

# Numerical Simulation of single bubble behavior on heating

A project under

Prof. Abhilash Kumar Tilak

Sharnam Singhwal	2018A4PS0067G
Arya Shah	2018A4PS0425G
Deven Paul	2018A4PS0047G
Utpal Narvekar	2017B5A40141G
Shreyansh Gautam	2017B4A40807G
Shivam Kashyap	2017B3A40554G



# Introduction

---

Boiling is one of the most effective heat transfer methods, and it is widely used in applications such as propulsion, electricity, electronic cooling, and chemical processes, among others. Significant progress has been made in enhancing our understanding of the boiling mechanism over the last few decades. Among these methods, single bubble nucleate boiling heat transfer is a fundamental concept that has long been studied in engineering science.

Over the last two decades, substantial progress has been made in this field through studies and numerical simulation. Measurement techniques and experimental methods have advanced rapidly in experiments, allowing for the measurement of local and instantaneous quantities such as the local wall temperature under a vapour bubble or transient heat transfer between solid and fluid.

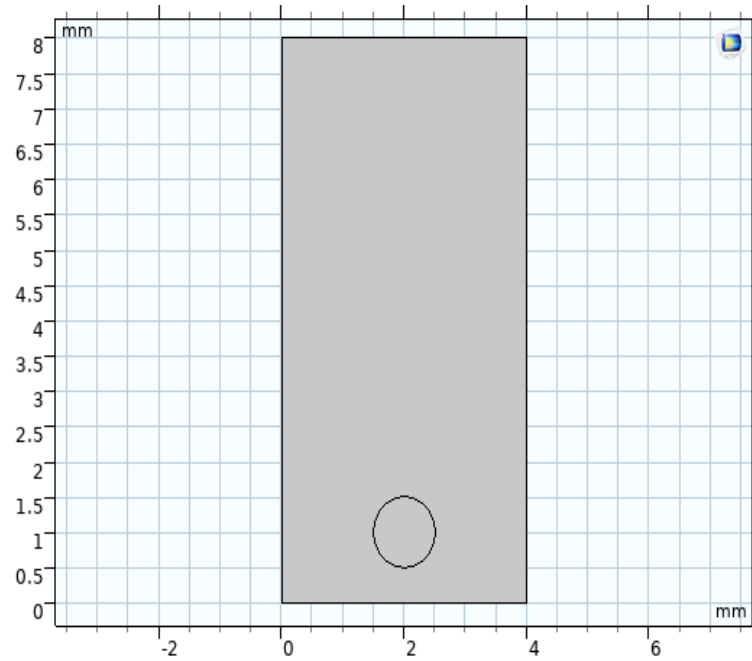
With the advancement of computational power and modelling techniques in the field of numerical simulation, progress in modelling single bubble nucleate boiling has been achieved. The Navier-Stokes and energy equations were solved in two-dimensional symmetric form in one of the first simulations of single bubble nucleate boiling. Finally, during the nucleate pool boiling, the velocity field, pressure, and temperature profile were obtained.

In this paper, we will use the Level Set methods with a test function in the COMSOL multiphysics software to simulate the single bubble boiling phase. The new model method can be used to refine heater configurations in a more realistic and rapid manner using this approach.

# Geometry

---

- A 2-D model was selected with a rectangle as the vessel and water as the base fluid.
- The circle created represents the bubble (air) to be analysed



*Fig1. Model Geometry*

# Physics used

---

## Laminar Flow

Fluid flows can be divided into two different types: laminar flows and turbulent flows. Laminar flow occurs when the fluid flows in infinitesimal parallel layers with no disruption between them. In laminar flows, fluid layers slide in parallel, with no eddies, swirls or currents normal to the flow itself. This type of flow is also referred to as streamline flow because it is characterized by non-crossing streamlines

- **Fluid Properties**
  - Common model input was taken for temperature
  - The reference temperature ( $T_{ref}$ ) was user defined as 293.15K
- **Initial values**
  - The velocity field was kept 0 as we are considering Pool boiling here
- **Wall 1**
  - No slip condition was maintained on upper and lower boundaries of the vessel
- **Gravity 1**
  - Gravity was included in the analysis with standard value 'g\_const' (system defined)
- **Pressure Point Constraint 1**
  - A necessary condition is to add a pressure constraint in order to satisfy the Laminar flow equation. This  $P_o=0$  is constrained at the bottom right corner of the vessel.

## Level Set

The level set method is a technique to represent moving interfaces or boundaries using a fixed mesh. It is useful for problems where the computational domain can be divided into two domains separated by an interface. Each of the two domains can consist of several parts.

The Level Set interface provides the equations and boundary conditions for using the level set method to track moving interfaces in fluid-flow models, solving for the level set function.

## Level Set Model

- **Initial Values 1**  
The water domain is chosen here as Fluid 1 (phase 0)

No flow condition

- **Initial values 2**  
The air (bubble) domain is chosen here as Fluid 2 (phase=1)
- **Initial Interface**  
Initial boundary is selected which separates the two phases selected

## Heat Transfer in Fluids

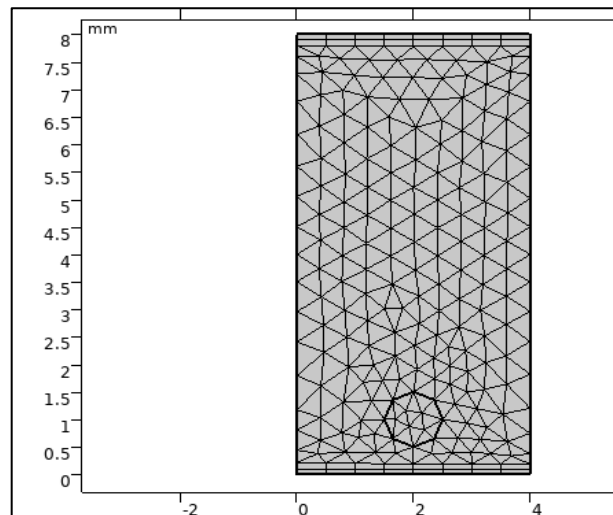
- **Fluid 1**  
A common model input is taken for all the parameters
- **Initial values 1**  
Initial temperature is user defined as standard 293.15K
- **Thermal Insulation**  
The left, top and right boundaries of the vessel are thermally insulated
- **Boundary Heat source 1**  
The bottom boundary is kept at general source with common model input
- **Boundary Heat source 2**  
Same for the bubble boundaries
- **Heat Flux 1**  
Heat flux is applied at the bottom boundary of the vessel and the heat rate is varied from 2W, 5W and 10W to analyse the bubble behavior with changing heat flux

# Meshing

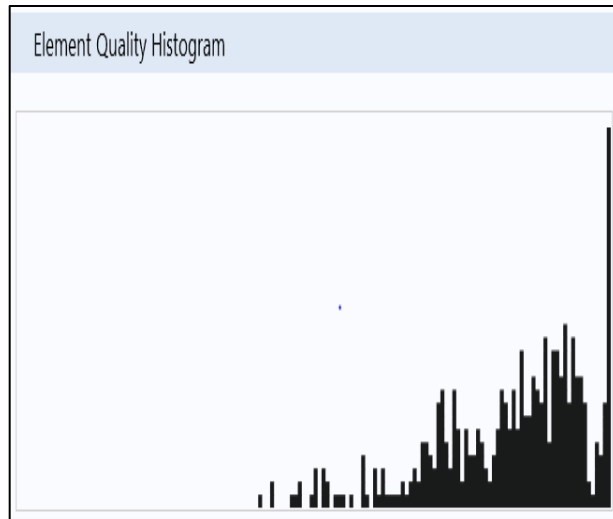
- COMSOL inbuilt physics controlled 'Extremely Coarse' Mesh was selected for the geometry.
- Meshing was changed to 'Course' for 10W heat rate case to ensure the last time step is converged.

