Droplet impact on liquid pool: Secondary droplets formation from Rayleigh jet breakup and crown splash

A PROJECT REPORT

Submitted in partial fulfillment of the

requirements for the award of the degrees

of

BACHELOR OF TECHNOLOGY

In

MECHANICAL ENGINEERING

Submitted by:

Aditya Berojya (180003004)

Guided by:

Dr. Ankur Miglani, Assistant Professor

INDIAN INSTITUTE OF TECHNOLOGY INDORE

May 2022

CANDIDATE'S DECLARATION

I hereby declare that the project entitled "**Droplet impact on liquid pool: Secondary droplets formation from Rayleigh jet breakup and crown splash**" submitted in partial fulfillment for the award of the degree of Bachelor of Technology in **Mechanical Engineering** completed under the supervision of **Dr. Ankur Miglani**, **Assistant Professor**, **Department of Mechanical Engineering**, IIT Indore is an authentic work.

Further, I/we declare that I/we have not submitted this work for the award of any other degree elsewhere.

27/05/22

Signature and name of the student(s) with date

Aditya Berojya

CERTIFICATE by BTP Guide(s)

It is certified that the above statement made by the students is correct to the best of my/our knowledge.

Dr. Ankur Miglani
Assistant Professor
Department of Mechanical Engineering
Indian Institute of Technology Indore
Simrol, Khandwa Road, Indore 453 552, India

27 May 2022

Signature of BTP Guide(s) with dates and their designation

Dr. Ankur Miglani

Assistant Professor

Preface

This report on"Droplet impact on liquid pool: Secondary droplets formation from Rayleigh jet breakup and crown splash" is prepared under the guidance of Dr. Ankur Miglani.

Through this report, I have studied the influence of the physical properties of fluids on the Rayleigh jet formation and crown splash formation.

I have tried to the best of our abilities and knowledge to explain the content in a lucid manner. I have tried to provide images and references wherever needed to help the reader get a complete understanding of the topic.

Aditya Berojya

B.Tech. IV Year

Discipline of Mechanical Engineering

IIT Indore

ACKNOWLEDGEMENTS

I wish to thank Dr. Ankur Miglani for his kind support and valuable guidance. He has always believed in me and encouraged me to achieve my goals. It was his belief in me that I was able to make this much progress in this project in the given period of time. He has always supported and motivated me to learn new things and hence helped me build a better perspective of this discipline. I would also like to thank Shrirang Shivankar, PhD student under Dr. Ankur Miglani for his guidance and constant support throughout my BTP.

Further, I would like to thank my family and friends who motivated me to achieve my goals and bear the ignorance due to my dedication to the project. Without their support, this report would not have been possible.

Aditya Berojya

B.Tech. IV Year

Discipline of Mechanical Engineering

IIT Indore

Abstract

The study of droplet interaction with liquid films has immense importance in the cleaning of oil spills. India sits on one of the busiest sea trade routes in the world, almost the entire oil supply to south-east Asian countries and China happens through Indian waters. Although this has benefitted India economically, the oil spill accidents have caused massive damage to the ocean ecosystem. The recent Ennore oil spill that occurred near the shores of Chennai, Tamil Nādu is one of such case where a huge loss of aquatic life has been observed due to oil spilling in seawater. In this work, we are trying to understand impact of low-density droplets on high-density fluid pools. The setup for the experiment involves a high-speed camera collecting images of the droplet impacting on the pool, image processing yields images describing the state of the interface at every timestamp. We observed that the density and viscosity of the fluids play a crucial role on the post-impact behavior. At a high Weber number, the probability of a Rayleigh jet breakup increases. However if the viscosity of the pool liquid is higher than the droplet, in that case penetration of the interface becomes difficult and the probability of a crown splash increases, in such cases the probability of obtaining a crown splash increases as the Weber number increases. At a high Ohnesorge number, no jet eruption is observed and if in such a case the Weber number is increased then the crown splash phenomenon becomes more prominent to occur.

TABLE OF CONTENTS

List of figures

Introduction

The phenomenon of a droplet impacting over a pool of liquid is very common in nature and our surroundings, that's the reason why it has not slipped through people's interest. Research work has been done in the field to get a better understanding of the interaction, the behavior is still not completely understood. The objective of this research paper is to take a note of the circumstances under which the Rayleigh jet(also known as Worthington jet) breaks up, leading to the formation of secondary or daughter droplets and crown splash formation due to a liquid drop impacting on a pool of different fluid, as well as to investigate the dependence of the phenomena on the physical properties of the droplet and pool liquid.

A droplet's impact may be categorized based on the target surface, which includes dry solid surfaces, thin liquid films, and deep liquid pools. Droplet size, liquid physical parameters such as density, viscosity, surface tension, and impact velocity are all factors that influence the result of a droplet impact occurrence. Only the influence of droplets on a deep liquid pool is the subject of this research.

The oil spill incidents in history have made the study of droplet impact on deep liquid pool very important since it helps in figuring out methods to clean the oil slick from the surface of the ocean. Other applications of the work include spray cooling in electronic devices, inkjet printing, agriculture and many more.

Figure 1-1: Oil spill near Chennai on Jan 28 2017

The phenomenon occurs in spray cooling of electronic devices where a jet sprinkles drops of cooling liquid over the surface of hot electronic devices. Initially, the phenomenon is droplets impacting on a solid surface but as a thin liquid layer forms over the surface of the device it becomes the case that we are studying “Droplet impact on the liquid pool”.

