increase the regression rate by up to 200% as the swirl number is increased [2]. Nonetheless, the disadvantages of the unidirectional swirling flow technique are twofold. First, the angular momentum of the gas produces a torque on the rocket motor, which will have to be corrected using either active controls, or passive control surfaces on the airframe. Second, swirl may induce instabilities in the nozzle throat area where the angular momentum can cause the mass flow through the nozzle to decrease. This temporary disruption in effective mass flow can destabilize the vortex and lead to a pulsating motion throughout the motor. Third, by virtue of centrifugal action, the colder and, therefore, heavier gases are pushed outwardly against the grain walls whereas the lighter, hotter gases are pulled inwardly. This temperature stratification process reduces the flame temperature and, with it, the rate of heat transfer to the fuel grain. Naturally, suppressing the rate of heat transfer to the grain leads to a more sluggish burning rate.

1.2. Quadrupole vortex method

The method that is being investigated in this paper is a variation of the swirl/vortex method mentioned above (Figure 2). Instead of a single vortex, four vortices are set up within the flow in such a way that vortices are interlaced side-by-side to produce the same downwash at the interface between them (Figure 3). In this setup the angular momentum of each vortex is balanced by the presence of the other three, which eliminates the control issue and possibly the instability issues that are seen in single

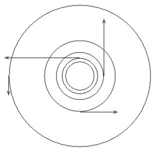

Figure 2. Monopole vortex.

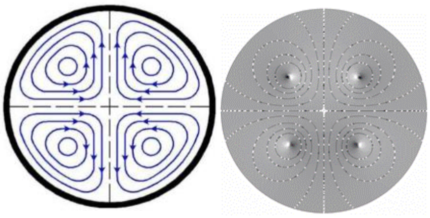

Figure 3. Quadrupole flow field.

vortex motors. It has also been shown that heat transfer rates can be improved by the introduction of circulatory motion within the fluid body, thus pulling gasses into the center of the chamber and allowing the hot gaseous products and, in some cases, the incoming oxidizer to mix vigorously before making contact with the walls of the chamber. As shown by Dake and Majdalani [3], heat transfer rates inside electronics enclosures can be visibly enhanced when the airflow across heat sinks is circulated using a quadruple vortex arrangement.

1.2.1 Motor design
A cold flow test article is currently being constructed to investigate the effectiveness of the quadrupole injection technique (Figure 4), especially that further analysis of the stability of the flow structure within the test article is needed. The experimental motor that is being evaluated is designed with 4 tangentially injected monopole vortex generators that are then inserted into the chamber in such a way to set up the quadruple vortex motion that we call “quadrupole.” Through this arrangement, the flow first enters a reservoir in the injector assembly (Figure 5) so that a constant pressure can be maintained through the injector holes. The injector holes, located on the injector plate (Figure 6), are used to inject the oxidizer tangentially into the monopole vortex tubes. The latter consist of cylinders where the monopole vortex is allowed to develop. The vortex generator outlets into the main combustion

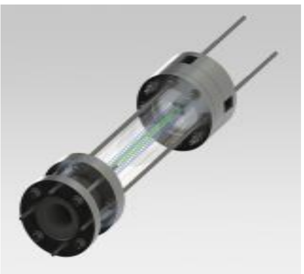

Figure 4. Motor design.

Figure 5. Injector assembly.

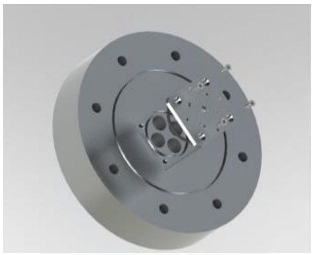

Figure 6. Injection plate.

chamber so that vortices, which are produced in a quad formation, rotate in opposite directions. This configuration will ensure that the interface between each vortex flows in the same direction, thus giving rise to a case of common downwash. The gas stream then flows across the combustion chamber where it will spiral around, burn with the fuel grain, and finally exit through the nozzle.

2. CFD methodology

2.1. Geometry
The geometry is generated by importing a simplified representation of the flow passages into the full motor design above into Pointwise (Figure 7). Surfaces are then created at the inlets and outlets of the flow path in order to close the path and generate a volume for the space that fills the flow passage. The rest of the structure is then removed from the geometric file since it is no longer needed in the remainder of the analysis. The resulting volume (Figure 8) is subsequently used to generate the mesh.

