

第一页为封面页

参赛学生姓名：袁昊琨 嵇海锐

中学：北京十一学校

省份：北京市

国家/地区：中国/北部赛区

指导老师姓名：于小雨

指导老师单位：北京十一学校

论文题目：Theoretical modeling and CFD-based refinement of first-stage thrust characteristics in a two-stage water rocket, supported by experimental measurements

Theoretical modeling and CFD-based refinement of first-stage thrust characteristics in a two-stage water rocket, supported by experimental measurements

Abstract

The water-discharge process of a water rocket is a transient phenomenon with various processes occurring simultaneously, such as compressed-air expansion, liquid acceleration, gas-liquid interface change, and variable-mass motion. The conventional models typically use a gas-expansion relation, Bernoulli equation and mass conservation to calculate pressure inside the bottle, nozzle velocity and mass flow rate. They reasonably capture the overall trends, although treatment of the unsteady flow during start-up and the gas-liquid mixing process at later times is relatively simple. In this study, a two stage water rocket was used as an experimental platform. An in-house flight data logging system was designed to capture data from acceleration, angular velocity and air pressure sensors, at a rate of $\sim 50\text{Hz}$, during actual flight. A transient VOF model of the gas-liquid flow inside the first stage was then constructed in Fluent and quantities which are difficult to measure directly in the experiment, such as the pressure, the mass of water, the mass flow rate and the velocity were obtained. Finally, the CFD results are compared to the conventional simplified formula, relation-wise. During first-stage water discharge, results indicate clearly change between different phases. The liquid velocity requires some time to build up during start-up and the classical quasi-steady relations are deviating considerably. The numerical results agree quite well with the conventional velocity and mass flow rate relations during the continuous liquid-phase flow period, with minor corrections. As soon as the gas-liquid transition starts, the outlet velocity – mass flow rate correlation drastically changes and the single liquid phase approximation is no longer possible. The study demonstrates that it is not possible to test for the correctness of a model just by comparing its acceleration at the end of the calculation chain. A better way to do this is to go back through the steps of pressure-velocity-mass flow rate-thrust-acceleration and see where the error is injected.

Keywords: water rocket; gas-liquid two-phase flow; VOF; sensor measurements; unsteady flow; simplified theoretical model

1 Introduction

A water rocket uses the energy stored in compressed air inside a bottle to drive water out of a nozzle at high speed, and obtains propulsion from the reaction force produced by the jet. Its structure does not look complicated, but many processes actually take place inside it during water discharge. As water continues to leave, the air volume increases and the pressure falls; the liquid inside the bottle gradually accelerates from rest, while the free surface keeps moving and deforming. Toward the end of water discharge, air may also enter the nozzle before all the water has left and flow out together with the remaining liquid. Thus, a propulsion process lasting less than one second actually involves gas expansion, liquid flow, changes in the gas-liquid interface, and a continuous decrease in rocket mass at the same time.

The conventional calculation methods commonly used for water rockets are relatively simple. Typically, the pressure inside the bottle is first calculated from the change in gas volume. The Bernoulli equation is then used to obtain the nozzle velocity from the pressure difference, and continuity relations are used to calculate mass flow rate and thrust. This approach has clear formulas, is easy to calculate, and correctly gives the overall trend of gradually decreasing pressure and water-discharge velocity. However, it usually treats the flow inside the bottle as quasi-steady, as though the flow were already established at every instant, and assumes that the outlet velocity can be represented by a single mean value. Whether the actual water-discharge process satisfies these conditions throughout cannot be judged solely from the acceleration curve.

This issue is even more important for a two-stage water rocket. The first stage not only provides the initial propulsion but also determines the velocity and state of motion at subsequent separation and second-stage startup. If the first-stage model contains errors in the time histories of pressure, mass flow rate, or thrust, a final result that appears close to the experiment may simply reflect the cancellation of different errors. This study therefore focuses on the connections between physical quantities and the propagation of errors: how the pressure inside the bottle changes, how that pressure produces the

water-discharge velocity, how this velocity determines mass flow rate and thrust, and how these quantities ultimately affect rocket acceleration.

To investigate this process, this study follows an approach combining experimental measurements, Fluent numerical simulations, and tests of conventional theory. In the experimental part, a flight data logging system was developed in-house to record acceleration, angular velocity, and air pressure during actual flight. In the numerical part, Fluent and the VOF method were used to observe the time evolution of the gas-liquid interface, pressure, velocity, and mass flow rate inside the first stage. In the theoretical part, the conventional formulas were divided into several steps and compared individually with the corresponding CFD data, rather than fitting only the final acceleration output.

This paper mainly addresses three questions. First, can first-stage water discharge be divided into several phases with different characteristics? Second, at which step in the calculation of pressure, velocity, mass flow rate, or thrust do the main errors in the conventional formulas arise? Third, which corrections are effective only during a particular phase, and which relations remain valid over a longer time interval? Because the high-time-resolution CFD data currently available mainly concern the internal flow of the first stage, and because the actual pressure at the start of second-stage discharge cannot be measured effectively, the numerical analysis and theoretical corrections in this paper also focus mainly on first-stage propulsion.

2 Literature Review

2.1 Lumped-Parameter Theory and Conventional Approximations

Prusa organized the conservation relations for water rockets from a fluid-mechanics perspective, showing the considerable analytical value of simplified models with only a few parameters [1]. Finney analyzed water propulsion as a physics problem and used an isothermal approximation for air expansion inside the bottle [2]. These studies reveal that a conventional water rocket model is not a single, unique set of equations: various studies can adopt different assumptions regarding air expansion, drag models and approaches to mass change. In the current paper, conventional model is primarily a method in which the volume of air is calculated based on the volume of water left, pressure is calculated based on a gas-expansion relation and the outlet velocity is calculated based on a relation of the type of Bernoulli.

Its benefits are that it requires only a small number of equations and the physical interpretation of each step is easily understood and thus it is quite suitable to estimating the overall movement of a water rocket. It, however, also makes the spatial variation of the actual flow field reduced to a few mean quantities. To illustrate, one pressure corresponds to the state of the entire bottle and one outlet velocity corresponds to the entire cross-section of a nozzle, and the geometry of the gasliquid interface typically does not appear anywhere in the equations. The paper thus analyzes independently whether the pressure, velocity and mass flow rate equations can be applied in various stages.

2.2 Unsteady Flow and Gas Thermodynamics

Gommes examined the moist-air expansion and transient flow and the inertia of liquids in more detail and indicated that various corrections can neutralize each other and that coincidence between a final result and an experiment can sometimes be due to cancellation of errors [3]. Romanelli et al. examined the air expansion in water rockets, which is a reference of the isothermal and adiabatic processes and the more general polytropic processes [4]. These experiments indicate that variations in pressure within the bottle and the movement of the liquid can be most effectively studied independently, and not by explaining all the deviations by the same empirical coefficient.

In a more general analysis of flow passages, Johnston wrote about the effect of inertia of fluids on the flow in nonuniform passageways [11]. The conditions in that study are not quite identical with those in a water rocket but one significant concept is applicable here: the velocity of the liquid does not immediately arrive at a constant value when pressure is applied; the flow velocity itself takes time to develop. In the context of startup, this work thus gives special attention to the question of whether the quasi-steady relation assumes that the velocity of the flow has already been generated too soon, and the issue of liquid inertia versus flow losses in general.

2.3 The Gas–Liquid Interface and Outlet Flow

The VOF method uses volume fractions to represent the regions occupied by different fluids and is a common method for calculating two-phase flows with free surfaces. Its basic ideas were systematically presented by Hirt and Nichols [5]. The CICSAM method proposed by Ubbink and Issa maintains a sharp gas–liquid interface during volume-fraction transport [6] and corresponds to the volume-fraction discretization scheme actually used in this paper. Compared with conventional models that provide only mean pressure and mean velocity, VOF has the advantage of allowing direct observation of how the water–air interface moves and when air begins to enter the outlet. In addition, the SST $k-\omega$ turbulence model used in this paper was proposed by Menter to describe turbulent effects in near-wall regions and external flows [7].

Previous water rocket studies have also found that gas may already have entered the nozzle while a small amount of liquid remains inside the bottle [9]. This means that water remaining inside the bottle does not imply that only water is still leaving the nozzle. Once gas–liquid mixing begins, the relationship used to calculate mass flow rate from a fixed water density and a single mean velocity may change. This paper therefore uses the state of the gas–liquid interface as an important basis for determining the range of applicability of the conventional formulas.

2.4 Limits of the Observational Evidence

Marranghello et al. demonstrated methods for analyzing the propulsion phase of water rockets using sensors and video [8]. This highlights the fact that the three IMU axes do not necessarily coincide with the rocket’s longitudinal axis. The magnitude of the three-axis acceleration therefore cannot simply be treated as the axial propulsion acceleration. When comparing experiment and CFD, the positive direction, treatment of gravity, and time origin must also be made consistent. Accordingly, the experimental part of this study specifically includes sensor bias calibration and extraction of the axial direction.

2.5 Two-Stage Studies and the Position of This Work

Chang and Weng have discussed the flight dynamics of two-stage water rockets [10]. This study also uses a two-stage water rocket as an experimental platform. However, the most complete CFD data currently available concern gas–liquid flow inside the first stage, and the internal pressure at the start of second-stage discharge cannot be obtained accurately. This paper therefore takes the first stage as its main object of analysis.

2.6 Research Gap and Approach of This Study

Taken together, previous studies show that conventional theory, gas expansion, VOF numerical methods, and sensor measurements have all received considerable attention, but these approaches are often discussed separately. For the same set of water rocket experiments, it remains worthwhile to investigate how actual flight data, CFD results for the internal flow, and conventional formulas can be compared along a single physical sequence, and how the earliest step at which errors arise can then be identified.

Accordingly, this study does not aim to fit the final acceleration curve with a large number of parameters. Instead, it makes stepwise comparisons following the physical process: first the relationship between air volume and pressure, then pressure difference and outlet velocity, then velocity and mass flow rate, and finally how these flow quantities produce thrust and acceleration. If a correction is effective only during a particular phase, it is used only in that phase; if a phenomenon has several possible explanations, those possibilities are retained.

3 Experimental Measurements and Data Processing

3.1 Overall Research Framework

This paper is divided into three sections, including experimental measurements, numerical simulations and theoretical analysis. The experimental section measures the acceleration, angular velocity, air pressure and other measurements as the water rocket actually flies. The numerical analysis employs Fluent to get the pressure field, velocity field, gasliquid interface and outflow mass flow rate within the bottle, which cannot be directly measured during the experiment. These data are then used in the theoretical section where the applicability of the conventional formulas will be analyzed. These three

sections are related but have various purposes: the experiment demonstrates how the rocket really moves, CFD assists in demonstrating how the fluid inside the bottle varies, the traditional formulas explain the connections between those variations.



Figure 3-1. Water rocket experimental platform and sensor payload

The experimental platform is a two-stage water rocket. To record the full flight, a small logging system capable of independent power supply and local data storage was developed in-house around an ESP32 microcontroller. The MPU6050 measures three-axis acceleration and three-axis angular velocity, the BMP388 measures ambient pressure and temperature, and the MicroSD card stores the complete time series. The ESP32 handles sensor initialization, timed sampling, data calibration, buffer management, and data writing. The materials are listed in Appendix A.

3.2 Experimental Platform and Sensor System

3.2.1 Water Rocket Experimental Platform

The water rocket experimental platform consists mainly of four parts: the launch base, the first stage, the second stage, and the nose-cone parachute. The first stage consists of two beverage bottles, each with a nominal volume of 1.25 L, and the second stage consists of one beverage bottle with a nominal volume of 1.25 L. A pressure-triggered separation mechanism connects the first and second stages. Under the experimental conditions, the initial water volumes were 625 mL in the first stage and 400 mL in the second stage. The experimental records cover the complete flight of both stages, while the subsequent CFD analysis focuses mainly on first-stage water discharge.



Figure 3-2. Launch base

During the preparation of experiments, the rocket is maintained in a stable position by the launch base. In this paper, the theoretical model does not take into account the base per se since it is devoted to flight data and internal flow.

The size, amount of water and weight of the first and the second stages are significant inputs to the later theoretical calculations and CFD setup. Table 3-1 gives their measured masses.

The nose cone has a parachute to minimize the impact upon landing. However, it did not have the same cushioning effect during the actual test flights.

Table 3-1. Main mass parameters of the water rocket experimental platform

Item	Mass / kg	Description
Launch base	0.5436	Experimental platform; excluded from flight mass
First-stage dry mass	0.2800	Dry structural mass of the first stage
Second-stage dry mass	0.1123	Dry structural mass of the second stage
First-stage wet mass	0.9050	First-stage dry structural mass plus water mass
Second-stage wet mass	0.5123	Second-stage dry structural mass plus water mass and separator

3.2.2 Sensor Assembly and Hardware Reliability

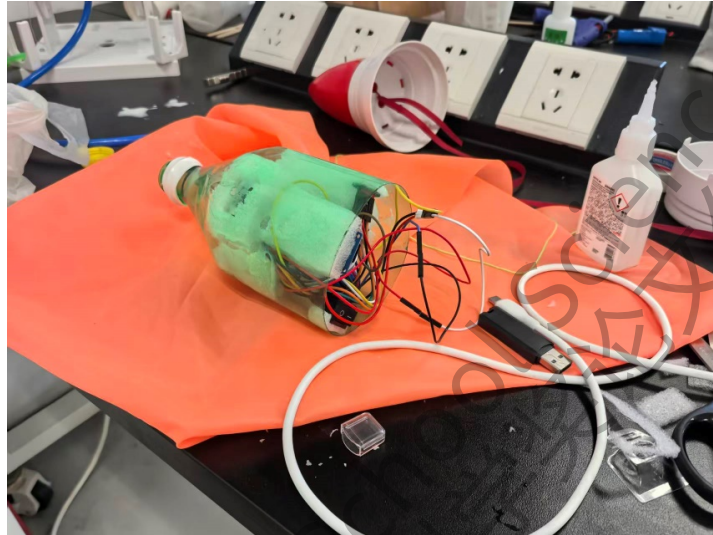


Figure 3-3. Final sensor assembly

The sensor system was not developed in a single attempt. The initial goal was simply to read all modules simultaneously, but installing them in the rocket introduced problems with power supply, wiring, and reliability under vibration, so the hardware underwent several revisions. In the final system, an 18650 lithium battery provides independent power, and the ESP32 supplies the individual modules. The MPU6050 and BMP388 communicate over the I²C bus, while the MicroSD card receives data over a separate SPI bus. This keeps sensor communication as separate as possible from the slower storage process.

The first version mainly used Dupont jumper wires and pin headers. It worked normally in static bench tests, but after installation in the rocket, shaking and impacts caused intermittent poor contact, leading to ESP32 resets, lost sensor connections, or interrupted recording files. To address this, critical power wires and some data wires were subsequently soldered, and mechanical restraints and cushioning materials were added to reduce relative wire movement under vibration. After these changes, the system maintained continuous power and data recording during flight tests.

This change shows that a rocket sensor system must not only operate on the bench but also maintain stable connections under vibration. Hardware reliability therefore became the first issue addressed before formal flight tests.