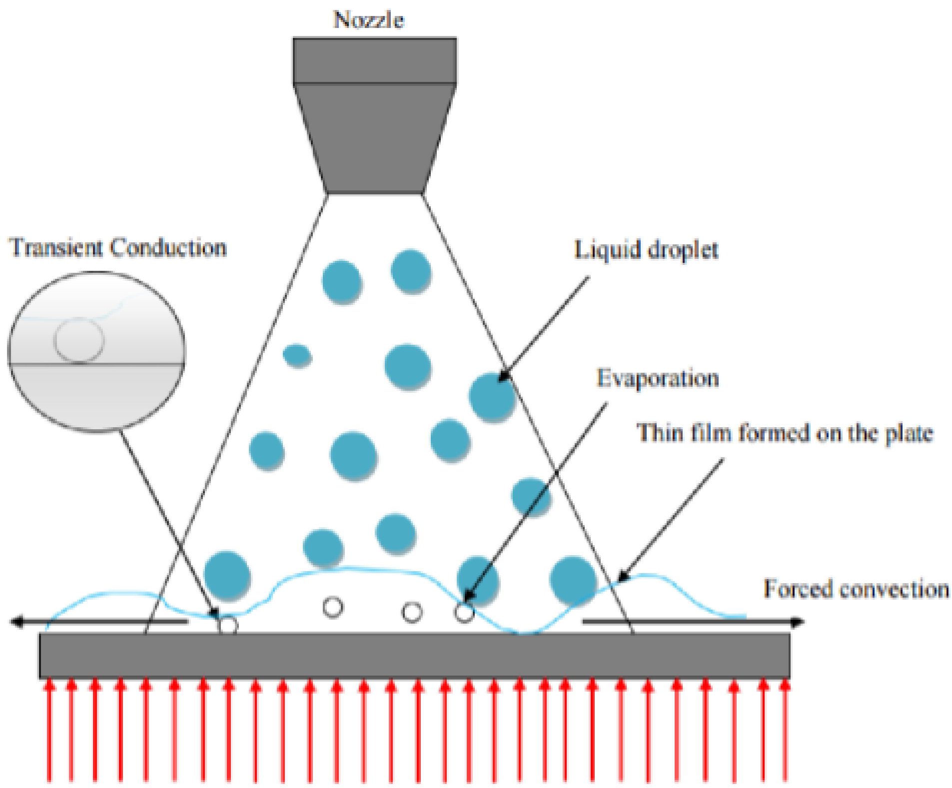


Figure 1-2: Spray cooling of electronic devices

In order to understand the Rayleigh jet phenomenon, we have considered fluids whose Ohnesorge number varies from 0.01 to 1.5. The droplets of low-density fluids were dropped on pools of other heavy fluids and then the interaction was studied. To verify results with numerical simulation, we generated a wedge geometry in Solidworks and did the 2D axisymmetric simulation in Openfoam using a multiphaseInterfoam solver which is an extension of interFoam solver for multiphase flows with more than two fluids. The interFoam solver is based on specie conservation principle and uses the VOF (Volume of Fluid) and CSF (Continuum Surface Force) methods to solve for flow problems.

1.1) What is VOF?

The VOF stands for Volume of Fluid, which is a robust method to solve multiphase problems. The method is used to locate the interface when 2 fluids interact with each other. It is based on the specie conservation principle. A variable α is used to determine the concentration of each phase in the domain. The α may vary from 0 to 1 and it is called the liquid volume fraction. The specie conservation equation is-

$$\frac{\delta\alpha}{\delta t} + u.\nabla\alpha = 0$$

Where u is the cell velocity.

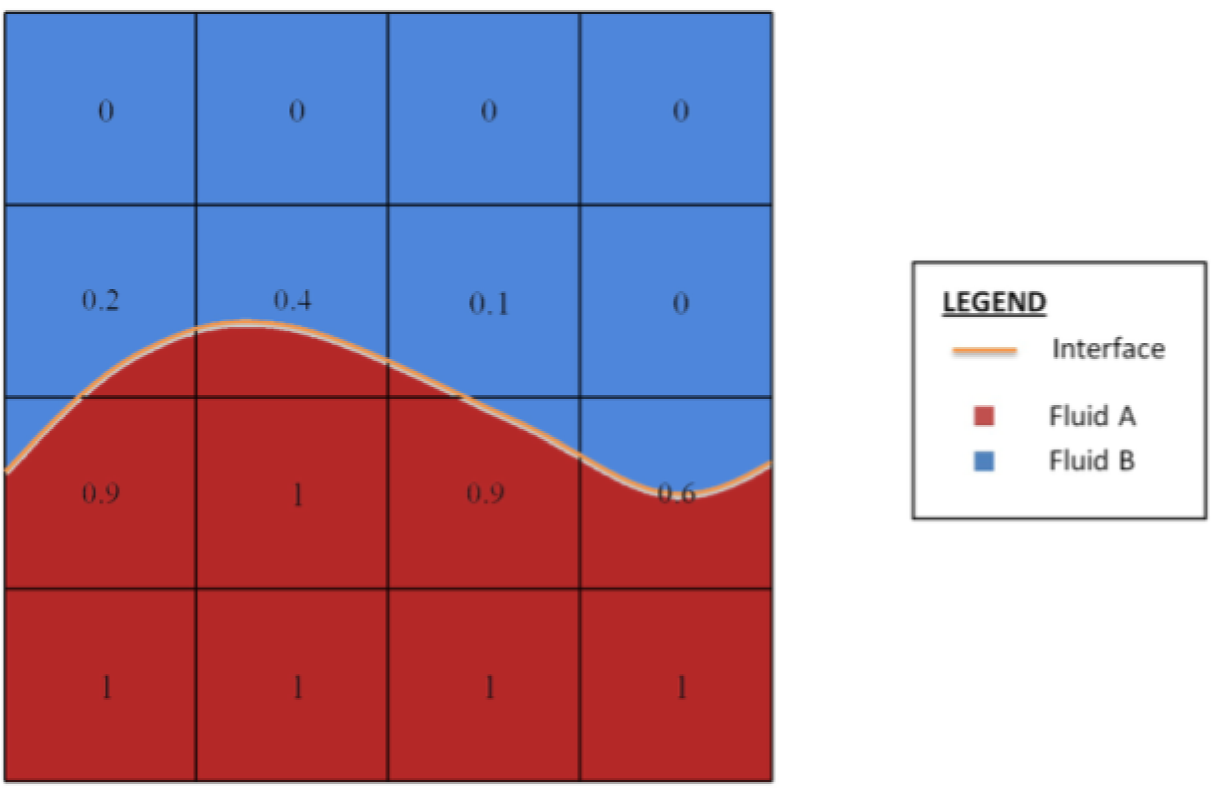


Figure 1-3: Volume of Fluid

1.2) What is CSF?

CSF stands for Continuum Surface Force, it is a model used to solve induce surface tension effects while solving for fluid flows where surface tension plays a crucial role in fluid motion. In the CSF model, the surface tension is considered as a continuous force acting proportionately to the curvature of the same color over the fluid surface. The surface tension is denoted by σ.

Work has been done to explore the outcomes of a droplet impacting a deep liquid pool and solid surfaces, those have been reviewed in the "Literature Review" section. The work however involves a droplet of a liquid impacting on the pool of the same liquid. The previous studies focus mainly on the impact velocity and viscosity of the liquid however in this study we get an understanding of the effect of other parameters like density on Rayleigh jet breakup and crown splash formation. We have studied the impact of all these parameters by systematic experimentation followed by numerical simulation. An axisymmetric simulation is done using OpenFoam, and the required refined mesh is generated after the grid convergence test. To vary the velocity, we vary the height from which the drop is released. The liquid properties such as viscosity and surface tension are varied by varying the pool and droplet liquids. The numerical simulations were conducted using the multiphaseInterfoam solver, which is based on Volume Of Fluid (VOF) and Continuum Surface Force (CSF) models. It is an extension of interFoam solver which is applicable only for two immiscible fluids.

