Studying the Oscillating Path Of A Wristwatch Falling Through Water

Mechanical Engineering Department, Embry-Riddle Aeronautical University
Leonard Farrell and Dr. Birce Dikici (Advisor)

Abstract

This research focuses on studying the oscillating motion of a wristwatch as it falls through water. The fluid imposes lift and drag forces on the body subsequently influencing its path as it descends. The body is also imposing forces on the fluid. The central question of this research is: Can Computational Fluid Dynamics (CFD) programs be used to model the system and recreate the observed motion? The method to analyze this problem is to model the system in two CFD programs. Ansys Fluent and OpenFoam are used. The desired outcome of this stage is to achieve a simulated model in which the oscillating motion is reproduced and acquired subsequent numerical data.

Experimental Procedure

The steps taken to produce the motion in (Figure 1):

1. Gather equipment
   - GoPro HERO7
   - Casio F-105 digital wristwatch
   - 10lb Dive Brick
2. Place the brick at the pool's bottom , 4 feet from the wall
3. Place the GoPro on top of the brick facing the wall
4. Look at the GoPro screen and determine frame boundaries
5. Begin recording
6. Hold the wristwatch in these conditions:
   - Watch face pointing up
   - Long axis facing camera
   - Wrist bands held up to form V shape
   - In the center of the frame
   - At the frame's upper limit
7. Release the watch

Figure 1: Motion of wristwatch descending through pool water

Figure 2: GoPro HERO7 on top of 10lbs dive brick

Introduction

This is a complex problem because both the fluid and object are in motion through the process, so the properties of both are transient from start to finish. Fluid Dynamic principles are central to the analysis of this system. It is important to consider the possible causes of this motion. Computational Fluid Dynamics (CFD) programs should be used for further analysis.

Preliminary Considerations

The wristwatch can be modeled as a simple pendulum (Figure 4) with several key differences:

- Amplitude increases from equilibrium point rather than beginning at max
- Amplitude becomes constant rather than decreasing

Three key considerations for the possible causes of this motion are:

- The uneven mass distribution of the watch (Figure 3)
- Incident vortex formations (Figure 5 & 6)
- Lift and Drag forces (Figure 6)

Figure 3: Casio digital watch circuit board.

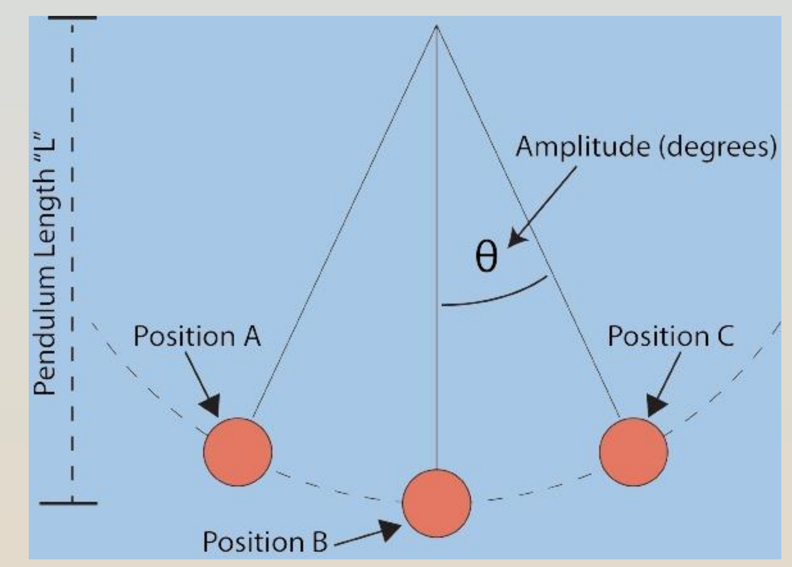

Figure 4: Annotated suspended pendulum diagram

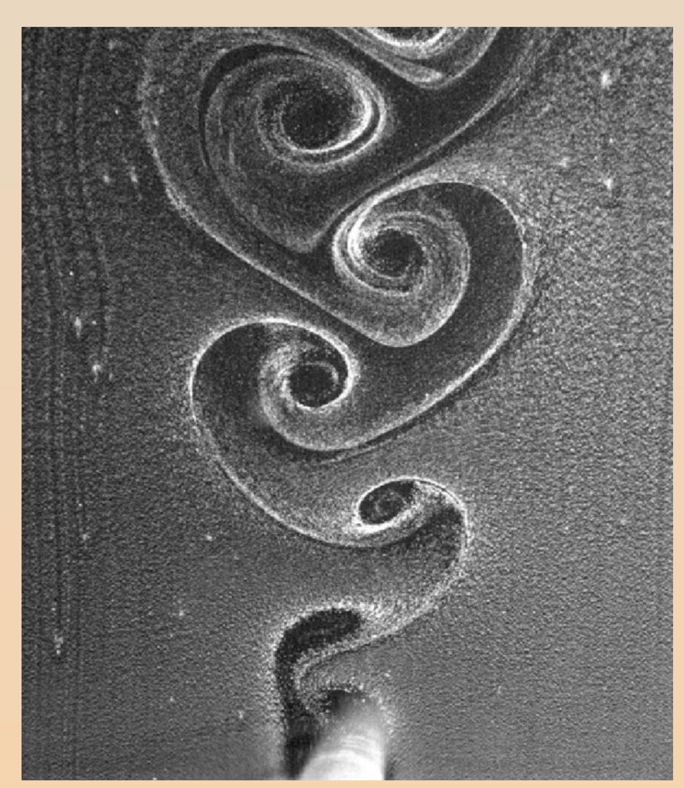

Figure 5: Von Kármán vortex street resulting from flow over a cylinder. (Huang, Luofeng & Benites, Daniela & Lyu, Shiyu & Smith, Tom & Li, Minghao & Shang, Yeru & Klettner, Christian. ,2019)