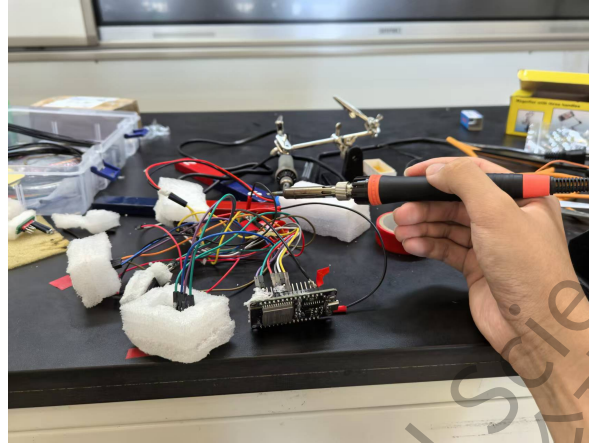


Figure 3-4. Reinforcing sensor connections by soldering

3.2.3 Data Acquisition Program and Storage Design

The sensor system software was written in the Arduino IDE and uploaded to the ESP32. After startup, the program sequentially performs device initialization, self-checks, sensor connection checks, MicroSD file creation, warm-up, and calibration before beginning formal data recording. During recording, sampling follows a fixed time reference to minimize the accumulation of program execution time in the sampling period.

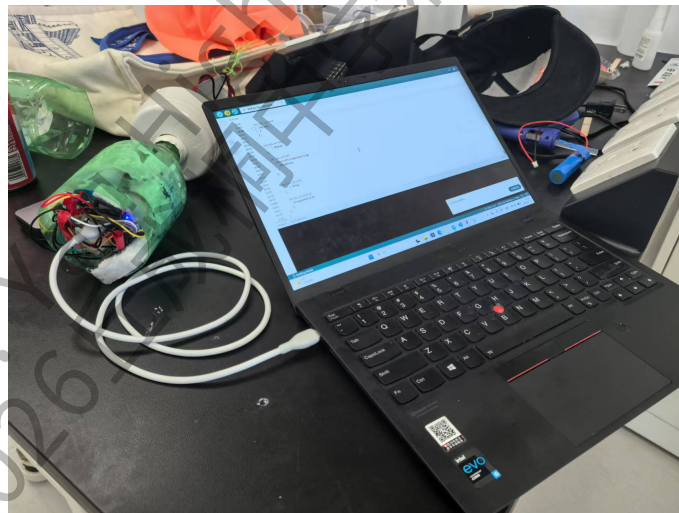


Figure 3-5. Uploading the program to the ESP32 development board through Type-C

MicroSD writing appeared to be the most noticeable issue during program development. The first program wrote to the SD card as soon as the sensors were read, which meant that the time taken to write was not predictable and sometimes caused tens of milliseconds of blocking. The result of sampling at 50 Hz is that every record must be made after 20 ms, so a single slow write may hold up the next sample, leading to some variation in the sampling interval, or even loss of data. The last program decouples sensor sampling and SD writing: the primary task reads the data at a periodic time, and puts them into a queue in memory, and another FreeRTOS task writes the records in the queue to the SD card in batches. It is this that enables the memory queue to absorb short delays in the event that SD writing is slowed down.

3.2.4 Warm-Up, Calibration, and Runtime Diagnostics

Another problem was MPU6050 bias and startup drift. In static tests, the three-axis acceleration and angular velocity did not always remain at exactly the same reference values, even when the sensor was motionless. Early in the study, a fixed set of zero offsets was estimated from a long static test. Differences were later found between power-up sequences in different environments, however, so each experiment was calibrated separately in the final procedure. The system first warmed up for approximately 60 s after startup, after which the rocket was kept stationary for approximately 20 s. The average of the data over this interval was taken as the zero reference for that experiment.

$$\bar{a}_i = \left(\frac{1}{N} \right) \Sigma a_i \quad (3-1)$$

$$\bar{\omega}_i = \left(\frac{1}{N} \right) \Sigma \omega_i \quad (3-2)$$

$$a'_i = a_i - \bar{a}_i \quad (3-3)$$

$$\omega'_i = \omega_i - \bar{\omega}_i \quad (3-4)$$

The acceleration used in post-processing is therefore the change relative to the stationary baseline before launch in that experiment, rather than the unprocessed raw sensor output.

Early test flights also included cases in which the power-on checks were normal but an anomaly suddenly appeared during flight. From the final acceleration curve alone, it was difficult to determine whether the anomaly arose from actual motion or from the recording system. The final program therefore also saves diagnostic variables including time_us, dt_us, dropped, and queue_depth. Here, time_us establishes a common time axis, dt_us checks the actual sampling interval, dropped records the number of records that could not enter the queue, and queue_depth indicates whether SD writing is causing a buffer backlog. These variables allow recording performance to be checked after the experiment.

Table 3-2. Main settings of the final data acquisition program; the complete program is provided in the appendix

Item	Final setting	Description
MPU6050 sampling frequency	50 Hz	Records three-axis acceleration and angular velocity
BMP388 sampling frequency	Target: 50 Hz	Target pressure and temperature update frequency; actual effective updates depend on sensor conversion time
Data logging frequency	Approximately 50 Hz	Each record carries a common timestamp
MPU6050 acceleration range	± 16 g	--
MPU6050 angular velocity range	± 500 °/s	--
MPU6050 bandwidth	Approximately 44 Hz	Digital filter setting in the program
I ² C interface	SDA GPIO21; SCL GPIO22	Shared by MPU6050 and BMP388
SD memory buffer	Approximately 8 KB	Reduces the effects of frequent card writes
Data queue length	Approximately 1600 records	Absorbs short delays in card writing
Batch writing	Approximately 100 records/batch	Reduces the number of SD write calls
Forced flush interval	Approximately 5 s	Prevents frequent flush operations from blocking the main sampling task
Startup warm-up time	Approximately 60 s	Reduces drift immediately after power-on
Static calibration time	Approximately 20 s	Calculates the zero reference for the current experiment
Total startup preparation time	Approximately 80 s	Sum of warm-up and calibration
MicroSD communication	SPI	Separate from the I ² C sensor bus
SD pins	CS 5; SCK 18; MISO 19; MOSI 23	Final ESP32 wiring
Diagnostic variables	time_us, dt_us, dropped, queue_depth	Check the time axis, dropped records, and buffer status

3.3 Flight Tests

The experiment with flying was conducted at an open field. The first-stage and second-stage water volumes were adjusted before each test condition in line with the given test parameters and rocket body, interstage structure, and sensor payload were assembled. Once the sensor system was started, it took about 60 s to warm up and then about 20 s when the rocket was stationary to calibrate the bias. Once calibration was done, the launch was done under the given conditions during that experiment, and all the flight was filmed on camera. The starting volumes of water were 625 mL in the first and 400 mL in the second stage.

To support comparisons among experimental measurements, CFD results, and theoretical calculations, the initial water amount, initial pressure, masses of the separate stages and sensor data files are also stored in the experimental records.

The ESP32 was then tested after each flight by ensuring it was still functioning normally and a full read of the CSV file on the MicroSD card was done. Flight data were only accepted to later analysis when it had been verified that the file was readable as usual, a time series was continuous, and that the diagnostic variables did not indicate any evident record faults.



Figure 3-6. Key frames from the flight experiment

The complete flight video is provided in Appendix C.

3.4 Sensor Data Post-Processing

The raw CSV files on the MicroSD card were then imported into Python after the experiment, to be processed uniformly. Originals were never deleted and only copied were time-window selection, plotting, sampling-quality checks, and extraction of the axial acceleration. This eliminates the possibility of loss of original information in the processing and simplifies the process of re-visiting individual steps of processing.

It took the entire recording about 160 s, with the flight being just a fraction of the recording. The main flight time window was chosen in post-processing as a result of the prominent motion events in the acceleration, angular velocity, and video records to see propulsion and separation better.

The final CSV mainly contains `time_us`, `dt_us`, `ax`, `ay`, `az`, `gx`, `gy`, `gz`, `temp`, `pressure`, `altitude`, `dropped`, and `queue_depth`. Here, `time_us` is the timestamp and `dt_us` is the actual interval between adjacent samples. The fields `ax`, `ay`, `az` and `gx`, `gy`, `gz` are the calibrated three-axis acceleration and angular velocity, respectively; the remaining fields record air pressure, temperature, relative altitude, and acquisition-system status. These time fields allow a direct check of whether the actual sampling frequency is close to the programmed value of 50 Hz.

One of the most important post-processing steps is extracting acceleration along the rocket's longitudinal axis. Because sensor installation was constrained by the space inside the rocket, the MPU6050 x, y, and z axes did not align exactly with the longitudinal axis. Neither a single sensor axis nor the magnitude of the three-axis acceleration could therefore be used directly. The direction of the rocket's longitudinal axis in the IMU coordinate system can be determined from the gravity components along the three sensor axes when the rocket is vertical and stationary.

$$e_r = (0.94456, 0.23837, -0.22580)$$

$$a_r = a \cdot e_r = 0.94456 a_x + 0.23837 a_y - 0.22580 a_z$$

Here, a_r is the acceleration component along the rocket's longitudinal axis. This projection avoids directly treating lateral vibration or sensor mounting-angle errors as acceleration in the propulsion direction.

The complete post-processing code is provided in Appendix D.

3.5 Experimental Data and Recording Stability

The sensor data and recording stability during actual flight are examined first below, before determining which quantities are suitable for subsequent comparison with CFD and theoretical models. The complete raw data are provided in Appendix E.

3.5.1 Acceleration and Angular Velocity

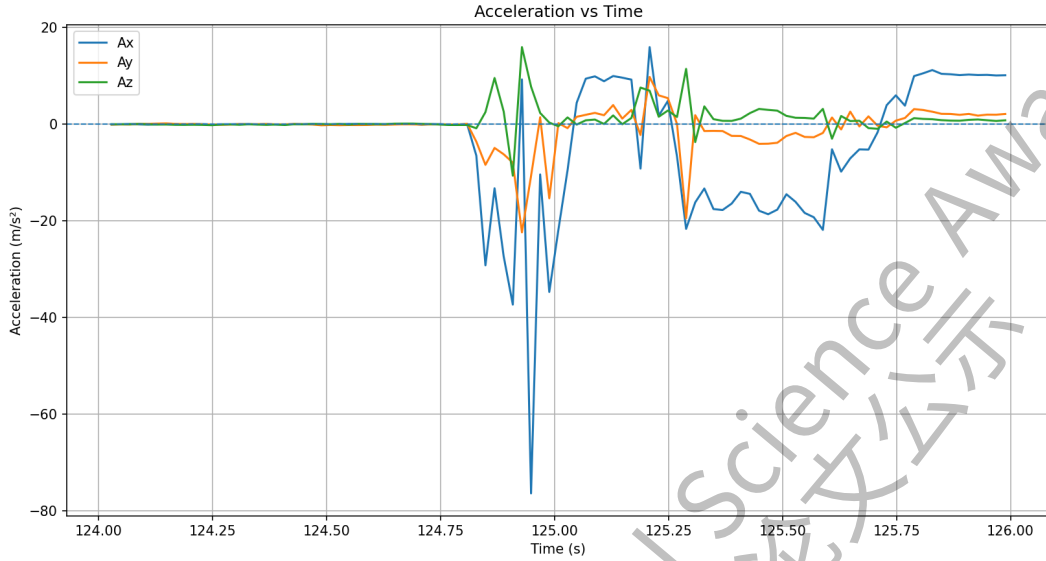


Figure 3-7. Three-axis acceleration records after subtraction of the stationary baseline

All three acceleration axes show clear changes during the main flight interval, with the largest change along the direction closest to the rocket's longitudinal axis. Two main propulsion phases can be identified in time. However, because the sensor orientation does not align exactly with the rocket body, these measurements cannot be treated directly as the rocket's true axial acceleration; comparisons are made after the projection described in Section 3.4. Note that bias calibration was performed with the rocket stationary before launch, so the values in the curves represent changes relative to this baseline.

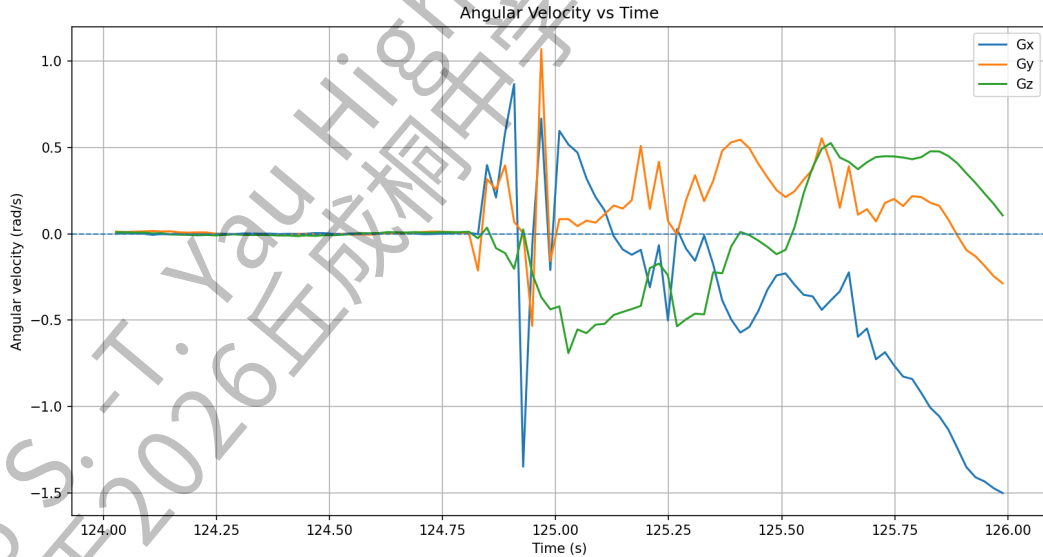


Figure 3-8. Three-axis angular velocity data

The angular velocity data show that the rocket's flight is not strictly one-dimensional motion without rotation. All three angular velocity components change noticeably around propulsion and separation, and some rotation persists afterward. This paper does not yet use the gyroscope to reconstruct the complete three-dimensional attitude. Instead, angular velocity is used mainly as supporting evidence for identifying flight states and illustrating the limitations of the one-dimensional model.