Literature Review

The paper that is reviewed goes by the topic name, “Droplet impact on liquid pools: secondary droplets formation from Rayleigh jet break-up and crown splash”, it is a research work by Eduardo Castillo Orozco, professor at the University of Central Florida. The work focuses on the impact of a droplet of one liquid on a pool of the same liquid. He has done an experimental and numerical analysis of cases where the droplet hits the pool surface with different velocities. The researcher has focussed on the impact velocity, surface tension, and viscosity of the liquid to derive conclusions from his work. His work suggested that the height to which the Rayleigh jet rises depends on the impact velocity of the impacting droplet. He concludes that the Ohnesorge number limits the Rayleigh jet phenomenon by damping the impact of the droplet. If Oh number of the pool liquid is very high then no Rayleigh jet breakup is observed and in some cases, even a jet does not erupt from the pool. In some cases, crown splash is observed when the Oh and We numbers have large values.

Experimental setup

The experimental setup is depicted schematically in Figure 3-1. The container was a round clear petri dish with a diameter of 80 mm and a thickness of 13 mm. The container is large enough to reduce the influence of the wall. The drops were made with a syringe pump (World Precision Instruments) with 100 L/min flow rate and needle a hypodermic needle gauge 23. with a nominal outside diameter of 0.64 mm. The events were caught on camera (i-Speed 2, Olympus) with a zoom lens (Navitar). With a resolution of 576 × 432 pixels and a frame rate of 2000 frames per second, The photos were taken. MATLAB programming was used to analyse the movies (included in Appendix A). The maximum ambiguity and minimal resolution.

A syringe pump, needle, and plastic tubing were used to make the droplets. A 1D stage with millimetre size precision was used to regulate the release height. To create a range of impact velocities, the release height was modified. Because the gravity force overcame the capillary pull, the size of droplets remained stable for a particular fluid. The syringe, tubing, and container were transferred between the two media after each test . Table 3-1 lists the parameters of several liquids as well as the droplet diameter, D_o, utilized in the experiment. All of the tests were carried out at 24.6°C +/-1°C and were performed at least three times to ensure that the results were consistent.

A syringe pump, needle, and plastic tubing were used to create droplets. A one-dimensional stage with millimetre accuracy was used to regulate the release height. To produce a range of impact velocities, the release height was modified

(0.1 m - 2.5 m). As the gravity force overcomes the capillary force, the size of droplets was constant for a particular fluid. The syringe, tubing, and container were all swapped out after each test.