Element Quality	
Quality measure:	Skewness ▾
Statistics	
<b>Complete mesh</b>	
Mesh vertices: 245	
Element type:	All elements
Triangles:	368
Quads:	32
Edge elements:	64
Vertex elements:	8
— Domain element statistics	
Number of elements:	400
Minimum element quality:	0.412
Average element quality:	0.83
Element area ratio:	0.2122
Mesh area:	32 mm <sup>2</sup>

*Fig2. Meshing statistics*



*Fig3. Final Mesh*



*Fig4. Mesh Quality*

# Multiphysics Used

---

- Two phase flow, Level Set
- Wetted Wall
- Nonisothermal flow

# Study

---

- Phase Initialization
- Time Dependent
- The computation time is 11 minutes and 22 seconds.

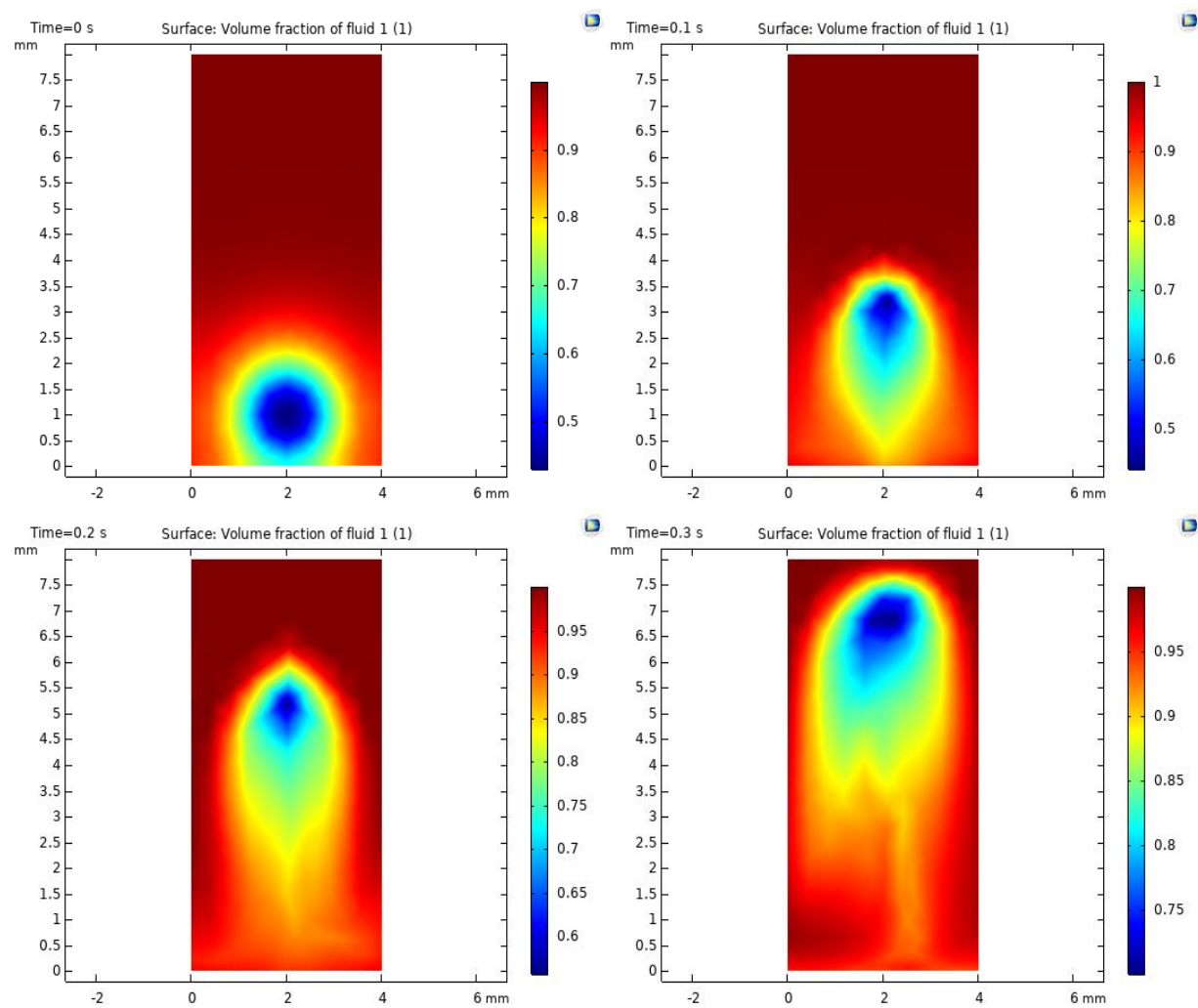


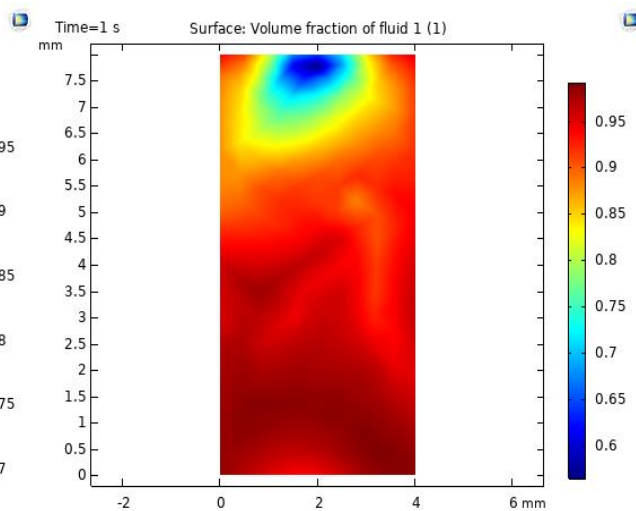
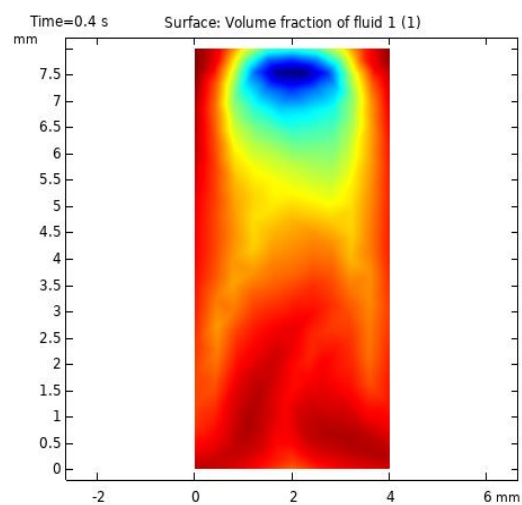
# Results

