

Cold flow simulation of vortex shedding in a segmented solid rocket motor

Durojaye R.O ^{1,*}, Ayegba T.A ², Aremo E.A ² and Gbajabiamila B.A ²

¹ Projects, Planning and Strategic Development Division, Bola Ahmed Tinubu Centre for Space Transport and Propulsion, Epe Lagos Nigeria.

² Rocket Systems Engineering Division, Bola Ahmed Tinubu Centre for Space Transport and Propulsion, Epe Lagos Nigeria.

Global Journal of Engineering and Technology Advances, 2026, 28(01), 061–075

Publication history: Received on 28 May 2026; revised on 05 July 2026; accepted on 09 July 2026

Article DOI: <https://doi.org/10.30574/gjeta.2026.28.1.0173>

Abstract

Combustion instabilities are a major problem in large solid rocket boosters. They are responsible for exciting undesirable oscillations in the propellant burn rate. It is believed that vortex shedding from upstream inhibitors is one of the phenomena responsible for the onset of these instabilities. Vortex shedding frequencies are time dependent due to the unsteady nature of the flow inside the rocket motor. When one of these frequencies coincides with a structural frequency, the oscillations can increase substantially. The protruding inhibitors are caused by their slow burn rate compared to that of the fuel. In the current research, cold flow simulations were carried out to understand the problem of vortex shedding. Results were generated for a fixed baffle diameter and distance between inhibitors while varying the flow velocity. Six different flow conditions were used starting from 70 gallons per minute to 120 gallons per minute, while vortex shedding distances corresponding to each flow configuration were measured in order to determine the Strouhal number. The Strouhal number for the various runs was found to be in the range of 0.19 to 0.28, which is slightly higher than that of previously published experiments of 0.185 to 0.20.

Keywords: Inhibitors; Oscillations; Propellant; Vortex; boosters; Baffles; Visualization

1. Introduction

In large solid rocket motors, propellant surfaces at the inter-segment joints are often covered with annular inhibitors which prevent burning at that face in order to get the desired thrust versus time characteristics. Initially, the inhibitors are almost flush with the propellant inner bore circumference. As the motor is fired, the inhibitors protrude into the exiting gas flow path because they do not burn as fast as the propellant. This eventually leads to the creation of vortices within the chamber cavity at different frequencies which could coincide with the acoustic frequency of the motor cavity leading to undesirable conditions in large solid rocket motors. These large solid rocket motors, in most cases, carry very sensitive payloads when used as launch vehicles or war heads when used as missiles, which could be destroyed as a result of these oscillations due to vortex interactions within the motor cavity.

“Water analogue are often used to simulate the flow of air and other gases around or through propulsion devices. Water is an easily obtainable, nontoxic fluid which lends itself readily to the introduction of dye and particles for flow visualization. Scaling laws are such that a relatively slow flow in water can accurately simulate a much more rapid gas flow allowing easy observation of time dependent phenomena” [1].

The Reynolds number is a primary scaling parameter used to ensure similarities between water flow models and their real gas counterparts. It expresses the ratio of inertia to viscous forces and determines the likelihood of a fluid existing in turbulent or laminar states in terms of microscopic-scale molecular motion [1].