3.5.2 Limitations of the BMP388 Pressure and Altitude Data

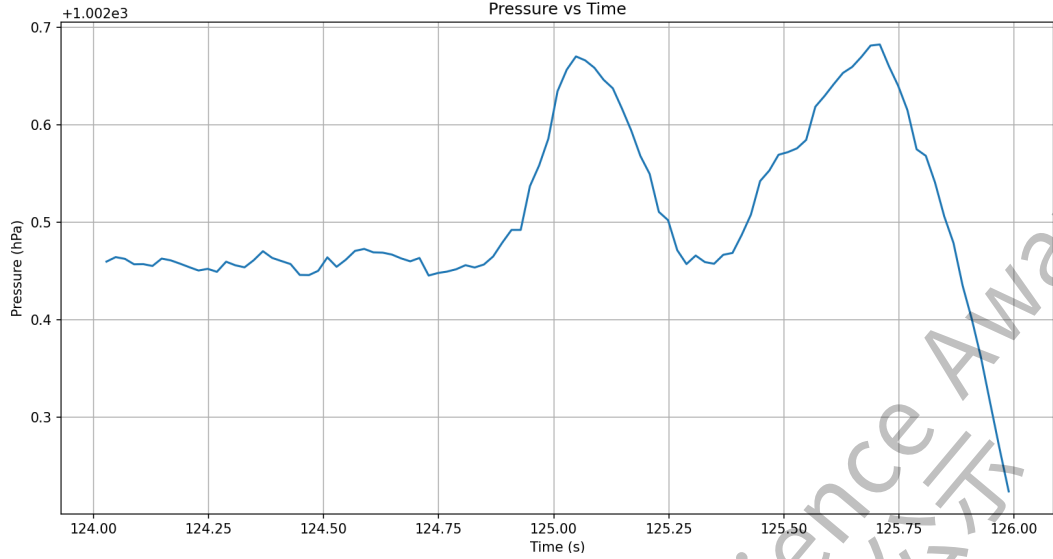


Figure 3-9. Pressure data

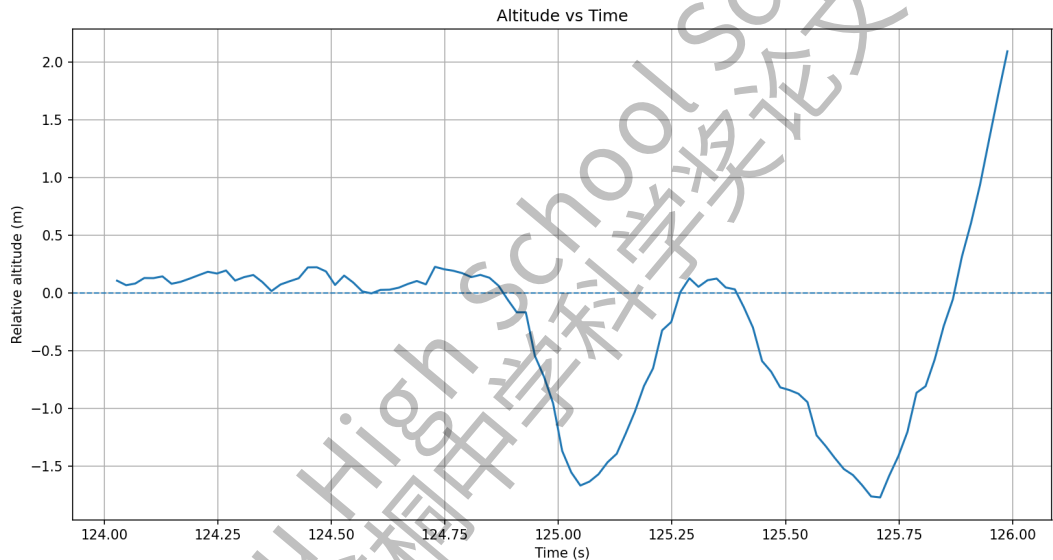


Figure 3-10. Altitude data

The BMP388 had an initial design of measuring relative altitude due to the variation in atmospheric pressure. But Figure 3-9 demonstrates that the pressure recorded in fact increases significantly at the two main propulsion phases, and the altitude based on it experiences rapid declines and rebounds which are not in accord with the actual movement. It means that the measurements of the BMP388 at propulsion cannot be easily considered as constant atmospheric static pressure. The area around the sensor can be influenced by the flow around the rocket body, local dynamic pressure, and attitude variations. At a given airflow velocity v , the order of magnitude of the dynamic pressure is $\frac{1}{2}\rho_{\text{air}}v^2$, and thus local flow can cause large changes in sensor reading even though the ambient static pressure varies insignificantly. Based on this, the paper presents no data of BMP388 at the propulsion stage to obtain the quantitative altitude measurements. The data is kept just as an experimental record, and as a guide to later structural improvements.

3.5.3 Sampling Stability and Data Integrity

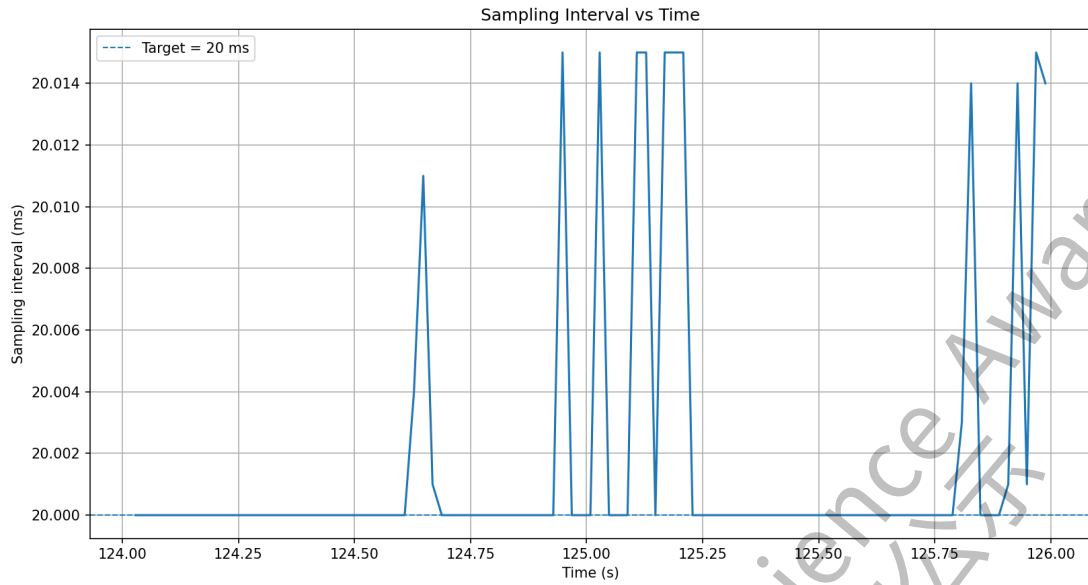


Figure 3-11. Actual sampling interval

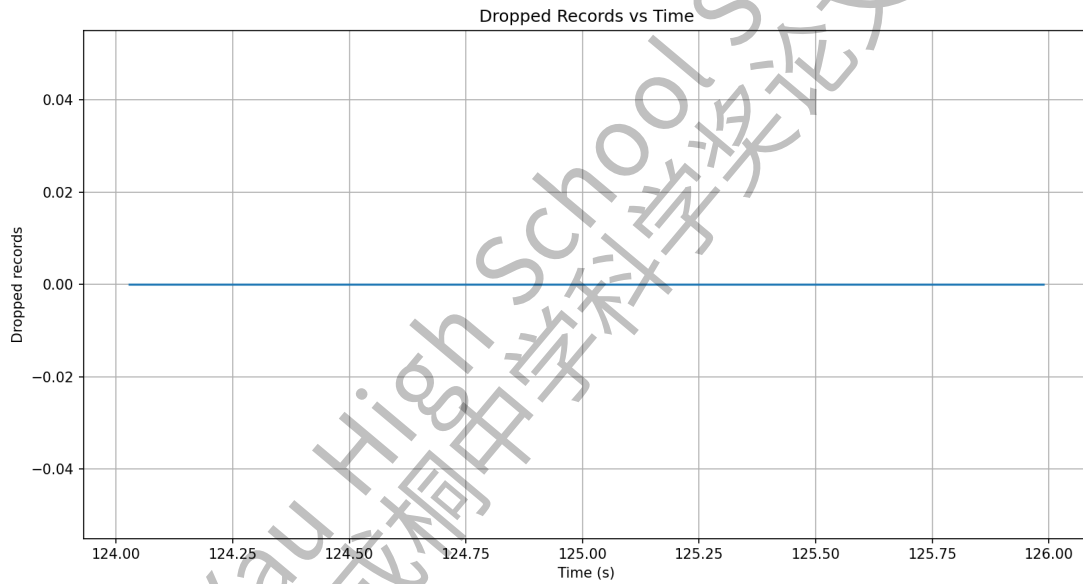


Figure 3-12. Number of dropped records

The sampling time of the sampling interval of us (dt_{us}) is near 20 ms, which is consistent with the desired frequency of 50 Hz, in the actual flight time window. In the meantime, dropped in the experimental record is 0, which means that there were no record losses in the program due to overflowing queues. This demonstrates that the asynchronous buffering that was implemented to solve the SD write blocking enabled continuous recording in this flight.

3.5.4 Rocket Longitudinal Acceleration

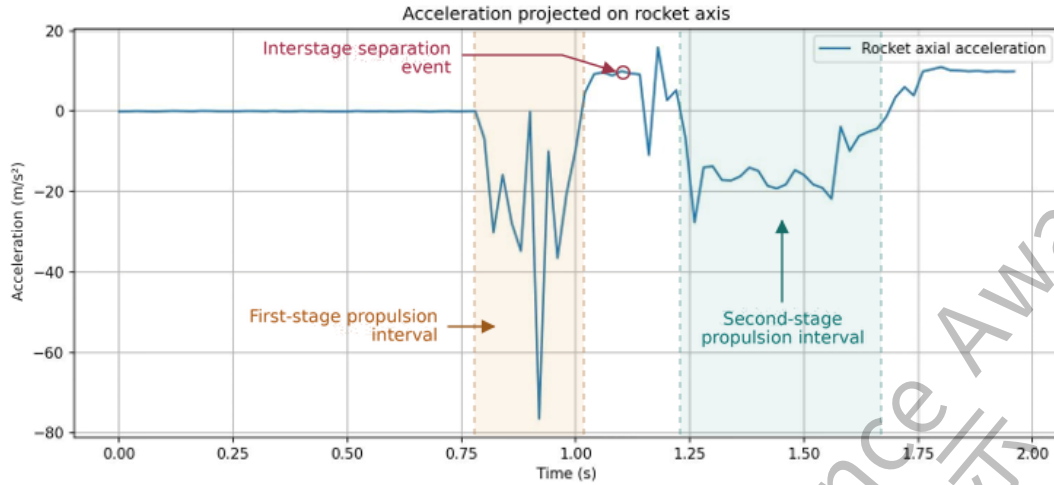


Figure 3-13. Extracted rocket longitudinal acceleration

Projecting the three-axis data onto the direction vector in Section 3.4 gives a rocket longitudinal acceleration more suitable for comparison with the one-dimensional theoretical model. Compared with using an arbitrary single sensor axis, this processing reduces the influence of the sensor mounting angle on the result. The curve contains identifiable features associated with first-stage propulsion, interstage separation, and subsequent changes in flight state.

It should also be noted that the experimental sampling period is approximately 20 ms, whereas the CFD time step is much smaller. The experimental data therefore cannot validate the very brief, high-frequency spikes in CFD point by point. Subsequent experiment–CFD comparisons focus more on propulsion duration, the overall acceleration trend, and integrated quantities.

4 Numerical Model and Computational Methods

4.1 Objectives of the Numerical Simulation and Model Simplifications

Experiments measure the acceleration and angular velocity of the entire rocket, and the ambient air pressure. During the few hundred millisecond water discharge, however, it is challenging to directly measure the pressure distribution inside the bottle, the motion of the gas–liquid interface, the fluid velocity at the nozzle outlet and the instantaneous mass flow rate. In this study, transient numerical simulation of gas–liquid flow inside the first stage is thus carried out using ANSYS Fluent 2026 R1 Student. The simulation does not intend to substitute flight tests, but it is designed to show internal processes, which cannot be observed directly in the experiment and to deliver data with relatively high time-resolution to be used for the tests of the conventional formulas afterward.

The numerical investigation begins at the moment of start of first-stage water discharge and runs until the main gas–liquid flow has significantly reduced. The geometry of the first-stage bottle body and the nozzle is close to being rotationally symmetric about the centerline, allowing the use of a two dimensional axisymmetric model to capture the essence of the geometry and to significantly decrease the computational costs. Besides the bottle interior, a region of air is kept outside the nozzle and the jet continues to develop upon exiting the nozzle.

A number of useful simplifications are used to make the real problem into a computational model. The bottle walls are assumed to be rigid (no elastic deformation under pressure), water is assumed to have a constant density, air is assumed to be an ideal gas, surface tension is not turned on and the gas–liquid interface is tracked by using VOF. The emphasis is on the general water-discharge process which is dominated by pressure, inertia and mass outflow. With these simplifications, the model can now be used to answer the main question which is the focus of this paper: Do the traditional pressure–velocity–mass flow relationships hold for the various stages of water discharge?

4.2 Geometry and Computational Domain

The first stage consists of two connected bottles, each with a nominal volume of 1.25 L. The total internal volume of the computational model is 2.7686 L. As shown in the model, this volume includes the

connecting section between the two bottles and the bottle-neck volumes, which are not included in the nominal 1.25 L capacity of each bottle. Its axial profile was first traced from photographs and measured dimensions of the actual bottles, and a two-dimensional axisymmetric computational domain was then constructed. The initial first-stage water volume in the experiment was 625 mL, so the same liquid-water volume was specified during numerical initialization, with air occupying the rest of the bottle. The first-stage axis is taken as the x direction, and the jet leaves the nozzle in the positive x direction.

The geometric model retains the connection between the two bottles, the contracting shoulder region, and the nozzle, and also includes an external air region outside the nozzle. This brings pressure release inside the bottle, acceleration through the nozzle, and the short-distance development of the emerging jet into a single calculation.



Figure 4-1. First-stage geometry modeling process

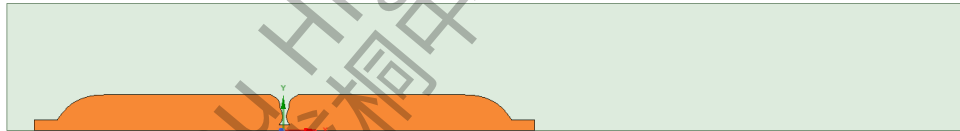


Figure 4-2. Final two-dimensional axisymmetric computational model of the first stage

Table 4-1. Main geometric parameters

Parameter	Value	Source
Total first-stage internal volume	2.7686 L	Full internal model geometry
Initial water volume	0.625 L	Experimental conditions
Initial air volume	2.1436 L	Difference between the two volumes
Nozzle diameter	24 mm	Measurement
Maximum bottle diameter	88 mm	Measurement
Total first-stage length	600 mm	Measurement

4.3 Mesh Generation

An unstructured mesh is used for the computational domain. The largest velocity gradients occur near the contracting bottle shoulder, the nozzle, and the jet, where the gas-liquid interface also moves rapidly. These regions are therefore locally refined. Larger cells are used in the bottle interior away from the nozzle and in the external far field to balance accuracy and computational cost.

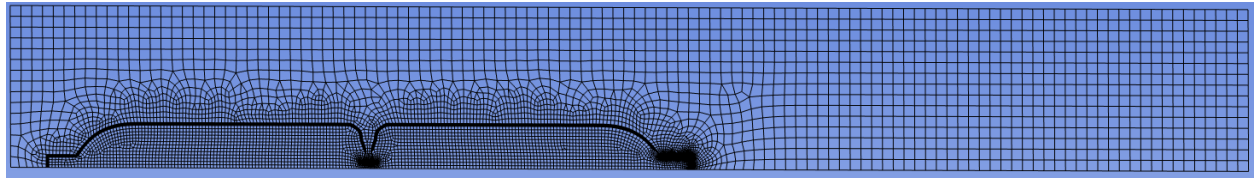


Figure 4-3. Overall computational mesh

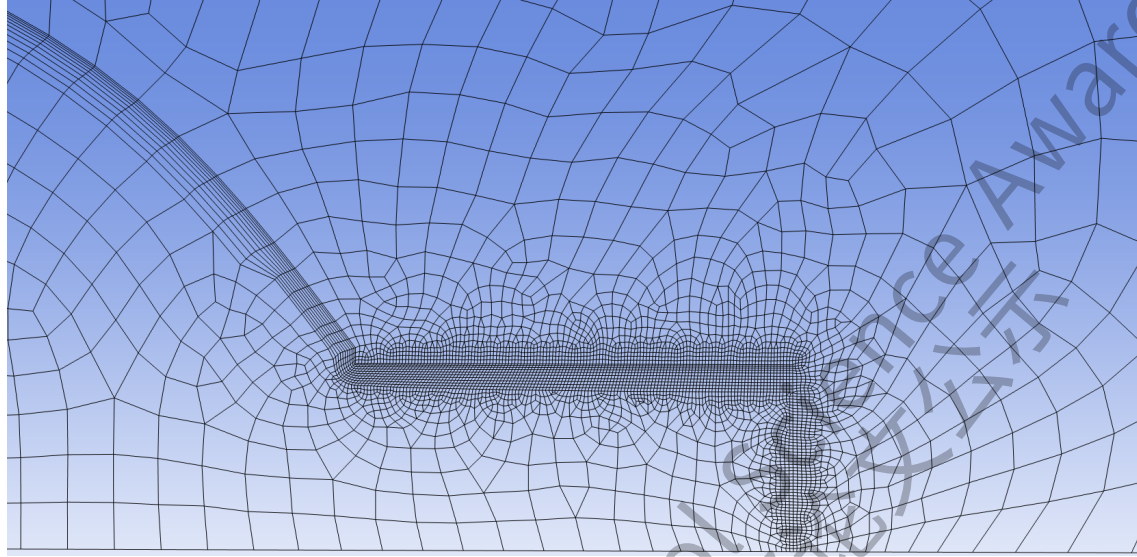


Figure 4-4. Locally refined mesh near the nozzle

Table 4-2. Basic mesh parameters and quality metrics

Item	Value
Total number of cells	12699
Total number of nodes	12850
Mesh size inside the bottle	3 mm
External mesh size	10 mm
Mesh size near the nozzle	0.2 mm
Maximum Skewness	0.84607
Mean Skewness	0.16723
Minimum Orthogonal Quality	0.24912
Mean Orthogonal Quality	0.94551

The final mesh contains 12 699 cells and 12 850 nodes. The mean Skewness is 0.16723 and the mean Orthogonal Quality is 0.94551, indicating that most cells have relatively regular shapes.