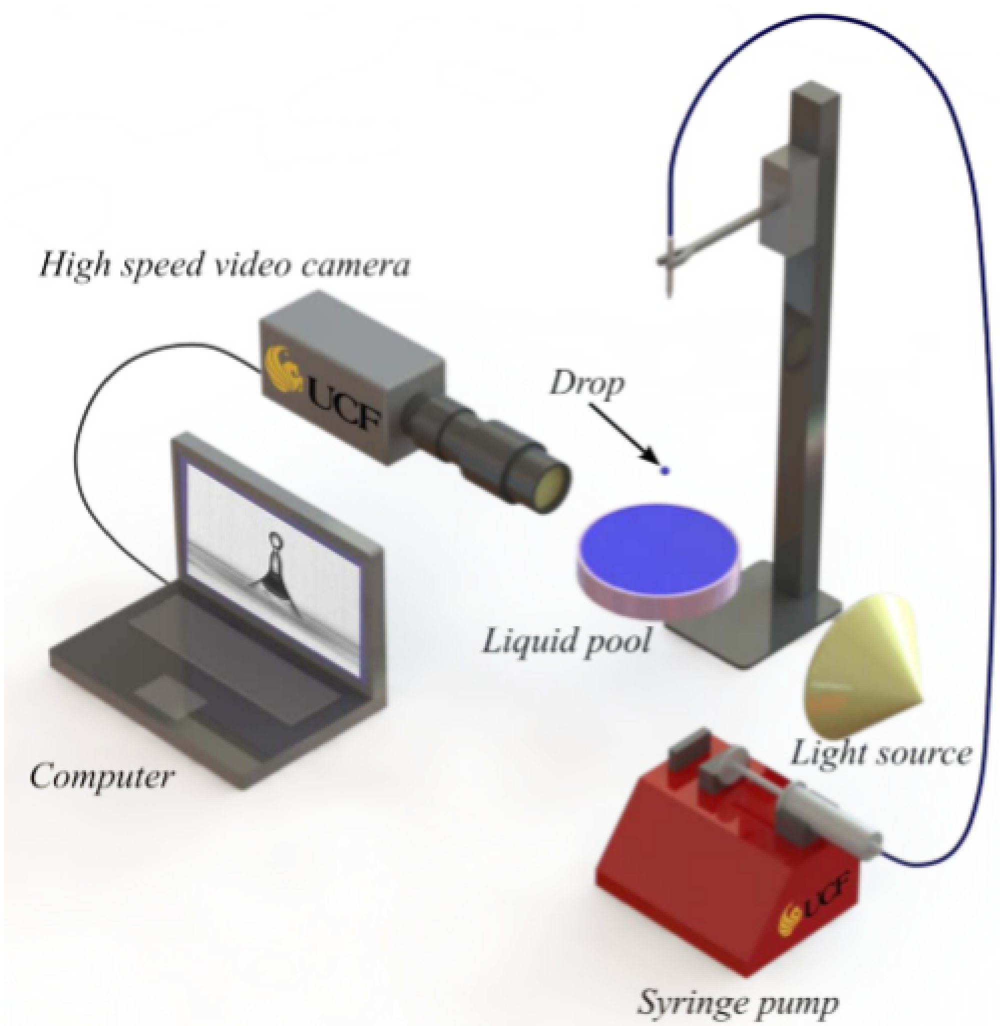


Figure 3-1: Experimental Setup

Numerical Formulation

4.1) Geometry

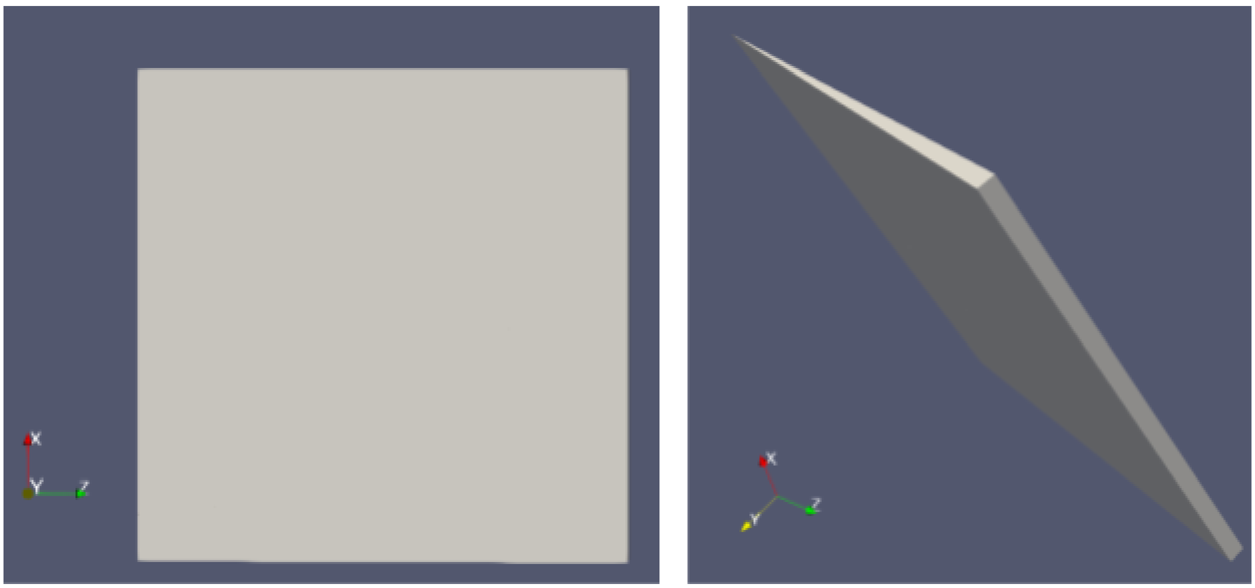


Figure 4-1: Geometry

The geometry that we consider for our simulation is a wedge of 5^o wedge. The size of the wedge is 40mm x 40mm. The geometry is symmetric in the XZ plane and the midplane is perpendicular to the Y-axis.

4.2) Mesh

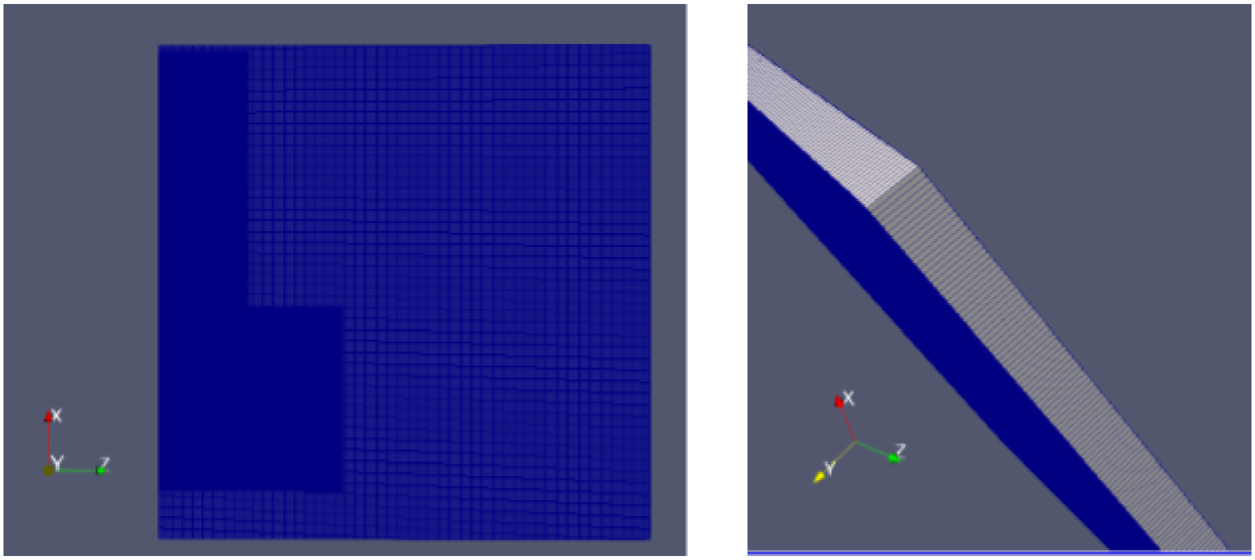


Figure 4-2: Mesh

We will be discussing the grid independence test while moving ahead, the test yields a mesh size of 0.02mm x 0.02mm to be the most optimal and accurate mesh size for the case. We use a mesh of size 0.02mm x 0.02mm in the vicinity of the droplet-pool interaction zone and 0.16mm x 0.16mm in rest of the domain. To join both the domains we construct two transition zones of mesh sizes 0.04mm x 0.04mm and 0.08mm x 0.08mm.

4.3) Solver details

The volume of fluid approach (VOF) is combined with the continuum surface force model (CSF) and incorporated in the OpenFOAM [31] open-source platform. On the grid, OpenFOAM uses a cell centre based finite volume approach.

The numerical simulation of the droplet impact process is carried out using the multiphaseInterfoam solver.The solver is an extension of interFoam solver in Openfoam for multiphase flows with more than two phases. For coupling between pressure and velocity in transient flow, the pressure implicit with splitting of operators (PISO) method [32] is employed as the solution process.

The VOF method considers a phase tracking variable α that represents the volume fraction of the phases present in the domain. One fluid has a volume fraction of (α) and other has (1-α) in the cell. The physical properties are defined as-

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2$$

$$\mu = \alpha\mu_1 + (1-\alpha)\mu_2$$

Where ρ,$\mu$are density and viscosity of the phases. The value of α while defining phases for initial conditions is 0 and 1. The value of α at interface is between 0 and 1.

The equations that are being used to solve the flow are the continuity, momentum and transport of volume fraction equations.The flow for this work is unsteady, incompressible and immiscible fluid flow. The flow considered here is laminar and axisymmetric in nature.

Mass conservation is described by the continuity equation-

$$\frac{\partial\rho}{\partial t} + \boldsymbol{\nabla}\cdot(\rho\boldsymbol{V}) = 0$$

V, here is the fluid velocity, t is time and ρ is the fluid density. Since fluid flow is incompressible we have the following condition-

$$\boldsymbol{\nabla} \cdot (\rho \boldsymbol{V}) = 0$$

The velocities are solved using the Momentum equation-

$$\frac{\partial(\rho \boldsymbol{V})}{\partial t} + \rho(\boldsymbol{V} \cdot \boldsymbol{\nabla})\boldsymbol{V} = -\boldsymbol{\nabla} p + \rho \boldsymbol{g} - \frac{2}{3}\boldsymbol{\nabla}(\mu \boldsymbol{\nabla} \cdot \boldsymbol{V}) + 2\boldsymbol{\nabla} \cdot (\mu \boldsymbol{S})$$

By adding stress tensor to above formulation, the above equation becomes-

$$\frac{\partial(\rho \boldsymbol{V})}{\partial t} + \rho(\boldsymbol{V} \cdot \boldsymbol{\nabla})\boldsymbol{V} = -\boldsymbol{\nabla} p + \rho \boldsymbol{g} + \boldsymbol{\nabla} \cdot \boldsymbol{T}$$

