

**T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL**

**MODELING AND SIMULATION OF THE
FUEL SLOSHING PHENOMENON IN FUEL
TANKS AND DESIGN OPTIMIZATION**

MERT AKKUŞ

**A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
APPLIED PROPULSION SYSTEM DESIGN
AND ENGINEERING FOR AEROSPACE
TECHNOLOGIES PROGRAM**

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

JULY 2025

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

**MODELING AND SIMULATION OF THE
FUEL SLOSHING PHENOMENON IN FUEL
TANKS AND DESIGN OPTIMIZATION**

MERT AKKUŞ

**A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
APPLIED PROPULSION SYSTEM DESIGN
AND ENGINEERING FOR AEROSPACE
TECHNOLOGIES PROGRAM**

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

JULY 2025

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

YAKIT TANKLARINDA ÇALKALANMA
FENOMENİNİN MODELLENMESİ VE SİMULASYONU
VE TASARIM OPTİMİZASYONU

MERT AKKUŞ

YÜKSEK LİSANS TEZİ
ENERJİ TEKNOLOJİLERİ ANABİLİM DALI
HAVACILIK VE UZAY TEKNOLOJİLERİNDE
UYGULAMALI İTKİ SİSTEMİ TASARIM
MÜHENDİSLİĞİ PROGRAMI

DANIŞMAN: DR. ÖĞR. ÜYESİ SALİH ÖZEN ÜNVERDİ

TEMMUZ 2025

MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by Mert Akkuş, defended on 04/07/2025 before the jury formed with the 30/06/2025 date and 2025/34 numbered decision of the GTU Graduate Administration Board, has been accepted as a MASTER of SCIENCE thesis in the Department of Energy Technologies, Applied Propulsion System Design and Engineering for Aerospace Technologies Program.

JURY

MEMBER

(THESIS ADVISOR) : Assist. Prof. Salih Özen ÜNVERDİ

MEMBER

: Assist. Prof. Osman OKUR

MEMBER

: Assoc. Prof. Halil BAYRAM

APPROVAL

Gebze Technical University Graduate Administration Board

...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL



*To my
Family...*

ABSTRACT

Fuel sloshing refers to the movement of liquid within partially filled fuel tanks that occurs due to dynamic motions such as acceleration, deceleration, cornering, and external environmental effects experienced by vehicles. This phenomenon can have adverse functional and structural consequences in engineering systems. Sudden and uncontrolled fuel movements exert irregular pressures on the tank walls, leading to increased mechanical stresses and reduced system performance.

In this study, the effects of fuel sloshing are modeled using numerical methods, and comparative analyses are conducted with respect to internal structural configurations. Simulations are performed for varying fill levels (33% and 66%) and different baffle configurations (solid and perforated) and lengths. The aim is to investigate the free surface dynamics, wall pressure distributions, and overall flow characteristics under these conditions, and to evaluate internal structural solutions that can mitigate the adverse effects of sloshing.

Numerical modeling is carried out using the ANSYS Fluent software, employing the Volume of Fluid (VOF) method, which is widely used for modeling multiphase flows. The VOF method effectively tracks the interface between air and fuel phases and accurately captures the time-dependent behavior of the free surface. A transient solution approach is adopted, incorporating the $k-\epsilon$ turbulence model to account for turbulent flow effects. Mesh sensitivity is especially emphasized in near-wall regions.

Various baffle designs are developed and integrated into simulations to reduce sloshing effects. These include both perforated and solid-surface baffles. The analysis results show that longer, perforated baffles effectively dissipate wave energy, reduce wave heights within the tank, and distribute wall pressures more evenly. Parameters such as shear force-time, pressure-time histories, and fuel-to-air ratio variations are analyzed in detail to understand sloshing behavior.

The findings indicate that internal baffle structures can significantly mitigate the effects of sloshing, offering important benefits in terms of both safety and structural integrity. Furthermore, a mesh sensitivity analysis is conducted, and simulations with varying cell counts are compared to ensure accuracy. Stability analysis through CFL calculations is also performed to verify the turbulence modeling.

This thesis not only focuses on modeling sloshing behavior but also provides design optimization insights. It lays the groundwork for the parametric optimization of internal baffle placement, opening size, and length, especially in fuel tanks used in aerospace applications.

Keywords: Fuel Sloshing, CFD, Volume of Fluid, Baffle, Fuel Tank

ÖZET

Yakıt çalkalanması, özellikle kısmen dolu yakıt tanklarında taşıtların hızlanma, yavaşlama, yön değiştirme ve çevresel etkilere maruz kaldığı dinamik hareketler sırasında meydana gelen sıvı yüzey dalgalanmasıdır. Bu olgu, mühendislik sistemlerinde hem işlevsel hem de yapısal açıdan olumsuz sonuçlar doğurabilmektedir. Yakıtın ani ve kontrolsüz hareketleri, tank duvarlarına düzensiz basınçlar uygulayarak mekanik gerilimleri artırmakta ve sistem performansında düşüşe yol açabilmektedir.

Bu çalışmada, yakıt çalkalanma olgusu sayısal yöntemler kullanılarak modellenmiş ve iç yapı düzenlemeleri açısından karşılaştırmalı analizler gerçekleştirilmiştir. Analizler, %33 ve %66 doluluk oranlarında, farklı dalgakıran konfigürasyonlarında (düz ve delikli) ve çeşitli boylarda yapılmıştır. Çalışmanın temel amacı, söz konusu koşullar altında oluşan serbest yüzey hareketlerini, duvar basınçlarını ve genel akış karakteristiklerini incelemek; ayrıca çalkanma etkilerini azaltabilecek iç yapı çözümlerini değerlendirmektir.

Sayısal analizler, ANSYS Fluent yazılımı aracılığıyla gerçekleştirilmiştir. Bu kapsamda, çok fazlı akışların modellenmesinde yaygın olarak tercih edilen Volume of Fluid (VOF) yöntemi kullanılmıştır. VOF yöntemi, hava ve yakıt fazları arasındaki arayüzü hassas biçimde izleyebilmekte ve zamanla değişen serbest yüzey davranışını doğru şekilde yansıtabilmektedir. Kullanılan akış modeli zamana bağlı (transient) çözümleme esasına dayandırılmış ve türbülans etkileri $k-\epsilon$ türbülans modeliyle hesaba katılmıştır. Sayısal ağ (mesh) yapısı oluşturulurken, özellikle duvar bölgelerinde ağ hassasiyeti artırılmıştır.