4.4 Physical Models and Governing Equations

This study uses Fluent's transient pressure-based solver. Since air and water coexist inside the bottle and their interface moves throughout water discharge, the VOF (Volume of Fluid) method is used to describe the gas-liquid two-phase flow. Only the basic equations directly relevant to the analysis in this paper are listed below.

4.4.1 Mass Conservation

The fluid satisfies mass conservation in the computational domain. For transient flow, this can be written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad (4-1)$$

This equation ensures correspondence between the decrease in fluid mass inside the bottle and the mass leaving through the nozzle and external boundaries. The later comparison of the decrease in water mass with the time integral of mass flow rate also follows from checking this conservation relation.

4.4.2 Momentum Conservation

The velocity field is obtained by solving the transient Navier–Stokes momentum equation, which can be written as:

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \otimes u) = -\nabla p + \nabla \cdot \tau + \rho g \quad (4-2)$$

Compared with the conventional Bernoulli formula, this equation retains the transient term describing velocity changes with time, spatial convection, and viscous effects. This is particularly important here because, at the start of water discharge, the liquid does not instantly reach a steady jet velocity but undergoes a distinct period of acceleration as the flow develops.

4.4.3 The Gas–Liquid Interface in VOF

The VOF method represents the proportions of water and air in each computational cell using volume fractions. Denoting the water volume fraction by α_w and the air volume fraction by α_a , we have:

$$\alpha_w + \alpha_a = 1 \quad (4-3)$$

Here, $\alpha_w = 1$ indicates pure water, $\alpha_w = 0$ indicates pure air, and cells with $0 < \alpha_w < 1$ lie near the numerical gas–liquid interface. The water volume fraction is transported with the flow:

$$\frac{\partial \alpha_w}{\partial t} + \nabla \cdot (\alpha_w u) = 0 \quad (4-4)$$

In mixed VOF cells, density is weighted by the volume fractions of the two phases:

$$\rho = \alpha_w \rho_w + (1 - \alpha_w) \rho_a \quad (4-5)$$

4.4.4 Air Compressibility and the Energy Equation

Water is modeled with a constant density of 998.2 kg/m³. Air is set to Ideal Gas, allowing its density to vary with absolute pressure and temperature:

$$\rho_a = \frac{p_{abs}}{RT} \quad (4-6)$$

The Energy Equation is also enabled to describe temperature changes during compressed-air expansion.

4.4.5 Turbulence Model

Large velocity gradients occur near the nozzle, so the SST k– ω turbulence model is used [7]. The turbulence model itself is not the subject of this paper. Its main role is to account for viscous and turbulent effects in the nozzle and external high-speed flow, bringing the numerical model closer to the actual flow than the completely inviscid Bernoulli approximation.

Table 4-3. Main physical models and material settings

Item	Setting
Solution type	Transient
Solver	Pressure-Based
Spatial formulation	2D Axisymmetric
Multiphase model	VOF
Primary phase	Air
Secondary phase	Water
Turbulence model	SST k– ω
Water density	998.2 kg/m ³
Air density	Ideal Gas
Surface tension	Not enabled
Gravity	9.81 m/s ² , in the +x direction
Energy Equation	Enabled

4.5 Initial and Boundary Conditions

At the initial time, water and air inside the bottle are specified according to the actual experimental conditions. The initial first-stage water volume is 625 mL, and the corresponding region is assigned a water volume fraction of $\alpha_w = 1$ using Patch. The remaining bottle interior and external computational region are initialized as air, with $\alpha_w = 0$. The external region is initially at ambient pressure.

Fluent represents pressure as operating pressure plus gauge pressure. In this paper, Operating Pressure is set to 101 325 Pa, and the initial air Gauge Pressure inside the bottle is 65 816 Pa. The initial absolute pressure inside the bottle is therefore:

$$p_{abs} = p_{op} + p_{gauge} = 101325 + 65816 = 167141 \text{ Pa}$$

The gauge pressure at the external Pressure Outlet is set to 0 Pa, corresponding to an ambient absolute pressure of approximately 1 atm. Distinguishing gauge pressure from absolute pressure is important because air is modeled as an Ideal Gas and its density is calculated using absolute pressure.

Table 4-4. Initial and boundary conditions

Region/parameter	Fluent setting	Value
Operating Pressure	Operating Pressure	101325 Pa
Initial air pressure inside the bottle	Gauge Pressure	65816 Pa
Corresponding absolute pressure	Calculated	167141 Pa
Initial water volume	Volume Fraction Patch	625 mL
External boundary	Pressure Outlet	Gauge pressure: 0 Pa
Bottle wall	Wall	No-slip
Axis of symmetry	Axis	—
Initial temperature	Initialization	300 K
Gravity	Enabled	9.81 m/s ² , +x direction

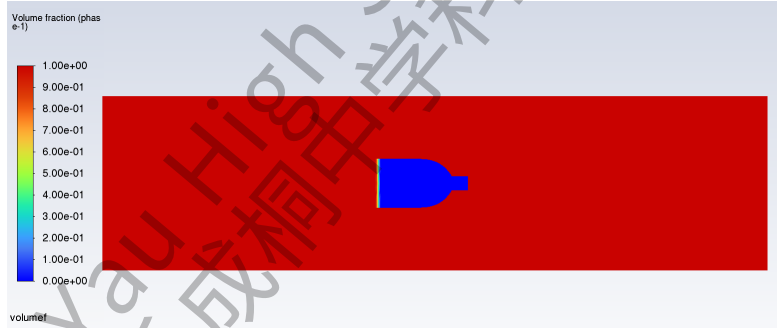


Figure 4-5. Initial air volume fraction distribution

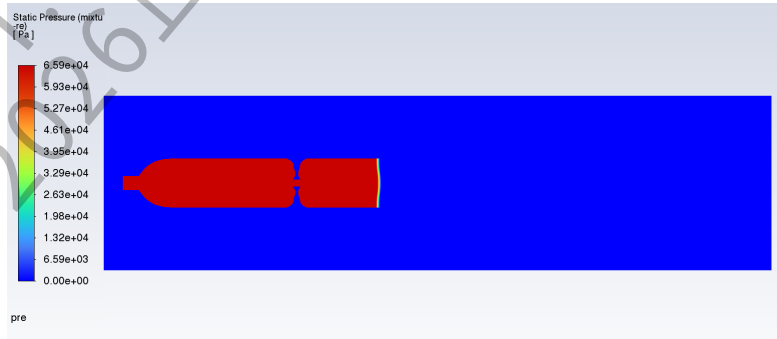


Figure 4-6. Initial static pressure distribution

Figure 4-5 is used to confirm the initial water volume and gas-liquid partition, while Figure 4-6 checks whether the pressurized region inside the bottle and the ambient pressure outside have been established as specified. Checking these two initialization results before the transient calculation prevents errors in geometric partitioning or pressure-reference settings from being carried into the calculation.

4.6 Solution Methods and Computational Settings

Water discharge from a water rocket changes rapidly over a short time, so a transient calculation is used. Pressure and velocity are coupled using PISO; pressure interpolation uses PRESTO!; the VOF volume fraction uses CICSAM [6] to keep the gas–liquid interface as sharp as possible; and the momentum, turbulence, and energy equations use second-order upwind discretization. Time discretization uses First Order Implicit, with a maximum of 30 iterations per time step.

Table 4-5. Numerical solution settings

Item	Setting
Pressure-Velocity Coupling	PISO
Gradient	Least Squares Cell Based
Pressure	PRESTO!
Volume Fraction	CICSAM
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind
Transient Formulation	First Order Implicit
Maximum iterations per time step	30

4.7 Time Step, Convergence, and Numerical Reliability

In the very short interval after water discharge begins, the liquid inside the bottle rapidly develops flow from rest, and the pressure, nozzle velocity, and gas–liquid interface position change most intensely. To improve the temporal resolution of this startup interval (200 time steps), an initial time step of approximately $1\ \mu\text{s}$ is used. Once the main flow has developed, the time step is adjusted to approximately $10\ \mu\text{s}$ to reduce the cost of the longer subsequent transient calculation.

Within each time step, both equation residuals and key monitored quantities are checked. Residuals below approximately 1×10^{-7} , except for continuity, are used as the main convergence reference. The pressure inside the bottle, outlet mass flow rate, and other quantities are also examined for appreciable drift during the iterations within the same time step. The calculation advances to the next time step only after both have stabilized.

4.8 Data Monitoring and Definitions of Output Quantities

To compare the CFD results with the conventional formulas relation by relation, a set of report quantities is saved continuously against Flow Time during the calculation. These quantities fall into two categories: those reported directly by Fluent, such as pressure, water mass, mass flow rate, and mean velocity, and those calculated from these basic quantities in post-processing, such as the thrust and acceleration used in this paper.

4.8.1 Mean Pressure Inside the Bottle and Remaining Water Mass

The pressure report inside the bottle is denoted by $P_r(t)$ and represents the mean static pressure over the specified report region. This quantity is recorded as gauge pressure. $P_r(t)$ is one of the main inputs used later to test the air-expansion relation and the pressure-difference–velocity relation.

The remaining water mass inside the bottle is denoted by $m_w(t)$. It is obtained by integrating the water volume fraction over the bottle interior:

$$m_w(t) = \int_{\Omega} \alpha_w \rho_w dV \quad (4-7)$$

This quantity directly reflects changes in the amount of water remaining during discharge and can also be compared with the time integral of mass flow rate to check conservation.

4.8.2 Outlet Mass Flow Rate and Mean Velocity

A report surface is defined at the nozzle cross-section to record the mass flow rate $qr(t)$ through it. The mass flow rate can be written as:

$$qr(t) = \int_{A_e} \rho(u \cdot n) dA \quad (4-8)$$

In Fluent, the mass flow rate sign is relative to the direction of the normal to the report surface. To make comparisons, outflow out of the bottle is always assumed to be positive in the plots and the following formulas.

The mean axial velocity at the nozzle outlet is denoted by $u_r(t)$. It defines a representative jet velocity, and compares it to the outlet velocity as calculated by the traditional Bernoulli formula. Remark that the actual velocity is not perfectly even throughout the nozzle, and therefore there is no single mean velocity which can possibly carry all the information regarding the spatial distribution of the velocity.

$$u_r(t) = \left(\frac{1}{A_e}\right) \int_{A_e} u_x dA \quad (4-9)$$

4.8.3 Physical Meaning of Thrust

For an ideal one-dimensional jet, a common expression for thrust is:

$$FT \approx \dot{m}u_e + P_{\text{ext}}A_e \quad (4-10)$$

Here, $P_{\text{ext}} = p_e - p_a$ denotes the nozzle outlet gauge pressure, where p_e and p_a are the outlet and ambient absolute pressures. The first term comes from the momentum carried by the discharged fluid, and the second from the difference between nozzle exit pressure and ambient pressure. For the nonuniform velocity distributions in CFD, a more rigorous momentum term requires integration over the entire nozzle cross-section, rather than simply multiplying two mean quantities.

To maintain consistency with the conventional model, the early data processing in this study used the available mass flow rate, representative velocity, and bottle-outlet pressure report to construct a thrust estimate Fr:

$$Fr(t) = |q_r(t)u_r(t)| + A_e P_{\text{ext}}(t) \quad (4-11)$$

In this case A_e represents the nozzle area and P_{ext} is the gauge pressure reported at the outlet of the bottle. Equation (4-11) uses this outlet gauge pressure; P_r remains the mean gauge pressure inside the bottle. The formula is used to add together the reported values to form a quantity that can be easily compared, rather than the resultant wall force of direct integration of all walls of the bottle in Fluent.

4.8.4 Derived Acceleration

To further examine how the decrease in water mass affects the overall response, this paper also calculates a derived acceleration $a_r(t)$ from Fr and the instantaneous total mass during first-stage water discharge:

$$a_r(t) = \frac{F_r(t)}{m_{\text{fixed}} + m_w(t)} \quad (4-12)$$

Here, m_{fixed} represents the structural and payload mass that does not change as water leaves during first-stage discharge (first-stage dry mass + second-stage wet mass + sensor mass, $m_{\text{fixed}} = 0.8205 \text{ kg}$), and $m_w(t)$ is the remaining water mass obtained from CFD. Gravity has not been subtracted from this acceleration, so it cannot be directly equated with the kinematic acceleration recorded by the flight sensors. Before the experiment-CFD comparisons later in the paper, the positive direction and treatment of gravity are made consistent.

5 Conventional Formulas and Methods of Comparison

The preceding experiments and simulations provide two types of information: the sensors record the motion of the rocket body, while CFD records changes in the internal fluid. We next use the CFD

pressure, velocity, mass flow rate, and water mass to examine each step of the conventional formulas. We seek to identify which assumptions become inappropriate, when this happens, and which corrections can explain the original deviations.

5.1 CFD Data and Notation

The final CFD report files contain 23 678 time points, covering 0–0.23497 s. Outlet velocity is denoted by u to avoid confusion with the flight velocity of the rocket body, and mass flow rate is denoted by q , in kg/s.

Table 5-1. Main symbols used in the following sections

Symbol	Meaning	Unit
P_r	Mean gauge pressure inside the bottle from CFD	Pa
P_{ext}	Nozzle outlet gauge-pressure report from CFD	Pa
P_g, P_a	Absolute gas pressure and ambient absolute pressure in the formulas	Pa
u_r, q_r	CFD velocity and mass flow rate reports	m/s, kg/s
$m_{w,r}$	CFD water mass	kg
V_g, V_b	Gas volume and total internal bottle volume	m ³
A, A_b	Nozzle area and representative bottle cross-sectional area	m ²
ρ_l	Liquid-water density	kg/m ³
F_r, a_r	Derived CFD thrust and acceleration report quantities	N, m/s ²

The subscript r indicates values read from the existing CFD files. Except for the outlet pressure report P_{ext} , quantities without this subscript are obtained from the corresponding formulas; other subscripts denote experimental values, conventional formulas, or corrected calculations.