---

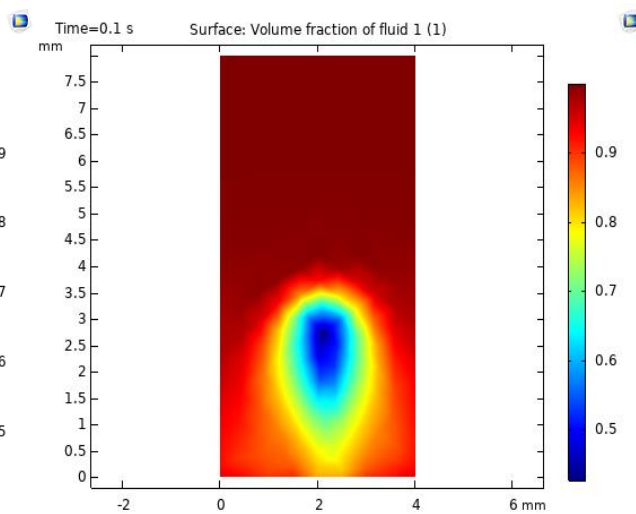
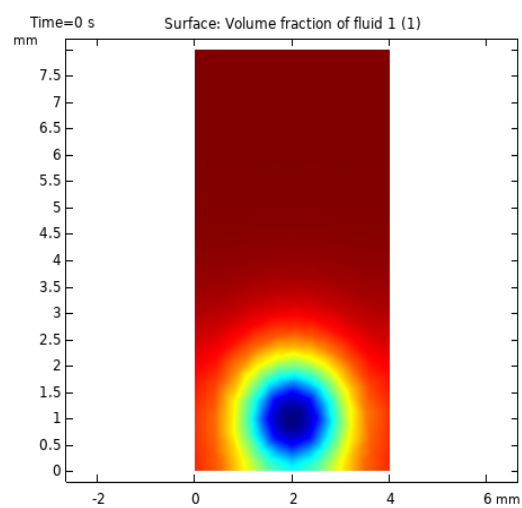
## Volume Fraction

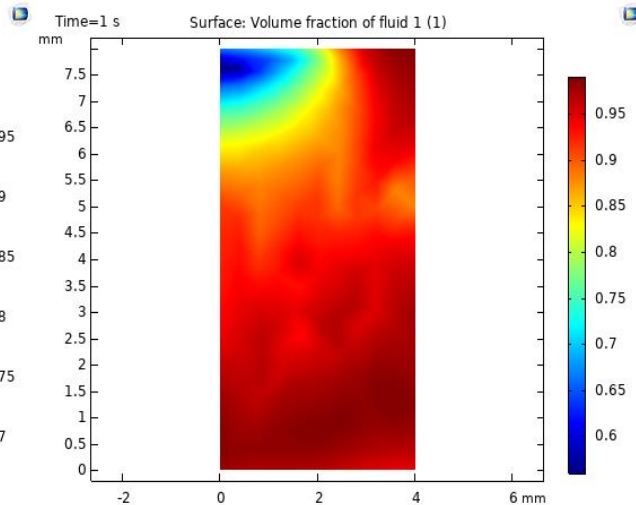
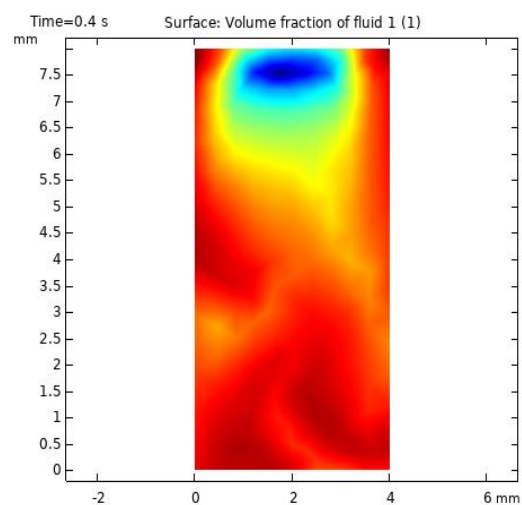
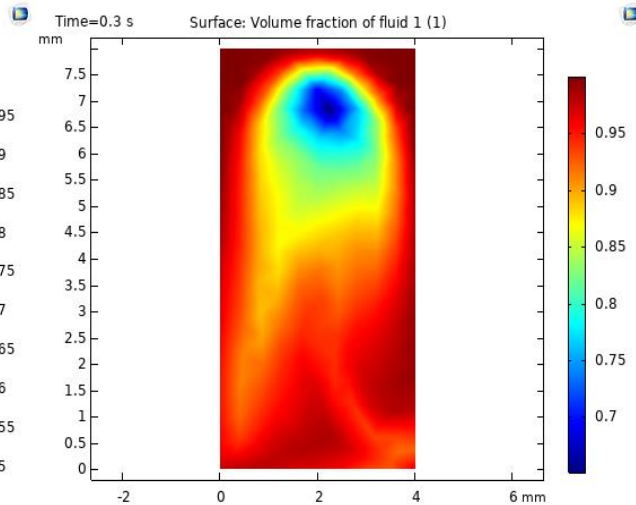
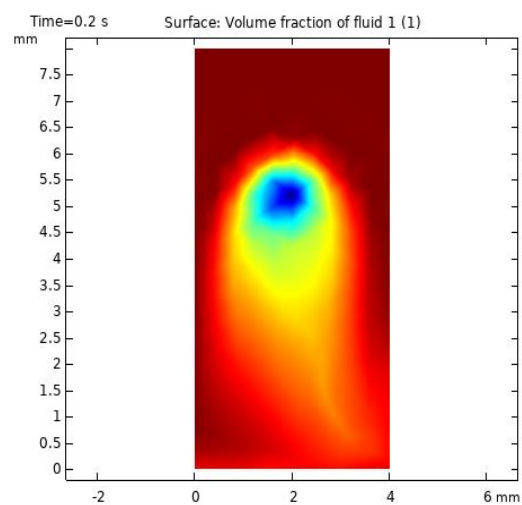
2W



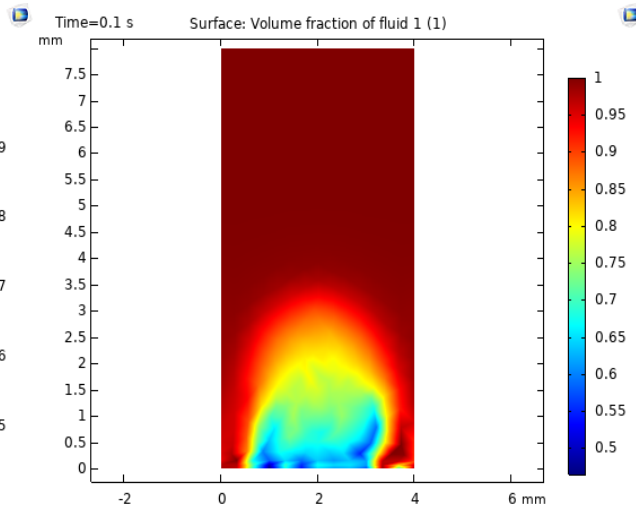
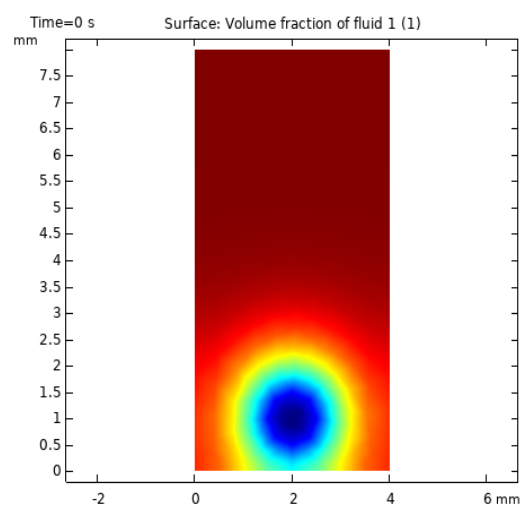