Farklı senaryolar dikkate alınarak, çalışmada yakıt tankı içerisindeki çalkalanma etkisini azaltmaya yönelik çeşitli dalgakıran tasarımları geliştirilmiş ve simülasyon ortamına entegre edilmiştir. Test edilen dalgakıran konfigürasyonları arasında delikli ve tam yüzeyli yapılar yer almaktadır. Yapılan analizler sonucunda, özellikle uzun ve delikli dalgakıranların sıvı dalgalarının enerjisini etkili biçimde dağıttığı, dalga yüksekliklerini azalttığı ve duvar basınçlarını daha dengeli hale getirdiği gözlemlenmiştir. Çalkanma davranışının değerlendirilmesinde; kesme kuvveti-zaman, basınç-zaman grafiklerinin yanı sıra yakıt-hava oranındaki değişimler gibi birçok parametre detaylı olarak analiz edilmiştir.

Elde edilen sonuçlar, iç yapı olarak kullanılan dalgakıranların çalkalanma etkilerini azaltmada etkili olduğunu ortaya koymuştur. Bu durum hem güvenlik hem de yapısal dayanıklılık açısından önemli kazanımlar sağlamaktadır. Ayrıca çalışmada ağ yapısının çözüme etkisi analiz edilmiş, farklı hücre sayılarında gerçekleştirilen simülasyonlarla çözüm doğruluğu karşılaştırmalı olarak değerlendirilmiştir. Türbülans çözümünün kararlılığı, CFL hesaplamalarıyla test edilmiştir.

Çalışma yalnızca çalkalanma davranışının modellenmesiyle sınırlı kalmamış, aynı zamanda tasarım optimizasyonu açısından da önemli katkılar sunmuştur. Özellikle havacılık uygulamalarında kullanılan yakıt tanklarının tasarımında; dalgakıran konumu, açıklığı ve boyu gibi parametrelerin optimize edilmesine yönelik temel altyapı bu çalışma ile ortaya konmuştur.

Anahtar Kelimeler: Yakıt Çalkalanması, CFD, VOF Yöntemi, Dalgakıran, Yakıt Tankı

ACKNOWLEDGEMENTS

I would like to share my gratitude to my thesis advisor, Assist. Prof. Dr. Salih Özen ÜNVERDİ for his considerable guidance and support throughout the period of this research. His expertise and feedbacks greatly contributed to the progress and quality of this thesis. I would also like to extend my thanks to the academic staff and personnel of the Energy Technologies Department / Applied Propulsion System Design and Engineering for Aerospace Technologies faculty members and Turkish Aerospace Academy, whose knowledge and dedication have shaped my academic journey and broadened my perspective in the field of aerospace technologies. Special thanks are due to my colleagues and teammates at Turkish Aerospace, whose technical insights and collaborative spirit enriched both the practical and theoretical aspects of this study. Their support in understanding real world aerospace subsystems significantly influenced the direction of my research. I am also grateful to my family and close friends for their patience and emotional support throughout this period. Finally, I would like to acknowledge the use of ANSYS Fluent software and other computational tools that enabled the simulation work presented in this thesis.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	vi
ÖZET	vii
ACKNOWLEDGEMENTS	viii
TABLE OF CONTENTS	ix
LIST OF SYMBOLS AND ABBREVIATIONS	xi
LIST OF FIGURES	xii
LIST OF TABLES	xv
1. INTRODUCTION	1
1.1. Background and Motivation	1
1.2. Problem Statement	1
1.3. Objectives and Contribution	2
1.3.1. Objectives of the Study	2
1.3.2. Scientific and Engineering Contribution	3
1.4. Structure of the Thesis	3
2. LITERATURE AND THEORY	5
2.1. Fuel Sloshing Phenomenon	5
2.2. Numerical Simulation of Sloshing	5
2.2.1. Governing Equations and the VOF Method	5
2.2.2. Turbulence Modelling	6
2.2.3. Applications in ANSYS Fluent	7
2.3. Baffle Design	7
2.4. Tank Geometry	7
2.5. Summary of Previous Studies and identified Gaps	8
3. SCOPE AND ASSUMPTIONS	9
3.1. Scope of the Study	9
3.2. Modelling Assumptions	10
3.2.1. Geometric and Physical Assumptions:	10
3.2.2. Numerical and Solver Assumptions:	11
3.3. Limitations and Simplifications	11
3.3.1. Neglected Physical Effects:	11
3.3.2. Computational Constraints:	12
4. NUMERICAL ANALYSIS	13
4.1. Geometry and Tank Configurations	13
4.1.1. Rigid Integrated Tank	13
4.1.2. Baffle Configurations	14
4.1.3. Modelling Considerations	14
4.2. Mesh Structure and Quality	14
4.2.1. Modelling Considerations	14
4.2.2. Mesh Independence Study	15
4.3. Time Step and Numerical Stability	17
4.4. Solver Implementation	17
4.5. Boundary Conditions and Initial Settings	18
4.5.1. Tank Wall Boundary Conditions	18
4.5.2. Free Surface and Phase Conditions	18

4.5.3. Initial Conditions	18
4.5.4. Dynamic Excitation Settings	18
4.5.5. Boundary Condition Summary	19
4.6. Solver Setup and Simulation Parameters	19
4.6.1. Solver Type	19
4.6.2. Pressure Velocity Coupling	20
4.6.3. Spatial Discretization Schemes	20
4.7. Baffle Configurations and Comparison Cases	20
4.8. Named Selections	23
5. RESULTS AND DISCUSSIONS	24
5.1. Effect of Fill Level	25
5.2. Effect of Baffle Quantity	27
5.3. Effect of Perforation	28
5.4. Effect of Baffle Height	29
6. CONCLUSION AND RECOMMENDATIONS	32
6.1. Summary of Findings	32
6.2. Contributions to Engineering Practice	34
6.3. Design Recommendations	35
6.4. Suggestions for Future Work	36
REFERENCES	38
BIOGRAPHY	40
PUBLICATIONS AND PRESENTATIONS FROM THE THESIS	41
APPENDICES	42

