Comparing the Thrust for Different Nozzle Shapes in Model Water Rockets

Haris Amin
Department of Physics, Wabash College, Crawfordsville, IN 47933
(Dated: March 15, 2007)

This paper illustrates the use of a numerical model with a finite-element method implementation to qualitatively describe the relationship between the length of a nozzle in a model water rocket and the thrust it exerts. We found that constricting the flow of the exiting water in the direction normal to the nozzle by increasing the length of the nozzle allows more thrust to be achieved.

Model water rockets are often used to demonstrate some basic principles of physics such as thrust, air drag, and projectile motion. A model water rocket traditionally consists of a chamber (bottle), filled with a certain amount of water, whose pressure is increased via an air pump till the thrust of the pressurized air and water exiting the nozzle force it to launch. What may seem as quite a simple system is actually fit to provide a wealth of knowledge about the physics of rockets and thrusts to elementary school and undergraduate physics studies alike. In this Letter, we will be providing a numerical model that we developed to observe the effects of the nozzle length on the thrust of a model water rocket. We also validate our numerical model by comparing its results to experimental data.

Our numerical model is based on Navier-Stokes fluid flow. Navier-Stokes equations seek to establish relations among the rates of change or fluxes. They establish that changes in momentum infinitesimal volumes of a fluid are simple the product of changes in pressure and dissipative viscous forces acting inside a fluid. Navier-Stokes equations are written as,

\[ \nabla \cdot \mathbf{u} = 0, \]  

(1)

\[ \mu_s \Delta \mathbf{u} + Re(\mathbf{u} \cdot \nabla \mathbf{u}) + \Delta p - \nabla \cdot \mathbf{\tau} = \mathbf{f}, \]  

(2)

where \( \mathbf{u} \) describes the velocity profile of the fluid, \( \mu_s \) describes the viscosity of the fluid, \( \mu_s \) describes the turbulence of the system, \( \mathbf{\tau} \) describes the stress tensor in the fluid, \( p \) describes the pressure in the system, and \( \mathbf{f} \) describes the forces acting on the system as a whole. Equation 1 mainly insures that mass is conserved in the system. These differential equations allow you to describe the balance of forces acting at any given region in the fluid. In the case of an ideal fluid, zero viscosity, these equations state that the accelerations is proportional to the derivative of internal pressure. A model water rocket is essentially such a system.

Our system is essentially a nonlinear system. Our numerical solver will involve a finite element method with least-squares implementation. This method allows us to approximate a finite space with piecewise polynomials.
This basically enables us to approximate a nonlinear system with polynomial approximations. We use quadratic polynomial approximation for our numerical model.

![Diagram of fluid flow contraction problem](image)

**FIG. 1:** This model illustrates a fluid flow contraction problem. The fluid is given an initial velocity profile. In this case we gave our fluid a parabolic velocity profile. The fluid passes from the first channel to the thinner second channel where its rate of flow increases. The numerical model also ensures that mass is conserved. Note that to simulate a water rocket with different nozzle lengths, the numerical model sampled velocity profiles at two locations, L and 1.5 L.

The domain of our numerical model consists of a large chamber and a smaller chamber resembling a model water rocket. The large chamber represents the body of the water rocket while the small chamber represents the nozzle of the rocket. Figure 1 illustrates our contraction problem. The thrust $T$ of a rocket due to the ejection of mass from nozzle is

$$T = |v_e \frac{dM}{dt}|,$$  

(3)

where $v_e$ is the exhaust velocity of the ejected mass and $\frac{dM}{dt}$ is the rate at which mass is ejected from the rocket from its frame of reference. Our numerical model allows us to gain a qualitative understanding of how the thrust of the rocket would vary when the nozzle (second chamber) is varied in length. The thrust correlates with the horizontal component of $v_e$. Comparing two nozzles by looking at the $v_e$ profiles at $L$ and at $1.5L$, as illustrated in the Figure 1, the $v_e$ profile with a smaller vertical component and larger horizontal component gives us more thrust. This enables us to gain a qualitative understanding of how the length of the nozzle actually correlates with the thrust of model water rocket.

Figure 2 and Figure 3 illustrate the absolute values of the $v_y$ profiles at $L$ and $1.5L$ respectively. As you can see the $v_y$ profile at $1.5L$ is a lot smaller in magnitude than...
the \( v_y \) profile at \( L \). From our numerical model, we were able to extract the \( v_x \) profiles at \( L \) and \( 1.5L \). Since all properties of our fluid such as density and viscosity were consistent in the nozzle at both \( L \) and \( 1.5L \), we can allow the thrust at both points to be directly proportional to the average of the respective \( v_y \) profiles. Taking this into account we found the thrust at \( L \), \( T_L \), to be \( 2.56 \times 10^{-2} \) and the thrust at \( T_{1.5L} \), to be \( 3.02 \times 10^{-2} \). The normalized \( T_L, T'_{1.5L} \), is \( 8.50 \times 10^{-1} \). These values agree with the aforementioned thought that smaller \( v_y \) values correspond to a larger \( v_x \) profile which in turn corresponds to a greater thrust. However, in order strengthen the foundation of our numerical model for its more qualitative perspective on the correlation between nozzle length and thrust of a water rocket, we need an experimental data and procedure.