5.2 Basic Assumptions and Calculation Relations of the Conventional Model

The conventional formulas first assume that the space inside the bottle is occupied only by water and air. Each volume of water discharged produces an equal increase in air volume. Thus, once the remaining water mass is given, the gas volume is

$$V_g = V_b - \frac{m_w}{\rho_l}. \quad (5-1)$$

Assuming that no gas leaves the bottle and that its expansion is approximately adiabatic gives

$$P_g = P_{g0} \left(\frac{V_{g0}}{V_g} \right)^\gamma \quad \gamma = 1.4. \quad (5-2)$$

Here, P_{g0} and V_{g0} are the initial absolute gas pressure and gas volume. As the air volume increases, the volume ratio in the formula decreases, giving a lower calculated pressure.

The second step uses the Bernoulli relation to convert the pressure difference into outlet velocity. The cross-sectional area correction between the internal bottle velocity and nozzle velocity, derived from the continuity equation, is retained here:

$$\beta_A = 1 - \left(\frac{A}{A_b} \right)^2 \quad u_B = \sqrt{\frac{2(P_g - P_a)}{\rho_l \beta_A}}. \quad (5-3)$$

Here, u_B is the outlet velocity calculated using the original formula. In this experiment, β_A is approximately 0.99447, close to 1, so this geometric correction is small. The more important assumption in Eq. (5-3) is that the flow velocity has already developed as soon as the current pressure difference is specified.

The third step treats the nozzle as a liquid cross-section with uniform velocity and density to obtain the mass flow rate, and then calculates the remaining water mass from the cumulative outflow:

$$q_B = \rho_l A u_B \quad m_w(t) = m_{w0} - \int_0^t q_B(\tau) d\tau. \quad (5-4)$$

The simplified thrust and acceleration of the conventional model are:

$$F_B = q_B u_B \quad a_B = \frac{F_B}{m_{w,B} + m_{fixed}}. \quad (5-5)$$

Here, $m_{fixed} = 0.8205 \text{ kg}$.

5.3 Model Decomposition and Analysis of Error Sources

If pressure, velocity, and mass flow rate are calculated together and only acceleration is checked, errors from the preceding steps become mixed. For example, a mass flow rate that is too high and a velocity that is too low may still produce a product close to CFD. We therefore examine the relations separately before attempting to combine them.

Table 5-2. Practical comparisons after separating the formulas

Relation tested	Input read from CFD	Quantity calculated from formulas
Gas-expansion relation	Water mass and corresponding gas volume	Pressure
Pressure-difference-velocity relation	Pressure	Outlet velocity
Velocity-mass flow rate relation	Outlet velocity	Mass flow rate
Startup inertia relation	Velocity change over a short interval	Required pressure difference
Combination of several formulas	Water mass as input to the pressure formula	Pressure, velocity, mass flow rate, and a cumulative water-quantity check

For example, when testing the velocity formula, we substitute the CFD pressure directly so that errors in the pressure formula itself are not attributed to the velocity formula. If the corrected velocity is close to CFD, this indicates an improvement in the pressure-velocity relation.

A number of candidate mechanisms were assembled in the preliminary study. This paper focuses on four types of candidate explanation directly related to the observed deviations. It explains the reasons, methods, and results of the corrections in detail around four main issues: startup inertia, liquid-phase coefficients, two-phase relations, and gas loss.

5.4 Evaluation of the Corrections

We first compare whether the curves are too high, too low, or follow different trends within the same phase, and then summarize the differences using numerical errors. The chosen measure is the root mean square error, RMSE, which can be understood as the typical difference between the calculated and CFD curves; smaller values are better. Since the CFD time step differs between the early and later parts of the record, the calculation is weighted by the actual duration of each interval:

$$RMSE_x = \sqrt{\frac{1}{T} \int_{t_a}^{t_b} [x_{calc}(t) - x_r(t)]^2 dt}, \quad T = t_b - t_a. \quad (5-6)$$

When combining multiple variables, the error for each quantity is first divided by its full-record range, the difference between its maximum and minimum values, before the errors are combined for comparison. This prevents pressure from receiving greater weight simply because of its larger numerical values. Equal weights are assigned to pressure, water mass, mass flow rate, and velocity, while lower weights are assigned to the force and acceleration derived from them.

A correction must satisfy two requirements at the same time: it must actually reduce the numerical error, and it must not introduce physical contradictions such as negative mass.

6 Water Discharge in CFD

6.1 Comparison of Experimental and CFD Acceleration

The experiment and CFD are generally consistent in terms of the duration of the first-stage response, and its general decay pattern, but there are local spikes in the experimental data. The video of the flight inspection showed no apparent sudden shifts in motion in relation to these spikes, and thus they are tentatively assumed to be due to IMU measurement noise.

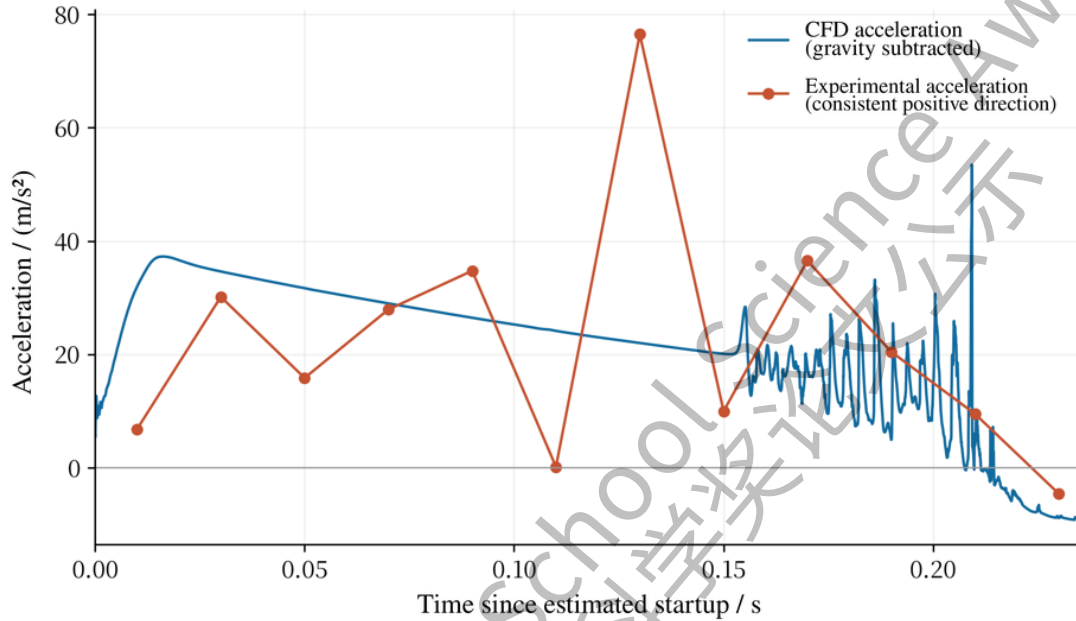


Figure 6-1. Experimental acceleration and CFD acceleration after gravity subtraction, with time aligned

Based on this, it can be inferred that, given the same initial conditions, the CFD simulation has the capability to give approximate first-stage water rocket discharge and can be used to further develop the conventional model.

6.2 Time Evolution of Flow Variables and Division into Phases

Figure 6-2 shows the original CFD curves. Initially, there is already an existing pressure, whereas velocity and mass flow rate gradually increase from values close to zero. This is followed by a comparatively smooth phase of liquid outflow: there is a reduction in pressure and the velocity and mass flow rate also lower gradually. The changes in the mass flow rate and velocity slowly increase in magnitude after about 0.152 s. Later in time, velocity increases are no longer necessarily accompanied by mass flow rate increases.

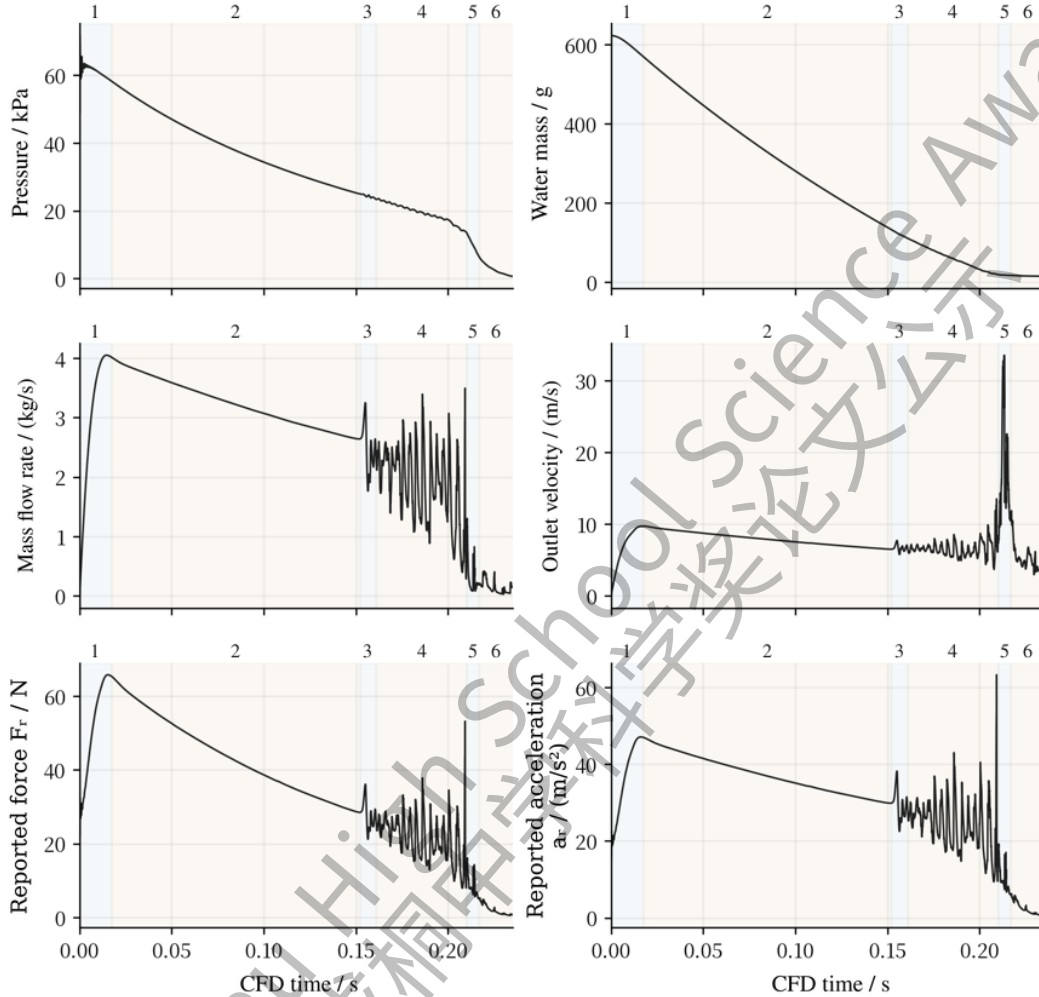


Figure 6-2. Original CFD pressure, water mass, mass flow rate, velocity, reported force, and reported acceleration. The background marks the six analysis phases

To examine these changes separately, we divide the analysis into six time intervals.

Table 6-1. Six phases of the current CFD time series

Phase	Time interval / s	Main features
1 Startup	0–0.0174	Pressure is established; liquid velocity and mass flow rate gradually increase
2 Continuous liquid-phase flow	0.0174–0.152	Pressure, velocity, and mass flow rate decrease relatively smoothly
3 Interface transition	0.152–0.161	The interface penetrates noticeably; the velocity–mass flow rate relation begins to change
4 Intermittent joint outflow	0.161–0.210	Mass flow rate and velocity fluctuate repeatedly; appreciable liquid outflow remains
5 High velocity with low mass flow rate	0.210–0.217	Velocity peaks while mass flow rate remains low
6 Tail of the record	0.217–0.23497	Overall decay; water and nonzero mass flow rate remain at the final recorded point

6.3 Phase-Dependent Changes in the Gas-Liquid Interface and Flow Field

The data curves show the overall changes, while the Fluent animations further reveal the spatial evolution of the gas-liquid interface. Figure 6-3 presents six representative times.

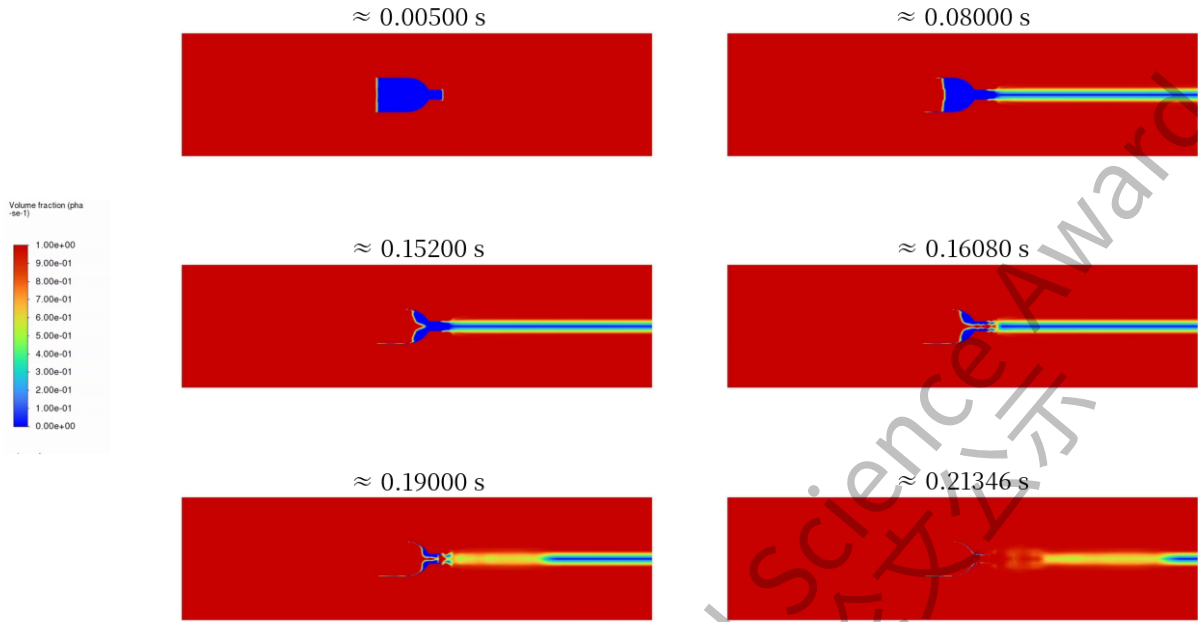


Figure 6-3. Key frames from the gas-liquid volume fraction video

The shrinkage of the water body occurs gradually at early times. Then later the air penetrates farther toward the outlet and the form of the left-over water is different. This indicates that a different condition exists between water left in the bottle and water that leaves the outlet. The traditional equations involve just the remaining amount of water and are unable to distinguish between a water constantly enveloping the outlet and already creating separate residual formations. This aligns with past studies which have indicated that the outlet can fill up with gas before all the water has been emptied [9].