LIST OF SYMBOLS AND ABBREVIATIONS

Δt	: Delta Time Step
ρ	: Density
μ	: Dynamic Viscosity
ε	: Epsilon
$\vec{g}$	: Gravitational Acceleration
Hz.	: Hertz
Δx	: Local Mesh Size
m.	: Meter
∇	: Nabla Operator
p	: Pressure
s	: Second
t	: Time
$a(t)$	: Time Dependent Acceleration
$\vec{u}$	: Velocity Vector
α	: Volume Fraction
BSc.	: Bachelor in Science
CFD	: Computational Fluid Dynamics
CFL	: Courant–Friedrichs–Lewy
Conf.	: Configuration
Diff.	: Difference
DNS	: Dynamic Numeric Solution
Ed.	: Edition
Et al.	: Et alii (others)
Frac.	: Fraction
GTU	: Gebze Technical University
LES	: Large Eddy Simulation
Max.	: Maximum
N/A	: Not Applicable
Perf.	: Perforated
pp.	: Pages
PRESTO	: Pressure Staggering Option
Quant.	: Quantity
Rad.	: Radian
RANS	: Reynolds Averaged Navier Stokes
SIMPLE	: Semi Implicit Method for Pressure Linked Equations
UAV	: Unmanned Air Vehicle
VOF	: Volume of Fluid
Vol.	: Volume
Vs.	: Versus

LIST OF FIGURES

	<u>Page</u>
Figure 4.1: Tank Geometry.	13
Figure 4.2: Edge Sizing.	15
Figure 4.3: Non Conformant Y+ Zones.	16
Figure 4.4: Configuration 0.	21
Figure 4.5: Configuration 1.	21
Figure 4.6: Configuration 2.	21
Figure 4.7: Configuration 3.	21
Figure 4.8: Perforated Baffle (Solution Domain).	21
Figure 4.9: Left Wall Selection.	23
Figure 4.10: Symmetry Plane.	23
Figure 6.1: Pressure vs. Configuration.	32
Figure 6.2: Shear vs. Configuration.	32
Figure 6.3: VOF Difference vs. Configuration.	33
Figure A.1: C0-F33 Plot of Volume Fraction on Left Wall.	42
Figure A.2: C0-F66 Plot of Volume Fraction on Left Wall.	42
Figure A.3: C1-S-H3-F33 Plot of Volume Fraction on Left Wall.	43
Figure A.4: C1-S-H3-F66 Plot of Volume Fraction on Left Wall.	43
Figure A.5: C2-S-H3-F33 Plot of Volume Fraction on Left Wall.	44
Figure A.6: C2-S-H3-F66 Plot of Volume Fraction on Left Wall.	44
Figure A.7: C3-S-H3-F33 Plot of Volume Fraction on Left Wall.	45
Figure A.8: C3-S-H3-F66 Plot of Volume Fraction on Left Wall.	45
Figure A.9: C3-P-H3-F33 Plot of Volume Fraction on Left Wall.	46
Figure A.10: C3-P-H3-F66 Plot of Volume Fraction on Left Wall.	46
Figure A.11: C3-P-H2-F33 Plot of Volume Fraction on Left Wall.	47
Figure A.12: C3-P-H2-F66 Plot of Volume Fraction on Left Wall.	47
Figure A.13: C3-P-H1-F33 Plot of Volume Fraction on Left Wall.	48
Figure A.14: C3-P-H1-F66 Plot of Volume Fraction on Left Wall.	48
Figure B.1: C0-F33 Plot of Maximum Shear Stress on Left Wall.	49
Figure B.2: C0-F66 Plot of Maximum Shear Stress on Left Wall.	49
Figure B.3: C1-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.	50
Figure B.4: C1-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.	50
Figure B.5: C2-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.	51
Figure B.6: C2-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.	51
Figure B.7: C3-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.	52
Figure B.8: C3-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.	52
Figure B.9: C3-P-H3-F33 Plot of Maximum Shear Stress on Left Wall.	53
Figure B.10: C3-P-H3-F66 Plot of Maximum Shear Stress on Left Wall.	53
Figure B.11: C3-P-H2-F33 Plot of Maximum Shear Stress on Left Wall.	54
Figure B.12: C3-P-H2-F66 Plot of Maximum Shear Stress on Left Wall.	54
Figure B.13: C3-P-H1-F33 Plot of Maximum Shear Stress on Left Wall.	55
Figure B.14: C3-P-H1-F66 Plot of Maximum Shear Stress on Left Wall.	55
Figure C.1: C0-F33 Plot of Maximum Pressure on Left Wall.	56
Figure C.2: C0-F66 Plot of Maximum Pressure on Left Wall.	56
Figure C.3: C1-S-H3-F33 Plot of Maximum Pressure on Left Wall.	57
Figure C.4: C1-S-H3-F66 Plot of Maximum Pressure on Left Wall.	57
Figure C.5: C2-S-H3-F33 Plot of Maximum Pressure on Left Wall.	58