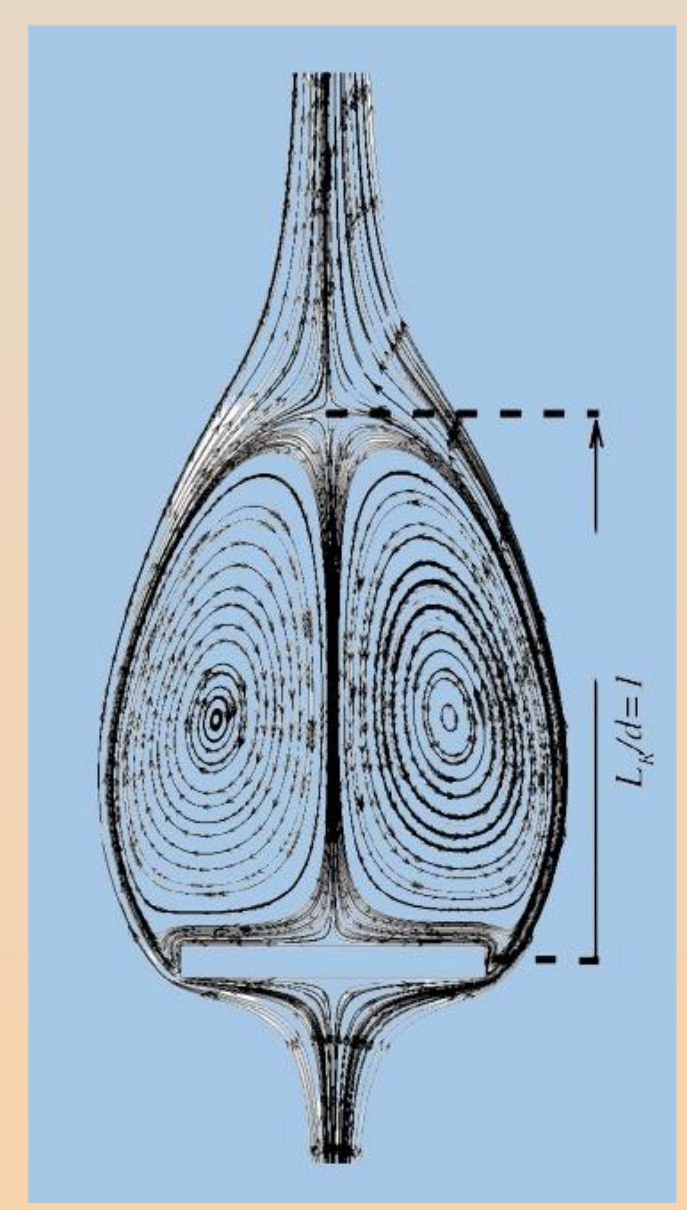

Figure 6: Incident vortices formed behind rectangular plate during normal flow. (Afgan, I., Benhamadouche, S., Han, X., Sagaut, P., & Laurence, D. , 2013)

Future Work

The next step is to simulate the system in a CFD solver. Ansys Fluent and OpenFoam are the chosen programs. Assuming ideal results after simulation, the main question is answered with an astounding "Yes!". Through testing variations of the original geometry, the following questions would be the future aims: In what ways does the object's geometry influence the path it takes? How can the geometry of the object be varied to produce different types of motion? Looking far into the future, the answers to these questions are invaluable to creating a program used to generate geometries to satisfy a user-defined descent path.

Navier-Stokes Equation

The Navier-Stokes equations come in many forms and derivations. They describe the motion of viscous fluids and can theoretically be applied to any fluid flow problem. This form provides a clear delineation between the main aspects involved in calculation. These equations are the core to CFD solvers.

$$\underbrace{\frac{\partial}{\partial t}\int_{\Omega} \rho v \, d\Omega}_{\text{VELOCITY RATE OF CHANGE}} + \underbrace{\int_{S} \rho v v \cdot n \, dS}_{\text{CONVECTIVE FLUX}} = \underbrace{\int_{S} \mathrm{T} \cdot n \, dS}_{\text{DIFFUSIVE FLUX}} + \underbrace{\int_{\Omega} \rho b \, d\Omega}_{\text{BODY FORCES}}$$

Figure 7: Navier-Stokes equation in integral form.

References

1. Afgan, I., Benhamadouche, S., Han, X., Sagaut, P., & Laurence, D. (2013). Flow over a flat plate with uniform inlet and incident coherent gusts. Journal of Fluid Mechanics, 720, 457-485. doi:10.1017/jfm.2013.25
2. Huang, Luofeng & Benites, Daniela & Lyu, Shiyu & Smith, Tom & Li, Minghao & Shang, Yeru & Klettner, Christian. (2019). UCL OpenFOAM Course Notes 2019. 10.13140/RG.2.2.18118.83529.
3. Patel, M. (2013). Hi-tech CFD. HiTechCFD. https://www.hitechcfd.com/cfd-knowledgebase/seven-stages-of-a-typical-cfd-simulation.html
4. Remmerie, W. AirShaper. (2019). Computational Fluid Dynamics Explained. https://www.youtube.com/watch?v=dyiREvdc4Gg&t=197s