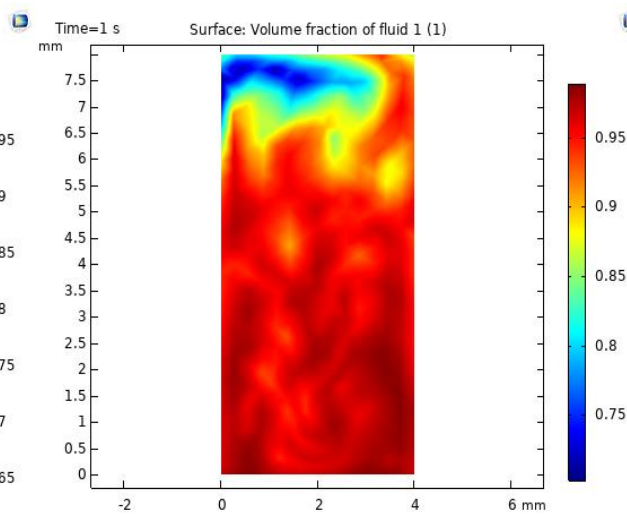
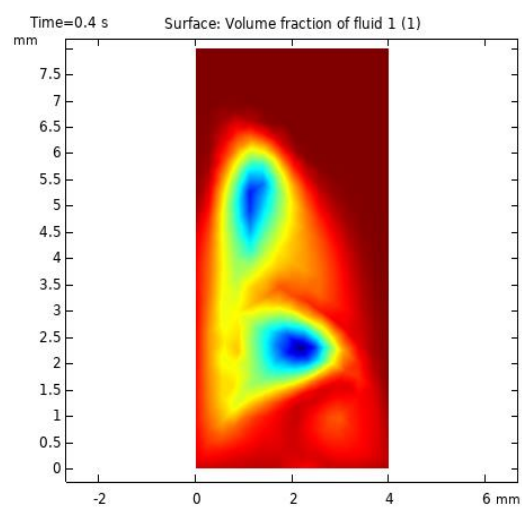
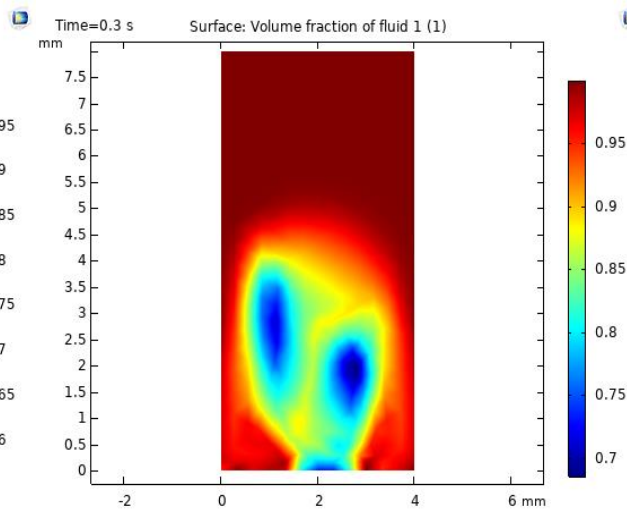
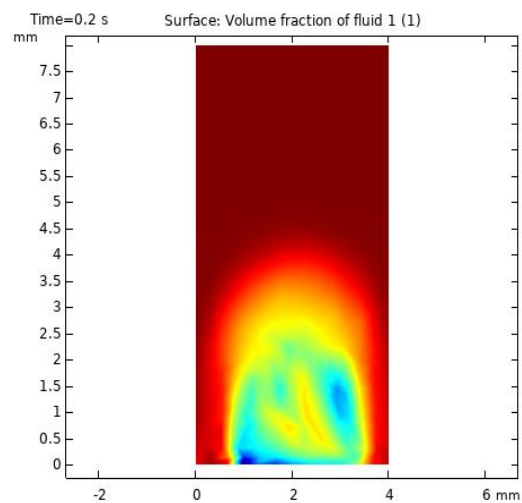
5W