Figure 7. Simplified geometry.

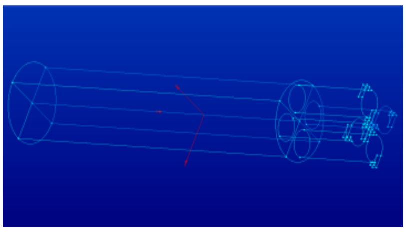

Figure 8. Geometry volume.

2.2. *Grid Generation*
The surfaces in the geometry are filled with unstructured grids (Figure 9). Baffles are used within the main chamber in order to refine the grid where the vortices interact (Figure 10). The walls in the analysis are assumed to be frictionless; therefore, an inviscid grid is generated within the volume. The resulting grid corresponds to an inviscid flow with frictionless walls (Figure 11).

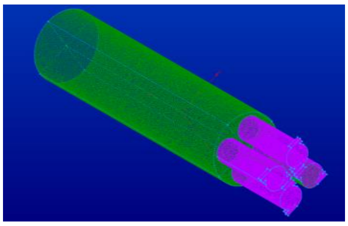

Figure 9. Surface grid.

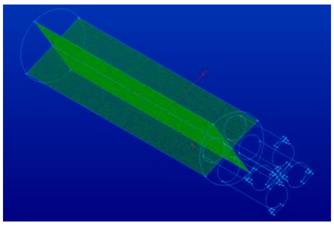

Figure 10. Baffle grid.

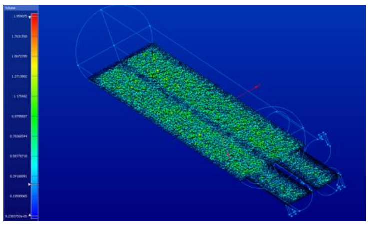

Figure 11. Grid slice.

2.3. *Boundary conditions*
The inlet to the problem is comprised of eight surfaces (Figure 12). The inlet condition consists of an inlet boundary, with a total temperature of 288K, a total pressure of 3.7MPa, and a velocity direction normal to the inlet surface. The outlet corresponds to the surface depicted at the end of the chamber tube (Figure 13). The outlet condition is set to be a pressure outlet with a prescribed static pressure of 3.5 MPa. The remaining surfaces are inviscid Euler walls.

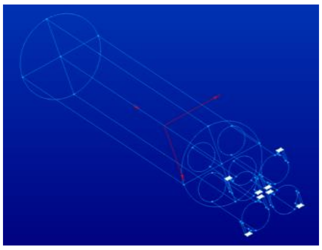

Figure 12. Inlet surfaces.

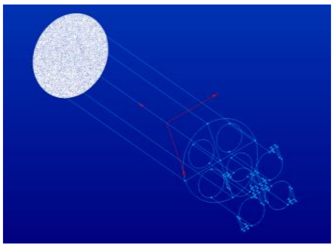

Figure 13. Outlet surface.

2.4. *Numerical analysis*
The second-order simulations were found to be too unstable to be solved directly from the initial conditions given above. Low pressure points would develop near the outlet, thus causing reverse flow through the outlet. For this reason, a first-order analysis was performed on the geometry using the ROE first-order method, and a CFL number of 4. The solution in this case was found to converge, with the residual decreasing by six orders of magnitude from its initial value, namely, with a final value of $10^{-11.2}$ for the Rho residual and $10^{-10.15}$ for the RhoE residual. The solution for the first-order case was then used as the initial condition for the JST Method with a CFL number of 4. This solution only converged to $10^{-7.83}$ for the Rho residual and $10^{-2.62}$ for the RhoE residual, due to the unsteady nature of the flow.

2.5. Data analysis

The data is analyzed using ParaView and the flow field is carefully examined to determine whether the desired flow configuration is achieved within the chamber. The difference in the *y*-component of the velocity on a cut plane perpendicular to the *y*-direction can be investigated to determine the intensity of the vortex at that point. Also vector plots of the velocity can be created on cut sections in the *z*-plain at different distances away from the inlet to the main chamber. These plots enable us to visualize the intensity and shape of the vortices within the chamber.