* Corresponding author: Durojaye R.O

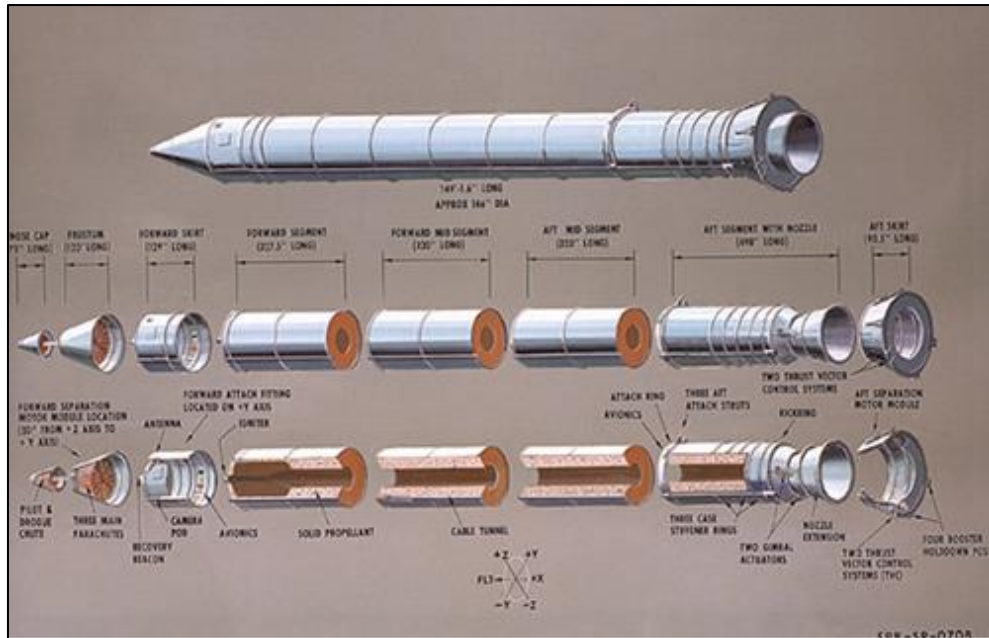


Figure 1 Internal Cavity of a Large Solid Rocket Booster [2]

2. Material and methods

A number of large solid propellant rocket motors, including the space shuttle, Titan and Arian 5 solid boosters, exhibit instabilities due to vortex shedding driven acoustic pressure oscillations [3]. This research work is to determine the effects of vortex shedding on the thrust oscillation of large solid rocket motors using cold flow simulation experiment techniques. Cold flow water tunnel facilities was used to simulate the internal cavity of a large solid propellant rocket motor in order to generate data for the flow simulation experiment.

Vortex shedding induced acoustic responses are observed from a variety of geometric and flow configuration combinations. Solid propellant rocket flow past two inhibitors is modeled in a cold flow cylindrical water tunnel test section with thin annular baffles used to simulate the inhibitors.

The objective of this research work is to determine the effects of vortex shedding on thrust oscillations of large solid rocket motors as a function of the vehicle internal geometry and velocity using a water tunnel facility to simulate the internal conditions of a large solid rocket booster.

The following variations can be made in the test configuration in order to generate data for different vortex shedding frequencies and velocity profiles in the test section.

- The spacing between baffles can be varied to excite different vortex shedding frequencies.
- Flow velocities past the baffles were varied to excite different vortex shedding frequencies.
- The location of baffle pairs in the test section can also be varied to excite different acoustic responses of the test cavity.
- The Baffle diameter can be varied to excite different vortex shedding frequencies.

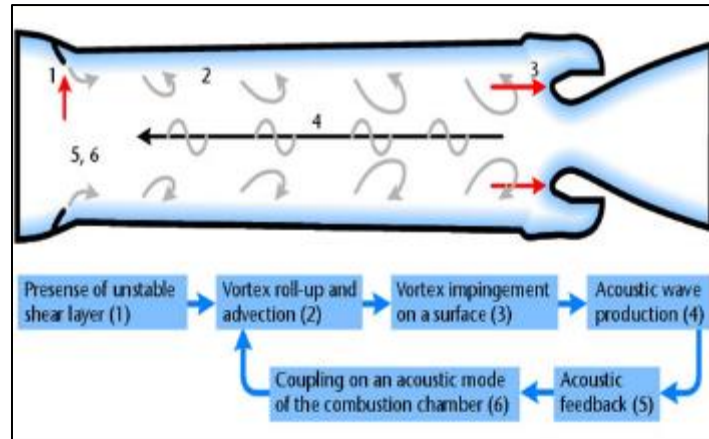


Figure 2 Vortex Interactions in Rocket Motor

Water Tunnels have been utilized in one form or another to explore fluid mechanics and aerodynamic phenomena for a very long time. Only in recent years, however, have Water Tunnels been recognized as highly useful facilities for critical evaluation of complex flow fields on many modern vehicles such as high performance aircraft [4,5].

In particular, Water Tunnels have filled a unique role as research facilities for understanding the complex flows dominated by vortices and vortex interactions. Flow visualization in Water Tunnels provides an excellent means for detailed observation of the flow around a wide variety of configurations. The free stream flow and the flow field dynamics are low-speed, allowing real time visual assessment of the flow patterns using a number of techniques, including dye flow through ports in the model, hydrogen bubble generation from strategic locations on the model, or laser light sheet illumination [4].

Comparison of flow visualization in water and wind tunnel experiments on a common model shows nearly identical vortex flow patterns, clearly illustrating that Water Tunnel observations have direct application to aerodynamics [4].

Water tunnel facilities of various sizes and capacities have been used to simulate the internal conditions of large solid rocket motors with a view to obtaining relevant data that could be used to validate CFD codes.

The water tunnel facility for the cold flow experiment comprises the following equipment and components.