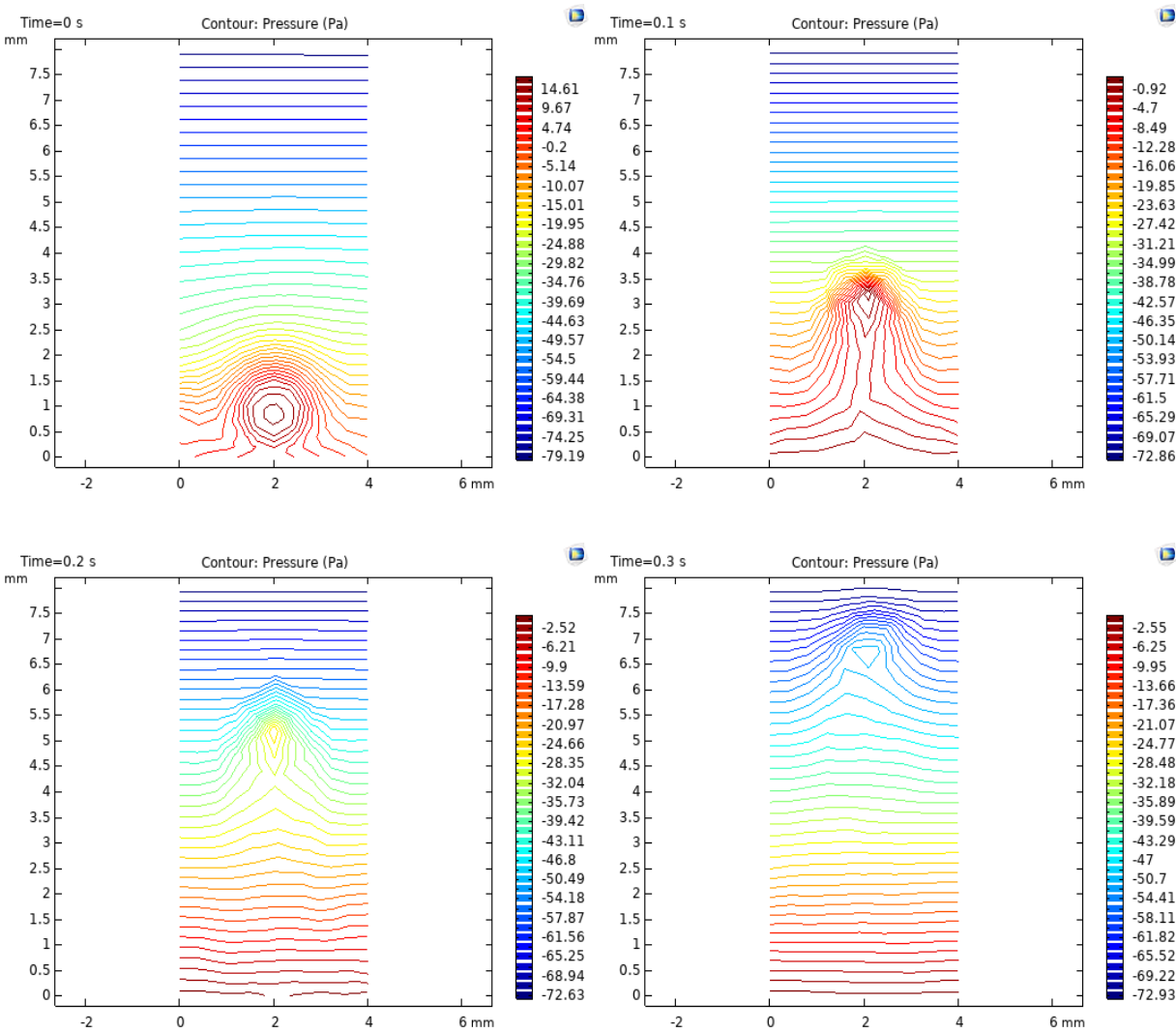
10W

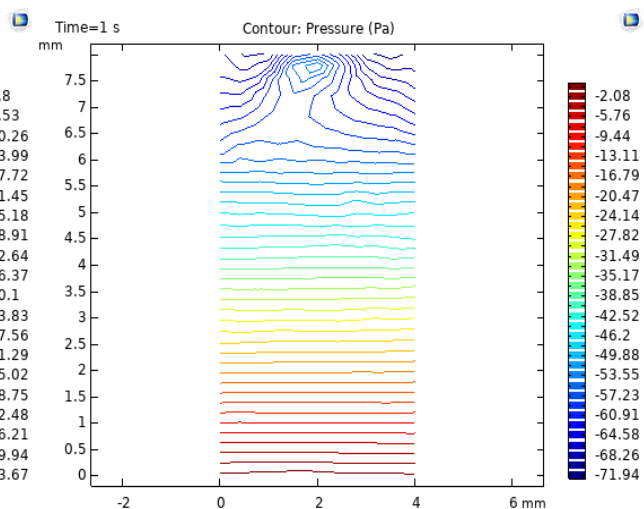
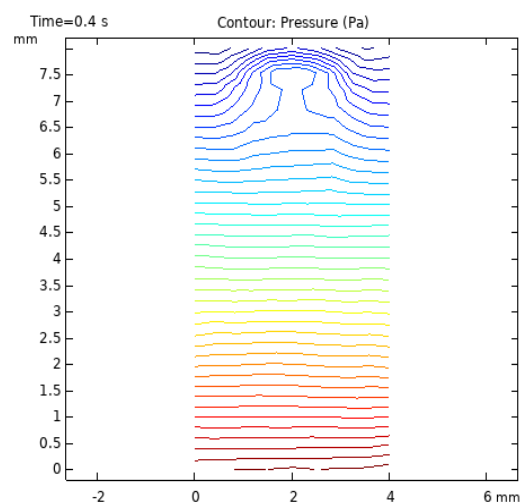




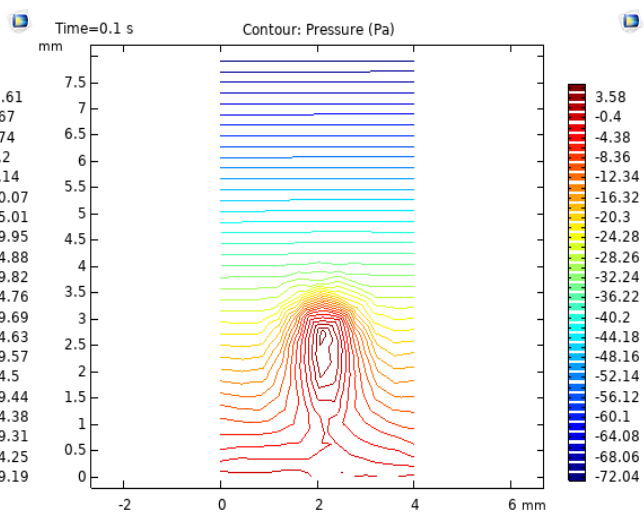
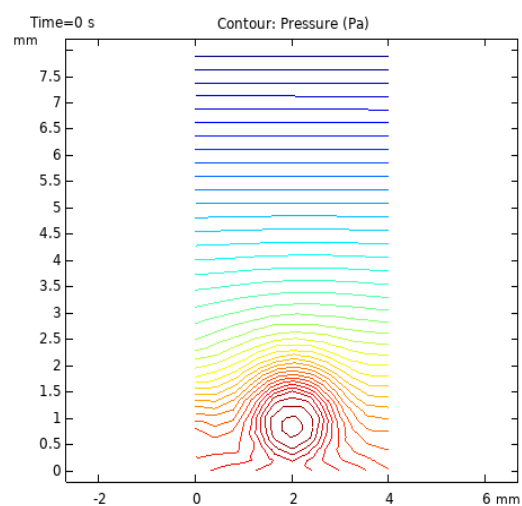
# Pressure

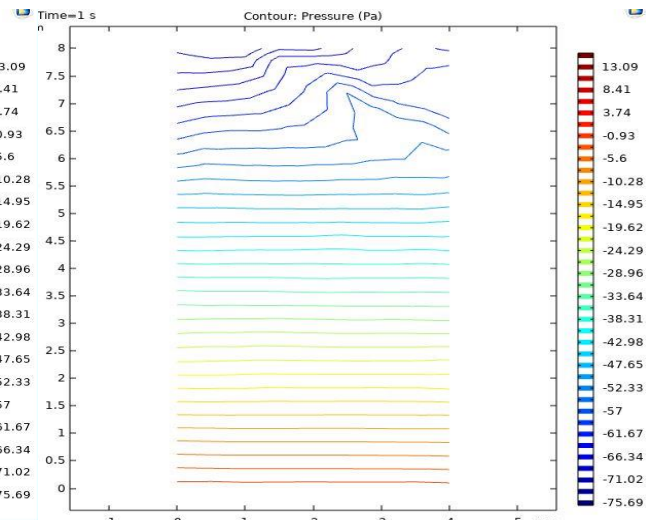
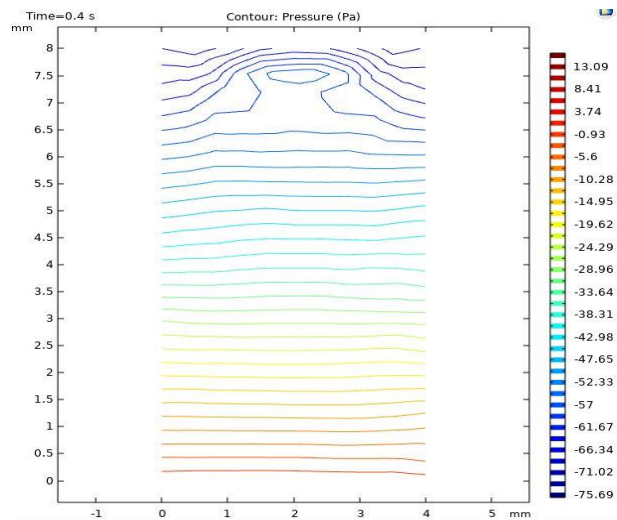
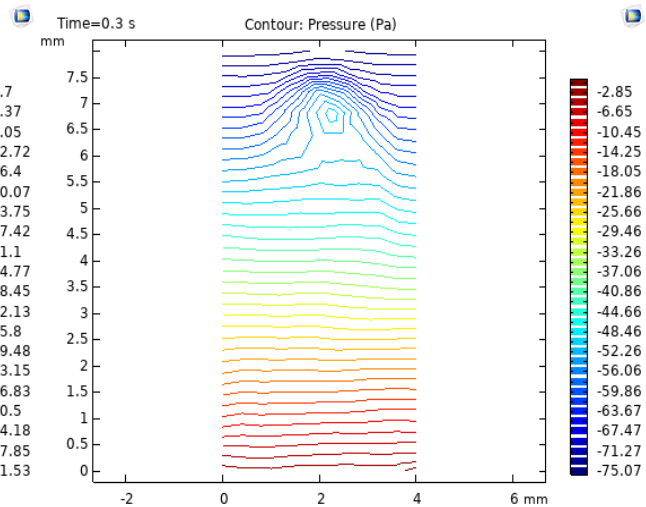
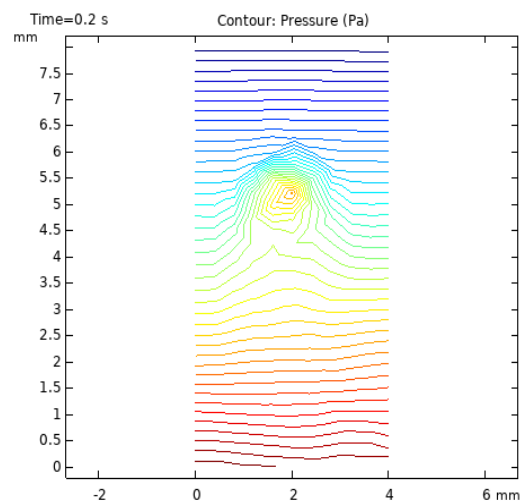
2W