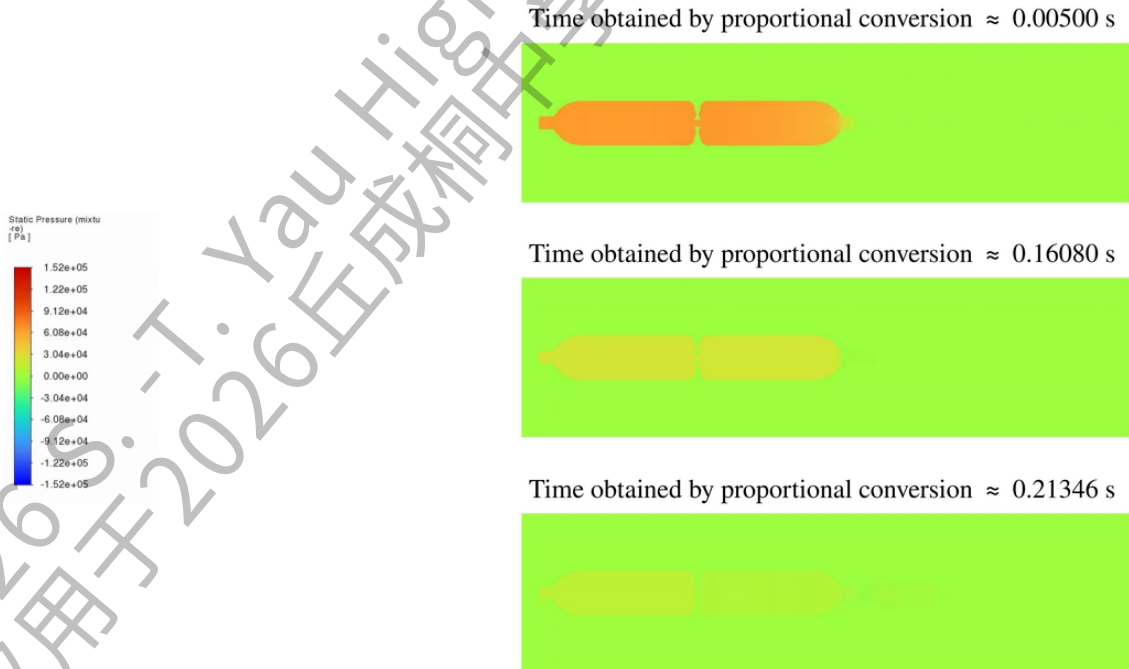


Figure 6-4. Key frames from the pressure video, in Pa. Pressure is not completely uniform in space

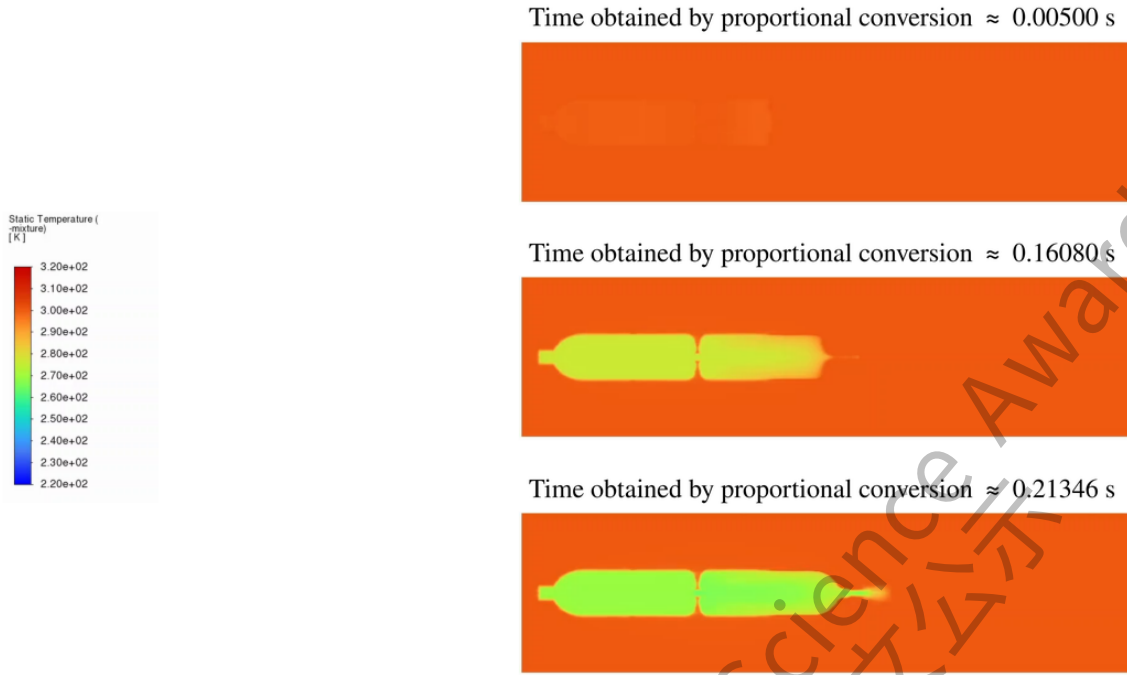


Figure 6-5. Key frames from the temperature video, in K

7 Corrections to the Formulas

7.1 Corrections for Liquid Inertia and Flow Losses During Startup

Near 0.001 s, the CFD pressure is approximately 59.76 kPa, while the velocity is only 1.35 m/s. Substituting the same pressure directly into the original Bernoulli formula gives approximately 10.96 m/s, which is clearly too high. The liquid is still accelerating at this point, but the original formula already treats it as though the flow velocity has developed.

This discrepancy points first to unsteady inertia, namely the pressure difference required to accelerate the liquid. We therefore replace the original direct correspondence between pressure difference and kinetic energy with a balance in which the pressure difference establishes the flow, overcomes losses, and maintains outlet kinetic energy. The relation between pressure difference and the known velocity is then

$$\Delta P \approx \frac{\rho_l}{2}(\beta_A + K)u^2 + I \frac{du}{dt}. \quad (7-1)$$

Only two terms have been added to the original relation. $K\rho_l u^2/2$ represents all flow losses combined: the larger K is, the greater the pressure difference required for the same velocity. The term $I du/dt$ represents liquid inertia and increases as the liquid accelerates more rapidly. Here, K is dimensionless and I has units of $\text{Pa} \cdot \text{s}^2/\text{m}$. This unsteady pressure balance is consistent with the basic idea of liquid-inertia analysis [3,11]; the two coefficients are fitted to the CFD data here.

Note that the physical meaning of this formula is to infer the required pressure difference from a known outlet velocity.

Differentiating a fluctuating velocity point by point can amplify noise. Time is therefore divided into short windows of 1 ms, and the mean pressure and velocity change are compared within each window:

$$\overline{\Delta P} \approx \frac{\rho_l}{2}(\beta_A + K)\overline{u_r^2} + I \frac{u_r(t_b) - u_r(t_a)}{t_b - t_a}. \quad (7-2)$$

The overbar denotes an average over the same time window.

A joint fit over 0–0.152 s gives $K = 0.26254$ and $I = 47.995 \text{ Pa} \cdot \text{s}^2/\text{m}$. Substituting these into Eq. (7-2), with water density approximated as 1000 kg/m^3 , gives:

$$\overline{\Delta P} \approx \frac{1000}{2}(0.99447 + 0.26254)\overline{u_r^2} + 47.995 \frac{u_r(t_b) - u_r(t_a)}{t_b - t_a}. \quad (7-3)$$

For example, during 1–2 ms, the CFD outlet velocity rises from approximately 1.354 m/s to 2.631 m/s, giving a mean rate of change of outlet velocity of approximately 1278 m/s². Substitution into the corrected formula gives an inertial pressure difference of approximately 61.32 kPa to accelerate the liquid, while the velocity-squared and flow-loss terms together correspond to approximately 2.61 kPa. The total driving pressure difference given by the corrected formula is therefore approximately 63.93 kPa, close to the CFD mean pressure difference of approximately 62.72 kPa. This shows that, early in startup, the outlet velocity is still low but the liquid is accelerating rapidly and requires a large inertial pressure difference. The original quasi-steady formula lacks this term and therefore predicts a pressure difference much smaller than CFD when using only the low outlet velocity.

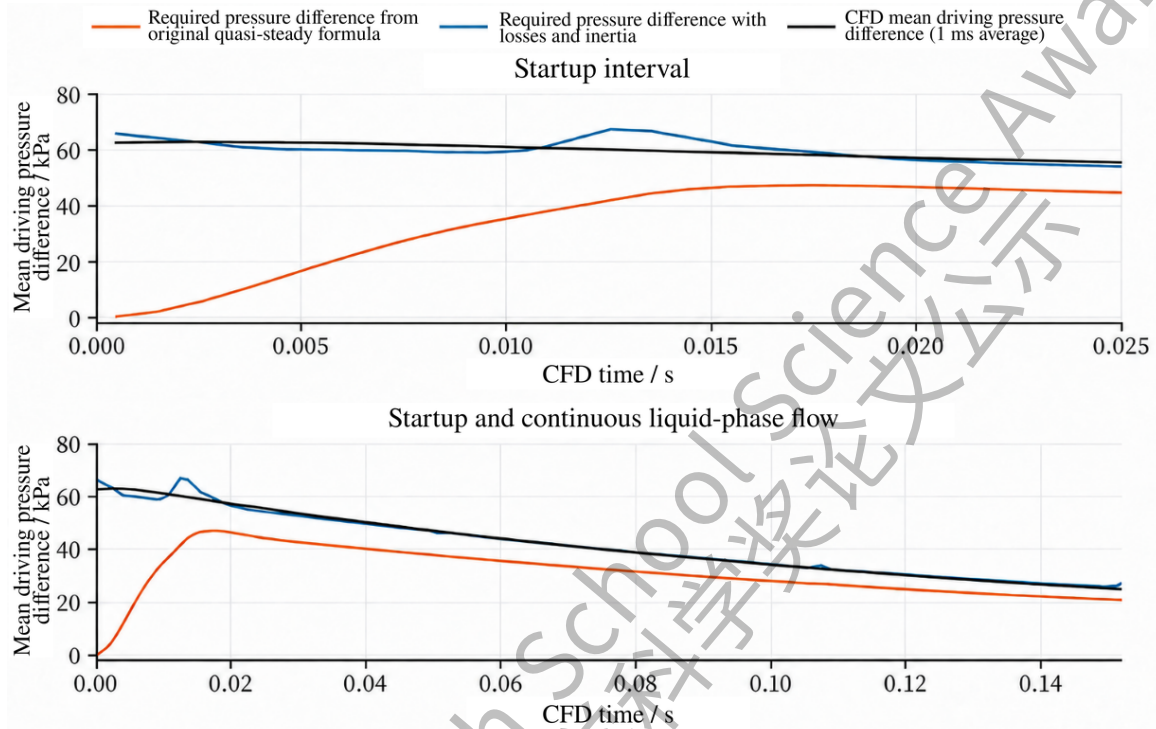


Figure 7-1. Comparison of the required pressure difference before and after adding inertia and losses. The upper panel enlarges the startup interval; the lower panel covers 0–0.152 s

Table 7-1. RMSE of the required pressure difference over 1 ms windows after adding different terms, in kPa

Fitting interval	Original quasi-steady formula	Losses only	Inertia only	Both terms
Startup phase 1	38.060	31.919	6.884	2.870
Startup and liquid-phase flow, 1 + 2	14.612	11.491	8.652	1.160

Each row is refitted over its corresponding interval, so the coefficients fitted to startup alone differ slightly from those in Eq. (7-3). The most direct conclusion is that adding flow losses alone cannot explain the slow development of flow during startup; liquid inertia is the main source of the error in this interval. Adding both terms reduces the pressure error over phases 1 + 2 by approximately 92.1%.

This result was also checked against the averaging duration. Changing the window from 0.5 ms to 4 ms changes the fitted I over phases 1 + 2 only from approximately 47.95 to 48.14, while K remains close to 0.2625. Applying the same pure-liquid relation to the full process does not produce a comparable improvement, however. The inertia correction is therefore clearly applicable only to the early flow.

7.2 Coefficient Corrections for Velocity and Mass Flow Rate During Continuous Liquid-Phase Flow

7.2.1 Outlet Velocity Coefficient Correction

During 0.0174–0.152 s, the velocity has developed and both pressure and velocity change relatively smoothly. In this interval, the original velocity formula mainly has an overall upward bias; the curve shape is not entirely wrong. We therefore retain the square-root relation and add only a velocity coefficient C_v :

$$u_{calc} = C_v \sqrt{\frac{2P_r}{\rho_l \beta_A}}, \quad C_v = 0.90169. \quad (7-4)$$

During the continuous liquid-phase interval in the CFD data, the reported velocity is approximately 90.2% of that calculated using the conventional velocity–pressure-difference relation. This may represent overall losses, contraction and velocity averaging; which one of these is the major component remains to be determined.

At 0.08 s, for example, the CFD pressure is 38.902 kPa. The original formula gives 8.845 m/s; multiplying by 0.90169 gives 7.976 m/s, while the CFD velocity is 7.986 m/s. The correction will not affect the general trend of deceleration associated with pressure drops, but will reduce most of the proportional bias.

7.2.2 Mass Flow Rate Coefficient Correction

Even with the use of the outlet velocity of the CFD directly, the original mass flow rate formula $q = \rho_l A u_r$ is slightly high in this interval. Thus, there is a need for a separate correction of the mass flow rate relation. In this case the flux coefficient is referred to by C_q :

$$q_{calc} = C_q \rho_l A u_r, \quad C_q = 0.90487. \quad (7-5)$$

Again taking 0.08 s, substituting the CFD outlet velocity into the original mass flow rate formula gives 3.613 kg/s; multiplying by 0.90487 gives 3.269 kg/s, while the CFD mass flow rate is 3.268 kg/s. This correction shows that the uncorrected calculated value at this time is approximately 10% higher than the CFD report.

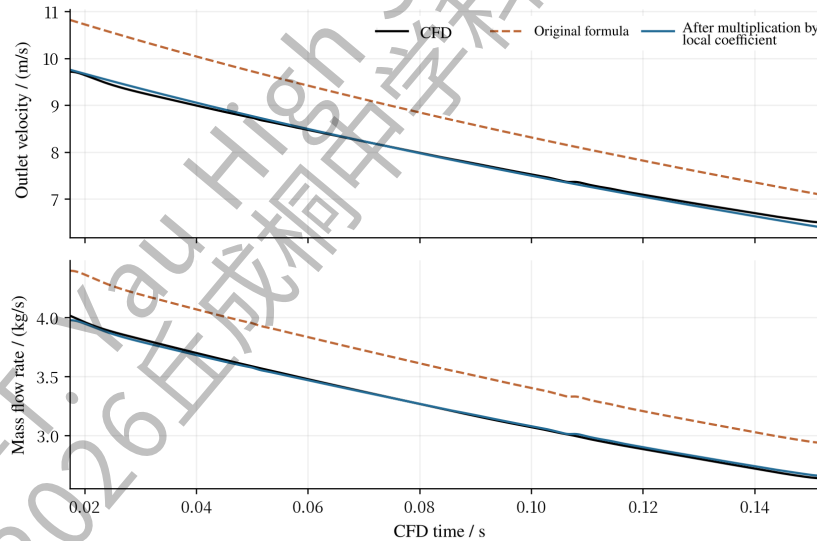


Figure 7-2. The upper panel calculates outlet velocity from CFD pressure; the lower panel calculates mass flow rate from CFD outlet velocity

Table 7-2. Errors after separate corrections to velocity and mass flow rate

Quantity tested	Original RMSE	Corrected RMSE	Error reduction	Corrected R^2
Velocity / (m/s)	0.8723	0.0480	94.5%	0.9973
Mass flow rate / (kg/s)	0.3441	0.01485	95.7%	0.9986

So, during the liquid-phase interval the most reasonable way is the use of the original physical framework and separate velocity and mass flow rate coefficients representing two effective coefficients in all. They are relatively stable in this part of the record with extensive time-block resampling yielding a velocity coefficient range of ~0.9000–0.9032 and a mass flow rate coefficient range of ~0.9037–0.9061.