3. Results

Strong vortices are generated in the monopole vortex tubes as shown by the difference in the axial *x*-velocity on a section cut in the *x*-plane (Figure 14). However, there is dissipation of the vortices before they empty into the main chamber. We thus find that shortening of the monopole tubes will allow for a stronger vortex to enter the main chamber. However, the vortex generated in the monopole

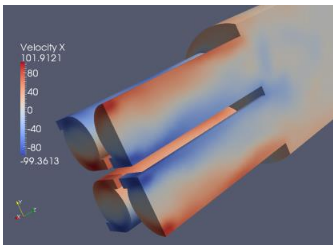

Figure 14. Axial velocity on section cut.

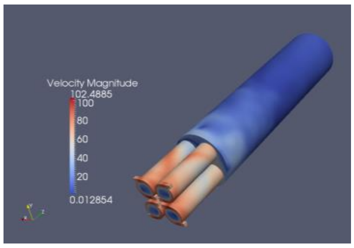

Figure 15. Velocity magnitude on surface.

tubes does not seem to be consistent due to the limited number of injectors creating a vortex at the injection tube outlet where the flow goes through a sudden expansion (Figure 15). This arrangement leads to instabilities in the flow after it has emptied into the main chamber, thus distorting the flow from its intended configuration. The dissipation of the quadrupole vortices soon after entering the main chamber can be seen in the destabilization of the individual vortices in cross-streamwise slices of the main chamber (Figure 16).

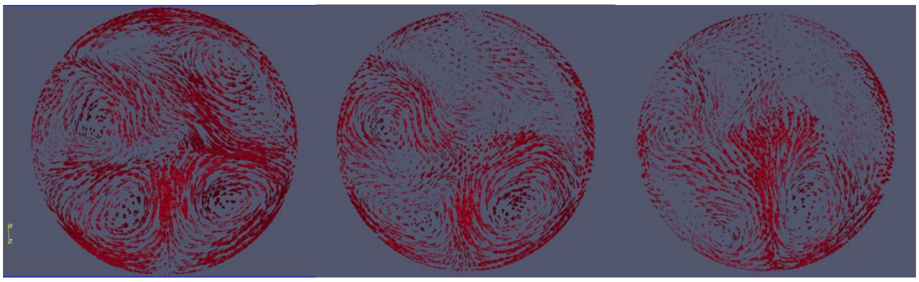

Figure 16. Flow pattern at 5mm, 10mm, and 15mm from chamber inlet.

4. Conclusions

In this paper, a basic simulation approach was considered to model the flow field associated with an innovative, quadrupole motor design. More work needs to address the geometry and boundary conditions that produce more stable quadrupole vortices that will remain coherent throughout the chamber. A significant amount of work must be performed to simulate this hybrid rocket motor thoroughly.

Future work will include modifying the geometry in such a way to produce smoother monopoles, while shortening the monopole tubes to create more intense monopoles. Then viscous walls will be added to the simulation to determine the effect of wall losses on the flow. Subsequently, the effects of wall pumping due to the sublimation of the fuel grain will be added by making the chamber walls porous, thus capable of injecting gas across the simulated surface boundary. A reacting flow simulation can also be carried out to study the heat transfer rate to the surface of the fuel grain. Then the porous boundary layer can be modified to capture the heat transfer rate that determines the regression rate and hence the mass flow rate at which the fuel may be injected into the chamber across the grain surface. Finally, the boundary layer analysis can used to model the physical regression of the fuel grain.

Acknowledgments
This material is based on work that was supported partly by the National Science Foundation and partly by the Auburn Alumni Engineering Council Endowed Professorship, Auburn University.

References

[1] Chiaverini M J, Serin N, Johnson D K, Lu Y-C, Kuo K K and Risha G A 2000 *Journal of Propulsion and Power* **16** 125
[2] Lee C, Na Y, Lee J-W and Byun Y-H 2007 *Aerospace Science and Technology* **11** 68
[3] Dake T J and Majdalani J 2009 Improving flow circulation in heat sinks using quadrupole vortices *Proceedings ASME 2009 InterPACK Conference collocated with the ASME 2009 Summer Heat Transfer Conference and the ASME 2009, 3rd International Conference on Energy Sustainability*: *American Society of Mechanical Engineers* pp 463-70