The **T** here is the stress tensor and S is the mean stress tensor. ρ is the density, μ is viscosity and σ is the surface tension per unit area. Let F_σ denote the surface tension force, taking it into account the above momentum equation becomes-

$$\frac{\partial(\rho \boldsymbol{V})}{\partial t} + \rho(\boldsymbol{V} \cdot \boldsymbol{\nabla})\boldsymbol{V} = -\boldsymbol{\nabla} p + \boldsymbol{\nabla} \cdot (\mu[\boldsymbol{\nabla V} + (\boldsymbol{\nabla V})^T]) + \rho \boldsymbol{g} + \boldsymbol{F_\sigma}$$

The CSF model is used to evaluate the surface tension force and then we can substitute it in the momentum equation-

$$\boldsymbol{F_\sigma} = \sigma k \boldsymbol{\nabla} \gamma$$

Where k represents the the surface curvature and γ denotes the volume fraction. Since γ attains values other than 0 and 1 only at interfaces, the term $\nabla\gamma$ is applicable only at interfaces and hence the surface tension force acts only at the fluid interfaces. The curvature is denoted as-

$$k = -\nabla \cdot \left(\frac{\nabla\gamma}{|\nabla\gamma|}\right)$$

Since the VOF method is based on the principle of conservation of specie, the volume fraction of specie is solved simultaneously with the continuity equation. The volume fraction equation becomes-

$$\frac{\partial(\gamma)}{\partial t} + \nabla \cdot (V\gamma) = 0$$

4.4) Initial and Boundary Conditions

The flow that we consider here is axisymmetric in nature. This is because we assume that the droplet is spherical when it hits the pool surface, however the shape may vary a bit due to drag from air while falling. Since Openfoam doesn't allow 2D simulation, we have constructed a wedge with wedge angle of 5 degrees. The wedge is given an edge length of 40mm. The wedge is symmetric in XZ plane

The upper boundary is set to open atmosphere and the pressure there is same as atmospheric pressure and the fluids can flow freely across it. The boundaries to the left and right of the domain are set to walls with no-slip condition. The slant faces of the wedge are given the "wedge" boundary condition, which is crucial to register our case as axisymmetric cylindrical.

At the beginning of simulation, the fluid droplet and pool are assigned their respective fluids. The droplet is given the velocity and rest of the domain is set to still air. Gravity value is taken to be 9.81m/s^2 everywhere in the domain. The initial pressure at walls and the top boundary is offset by $\rho g z$ so that the initial pressure value everywhere in the domain is set to 0.

$$\Delta p = \frac{2\sigma}{R}$$

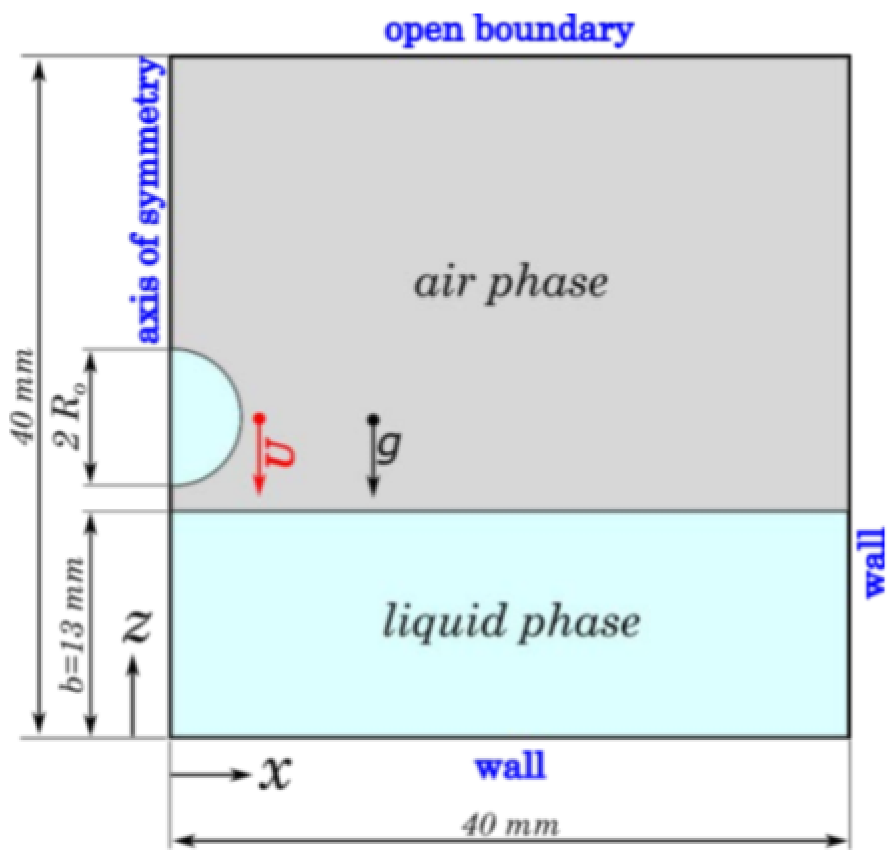


Figure 4-3: Domain for CFD simulation in Openfoam

4.5) Boundary conditions in Openfoam

1) **No-slip**: The velocity a the wall is set to zero, used to signify the shear forces at the wall.
2) **zeroGradient**: The gradient perpendicular to the boundary is set to zero, which means that the property will not change in the direction perpendicular to the face.
3) **pressureInletOutlet velocity**: Used to define velocity on a face where the value of pressure is known. In our case the value of pressure at the open face is known to be atmospheric pressure.
4) **fixedValue**: Used to give a fixed value of a property on a face, this value acts as a base value for rest of the domain. In our case the value of velocity at the wall is 0 at all times. This can be set using fixedValue condition and then this value can help solve the flow in rest of the domain.
5) **inletOutlet**: It is used similar to zeroGradient with just one change that it shift to the provided fixed value whenever the flow reverses its direction. It is used to prevent backflow anomaly in fluid flows.
6) **totalPressure**: Used to define total pressure at a face in the geometry.
7) **cyclicAMI**: AMI stands for Arbitrary Mesh Interface, the cyclicAMI condition is used when cells on both sides of a face don’t match fully with each other. It helps the cells to communicate across the interface using a weighted approach.

4.6) Grid Independence Test

Need for grid independence test?

The results of a CFD simulation depend on the grid size of the domain. The more refined the mesh is, the more accurate the results we get but at the same time the computational cost increases with the number of elements. So an optimal mesh size has to be determined that can be used to run the simulations with accurate results and minimum computational cost.