The run tank is a 300 gallons capacity tank that holds water which runs through the test section. It is equipped with a stop switch which cuts off the return pump when it is full, in order to avoid overflow. A 25 hp capacity pump, Figure 4 (main pump), pumps water from the run tank through the test section. The flow rate of water flowing through the test section is measured using a magneto-hydrodynamic (MHD) flow meter in gallons per minute, Figure 6. The water then flows past the inhibitors in the test section, Figure 9. The water exiting the test section goes directly into the catch tank which is a 600 gallon capacity tank below the test stand. A 1.5 Hp return pump, Figure 8, is used to pump water from the catch tank back into the run tank, making the system a closed cycle system. The filter, Figure 7, is used to remove unwanted particles and dirt from the entire system so as to ensure running the test with clean water. The video camera was also interfaced with the computer, and it was used to capture images and record flow conditions while carrying out an experiment. The computer was used for computation and data analysis, and also an interface between equipment within the test facility.

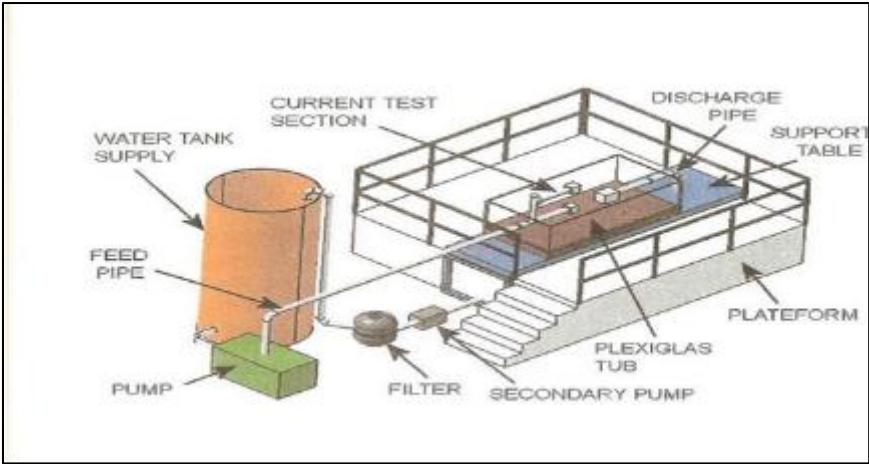


Figure 3 Water Tunnel Facility Schematic [5]



Figure 4 Test Facility



Figure 5 Outlet Valve



Figure 6 MHD Flow Meter



Figure 7 Filter



Figure 8 Return Pump



Figure 9 Test Section



Figure 10 Discharge Valve



Figure 11 Pump Control Switch

Before an attempt was made to carry out the experiment, it was ensured that a physical observation and inspection of facilities for any abnormalities within the facilities and surroundings was carried out, e.g., water leakage, improper placement of tools, oil spillage etc.

2.1. Experiment

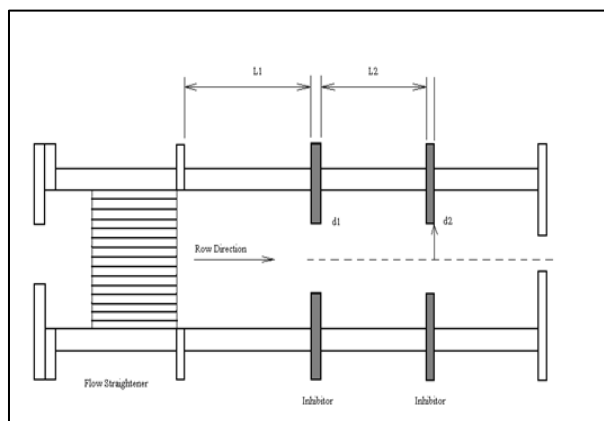


Figure 12 Schematic Diagram of the Test Section

Water flows past the MHD flow meter at a certain flow rate measured in gallons per minute through the flow straightener before coming across the upstream baffle. The pump speed regulator is adjusted high or low in order to increase or decrease the flow rate of water passing through the section. Figure 13 shows the image of the flow pattern with dye injection, while Figure 14 shows the image of the flow pattern with particle injection.

The flow rate of water in gallons per minute is then converted to velocity using the geometry of the test section by dividing the volume flow rate by the test section's area. The experiment was carried out in sequence, starting with 70 gallons per minute up to 120 gallons per minute with an interval of 10 gallons per minute.

Flow pattern images are obtained using dye and fine particles at different velocities. The laser is turned on to produce a red light while a filter is attached to the camera when injecting particles. The use of filter is not necessary when injecting dye into the test section. Particles unlike the dye are more visible at high velocities and are more preferred to the dye since the dye tends to vanish faster at high velocities. Vortex shedding distances are measured in order to determine the shedding frequency corresponding to the flow velocity, this is then used to obtain the Strouhal number corresponding to the flow velocity.