7.3 Breakdown of the Pure-Liquid Approximation After the Gas–Liquid Transition

7.3.1 Testing Mass Flow Rate at Later Times

When the CFD outlet velocity at its peak is substituted into the liquid-phase mass flow rate relation (0.21346 s, phase 5), the CFD velocity is 33.522 m/s. The formula gives a mass flow rate of approximately 15.16 kg/s; even after multiplication by the liquid-phase correction coefficient, it remains approximately 13.72 kg/s. The CFD report is only 0.111 kg/s, a difference of more than two orders of magnitude.

This difference shows that the velocity–mass flow rate relation in this phase can no longer be described by a fixed proportional coefficient. The material flowing at high speed may be predominantly low-density gas, or the mean velocity may no longer represent the water carrying most of the mass. Continuing to treat the entire outlet area and the entire mean velocity as pure-water flow will severely overestimate mass flow rate.

7.3.2 Differences in the Mass Flow Rate Coefficient Between Phases

To show this change, we rearrange Eq. (7-5) and calculate

$$C_{q,obs}(t) = \frac{q_r(t)}{\rho_l A u_r(t)}. \quad (7-6)$$

This is a ratio formed from two existing CFD report quantities. If the original approximation of uniform pure-water flow holds, it should be close to 1. If a fixed loss coefficient is sufficient to describe the relation, the ratio should remain at the same constant value over an extended interval.

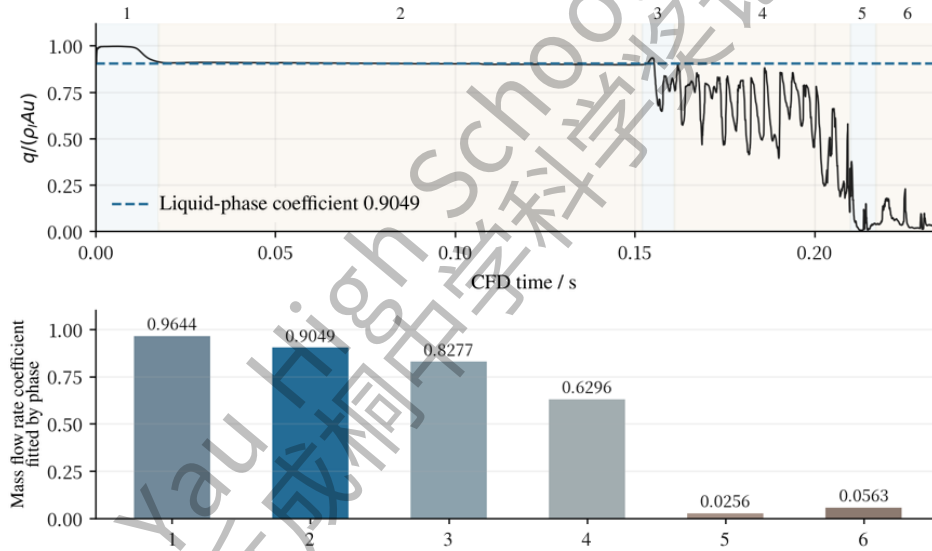


Figure 7-3. The upper panel shows the ratio of the CFD-reported mass flow rate to the value calculated from the pure-water relation, $q_r/(\rho_l A u_r)$; the lower panel shows the mass flow rate coefficients obtained separately for the six phases

The coefficient is approximately 0.9049 in phase 2, falls to approximately 0.6296 in phase 4, and is only approximately 0.0256 in phase 5. Clearly, C_q at later times is no longer the same constant as at early times.

7.3.3 Limitations of a Constant Effective Density

This study also tried substituting the same constant effective density into both the velocity and mass flow rate formulas. A combined fit gave approximately 588.5 kg/m³, reducing the full-record mass flow rate RMSE from 1.297 to 0.801 kg/s, but increasing the velocity RMSE from 2.890 to 4.319 m/s. In other words, lowering the density reduced the overprediction of mass flow rate while simultaneously increasing the calculated velocity and worsening its error.

Moreover, without other corrections, density cancels from the following product:

$$u^2 = \frac{2\Delta P}{\rho_{eff}\beta_A}, \quad qu = \rho_{eff} A u^2 = \frac{2A\Delta P}{\beta_A}. \quad (7-7)$$

Thus, the product of mass flow rate and velocity alone cannot determine whether the density is reasonable. Conversely, fitting density using only liquid-phase velocity data gives approximately 1230 kg/m³, which is mathematically equivalent to the velocity coefficient 0.90169. These results show that effective density can help test the formulas, but the current evidence is insufficient to treat it as the actual two-phase mixture density.

7.4 Pressure Correction Accounting for Changes in Gas Mass

7.4.1 Effect of Adjusting the Expansion Exponent

The original pressure relation reduces pressure only through an increase in gas volume and always treats gas mass as constant. The most direct correction is to replace the fixed exponent 1.4 with an exponent n to be fitted:

$$P_g = P_{g0} \left(\frac{V_{g0}}{V_g} \right)^n. \quad (7-8)$$

However, setting n to 1, corresponding to an isothermal approximation, increases the full-record pressure RMSE from 4.246 kPa to 11.515 kPa, making the result worse. Restricting n to 1–1.4 during fitting still gives a value near 1.4, with almost no benefit. When this range is relaxed, the best value is approximately 1.48455 and the full-record error falls to 3.837 kPa, but the phase 2 pressure error rises from 1.043 to 1.634 kPa.

Simply fitting the exponent therefore produces a compromise between phases without a clear physical interpretation.

7.4.2 Gas-Loss Correction and Its Effect

After the gas–liquid transition, gas may leave through the outlet. Pressure then falls for two reasons: gas volume increases and gas mass inside the bottle decreases. For the simplified case of a uniform ideal gas undergoing approximately adiabatic discharge, the mass and energy relations give

$$\frac{P_g}{P_{g0}} = \left[\frac{m_g}{m_{g0}} \frac{V_{g0}}{V_g} \right]^\gamma. \quad (7-9)$$

Compared with the original formula, this adds only the gas mass ratio. When $m_g = m_{g0}$, Eq. (7-9) reduces to the original adiabatic formula; when gas mass decreases, it predicts a faster pressure decline.

The current CFD setup has no separate report for gas-phase mass outflow. During the study, the difference between total discharged mass and discharged water mass was therefore selected as the discharged gas mass:

$$B(t) = \int_0^t q_r(\tau) d\tau - [m_{w,r}(0) - m_{w,r}(t)]. \quad (7-10)$$

This difference can reflect gas loss, but inconsistencies in report definitions and numerical errors may also introduce other contributions. We therefore do not directly treat $B(t)$ as the discharged gas mass, but introduce an effective attribution coefficient η to obtain the corrected expression

$$P_{calc} = P_{g0} \left[\frac{V_{g0}}{V_g(t)} \left(1 - \eta \frac{B(t)}{m_{g,ref}} \right) \right]^n - P_a. \quad (7-11)$$

Here, $m_{g,ref} = 0.00433659 \text{ kg}$ is the initial gas mass established under the condition that the initial gas temperature equals the ambient temperature. Fixing $n = 1.4$ and fitting only η gives $\eta = 0.83406$.

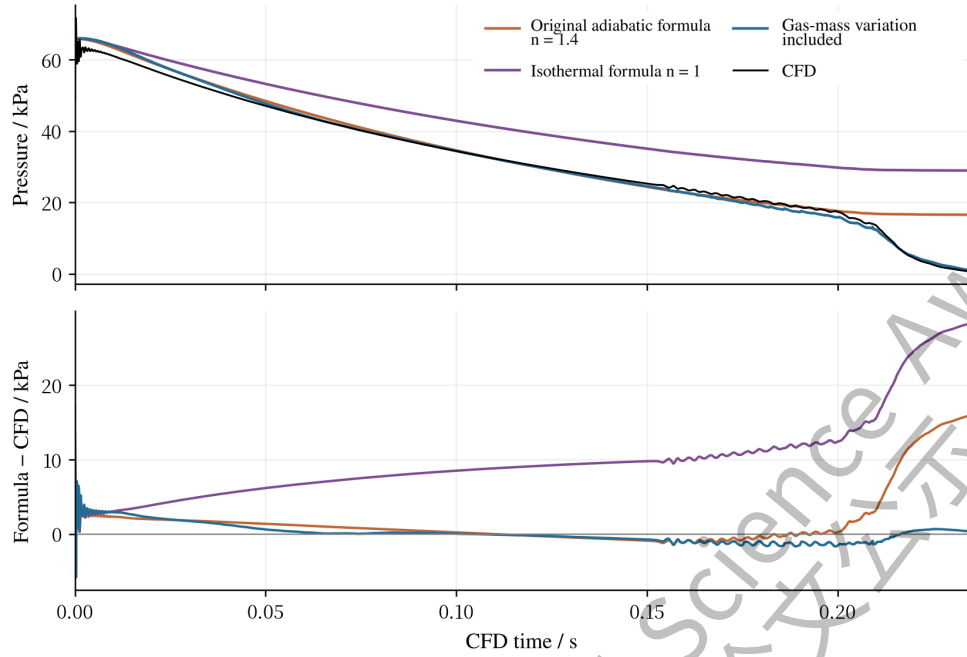


Figure 7-4. Comparison of pressure from the original adiabatic formula, the isothermal formula, and the gas-loss correction. The lower panel shows the formula result minus CFD

Table 7-3. Specific modifications to the pressure formula and their results

Formula	Effective parameters	Full-record pressure RMSE / kPa	Phase 2 RMSE / kPa
Original adiabatic formula	Fixed $n = 1.4$	4.246	1.043
Isothermal formula	Fixed $n = 1$	11.515	7.737
Fitted exponent	$n = 1.48455$	3.837	1.634
Gas-loss term added	$n = 1.4, \eta = 0.83406$	1.248	0.840
Gas-loss term and exponent fitted together	$n = 1.37879, \eta = 0.89353$	1.185	0.927

Adding only the gas-loss term reduces the full-record pressure error by approximately 70.6%. Fitting the exponent as well reduces the error by only a further approximately 5.0%, while worsening the phase 2 error. Fixing the exponent at 1.4 and adding a single loss coefficient therefore gives a more readily interpretable pressure correction.

7.5 Combined Corrections and Multivariable Tests During Liquid-Phase Flow

The preceding velocity test uses CFD pressure directly, while the mass flow rate test uses CFD velocity directly. The next question is whether several quantities can remain close to CFD simultaneously when these corrections are combined during the continuous liquid-phase interval. This correction fits two coefficients, one for velocity and one for mass flow rate. The formulas are:

$$\begin{aligned} V_g(t) &= V_b - \frac{m_{w,r}(t)}{\rho_l}, \\ P_{calc}(t) &= P_{g0} \left[\frac{V_{g0}}{V_g(t)} \right]^{1.4} - P_a, \\ u_{calc}(t) &= 0.88940 \sqrt{\frac{2P_{calc}}{\rho_l \beta_A}}, \\ q_{calc}(t) &= 0.90964 \rho_l A u_{calc}(t). \end{aligned} \quad (7-12)$$

At the beginning of the liquid-phase interval, $t_s = 0.0174$ s, the known CFD water mass of 0.5702026 kg is used as the starting value for integration:

$$m_{w,check}(t) = 0.5702026 - \int_{0.0174}^t q_{calc}(\tau) d\tau. \quad (7-13)$$

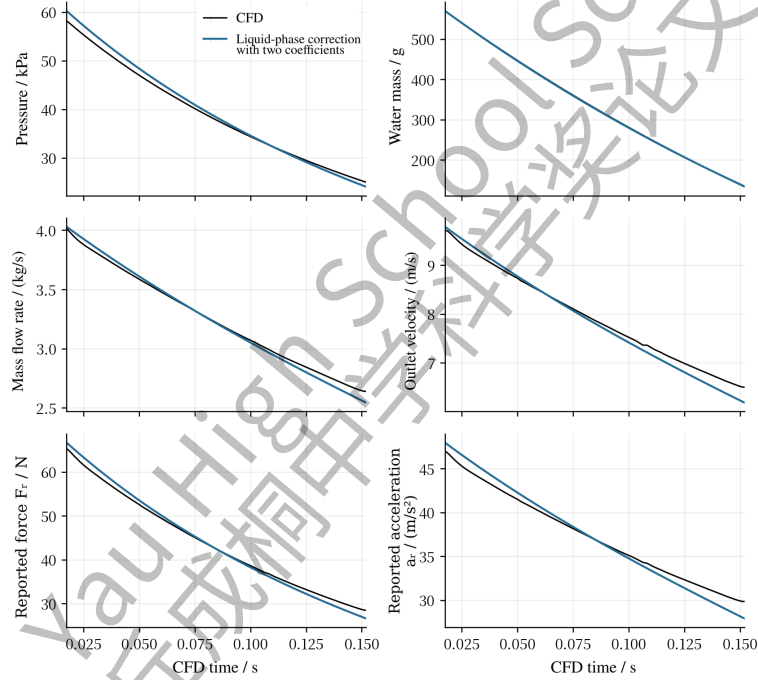


Figure 7-5. Comparison of the liquid-phase correction with two coefficients against CFD during continuous liquid-phase flow

Table 7-4. Results of the liquid-phase correction with two coefficients over 0.0174–0.152 s

Variable	RMSE	R ²
Pressure	1.043 kPa	0.9879
Velocity	0.1350 m/s	0.9784
Mass flow rate	0.03703 kg/s	0.9910
Water mass	1.051 g	0.99993
F_r	1.0207 N	0.9904
a_r	0.8933 m/s ²	0.9662

At the end of phase 2, this combined correction gives a water mass of 134.360 g, compared with 133.199 g in CFD, a difference of approximately 1.161 g. The mass flow rate integral for this phase is approximately 0.267% lower than CFD, while the F_r integral is approximately 0.121% higher. These results show that, during continuous liquid-phase flow, two coefficients can already describe the relationships among several interconnected quantities reasonably well at the same time.

7.6 Applicability of Constant-Coefficient Combinations Over the Full Process

This study also tried applying the pressure exponent, velocity coefficient, and mass flow rate coefficient together over the full process. The full-record correction without a water-quantity constraint gave $n = 1.54538$, $C_v = 0.92449$, and $C_q = 0.83626$. It brought some curves closer to CFD, but integrating to the final recorded point gave a water mass of -6.415 g. It therefore cannot be accepted as a correction.

The fit was then repeated with the requirement that the cumulative discharged water quantity not exceed the initial water quantity, giving a full-record correction with a water-quantity constraint. Its parameters are $n = 1.55106$, $C_v = 0.92972$, and $C_q = 0.82493$. Although negative water mass no longer occurs, the final value is approximately zero, whereas CFD still contains 14.777 g. The late velocity peak also remains unexplained.