An initial mesh size of 0.16mm x 0.16mm is used to create the mesh for our domain. We had earlier taken a physical reading of the case when a silicone 5cSt droplet impacts a silicone 5cSt pool. This case is used for the grid convergence test, the droplet hits the liquid pool causing a Rayleigh jet to erupt from the pool and a subsequent droplet pinch-off that follows. These phenomena are used to check the correctness of mesh. The mesh size is reduced to half, every time the mesh doesn't pass the test. The mesh size goes from By this way we boil down to a mesh size of 0.02mm.

Mesh Size (mm)	Droplet pinchoff	Droplet separation height matched
0.16mm x 0.16mm	✗	✗
0.08mm x 0.08mm	✗	✗
0.04mm x 0.04mm	✓	✗
0.02mm x 0.02mm	✓	✓
0.01mm x 0.01mm	✓	✓

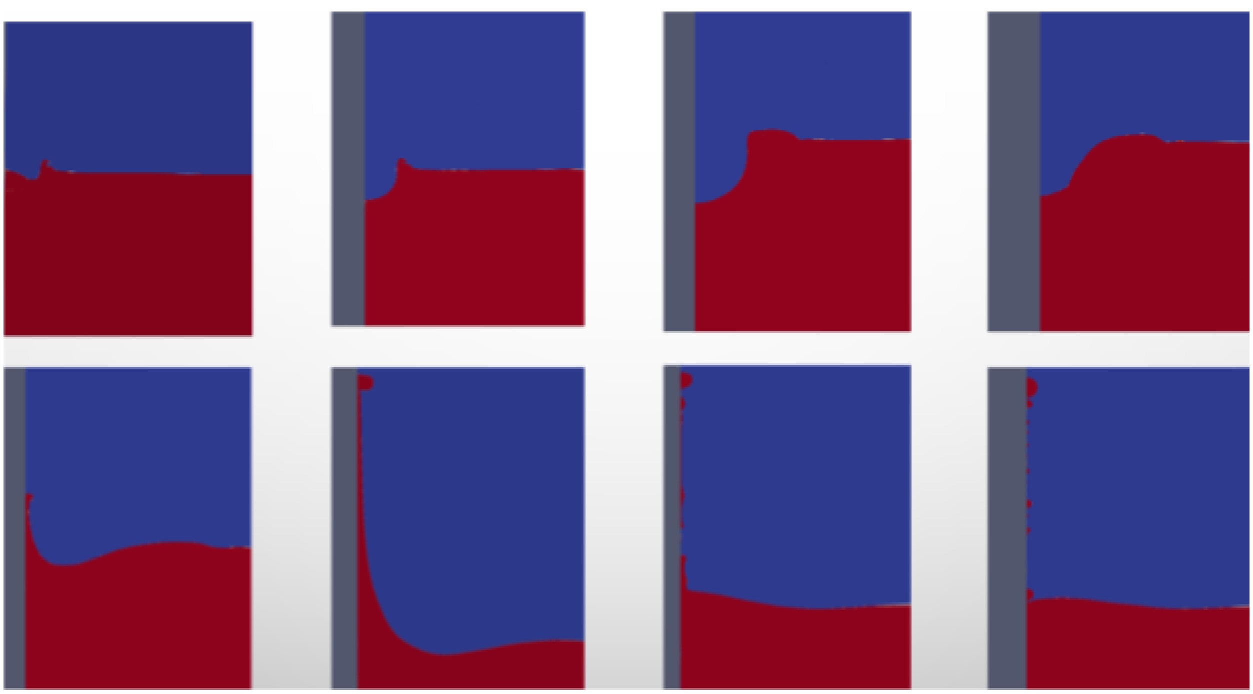


Figure 4-4: CFD simulation of Si 5cSt droplet impacting on Si 5cSt pool

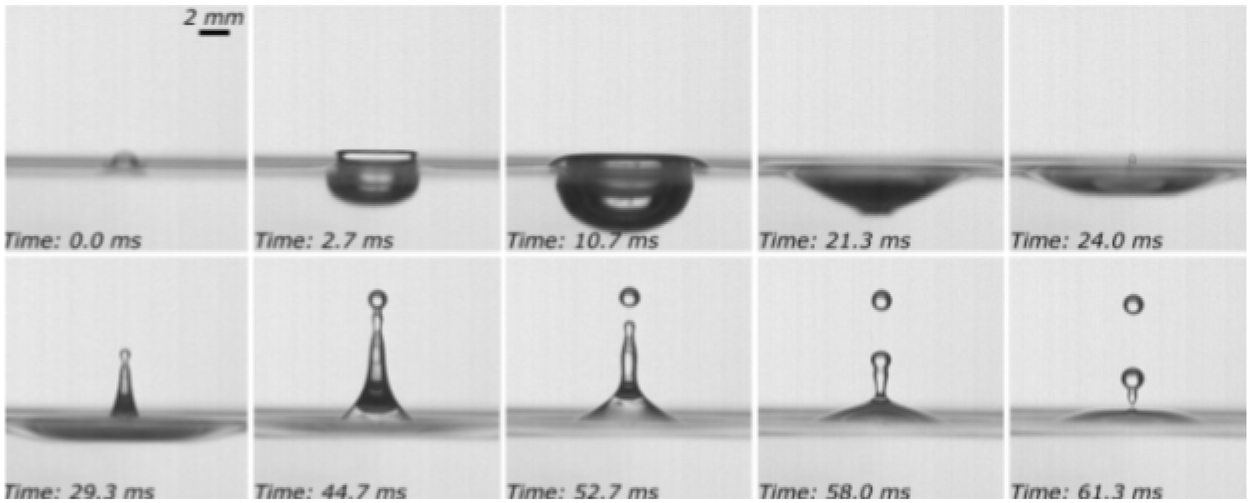


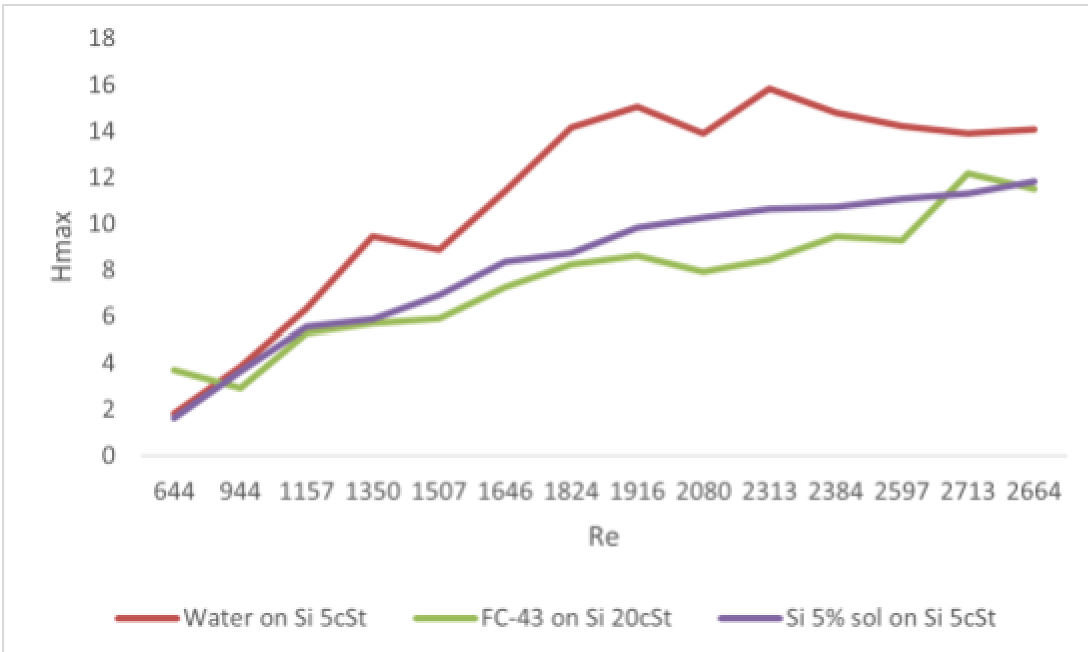
Figure 4-5: Experimental reading of Si 5 cSt droplet impacting on Si 5 cSt pool with V=2.8m/s

The above case is the case where a silicone oil droplet impacts on silicone 5cSt pool with a velocity of 2.1m/s, that is a We = 184. In this case, a crater is created and then a small jet erupts from the crater. This is the Rayleigh jet that breaks up to form secondary droplets. The simulation for the above case is done in Openfoam and it is used to determine the optimal mesh size for the domain.

Results and Discussions

5.1) Relationship between Re,Oh,We and Rayleigh jet breakup