![FIG. 2: This is the magnitude of \( v_y \) sampled 20 times at \( L \).](image)

![FIG. 3: This is the magnitude of \( v_y \) sampled 20 times at \( 1.5L \)](image)

Figure 4 illustrates the setup four model water rocket. It consists of a large PVC pipe chamber with an air nozzle attached to it and a hole for the pressurized water to escape at the bottom. This large chamber is attached to a force plate that will measure the force exerted on it when pressurized water escapes the chamber. In order to compare the results obtained from this setup to our numerical model, we have a smaller PVC pipe that behaves as the nozzle for our water rocket. We can then measure the force exerted on the force plate with this smaller chamber attached to simulate our numerical model at \( 1.5L \), and without the smaller chamber to simulate our numerical
model at $L$.

FIG. 4: Pressurized air is pumped through the air nozzle. As the pressure in the larger PVC chamber increases, the water exits the hole in the first chamber. As the water exits the first chamber it exerts a force against the force plate. Two scenarios were considered here. One where the water simply exits the first chamber into the atmosphere. Second where the water exits the first chamber and then enters the second smaller PVC pipe chamber before being released into the atmosphere.

After taking several measurements with and without the small chamber with the same volume of water and at the same pressure, we can fit the force versus time graph to analyze how the water exits the rocket. Knowing that the force-time graph represents,

$$F = gM - T, \quad (4)$$

where force $F$, thrust $T$, and mass $M$ are functions of time, we can obtain $T$ and an average for the thrust exerted by the rocket. Recalling Equation 3 and knowing that,

$$v_e = \frac{1}{\rho A_{\text{nozzle}}} M', \quad (5)$$

where $\rho$ is the density of the fluid, water, and $A_{\text{nozzle}}$ is the cross-sectional area of the nozzle through the water exits we can rewrite Equation 4 as

$$F = gM - \frac{1}{\rho A_{\text{nozzle}}} M'^2, \quad (6)$$

where

$$T = \frac{1}{\rho A_{\text{nozzle}}} M'^2. \quad (7)$$
Then all that is left to do is to solve the differential equation, Equation 7, with an easy numerical solver to determine the thrust.

Figure 5 and Figure 6 represent the running average force with respect to time of our model water rocket without the small chamber and with the small chamber respectively. The peak of each graph represents when the all of the water had exited the model rocket. The force approaching the peak represents the thrust of the rocket in each case as the mass (water) exits the rocket with respect to time. Solving Equation 7 for both cases, we found the thrust in the rocket without the small chamber, $T_1$ to be $5.42 \pm 0.89$ N and the thrust in the rocket with the small chamber, $T_2$ to be $16.27 \pm 1.10$ N. An important fact to note is that the inner radius of the small PVC chamber is smaller than the radius of the hole in the large chamber as illustrated in Figure 4. This was taken into account in solving our differential equation for each case.

The normalized $T_1$, $T'_1$ is 0.33. This is not very similar to $T'_L$ at all. This can be due to the fact that the data obtained from our numerical model does not deal with different size radii for the water to exit through. Though our numerical model did not take different sized exit holes into account, our experimental results still reinforce the key qualitative analysis provided by our numerical model. Our numerical model was successful at illustrating how the length of the nozzle is able to restrict the flow of the water to the direction normal to he nozzle.

The success of our numerical model to provide a qualitative perspective on water rockets and nozzles is not at its end. This model could very well be modified to examine the effect of different nozzle shapes on the thrust of a water rocket. Such shapes could include a conic, parabolic, or even an hourglass-shaped nozzle. Similarly, one can check the validity of the numerical model qualitatively by comparing it to experimental data. This model allows us to further examine simple model water rockets in a more depth. Such analysis fosters a better understanding of the physics of rockets.

\[ \text{Prusa, Joseph M., Hydrodynamics of a water rocket, SIAM Review Vol.42, No.4, 2000} \]
\[ \text{Yawn, James, Model Rocket Motors, http://www.jamesyawn.com/modelrocket/drill/index.html} \]
FIG. 5: Water exits the first chamber into the atmosphere. Notice that the force-time relationship is linear as the water exits the first chamber. A running average of 50 runs was taken to reduce the simple harmonic oscillating effect of the water rocket system when the pressurized water was exiting the system.
FIG. 6: Water exits the first chamber and into the second smaller chamber before entering the atmosphere. Notice that the force-time relationship is quadratic as the water exits the rocket. Again a running average of 50 runs was taken to reduce the simple harmonic oscillating effect of the water rocket system when the pressurized water was exiting the system.