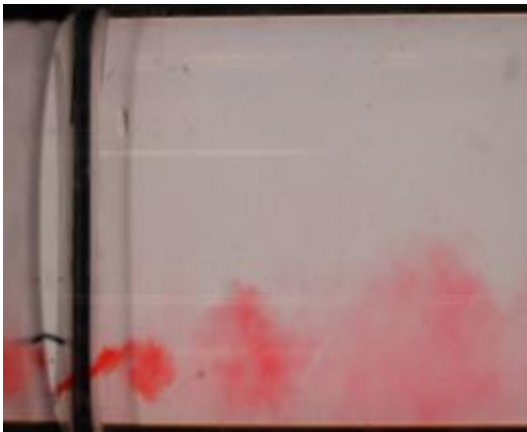
	
Figure 13 Internal Cavity of Test Section with Dye Injection for Flow Pattern Visualization	Figure 14 Flow Pattern Image Obtained Using particle injection for Flow Pattern Visualization



Figure 15 20 mw Coherent Laser

The laser is a 20 mw capacity, and it was used to produce a red light through the test section for the purpose of flow pattern visualization.

2.2. Image Visualization

Visualizing images and flow patterns in the test section can be quite difficult and cumbersome due to light scattering effects. Images are clearer and more accurate when viewed at an angle compared to when viewed from a vertical

position at 90°. The use of a high powered laser was employed in order to capture a clear image of flow pattern and vortices along the flow line. The power of the HeNe (2mW) laser that was initially used was not enough to produce a clear and visible image of the flow pattern. The laser was then changed to a 20mW coherent laser that was able to produce a clearer image. Basically, the use of the dye injection and particle injection was adopted in this experiment for flow pattern visualizations.

2.3. Light Scattering

Scattering is a general process where some forms of radiation such as light, sound, or moving particles are forced to deviate from a straight trajectory by one or more localized non-uniformities in the medium through which they travel. When reflections undergo scattering, they are called diffused reflection while un-scattered reflections are called specular (mirror-like) reflection [6].

The scattering effect is one of the major problems in the flow visualization process as it always affects the clarity and accuracy of the image obtained [7,8]. Basically, there are three main light scattering phenomena based on a dimensionless size particle, α , which is defined as;

$$\alpha = \pi D_p / \lambda \quad \dots\dots\dots (1)$$

Where πD_p is the circumference of a particle and λ is the wave length of incident radiation. Based on the value of α , these three domains are:

$\alpha \ll 1$: Rayleigh scattering where particles are small compared to the wave length of light.

$\alpha \approx 1$: Mie scattering where particles are about the same size as the wave length of light.

$\alpha \gg 1$: Geometric scattering where particles are much larger than the wave length of light.

The degree of scattering varies as a function of the ratio of the particle diameter to the wavelength of the radiation, along with other factors including polarization, angle, and coherence [9].

$$f = \frac{S_t V}{d} \quad \dots\dots\dots (2)$$

$$S_t = \frac{f d}{v} \quad \dots\dots\dots (3)$$

where

f is the Vortex shedding frequency (Hz)

S_t is the Strouhal number (dimensionless)

v is the flow velocity and

d is the baffle diameter (characteristic length)

$$v = d / t \quad \dots\dots\dots (4)$$

$$v / d = 1 / t$$

$$v / d = 1 / t = f \text{ (Hz)} \quad \dots\dots\dots (5)$$

3. Results and discussion

Several tests were carried out using the water tunnel facilities at the Propulsion Research Center; results and data obtained were analyzed; various graphs and charts were also plotted in order to verify if results obtained match existing analytical or experimental solutions.

3.1. Water Tunnel Configuration

The water tunnel facility used for this research work has the configurations listed in Table 1.

Table 1 Test Facility Configurations

Total length of test section	76 inches	1.93m
Diameter of test cylinder	6.5 inches	0.1651m
Diameter of upstream baffle	3 inches	0.0762m
Diameter of downstream baffle	3 inches	0.0762m
Distance between upstream and downstream baffles	12 inches	0.3048m

3.2. Flow Conditions

Water is admitted into the test section at a regulated volume flow rate in gallons per minute. This is then converted into velocity per second using the geometry of the water tunnel. Flow velocities past the baffles were varied in order to excite different vortex shedding frequencies.

3.3. Strouhal Number

The Strouhal number is a non-dimensional quantity that can be determined using the measured vortices generated across the upstream baffle (inhibitor), and the frequency generated during a particular flow condition.

3.4. Flow Rates and Velocity

The main pump pumps in water through the test section in gallons per second as measured by the flow meter; this is then converted to velocities using the measured vortices and the test section geometry.