Figure C.6: C2-S-H3-F66 Plot of Maximum Pressure on Left Wall.	58
Figure C.7: C3-S-H3-F33 Plot of Maximum Pressure on Left Wall.	59
Figure C.8: C3-S-H3-F66 Plot of Maximum Pressure on Left Wall.	59
Figure C.9: C3-P-H3-F33 Plot of Maximum Pressure on Left Wall.	60
Figure C.10: C3-P-H3-F66 Plot of Maximum Pressure on Left Wall.	60
Figure C.11: C3-P-H2-F33 Plot of Maximum Pressure on Left Wall.	61
Figure C.12: C3-P-H2-F66 Plot of Maximum Pressure on Left Wall.	61
Figure C.13: C3-P-H1-F33 Plot of Maximum Pressure on Left Wall.	62
Figure C.14: C3-P-H1-F66 Plot of Maximum Pressure on Left Wall.	62
Figure D.1: C0-F33 Plot of Volume Fraction on Symmetry Plane.	63
Figure D.2: C0-F66 Plot of Volume Fraction on Symmetry Plane.	63
Figure D.3: C1-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.	64
Figure D.4: C1-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.	64
Figure D.5: C2-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.	65
Figure D.6: C2-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.	65
Figure D.7: C3-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.	66
Figure D.8: C3-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.	66
Figure D.9: C3-P-H3-F33 Plot of Volume Fraction on Symmetry Plane.	67
Figure D.10: C3-P-H3-F66 Plot of Volume Fraction on Symmetry Plane.	67
Figure D.11: C3-P-H2-F33 Plot of Volume Fraction on Symmetry Plane.	68
Figure D.12: C3-P-H2-F66 Plot of Volume Fraction on Symmetry Plane.	68
Figure D.13: C3-P-H1-F33 Plot of Volume Fraction on Symmetry Plane.	69
Figure D.14: C3-P-H1-F66 Plot of Volume Fraction on Symmetry Plane.	69
Figure E.1: C0-F33 Contour of Volume Fraction on Symmetry (0-5s).	70
Figure E.2: C0-F66 Contour of Volume Fraction on Symmetry (0-5s).	70
Figure E.3: C1-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).	71
Figure E.4: C1-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).	71
Figure E.5: C2-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).	72
Figure E.6: C2-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).	72
Figure E.7: C3-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).	73
Figure E.8: C3-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).	73
Figure E.9: C3-P-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).	74
Figure E.10: C3-P-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).	74
Figure E.11: C3-P-H2-F33 Contour of Volume Fraction on Symmetry (0-5s).	75
Figure E.12: C3-P-H2-F66 Contour of Volume Fraction on Symmetry (0-5s).	75
Figure E.13: C3-P-H1-F33 Contour of Volume Fraction on Symmetry (0-5s).	76
Figure E.14: C3-P-H1-F66 Contour of Volume Fraction on Symmetry (0-5s).	76
Figure F.1: C0-F33 Contour of Volume Fraction on Left Wall (0-5s).	77
Figure F.2: C0-F66 Contour of Volume Fraction on Left Wall (0-5s).	77
Figure F.3: C1-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).	78
Figure F.4: C1-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).	78
Figure F.5: C2-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).	79
Figure F.6: C2-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).	79
Figure F.7: C3-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).	80
Figure F.8: C3-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).	80
Figure F.9: C3-P-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).	81
Figure F.10: C3-P-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).	81
Figure F.11: C3-P-H2-F33 Contour of Volume Fraction on Left Wall (0-5s).	82
Figure F.12: C3-P-H2-F66 Contour of Volume Fraction on Left Wall (0-5s).	82
Figure F.13: C3-P-H1-F33 Contour of Volume Fraction on Left Wall (0-5s).	83



LIST OF TABLES

	<u>Page</u>
Table 4.1: Mesh Level vs. Maximum Pressure.	16
Table 4.2: Boundary Conditions.	19
Table 4.3: Spatial Discretization.	20
Table 4.4: Comparison Cases.	22
Table 5.1: Volume Fraction Results.	24
Table 5.2: Wall Pressure and Shear Results.	24
Table 5.3: Pressure and Shear vs. Fill Level.	25
Table 5.4: Volume Fraction Difference vs. Fill Level.	26
Table 5.5: Pressure and Shear vs. Baffle Quantity.	27
Table 5.6: Volume Fraction vs. Baffle Quantity.	27
Table 5.7: Pressure and Shear vs. Perforation.	28
Table 5.8: Volume Fraction Difference vs Perforation.	28
Table 5.9: Pressure and Shear vs. Baffle Height.	30
Table 5.10: Volume Fraction Difference vs. Baffle Height.	30

1. INTRODUCTION

1.1. Background and Motivation

Recently, the need for lightweight, efficiency in terms of structure, and dynamically stable fuel systems has become increasingly critical in aerospace and automotive industries. As vehicles undergo sudden maneuvers such as braking, cornering, or takeoff and landing operations, the fuel inside partially filled tanks exhibits complex free surface motion known as sloshing. This fluid movement, which results from inertia and gravitational restoring forces, can exert significant dynamic loads on the internal walls of fuel tanks, potentially leading to structural fatigue, noise generation, and even functional failure of fuel delivery systems.

Fuel sloshing becomes especially problematic in vehicles with frequent acceleration and deceleration cycles or in systems with large flat bottom tanks and high fill ratios. The impact of sloshing is not limited to mechanical aspects alone; it can also compromise vehicle stability by shifting the center of gravity, cause measurement errors in fuel sensors, and reduce overall operational reliability. In aerospace applications, where continuous and uninterrupted fuel flow is vital, uncontrolled sloshing may endanger flight performance and safety.

Traditional design approaches have often treated sloshing effects empirically or assumed simplified flow models that fail track transient movements and the nature of fuel behavior. Nevertheless, progress in computational fluid dynamics offers engineers to simulate multiphase flows using VOF. This approach allows to observe wall pressures, shear pressures and wave movement by tracking effect of baffles.

The baffled fuel tanks is shown to be effective in terms of preventing large slosh amplitudes and reduce energy levels that liquid holds due to acceleration.

1.2. Problem Statement

Fuel sloshing in partially filled fuel tanks creates an engineering problem in modern vehicles, especially under dynamic loading. The unsteady movement due to braking,

acceleration causes pressure fields, free surface movements. These effects may cause fatigue, vibration and inaccurate fuel quantity readings.

Although the demand for slosh prevented fuel systems, many designs rely on laboratory tests and trial-error investigations. This problem may end up having ineffective or over-conservative design solutions. Outcome of those design may lead to weight increase or complex models harder or time-inefficient production periods.

Another crucial gap lies in the optimization of internal slosh suppressing devices such as baffles. While baffles are commonly employed to mitigate sloshing, the optimal design parameters including orientation, perforation, spacing, and size are highly dependent on specific application scenarios. Existing studies often analyze limited baffle configurations, without exploring their comparative effectiveness across different tank geometries.

Given these limitations, there is a need for a detailed, simulation driven investigation that incorporates high fidelity CFD modeling techniques such as the VOF method to evaluate sloshing induced behavior in integrated tanks. Such a study must assess how internal baffle configurations influence sloshing suppression, pressure distribution, and fluid stability under varying fill levels and baffle locations.

This thesis addresses this gap by systematically analyzing the sloshing behavior of representative fuel tank architectures using ANSYS Fluent, and by evaluating the performance of multiple baffle configurations. The outcome is expected to provide practical insights into slosh mitigation strategies and to support the design of safer, more reliable fuel storage systems.