5.1.1) Impact of Re



The above is the graph of the maximum height of the jet before breakup plotted against the Reynolds number. It can be clearly seen that the height increases as the Reynolds number increases. This is because Re is the indicator of the inertia of the droplet and as the inertia of the droplet increases so does its kinetic energy hence there is more energy available for jet eruption.

5.1.2) Impact of We

We signify the dominance of inertia force over surface tension force. Therefore a high We means the inertia forces dominate the surface tension force. The jet then erupts to a higher altitude as the droplet brings in more energy however the surface tension force is not enough to sustain the weight of a large droplet hence a droplet pinch-off occurs but the droplet breaks into multiple small secondary droplets.

5.1.3) Impact of Oh

The Oh is the indicator of the dominance of viscous force over surface tension force. A higher value of Oh indicates that viscous force has dominated the surface tension force. If the Oh of the pool liquid is high, this indicates that the pool can dampen the impact of the droplet. A high Oh number also delays the pinch-off. If the We is high when Oh of the pool is high, crown splash can occur as the high available kinetic energy will get dissipated over the liquid surface.

5.2) Physics behind droplet pinch-off

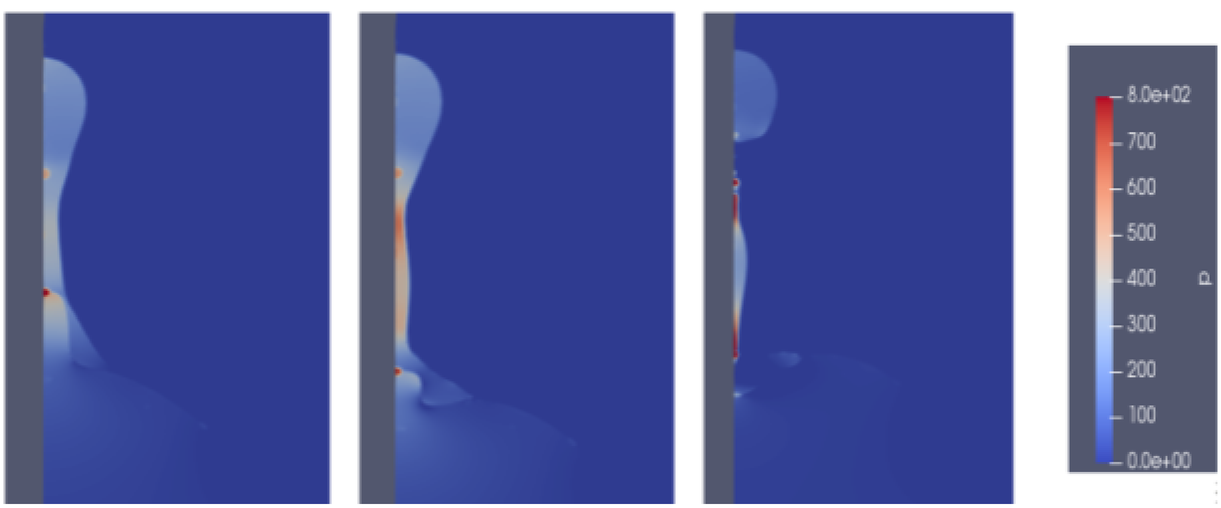


Figure 5-1: Pressure reading during pinch-off when a water droplet impacts a Silicone 5cSt pool