Area of the test section is

$$A = \frac{\pi d^2}{4} \dots \dots \dots (6)$$

where A = area of the test section

d = diameter of the cylinder = 0.1651m

$$A = 0.0214m^2$$

$$v = \frac{Q}{A} \dots \dots \dots (7)$$

3.5. Flow meter readings at different flow rate.

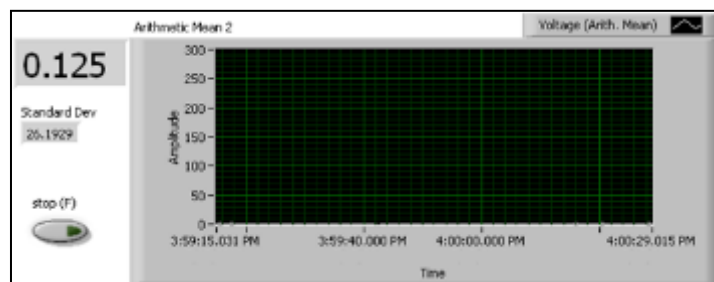


Figure 16 Flow Meter Digital Reading 0 Gallons per Minute

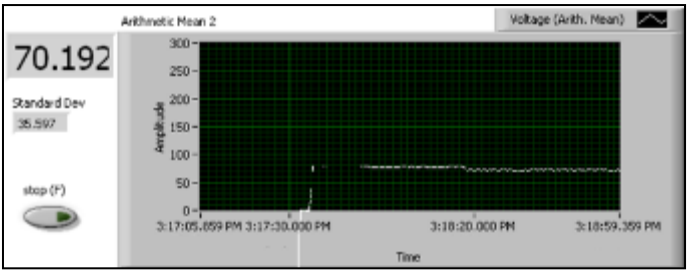


Figure 17 Flow Meter Reading 70 Gallons per Minute

This is where the particle or die injection is done when reading stabilizes on the desired flow level before the measurement is taken.