1.3. Objectives and Contribution

1.3.1. Objectives of the Study

The main aims of this thesis are as follows:

- To evaluate the effects of different fill levels on sloshing characteristics, including shear, wall pressure, and fluid displacement.
- To implement and assess various internal baffle configurations and shapes aimed at minimizing sloshing induced pressure fluctuations and improving fuel stability.

- To provide practical design recommendations for improving the performance, reliability, and safety of fuel storage systems in dynamic environments.

1.3.2. Scientific and Engineering Contribution

This thesis provides a systematic and comparative CFD based analysis of sloshing in flexible and rigid fuel tanks, which is a relatively underexplored area in the literature. By employing the VOF method within ANSYS Fluent, the study enables high fidelity simulation of transient, multiphase interactions under realistic motion profiles.

Moreover, the study introduces a configurational evaluation of internal baffle configurations, considering their geometric and positional variations. This allows for the identification of effective slosh mitigation strategies that are adaptable across different tank types and operational conditions. The findings contribute to the ongoing development of safer and more efficient fuel system designs in aerospace and heavy-vehicle engineering.

By integrating theoretical understanding with practical simulation outcomes, this research bridges a critical gap between conceptual modeling and real world fuel system design and proposes an adaptable framework for future sloshing analysis and tank optimization efforts.

1.4. Structure of the Thesis

This thesis consists of six chapters, each addressing a specific component of the research process and contributing to the overall understanding of the fuel sloshing phenomenon and its mitigation:

- **Introduction:** Provides the background, motivation, problem definition, objectives, and overall organization of the thesis.
- **Literature and Theoretical Background:** Reviews the fundamental concepts of fuel sloshing, CFD modeling, VOF methodology, and previous studies relevant to baffle design and tank geometry.
- **Scope and Assumptions:** Defines the boundaries of the study, outlines the modeling assumptions, and clarifies the limitations and constraints adopted in the simulations.

- **Numerical Analysis:** Describes the simulation methodology, including tank modeling, mesh generation, boundary conditions, solver settings, and the different test cases involving baffle configurations.
- **Results and Discussion:** Provides and evaluates the simulation results with a focus on fluid motion, pressure-shear distribution, volume fraction and the performance of baffle designs under different conditions.
- **Conclusions and Recommendations:** Sums up the important outcomes of this study and provides suggestions for future studies and potential design improvements.



2. LITERATURE AND THEORY

2.1. Fuel Sloshing Phenomenon

Sloshing refers to motion of liquid's free surface inside container due to any disturbance. Depending on the type of disturbance and shape of the container, fluid surface movement varies. The slosh problem includes pressure distribution, forces, natural frequencies and moments [1].

Sloshing movement is defined by nonlinearity and randomness, which are accompanied by large deformations due to breaking and movement of free surfaces [2]. In order to reduce sloshing in tanks, baffle usage is very popular solution as a passive control method, which reduces energy levels, dividing flows in tank into sub-flows [3].

Analysis of liquid sloshing offers challenge due to complex physics. However, experimental investigation is not cost effective. Simulations could play an important role in obtaining critical information on the design of fuel tanks at earlier durations to reduce manufacturing times [4].

2.2. Numerical Simulation of Sloshing

2.2.1. Governing Equations and the VOF Method

The volume of fluid method is powerful method to simulate free surface. It captures the complex surface geometries [5].

The motion of incompressible, Newtonian fluids is governed by the Navier–Stokes equations, which express the conservation of mass and momentum. These are written as:

- **Continuity equation (mass conservation):**

$$\nabla \cdot \vec{u} = 0 \quad (2.1)$$

- **Momentum conservation equation:**

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla \cdot (\rho\vec{u}\vec{u}) = -\nabla p + \nabla \cdot (\mu\nabla\vec{u}) + \rho\vec{g} \quad (2.2)$$

$\vec{u}$ is the velocity vector, p is the pressure, ρ is the fluid density, μ is the dynamic viscosity, and $\vec{g}$ is the gravitational acceleration vector [6].

To simulate the interface between two fluids (fuel and air), the volume of fluid method is commonly employed. VOF tracks the volume fraction of each fluid in computational cells:

$$\frac{\partial\alpha}{\partial t} + \vec{u} \cdot \nabla\alpha = 0 \quad (2.3)$$

where α is the volume fraction of the primary phase (fuel), with $\alpha = 1$ indicating a cell fully filled with fuel, $\alpha = 0$ indicating air, and $0 < \alpha < 1$ indicating a free surface region [7].

VOF assumes a shared velocity field and solves a single set of momentum equations for the entire domain. The physical properties at each location are calculated based on a volume fraction weighted average. This makes it ideal for modeling sloshing, where sharp free surfaces and large interface deformations are present.

2.2.2. Turbulence Modelling

Due to instability of the sloshing, turbulence modeling is essential for capturing realistic wave motions and pressure fluctuations. Among available models, the realizable $k-\varepsilon$ turbulence model provides a good balance between accuracy and computational cost, especially for internal flow problems with recirculation. This model solves transport equations for the turbulent kinetic energy k and its dissipation rate ε , incorporating a variable turbulent viscosity coefficient to satisfy the realizability condition.

In this study, the realizable $k-\varepsilon$ model is used in conjunction with the VOF method to accurately represent the turbulent multiphase flow within the tank [4].

2.2.3. Applications in ANSYS Fluent

ANSYS Fluent provides a robust framework for VOF-based sloshing analysis. It supports unsteady, transient simulations with adaptive time-stepping, surface tension modeling, and interface sharpening. In Fluent, the geometric reconstruction scheme is typically selected for better interface resolution in VOF applications [8].

Tank geometries are discretized using structured or hybrid meshes with finer cells near the free surface to enhance accuracy. Boundary conditions are defined based on the motion scenario, such as inertial acceleration inputs or moving reference frames, to replicate braking or swaying conditions.

This software enables the investigation of critical parameters such as free surface motion, wave elevation, pressure fluctuations, and the effectiveness of baffle configurations in slosh conditions.

2.3. Baffle Design

Baffles are designed to interrupt sloshing movement in fuel tank. Several studies conducted in literature. It has been published that baffle height is important factor to reduce sloshing induced effects. Fill level is also crucial parameter to track on volume of fluid method [9].

Baffles act as internal resistive surfaces that reflect and disperse incoming waves. When optimized, they reduce peak wall pressures and stabilize free surface elevation, especially under dynamic excitation conditions [10].