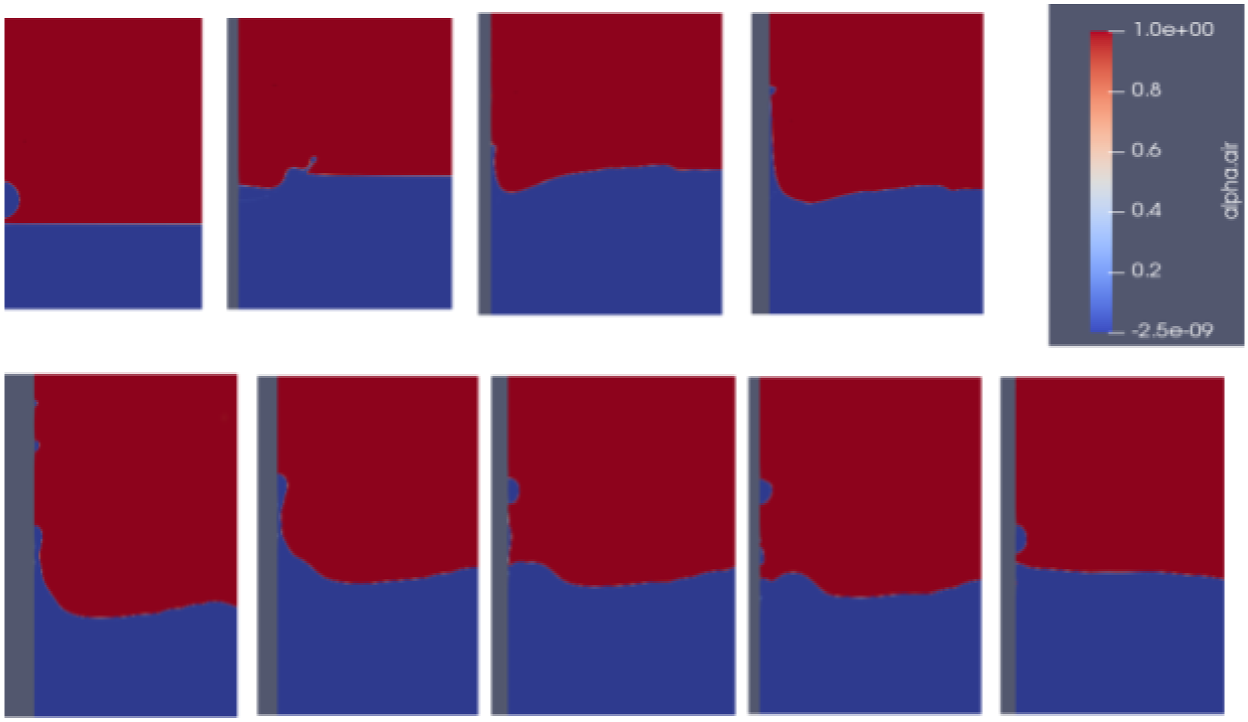


Figure 5-2: CFD simulation of water droplet on Si 5cSt pool with V=2.8m/s

When a Rayleigh jet erupts from a pool of liquid it destabilizes because of Rayleigh-plateau instability, as the jet height increases wavy structure is formed at the surface of the jet. At the top of the jet these wavy structures form a bottleneck of pressure and the pressure at the neck becomes very high subsequently leading to a droplet pinch-off.

5.3) Other phenomena observed

5.3.1) Backflow

When a Rayleigh jet is rising, it keeps losing velocity against gravity. At certain point the fluid in the droplet stops rising and starts to flow back into the pool. If the Oh is high then it is possible that all the fluid flows back to the pool before the separation could occur. This is because Oh slows down the pinch-off of the droplet and hence more time is available for backflow.

5.3.2) Cavity in pool

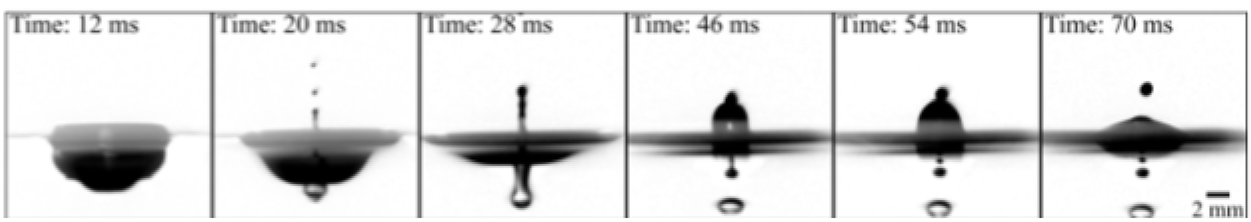


Figure 5-3: Experimental reading of Water droplet impacting on silicone 10cSt pool

When a droplet of a fluid of high density impacts a pool of liquid with low density, a cavity or air bubble is formed in the pool. However, if a low-density droplet impacts a high-density liquid pool, there is a possibility of a crown splash occurring. This is because the droplet won't be able to dampen all energy in the pool and hence a lot of energy will have to be dissipated over the surface.

5.4) Impact of velocity, viscosity and surface tension

The velocity of the droplet plays a crucial role in Rayleigh jet eruption since it is the source of energy for the rising jet. A higher velocity means more energy available and hence higher the jet will rise. Viscosity of the pool on the other hand has the exact opposite impact, the higher the viscosity of the pool the more would be the damping effect and hence the jet would erupt to a smaller height. If the viscosity is very high then no pinch-off can also occur or a crown splash can also occur if the droplet impacts the pool with a very high velocity. The surface tension is a measure of the maximum weight that a droplet can hold, which means that a higher surface tension indicates that a larger droplet can be formed as the surface forces could withstand a larger volume of fluid. The same way a smaller surface tension would mean that the big droplet cannot be sustained and hence number of small secondary droplets would be formed if a pinch-off occurs.

Conclusions and Future Scope

As discussed above, we now understand the impacts of physical properties on Rayleigh jet formation and crown splash formation. So we can conclude that if we want to design a mechanism where we need more turbulence like in spray cooling we want a Rayleigh jet erupting from the pool and subsequently breaking into smaller secondary droplets. This can be achieved by high impact velocity and low Oh number of the pool fluid. We also conclude that whenever a high speed droplet impacts on the pool surface where the pool liquid has very high Oh number, in such cases a crown splash can occur. If the surface tension of the fluids is less then it is possible to have a large number of secondary droplets that are smaller in size rather than a big secondary droplet.

We have discussed earlier about hazardous impacts of oil spills on our environment. The project will help in deriving methods to effectively clean the oil spills within a short span of time. Study of droplet also helps in agriculture purposes where spraying fertilizers and irrigation can be optimized to benefit the farmers.

References

1) O. Reynolds, On the action of rain to calm the sea, Nature 11, 279-280 (1875).

2) A.M. Worthington, A Study of Splashes, Longmans, Green, London, 129 (1908).

3) M. Rein, Phenomena of liquid-drop impact on solid and liquid surfaces, Fluid Dyn. Res. 12, 61 (1993).

4) R. Rioboo, C. Tropea, M. Marengo, Outcomes from a drop impact on solid surfaces, Atomization Sprays 11, pp. 155-165, (2001).

5) C. Clanet, C. Béguin, D. Richard, and D. Quéré, Maximal deformation of an impacting drop, J. Fluid Mech. 517, pp. 199-208, (2004).

6) S.L. Manzello, J.C. Yang, An experimental study of a water droplet impinging on a liquid surface, Exp. Fluids 32, 580 (2002).

7) K.L. Pan and C.K. Law, Dynamics of droplet-film collision, J. Fluid Mech. 587, pp. 1-22 (2007).

8) L.V. Zhang, P. Burnet, J. Eggers, R.D. Deegan, Wavelength selection in the crown splash, Phys. Fluids 22, 122105 (2010).

9) D. Banks, C. Ajawara, R. Sanchez, H. Surti, and G. Aguilar, Effects of Drop and Film Viscosity on Drop Impacts onto Thin Films, Atomization Sprays, 23: 555-570 (2013).

10) T. Tran, H. de Maleprade, C. Sun, D. Lohse, Air entrainment during impact of droplets on liquid surfaces, J. Fluid Mech. 726, R3 (2013).