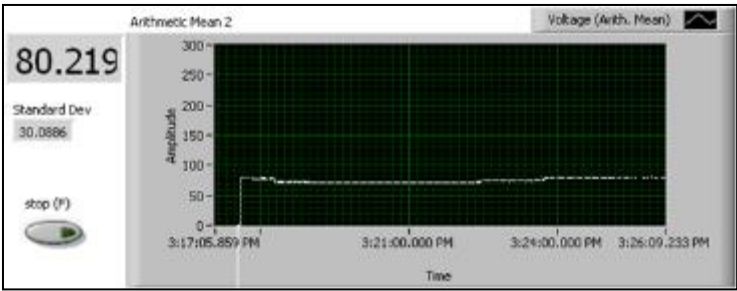


Figure 18 Flow Meter indicating a Volume Flow Rate of 80 gallons per Second

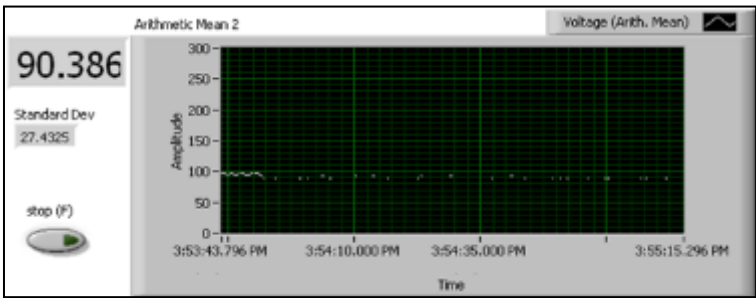


Figure 19 Flow Meter Reading at a Volume Flow Rate of 90 gallons per minute

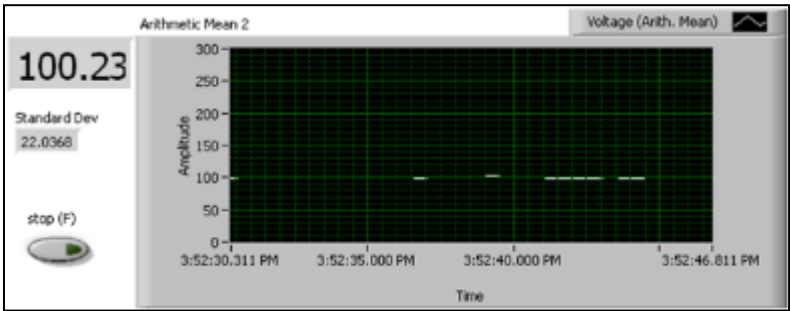


Figure 20 Flow Meter Reading at a Volume Flow Rate of 100 Gallons per Minute

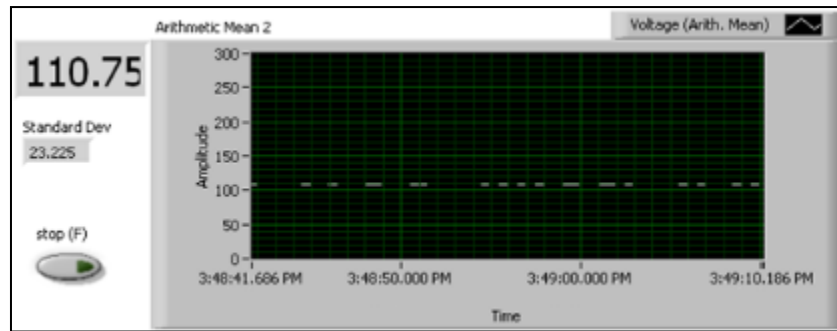


Figure 21 Flow Meter Reading at a Volume Flow Rate of 110 Gallons per Minute

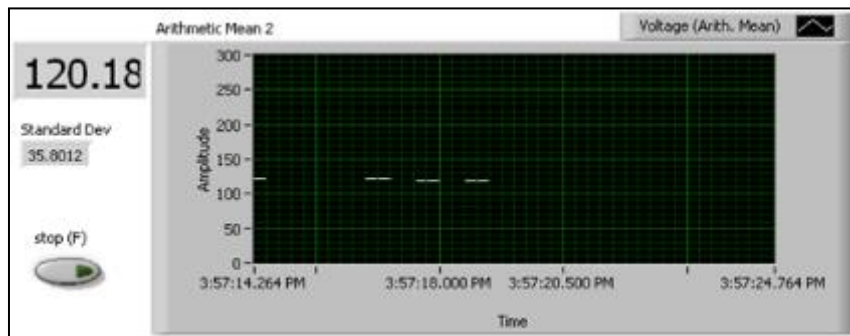


Figure 22 Flow Meter Reading at a Volume Flow rate of 120 Gallons per Minute

3.6. Flow pattern corresponding to the different flow rate are shown in the diagrams below.

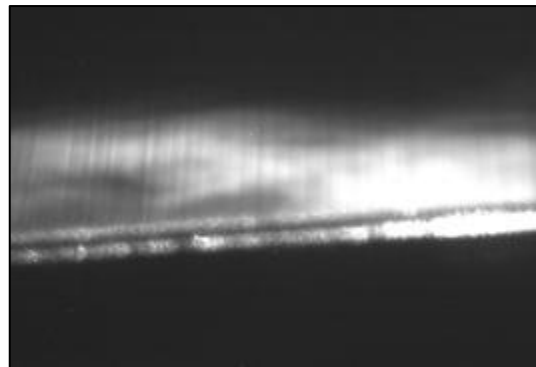


Figure 23 Flow Pattern at a Flow Rate of 70 gallons per Minute

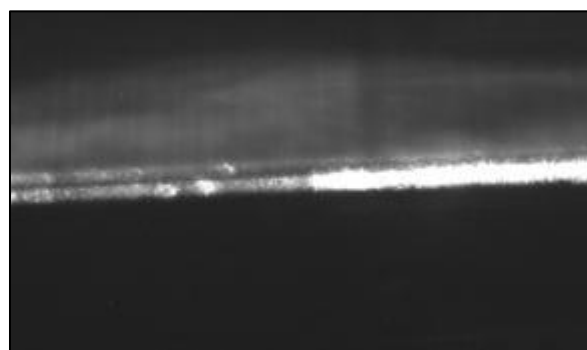


Figure 24 Flow Pattern at a Flow rate of 80 Gallons per Minute

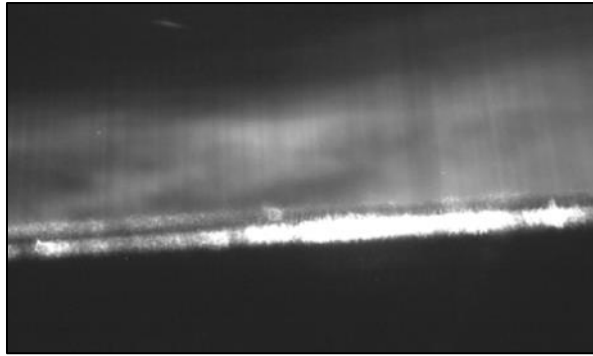


Figure 25 Flow Pattern at a Flow Rate of 90 Gallons per Minute

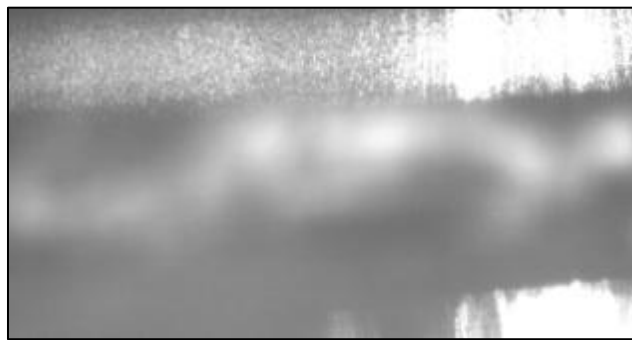


Figure 26 Flow Pattern at a Flow Rate of 100 Gallons per Minute



Figure 27 Flow Pattern at a Flow Rate of 110 Gallons per Minute



Figure 28 Flow Pattern at a Flow Rate at 120 Gallons per Minute

$$v_1 = \frac{4.414 * 10^{-3} \text{ m}^3}{0.0214 \text{ m}^2 \text{ s}}$$

$$v_1 = 0.21 \text{ m/s.}$$

Using equation (5),

$$f = v / d$$

where

f is the frequency

d is the measured vortex length

and

v is the flow velocity

$$f_1 = \frac{0.21 \text{ m}}{0.27 \text{ s}} * \frac{1}{\text{m}} = \frac{1}{\text{s}}$$

$$f_1 = 0.78 \text{ Hz}$$

$$S_{t1} = \frac{fd}{v}$$

$$S_{t1} = 0.28$$