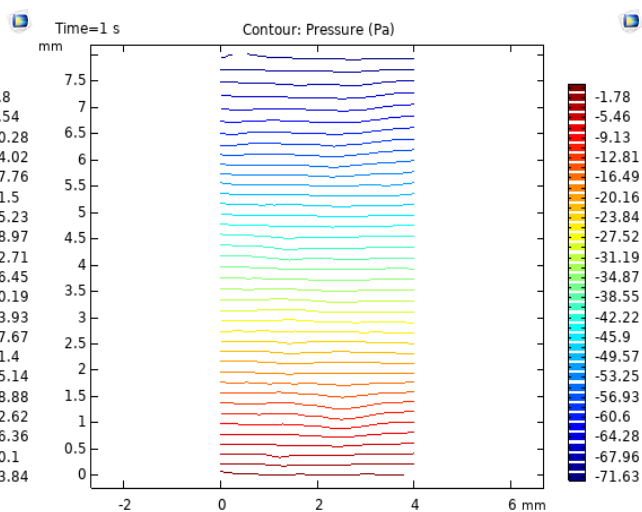
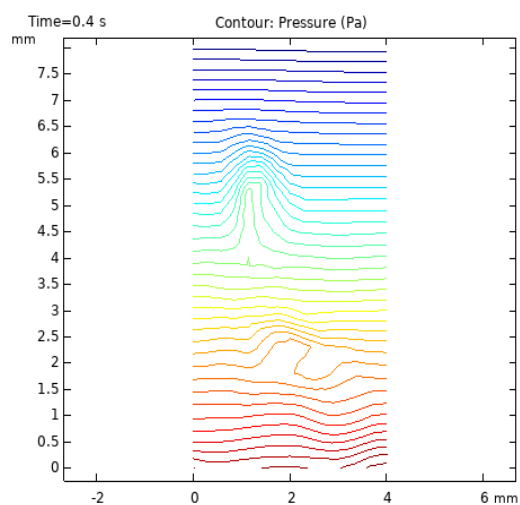
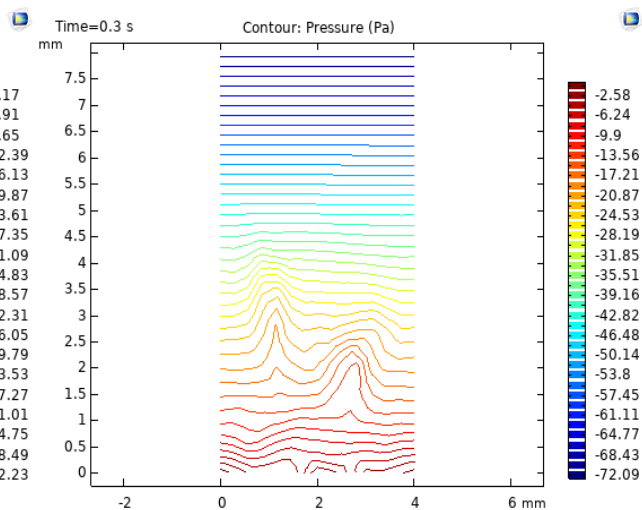
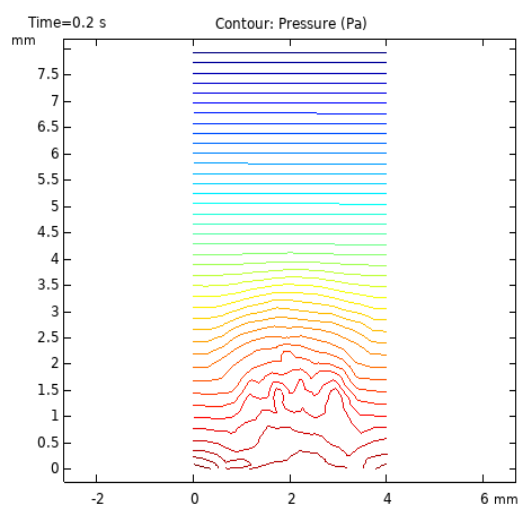
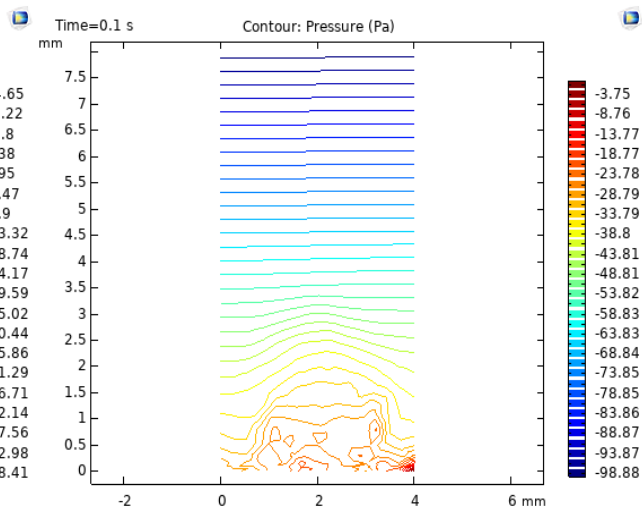
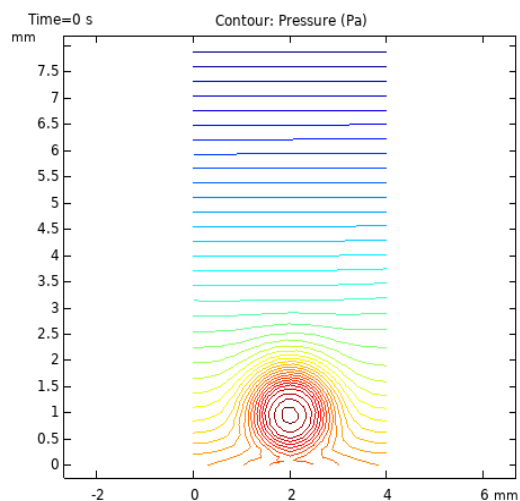