2.4. Tank Geometry

The geometry of the fuel tank plays a important role in sloshing behavior. Rectangular tanks, for example, are more susceptible to large wave amplitudes due to flat vertical walls and wide free surfaces. In contrast, cylindrical and curved-wall tanks tend to induce smoother wave propagation and reduced peak pressures [11].

Response to sloshing and the effect of internal baffles are quantitatively compared in terms of free surface motion, pressure distribution, and flow stabilization.

2.5. Summary of Previous Studies and identified Gaps

Numerous studies have been conducted to investigate sloshing phenomena in partially filled tanks using both experimental and numerical methods. CFD based approaches, particularly those employing the VOF method, have enabled accurate simulations of multiphase fluid interactions under dynamic excitations. There are studies focusing on fill level, tank geometry, and baffle placement [12] on sloshing behavior.

Significant attention has also been devoted to baffle design as a slosh suppression strategy. Research has shown that perforated and solid baffles can reduce wave heights, surface deformation, and dynamic wall pressures. However, most of these investigations have limited their scope to either rigid tank structures or simplified rectangular geometries without considering the interplay between different tank types and internal structures under realistic conditions.

In the context of turbulence modeling and solver accuracy, many previous studies have used RANS models such as standard $k-\epsilon$. Furthermore, mesh sensitivity and solver stability are often addressed briefly, with limited parametric evaluation of simulation quality.

This thesis addresses the above gaps by providing:

- An evaluation of various baffle geometries across different fill levels,
- A high resolution VOF simulation framework with mesh sensitivity checks,
- A more realistic understanding of rigid tank sloshing behavior under literature-based acceleration function
- The findings aim to enhance the design of slosh resistant fuel systems, especially in aerospace applications where lightweight fuel tanks are increasingly adopted.

3. SCOPE AND ASSUMPTIONS

3.1. Scope of the Study

The scope is specifically limited to the following framework:

- Tank type: A rigid integrated tank, commonly used in land vehicles and planes.
- Simulation method: A three-dimensional CFD simulation framework is developed using ANSYS Fluent, employing the VOF method to capture the transient multiphase flow.
- Tank excitation: Simulations are conducted under external excitations simulating longitudinal acceleration, representing realistic conditions experienced in operational scenarios.
- Baffle configurations: Different types of internal baffles (solid and perforated) with varying geometric parameters (number, spacing, position, height) are implemented to assess their influence on sloshing suppression.
- Parameters of interest: The analysis evaluates:
 - Volume fraction of fuel induced by slosh movement,
 - Transient pressure distribution on tank walls,
 - Shear stress on tank wall,
 - Effectiveness of baffle geometry in minimizing peak loads.
- Fill levels: The effect of different fill ratios (33% and 66%) is studied to understand sloshing severity across a range of operating conditions.

This study does not include:

- Thermal effects or phase change phenomena, assuming isothermal and incompressible fluid conditions.
- Multicomponent fuel modeling, focusing instead on single-phase Newtonian liquid behavior.
- Fluid structure interface model

- Structural analysis of tank walls

This thesis aims to ensure accurate interpretation of results and to provide a clearly scoped evaluation of sloshing mitigation strategies applicable to rigid tank architectures.

3.2. Modelling Assumptions

To construct a feasible and computationally efficient numerical model of the fuel tank sloshing problem, several assumptions are made in accordance with established practices in the CFD literature. These assumptions are necessary to reduce model complexity while retaining the dominant physical effects relevant to the study objectives.

3.2.1. Geometric and Physical Assumptions:

- The fuel tanks are modeled as closed and rigid-walled containers.
- The internal geometry includes baffles of varying configurations. Baffles are assumed to be 10mm thin solid plates or uniformly perforated geometries without structural deformation or dynamic response.
- Baffles are not investigated in narrow direction, due to less sensitivity to slosh and modelled as 200mm in width in each case.
- The fluid is Newtonian, incompressible, and isothermal. Jet A1 liquid properties are used for density and viscosity. (Ansys Library)
- The ambient air phase is included in the simulations to enable free surface tracking via the VOF method, but surface tension(constant) and evaporation effects are neglected.
- Acceleration is given in longitudinal and gravity direction. No acceleration is provided in narrow direction of tank, since this direction is less sensitive to slosh due to shape.

3.2.2. Numerical and Solver Assumptions:

- The Volume of Fluid is used to track the free surface interface, assuming a shared velocity field across phases [7].
- Gravity is applied in the vertical direction, while longitudinal acceleration simulates harmonic slosh motion via inertial reference frames.
- Turbulence is modeled using the realizable $k-\varepsilon$ model, suitable for internal, wall-bounded, transient flows.
- The tank wall is assumed to be adiabatic and no-slip, with zero wall deformation.
- Mesh refinement studies ensure numerical independence, but dynamic mesh motion is not included in the scope.
- Simulations are carried out as transient.

These assumptions align with common practices in previous CFD studies on sloshing and are sufficient to capture the dominant fluid dynamics and assess the impact of baffle design and tank type under realistic excitation.

3.3. Limitations and Simplifications

While this study provides a comprehensive numerical investigation of sloshing phenomena using CFD techniques, several simplifications are adopted to ensure computational feasibility and to focus on the primary objectives of the research. These limitations, although necessary, may influence the generalizability of the results and should be considered when interpreting the findings.

3.3.1. Neglected Physical Effects:

- Thermal effects and evaporation: The fluid is assumed isothermal. Heat transfer between the fluid and tank walls, as well as phase change (e.g., vaporization), are neglected, which may influence results in aerospace applications where thermal gradients exist.

- Surface tension and cavitation: Surface tension forces are constant in the VOF setup, and cavitation phenomena are not modeled, under the assumption that operating pressures remain above vapor pressure thresholds.

3.3.2. Computational Constraints:

- Turbulence modeling: While the $k-\epsilon$ model is sufficient for sloshing related turbulence, more advanced models (e.g., LES or DNS) are not used due to high computational cost and convergence complexity.
- Grid resolution: Despite mesh independence studies, grid refinement is limited to practical levels to maintain manageable simulation times. Extremely fine resolutions near free surfaces are avoided.
- Boundary conditions: External acceleration is applied via inertial reference frames. No external vehicle motion or coupled dynamics (e.g., suspension, pitch) are considered.
- Material non-idealities: The fuel is modeled Newtonian liquid as single phase with constant properties. Real fuel behavior under variable temperature or multi-component mixing is beyond the scope of this study.