The above calculations were carried out for the six different runs; the data obtained are tabulated in the tables below.

Table 2 Flow Velocity

v_1	0.21 m/s
v_2	0.24 m/s
v_3	0.27 m/s
v_4	0.30 m/s
v_5	0.32 m/s
v_6	0.35 m/s

Table 3 Frequency

f_1	0.78 Hz
f_2	0.80 Hz
f_3	0.82 Hz
f_4	0.85 Hz
f_5	0.86 Hz
f_6	0.89 Hz

Table 4 Stouhal Number

S_{t1}	0.28
S_{t1}	0.25
S_{t1}	0.23
S_{t1}	0.21
S_{t1}	0.20
S_{t1}	0.19

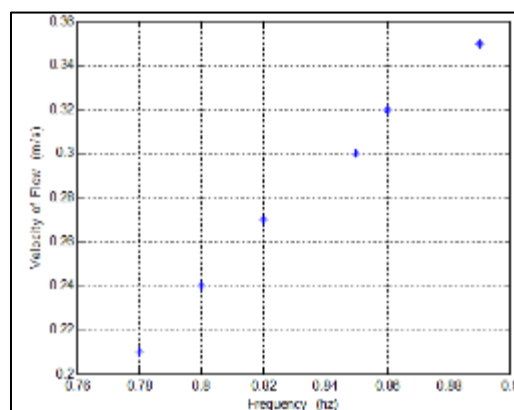
Table 5 Summary of Data Obtained

Flow rate (gallons/min)	Velocity (m/s)	Frequency (Hz)	Strouhal number
70	0.21	0.78	0.28
80	0.24	0.80	0.25
90	0.27	0.82	0.23
100	0.30	0.85	0.21
110	0.32	0.86	0.20
120	0.35	0.89	0.19

Table 5 shows the summary of the data obtained for the six runs. The flow rate of water through the test section was the controlled parameter, while the frequency was determined from the measured vortex shedding distance; the Strouhal Number was calculated from the relating equations.

3.7. Graphs

Various graphs were plotted to verify the relationship between the different parameters.

**Figure 29** Graph of Velocity against Frequency

This is a plot of velocity against frequency; it shows an almost linear relationship between velocity and frequency.

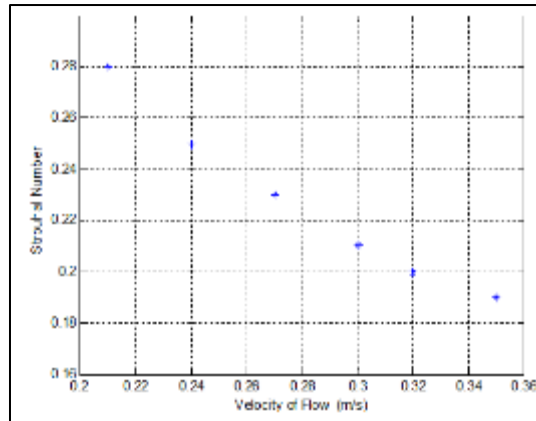


Figure 30 Graph of Strouhal Number against Velocity

Since the Strouhal number is a function of the Reynolds number, it means the vortex interaction is only pronounced at a certain range of Reynolds numbers, typically $20 < Re < 20,000$. This is why vortex shedding was not pronounced at lower velocities and at a velocity where the Reynolds number is greater than 20,000.