5W





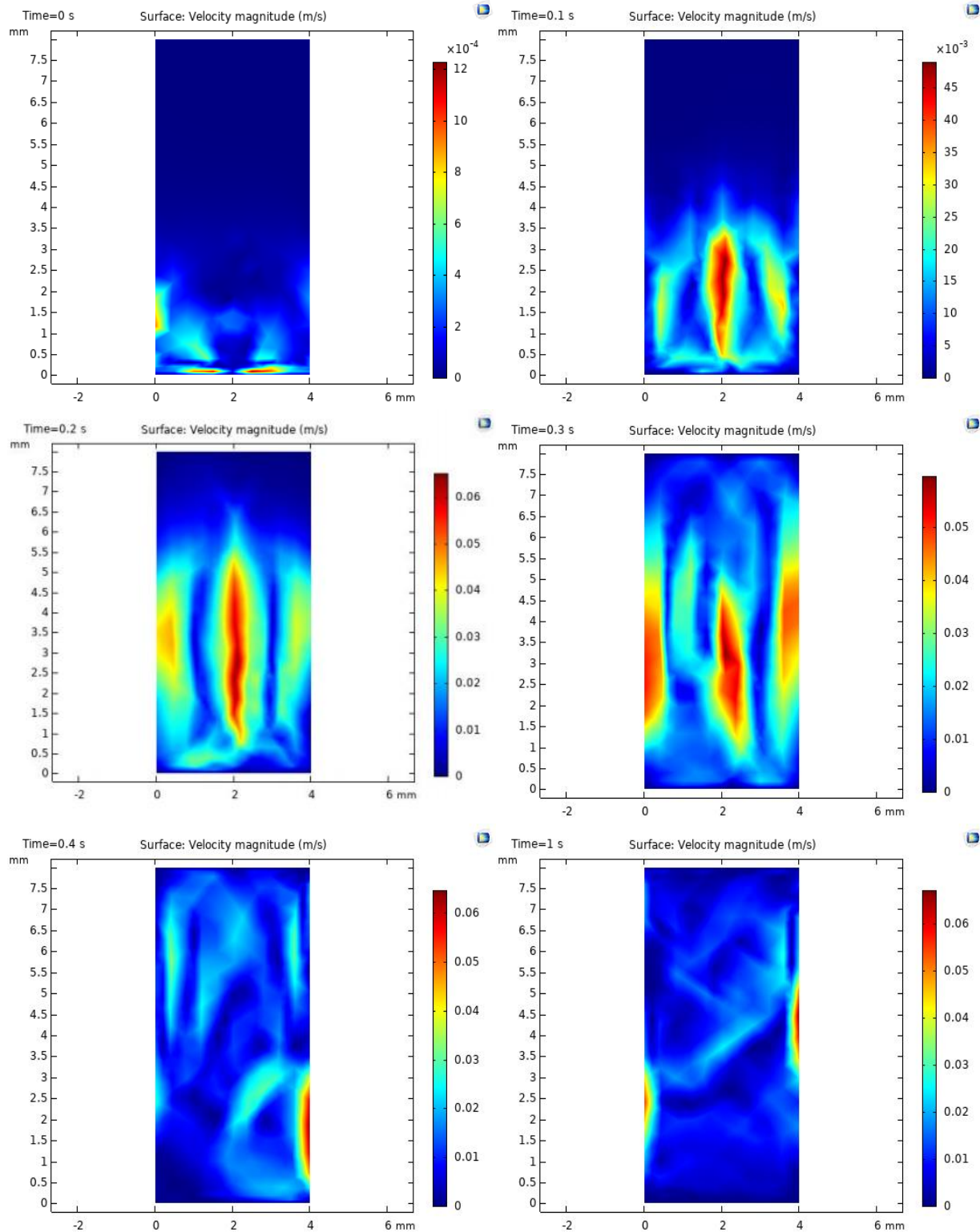
10W



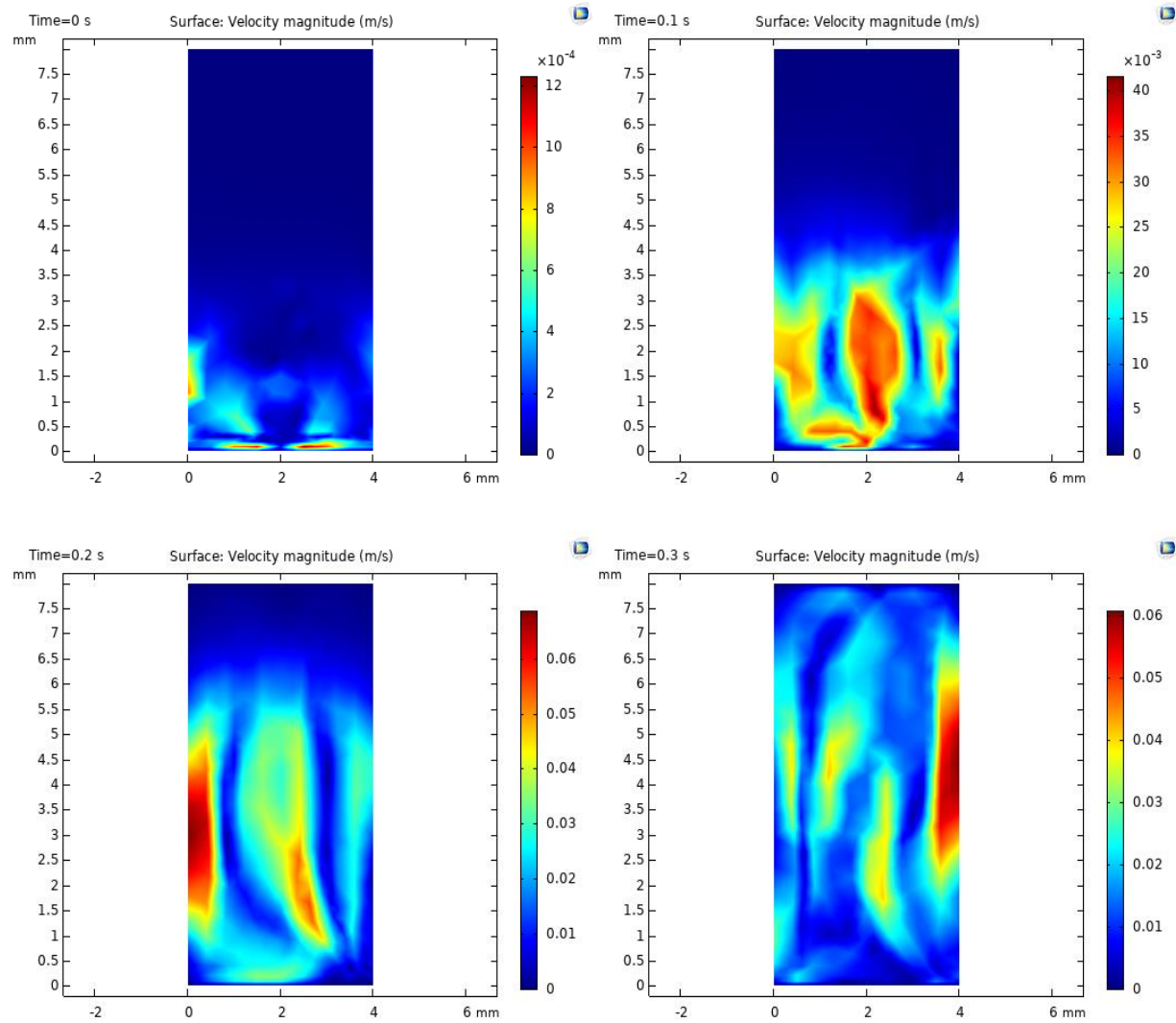


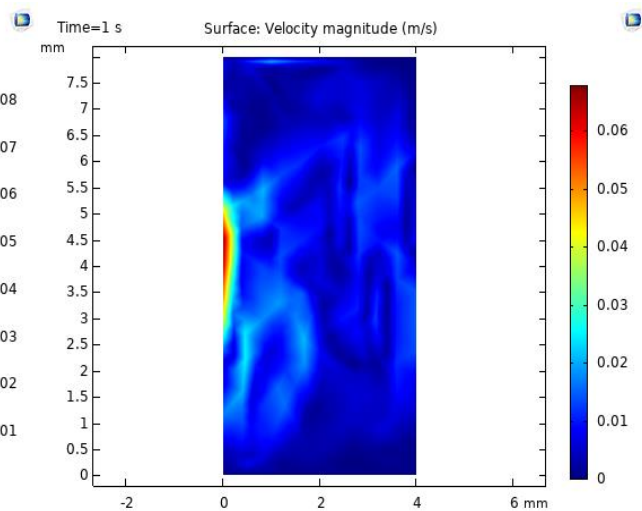
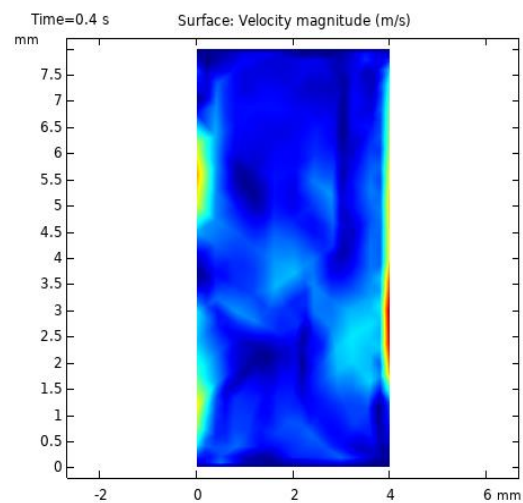
# Velocity