These simplifications are consistent with earlier CFD-based sloshing research and while they reduce model fidelity in certain aspects, they allow for focused analysis on the key parameters: tank geometry, baffle configuration, fill level, and free surface behavior under dynamic loading.

4. NUMERICAL ANALYSIS

4.1. Geometry and Tank Configurations

The geometry of the fuel tanks is important in sloshing dynamics, wave motion, and effectiveness of slosh mitigation strategies. In this study, two distinct tank configurations are analyzed under identical boundary conditions to assess the differences in fluid behavior and baffle performance.

4.1.1. Rigid Integrated Tank

The rigid integrated tank reflects typical automotive and terrestrial vehicle fuel systems. The walls of the integrated tank are completely inflexible under acceleration.

Key geometric features:

Tank dimensions: 0.8 m (length) $\times$ 0.3 m (width) $\times$ 0.2 m (height) (identical external dimensions for direct comparison) (See **Figure 4.1**)

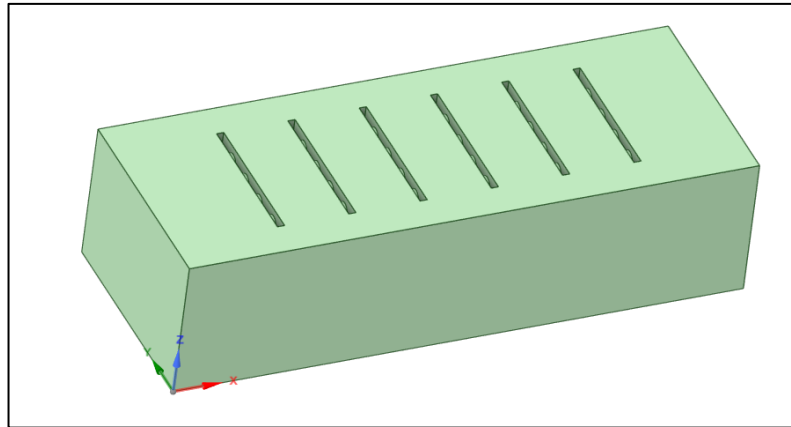


Figure 4.1: Tank Geometry.

- Wall conditions: Rigid, non-deforming boundaries.
- Fill levels: 33% and 66%
- Baffles: Flat and perforated
- Height: Full (100mm) and half (50mm)

- Perforation cut-outs (10mm in radius, uniform 16 circles and symmetrically placed, 19% open to air ratio of area)

4.1.2. Baffle Configurations

Various internal baffle configurations are implemented to analyze their effectiveness in slosh mitigation:

- Solid vertical baffles: Full-height plates perpendicular to the longitudinal axis, placed at specific distances from the tank walls.
- Perforated baffles: Plates with circular openings (porosity ratio approximately 19%) to allow partial fluid passage and energy dissipation.
- Number: 2 to 13 baffles placed bottom and upper half of fuel tank

4.1.3. Modelling Considerations

- Mesh design: Higher mesh density near baffle edges to capture wave breaking and localized pressure peaks.
- Initial conditions: Fluid assumed at rest with gravitational settling before the application of dynamic excitation.

These configurations enable a systematic assessment of how internal structural modifications influence sloshing intensity, wall pressure distributions, and overall fluid stability for both flexible and rigid tank architectures.

4.2. Mesh Structure and Quality

Mesh structure is a critical factor influencing the accuracy and stability of CFD simulations, especially when modeling transient free surface phenomena such as sloshing. Proper mesh design ensures that important physical features, such as interface deformation and wall pressure peaks, are accurately captured.

4.2.1. Modelling Considerations

In this study, the tank domain is discretized using a structured mesh wherever possible to ensure low numerical diffusion and stable interface tracking with the VOF method.

Near complex geometries, such as baffle edges, local refinement is employed to capture flow separation and energy dissipation.

Mesh characteristics:

- Edge sizing near the baffle zones and finer mesh geometries. (See **Figure 4.2**)

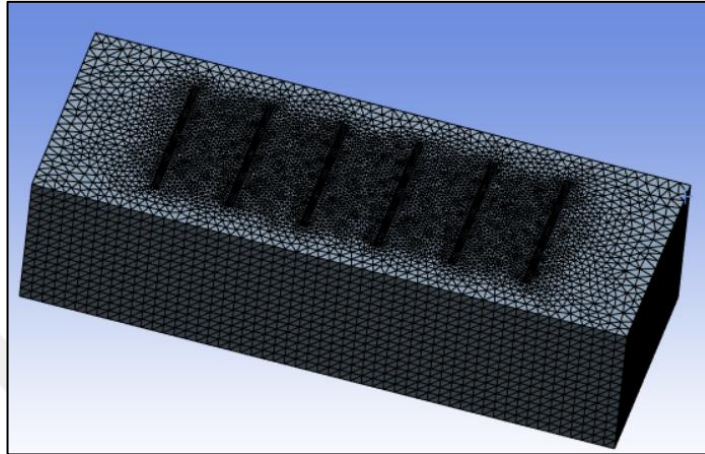


Figure 4.2: Edge Sizing.

- Coarser mesh zones in bulk liquid regions where gradients are less severe.
- Boundary layer refinement near tank walls to improve wall shear stress prediction and pressure accuracy.
- Adaptive meshing is not used due to its additional computational cost and convergence complexity under large transient deformations.

4.2.2. Mesh Independence Study

To investigate mesh effect on results, four mesh densities are tested:

- Coarse mesh: 114733 cells
- Medium mesh: 130098 cells
- Fine mesh: 146677 cells
- Finer mesh: 170242 cells

The average is calculated to compare mesh cases. Face meshing, edge and element sizing methods are utilized to distribute sufficient mesh to the needed surfaces. The

density increases around baffle zones. By changing edge and element sizing, mesh study is exercised.

Variation of selected parameters across mesh sizes is given in **Table 4.1**.

Table 4.1: Mesh Level vs. Maximum Pressure.