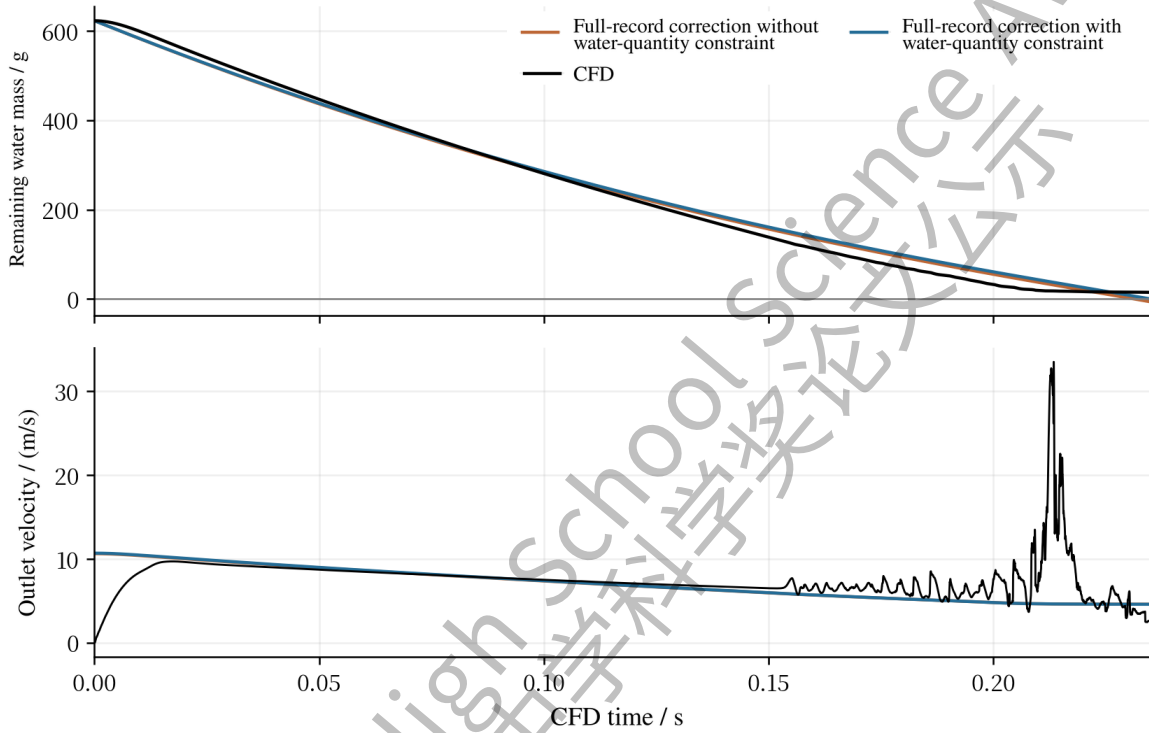


Figure 7-6. Comparison of two full-record constant-coefficient correction combinations with CFD

Table 7-5. Errors of the combined correction with a water-quantity constraint

Full-record variable	Conventional formula combination RMSE	Full-record correction with water-quantity constraint RMSE	Change
Pressure / kPa	4.246	4.087	Decrease of 3.7%
Water mass / g	130.377	16.884	Decrease of 87.0%
Mass flow rate / (kg/s)	1.297	0.746	Decrease of 42.5%
Velocity / (m/s)	2.890	2.936	Increase of 1.6%
F_r / N	14.992	8.458	Decrease of 43.6%
a_r / (m/s ²)	12.987	7.658	Decrease of 41.0%

The physical meaning of this failure is that early flow losses can be summarized by constants, but the flowing material and outlet velocity distribution change at later times, so the same set of constants cannot simply be fitted over the entire process. In particular, the worsening velocity error alongside improvements in other quantities shows that the fit is trading errors in some variables for improvements in others.

8 Discussion

8.1 Applicable Phases and Physical Meaning of the Corrections

The results can be summarized by physical process. Liquid inertia and flow losses need to be retained during startup. During continuous liquid-phase flow, the Bernoulli-type structure can be retained, with

separate velocity and mass flow rate coefficients summarizing the main deviations. At later times, pressure requires a correction for decreasing gas mass, rather than merely a change in the expansion exponent of a closed gas system.

Table 8-1. Conclusions for the individual corrections

Specific correction	Test result	How it is retained
Liquid inertia and flow losses added to the pressure formula	Marked improvement in early pressure agreement	An explanation based on early liquid inertia
Velocity multiplied by C_v ; mass flow rate by C_q	Phase 2 errors reduced by 94.5% and 95.7%, respectively	Local liquid-phase relations
Density reduced uniformly over the full record	Mass flow rate improves but velocity worsens	Not adopted
Gas mass ratio added to the pressure formula	Marked reduction in full-record pressure error	A relation for pressure at later times
Full process fitted with three constants	Negative water mass or late velocity mismatch	Not adopted

These results also show that the corrections K and C_v should not be superimposed, because they are different ways of describing similar losses. The combination of inertia and losses during startup and the constant corrections during liquid-phase flow can explain the data within their respective intervals of applicability. They have not yet been joined with the later two-phase flow into a tested relation covering the full process.

8.2 Unresolved Physical Mechanisms and Numerical Errors

The most difficult features to explain at present are the short pulses after the gas-liquid transition. Simple corrections to the conventional velocity and mass flow rate formulas clearly cannot describe the numerical oscillations in the CFD results. However, integrating the CFD mass flow rate and acceleration shows that, although outlet velocity and acceleration frequently oscillate and reach large values in phases 4 and 5, the peaks are extremely brief, so these phases contribute much less to the total integrals than the continuous liquid-phase interval. This suggests a direction for future work: further corrections for the late flow need not reproduce every feature of the CFD curves, provided that the integrals of the key quantities agree with CFD.

Table 8-2. Cumulative contributions by physical phase

Phase	Decrease in water mass / g	Thrust integral / (N · s)	Share of full-record thrust integral
1	52.912	0.9044	11.05%
2	437.004	5.8925	71.97%
3	22.447	0.2506	3.06%
4	91.686	1.0466	12.78%
5	1.960	0.0586	0.72%
6	2.328	0.0348	0.43%

9 Conclusions

This study is conducted in three directions – experimental records, Fluent results for the internal flow and simplified theory, wherein special emphasis is laid on the modification of the conventional calculation method. There are the following main conclusions.

Firstly, a quasi-steady pure-liquid approximation cannot be applied during the whole period of water discharge. Pressure is already there at start up time and the liquid velocity is still in progress. The RMSE of the 1 ms averaged pressure over 0-0.152 s for an addition of inertia and losses was reduced to 1.160 kPa from 14.612 kPa, representing that startup inertia is needed.

Secondly, the original formulations are still applicable in continuous flow of liquid phase. With CFD pressure being used directly, multiplying the velocity formula by approximately 0.9017 reduces the error by approximately 94.5% and with CFD velocity being used directly, multiplying the mass flow rate

formula by approximately 0.9049 reduces the error by approximately 95.7%. The merged result of several relations locally for the liquid-phase interval is an end-of-phase water mass deviating by about 1.161 g from CFD, indicating that just few coefficients are able to capture the principal changes in this interval.

Third, at later times, the mean outlet velocity can no longer be interpreted as the velocity of a uniform pure-water outflow. The formula for pure water yields a mass flow rate of about 15.16 kg/s at the velocity max, while CFD yields a mass flow rate of just 0.111 kg/s.

Fourth, using the expansion exponent does not explain late behavior of pressure. Adding the gas-loss term reduces the full-record pressure RMSE from 4.246 to 1.248 kPa, a greater reduction than adjusting the expansion exponent alone.

Fifth, cumulative results must be checked when selecting formulas. Some full-record constant-coefficient combinations with relatively low errors nevertheless give negative water mass and should be rejected. The integrated results show that full-record totals can be insensitive to some millisecond-scale fluctuations, but the mean levels during gas-liquid mixing and the cumulative contributions of the individual phases must still be retained.

The current outcome is therefore a set of correction relations with clearly defined phases of applicability, together with the rejection of several corrections that show clear limitations on this dataset. It identifies which parts of the original formulas can be retained and which require new information. A correction method that describes the complete flow process remains to be developed.

References

- [1] Prusa J M. Hydrodynamics of a water rocket[J]. SIAM Review, 2000, 42(4): 719-726.
- [2] Finney G A. Analysis of a water-propelled rocket: A problem in honors physics[J]. American Journal of Physics, 2000, 68(3): 223-227.
- [3] Gommès C J. A more thorough analysis of water rockets: Moist adiabats, transient flows, and inertial forces in a soda bottle[J]. American Journal of Physics, 2010, 78(3): 236-243.
- [4] Romanelli A, Bove I, González Madina F. Air expansion in a water rocket[J]. American Journal of Physics, 2013, 81(10): 762-766.
- [5] Hirt C W, Nichols B D. Volume of fluid (VOF) method for the dynamics of free boundaries[J]. Journal of computational physics, 1981, 39(1): 201-225.
- [6] Ubbink O, Issa R I. A method for capturing sharp fluid interfaces on arbitrary meshes[J]. Journal of computational physics, 1999, 153(1): 26-50.
- [7] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications[J]. AIAA journal, 1994, 32(8): 1598-1605.
- [8] Marranghello G F, Lucchese M M, Rocha F S. Analysis of the propulsion phase of water rockets using smartphone sensors and video analysis[J]. The Physics Teacher, 2022, 60(6): 437-440.
- [9] Watanabe R, Tomita N, Takemae T. Thrust Characteristics of Water Rocket and Their Improvement[J]. Japan Society of Aeronautical Space Sciences, 2003, 51(593): 314-320.
- [10] Chang F Y, Weng H C. Dynamic characteristic analysis of a two-stage water rocket in flight [J][J]. Journal of the Chinese Society of Mechanical Engineers, 2020, 41(5): 605-614.
- [11] Johnston D N. Prediction of fluid inertance in nonuniform passageways[J]. 2006.

Appendix A Record of Experimental Materials

Experimental platform: two-stage water rocket with parachute and pressure-differential separator

Sensor kit: ESP-32 development board with Type-C port and CH340

GY-521 MPU6050 module

BMP388 digital barometric pressure sensor with IIC interface terminals

Micro SD card module with SPI interface

TF memory card (8G)

18650 lithium battery, 3.7 V, 2400 mAh

KCD1 push-button switch

Dupont connector socket (2 rows $\times$ 5P pins)

Custom EPE pearl-foam board (length 1 m $\times$ width 50 cm $\times$ thickness 30 cm)

Appendix B Data Acquisition Program

test.ino

Appendix C Experimental Video Materials

rocket test.mp4

Note: The 2 atmospheres mentioned in the video is the pump pressure-gauge reading, which differs from the actual gauge pressure. When the rocket was pressurized with the pump and measured with a separate pressure gauge, a pump-gauge reading of 2 atm corresponded to an actual gauge pressure of 0.65 atm. This paper uses 0.65 atm as the initial gauge pressure.

Appendix D Data Post-Processing Programs

processdata.py

real a.py

Appendix E Flight Sensor Data

test1.csv

Appendix F Simulation Data and Animations

accel-rfile.out (CFD-derived acceleration)

average-p-rfile.out (mean pressure inside the bottle)

average-velocity-rfile.out (mean fluid discharge velocity)

flowrate-rfile.out (mass flow rate at the bottle outlet)

masswater-rfile.out (water mass inside the bottle)

p-gauge-rfile.out (gauge pressure at the bottle outlet, P_{ext})

thrust-rfile.out (derived thrust)

pressure.mov

temperature.mov

water.mov

致谢页

本研究起源于我们对航空航天的兴趣。我们平时经常关注火箭发射，也在课内物理学习中接触到了伯努利原理等流体力学知识。由此，我们开始好奇：能否利用已经学过的知识，建立一套描述水火箭运动的公式？更重要的是，这些公式究竟能在多大程度上描述真实水火箭的飞行过程？如果理论与实际存在差异，原因又是什么？带着这些问题，我们共同确定了研究选题。考虑到嵇海锐此前已有 CFD 仿真研究经验，我们决定结合实验、数值仿真与理论分析开展研究。

本研究的指导老师为于小雨老师，是本校物理教师，也是我们的 IYPT 指导老师，但并非我们的课内物理授课教师。老师为我们主要提供方向性意见，在论文撰写阶段未进行额外指导，指导为无偿指导。此外，在火箭飞行实验中，我们曾邀请徐梓桐同学协助实验操作。我们感谢老师和这位同学在研究过程中给予的帮助。

选题确定后，装置采购与火箭组装由我们共同完成。研究初期遇到的第一个困难，是火箭加压后出现漏气，无法正常开展实验。经过排查，我们发现所选瓶子的瓶口与两级连接器不匹配，更换后才解决这一问题。这让我们意识到，研究计划与实际实验之间，还隔着许多需要逐一处理的具体问题。

为了获取飞行数据，我们购买并组装了传感器系统，随后开展编程与调试。这部分工作主要由嵇海锐负责。过程中遇到的问题包括供电连接在振动条件下不稳定、IMU 零位漂移，以及数据写卡造成的程序阻塞等。我们分别通过调整连接方式、重新获取静止零位参考，以及调整数据读取和记录的程序结构进行处理，相关问题与处理过程已在论文第三部分中说明。

传感器调试与飞行实验交替进行，因为一些问题只有在实际飞行条件下才会暴露。袁昊琨主要负责火箭维护，试飞实验由我们共同完成。实验过程中，令我们印象最深的一次挫折，是火箭二级在飞行中突然偏转，挂在一棵很高的树上，当天未能取回。由于传感器全部安装在上面，我们一度担心研究会因此中断。直到第二天取回火箭，我们才发现，尽管经历了一夜雨淋，传感器仍在持续运行。这一结果让我们感到意外，也让我们对这套系统在此次意外中的表现有了更直观的认识。

获得一组可用于分析的飞行数据后，嵇海锐负责数据后处理；袁昊琨同步查阅文献，构建并核查传统理论计算方法。与此同时，嵇海锐开展 Fluent 仿真工作。这一阶段遇到的主要困难，是按照打气筒压力表的读数设置初始压力后，仿真结果与实验明显不符。我们最初记录的压力表读数为 2 atm，随后通过独立压力表复测，发现对应的实际读数约为 0.65 atm，并与商家确认了原压力表存在偏差。这一经历使我们认识到，测量设备给出的读数也需要核查，不能直接将理论与实验之间的所有差异归因于模型本身。

获得完整的仿真数据后，我们将其与后处理后的实验数据进行对比，评估二者的一致性，并在此基础上推进理论公式的修正与分析。公式修正工作主要由嵇海锐负责，袁昊琨则结合前期文献阅读与研究进展，起草文献综述。

理论分析完成后，我们开始集中撰写论文。袁昊琨负责文献综述初稿及前期相关草稿；嵇海

锐在这些草稿的基础上，完成实验、数值仿真和理论分析三部分的主体撰写；袁昊琨随后对论文内容进行审核、修正，并承担排版工作。上述分工贯穿研究与写作过程，双方在共同开展的工作中持续讨论，并根据实验和分析结果调整研究思路。

本研究使用的 AI 工具包括 ChatGPT 和 Gemini。我们在下文说明其具体用途：

在传感器系统搭建与调试阶段，由于我们缺乏电路连接和焊接经验，GPT-5.6 和 Gemini 3.6（和 3.1pro），在方案理解、接线问题及相关操作知识方面提供了重要帮助。我们也缺乏使用 Arduino IDE 编写程序的经验，因此，这两种工具为传感器程序编写提供了关键支持，并持续参与调试过程中有关程序问题的分析和修改。

在数据分析阶段，多份用于数据后处理和数据拟合的 Python 程序主要由上述 AI 工具辅助编写。具体涉及火箭轴向方向提取，原数据绘图等。这些工具在代码生成与修改方面承担了较多工作。

论文撰写阶段尝试过使用 GPT-5.6 检索论文并完成文献综述。但最终未采纳，检索到的文献对论文撰写构成参考。

在论文整理阶段，GPT-6 协助完成了 Word 文稿的 PDF 排版，包括图表、图注及相邻说明的分页调整，以及转换后的内容完整性检查。

我们感谢指导老师、协助实验的同学，以及研究过程中给予我们帮助的人。这项研究让我们体会到，从一个出于兴趣的问题出发，到获得数据、解释差异并形成论文，需要反复尝试，也需要如实记录那些未能奏效的尝试和得到的帮助。