2W

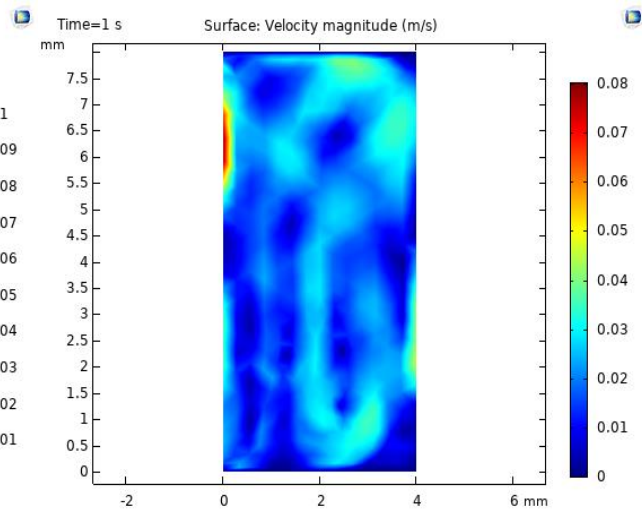
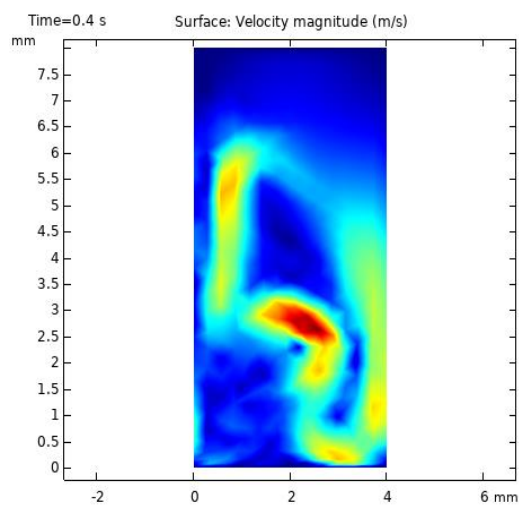
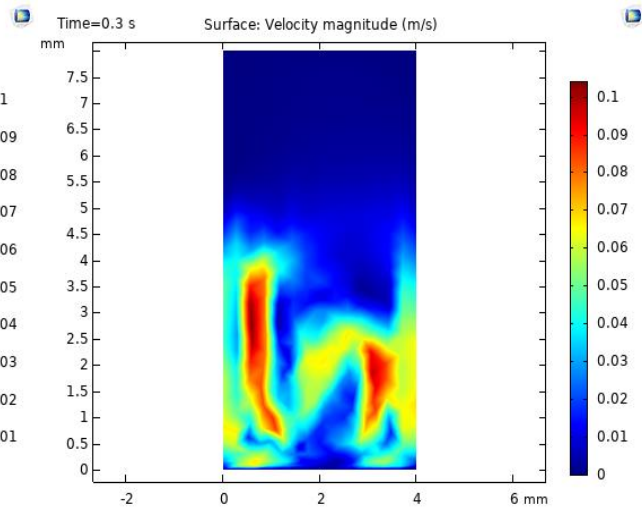
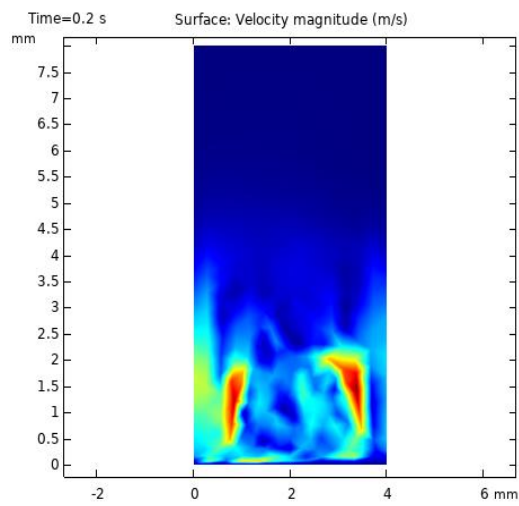
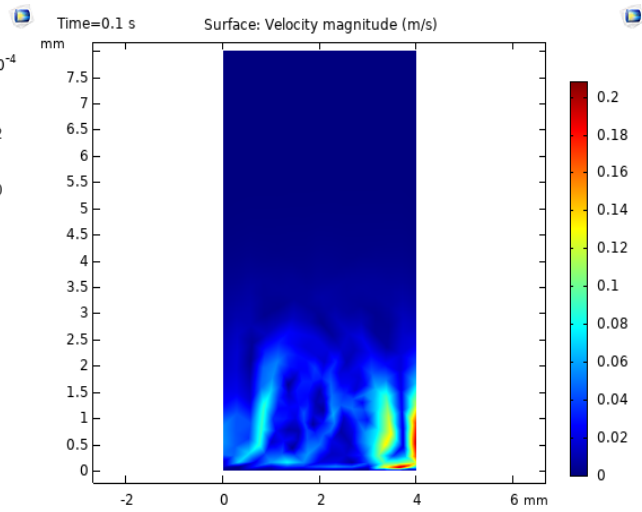
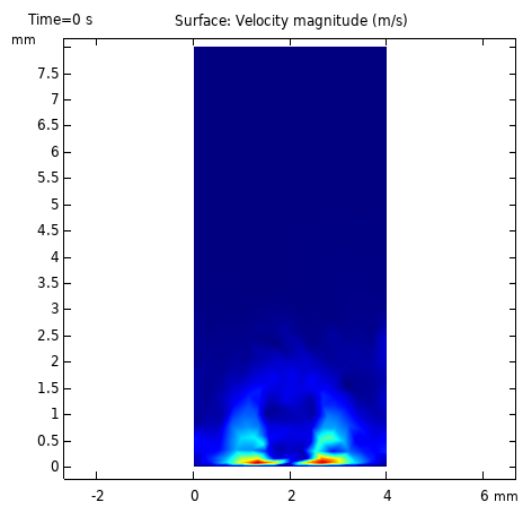


5W





10W



# Conclusion

---

- We learnt how to use the COMSOL Multiphysics software with Physics Laminar Flow, Level Set and Heat Transfer in Fluids i.e. for CFD.
- The flow patterns of the bubble are different for different values of the heat rate supplied at the bottom surface of the vessel which are clearly seen from the animation and snips attached.
- Further, this study can be carried out for various other values of heat rates as well in order to get an extensive view of the bubble dynamics.
- The study can also be extended to flow boiling conditions to see the effect that the base fluid velocity creates on the overall transport of the bubble.

# Documentation

---

- [Volume fraction \(2W\) animation](#)
- [Volume fraction \(5W\) animation](#)
- [Volume fraction \(10W\) animation](#)