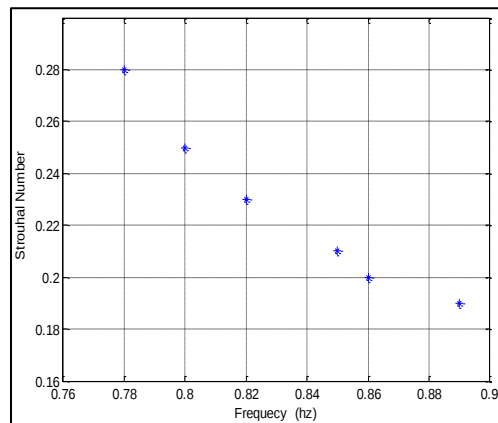


Figure 31 Graph of Strouhal Number against Frequency

Since frequency has an almost linear relationship with velocity, vortex shedding is expected at a frequency range that is similar to the velocities.

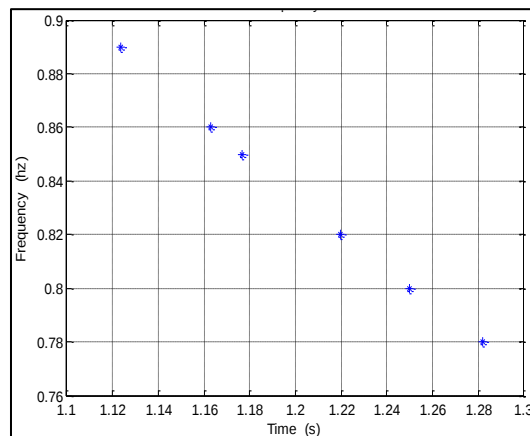


Figure 32 Graph of Frequency against Time

Figure 32 indicates a drop in frequency as time increases. The chamber cavity increases as the propellant burns out; this led to a decrease in velocity, which has an almost linear relationship with the frequency.

4. Conclusion

The process of visualizing flow pattern in a water tunnel experiment is not a simple process, as it involves the use of sophisticated and high powered lasers and lenses.

The facilities used for this experiment were able to generate data to justify numerically and experimentally determined data by researchers who have done similar work in the past. Some of the results from this research work have shown that vortex shedding only occurs at a certain range of Strouhal number, $185 < St < 0.20$. This means that there are certain velocities below and above which vortex interactions will not take place. This has also justified the relationship between Strouhal number and Reynolds number which is also a function of fluid velocity.

Compliance with ethical standards

Acknowledgments

We sincerely appreciate the contribution of Dr. Marlo Moser for his immense contribution and guidance on experimental set up during this research work.

Disclosure of conflict of interest

There are no conflicts of interest.

References

- [1] Waeche R.H, Woodward Sarrgent, William H. and Marchman J.F. Space Shuttle SRM Internal Flow, AIAA/ASME/SAE/ASEE, 24th Joint Propulsion Conference, July 11-13, 1988.
- [2] Alison F, VanMoorhem W.K. Vortex Shedding Induced Sound Inside a Cold-Flow Simulation of Segmented Chamber. Journal of Propulsion and Power, Vol.19, No.2, March-April, pp 287-295, 2003.
- [3] Alison F, VanMoorhem W.K. Prediction of Vortex Shedding Responses in Segmented Solid Rocket Motors. AIAA-90-2073, AIAA/SAE/ASME/ASEE 26th Joint Propulsion Conference, July16-18, 1990.
- [4] Rolling Hills Research Corporations, Water tunnel.
- [5] Oliver D. Water Tunnel Facility Design for External Flow Visualizations, Department of Mechanical and Aerospace Engineering, University of Alabama in Huntsville USA, 2010.
- [6] Brown R.S, Dunlap R, Young S.W, and Waugh R.C. Vortex Shedding as a Source of Acoustic Energy in Segmented Solid Rockets. Journal of Spacecraft, Vol.18, No.4, 1981, pp 312-319.
- [7] ISShu P.H, Sforzini R.H, and Foster W.A.Jr. Vortex Shedding from Solid Rocket Propellant Inhibitors, AIAA-86-1418, AIAA/ASME/ASEE 22nd Joint Propulsion Conference, 1986.
- [8] Flandro G.A. Vortex Driving Mechanism in Oscillatory Rocket Flows. Journal of Propulsion and Power, Vol. 2, No. 3, 1986, pp 206-214
- [9] <http://en.wikipedia.org/wiki/Scattering>.