Mesh Quantity	Facet Maximum Pressure (Pa) (50 time step)	Difference
114733	46889.66	N/A
130098	1709.03	-96%
146677	1600.22	-6%
170242	1623.62	+1%

The differences between the medium and fine mesh are within acceptable engineering limits (below 5%), justifying the use of the fine mesh for further analysis. To be conservative, although convergence is reached at three hundred thousand mesh, 146 thousand mesh is selected for analysis. Y^+ value is found to be 80.6 area weighted average. For $k-\epsilon$ model with standard wall function, this value should be above 30. Some areas are non-conformant shown in **Figure 4.3**. Due to computational cost, since zones are limited, it is accepted as successful solution.

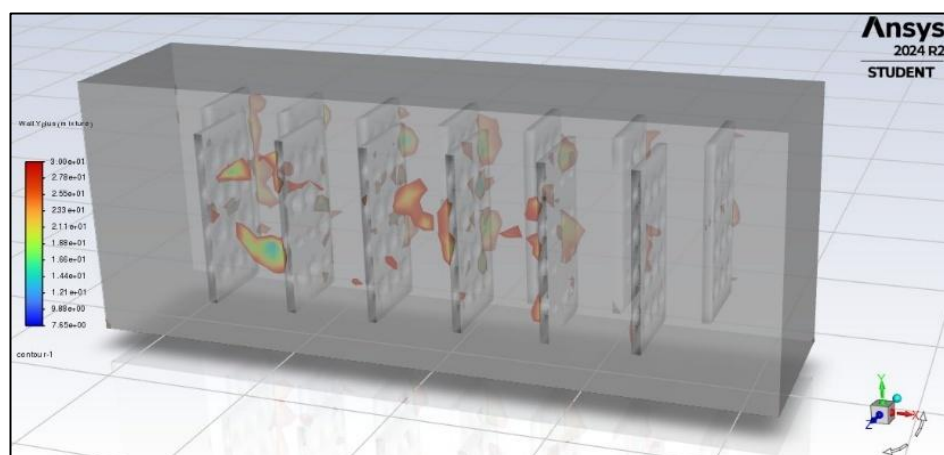


Figure 4.3: Non Conformant Y^+ Zones.

4.3. Time Step and Numerical Stability

In transient CFD simulations involving the VOF method, the selection of an appropriate time step is critical to ensure both the numerical stability and physical accuracy of the solution. A key parameter used in assessing time step stability is the Courant–Friedrichs–Lewy (CFL) number. For implicit schemes, maintaining CFL around 1 to 1.5 is generally required for stability, whereas explicit schemes, such as the one employed in this study, allow for slightly higher CFL values while maintaining convergence and accuracy.

A mesh resolution of 0.015 meters and a maximum observed flow velocity of approximately 2.01 m/s were used. To balance computational cost and accuracy, a time step size of 0.008 seconds was selected. The corresponding CFL number was calculated using the following formula:

$$\text{CFL} = \frac{u \cdot \Delta t}{\Delta x} \quad (4.1)$$

where u is the characteristic flow velocity, Δt is the time step size, and Δx is the local mesh size. Substituting the values into the formula, the CFL number was found to be approximately **1.07**. This value is considered acceptable for simulations using the implicit VOF method, providing a good balance between temporal resolution and numerical stability.

4.4. Solver Implementation

In this study:

- The geometric reconstruction scheme is used for interface capturing.
- Implicit body force formulation is applied to handle gravity and inertial accelerations.
- The second-order upwind method is used for momentum and volume fraction discretization to improve solution accuracy.

These settings provide a good balance between numerical stability and physical accuracy for highly transient multiphase flows such as fuel sloshing.

4.5. Boundary Conditions and Initial Settings

Proper implementation of boundary conditions ensures that the fluid behavior within the tank replicates real-world operating conditions as closely as possible.

4.5.1. Tank Wall Boundary Conditions

- Rigid tank walls: In the integrated tank configuration, all walls are modeled as stationary no-slip walls, fluid velocity at the wall is zero ($\vec{u} = 0$).
- Adiabatic walls: Heat transfer is neglected; all tank surfaces are treated as adiabatic (zero heat flux).

Wall shear stresses are computed by the turbulence model, and roughness effects are ignored to simplify the analysis.

4.5.2. Free Surface and Phase Conditions

- Top boundary: The tank is considered fully enclosed; there is no open venting surface. Air is trapped above the fuel, allowing realistic air compression and interaction with the liquid phase.
- Phase interaction: Surface tension effects are constant, and no phase change (evaporation/condensation) occurs.

The two-phase interface between fuel and air is captured using the volume of fluid method with geometric reconstruction for accurate interface sharpness.

4.5.3. Initial Conditions

- Fluid initialization: The tank is initially at rest, with the liquid phase positioned according to the specified fill level (33% and 66%)
- Velocity field: The initial velocity throughout the domain is set to zero ($\vec{u} = 0$).

4.5.4. Dynamic Excitation Settings

To simulate acceleration effects:

An inertial reference frame with a specified longitudinal acceleration is applied.

Acceleration function is given as

$$a(t) = -12.63[\text{m/s}^2] \cdot \sin(12.566[\text{rad/s}] \cdot t) \quad (4.2)$$

which is estimated by 0.08m amplitude [13] and 2 Hz frequency commonly used interface in helicopter cabin section [14].

Acceleration is treated as a body force added to the gravitational field.

This setup creates transient sloshing behavior without requiring explicit motion of the computational mesh.

4.5.5. Boundary Condition Summary

Summary is provided in **Table 4.2**.

Table 4.2: Boundary Conditions.

Boundary Type	Condition Applied
Tank Walls	No-slip, stationary, adiabatic
Free Surface (Fuel-Air)	Captured using VOF, surface tension constant
Initial Liquid Velocity	Zero
Dynamic Excitation	Gravitational acceleration and harmonic wave function along longitudinal axis

4.6. Solver Setup and Simulation Parameters

Accurate modeling of transient sloshing behavior requires careful selection of solver settings, numerical methods, and simulation parameters. In this study, the setup within ANSYS Fluent is chosen to ensure solution stability, accuracy, and computational efficiency.

4.6.1. Solver Type

- Solver: Pressure-based, transient solver.
- Time Stepping: Fixed, 0.008s, 625 time steps in total

- Multiphase Model: Volume of Fluid (VOF) with two phases: liquid (fuel) and gas (air).

4.6.2. Pressure Velocity Coupling

- Pressure–velocity coupling scheme: SIMPLE.
- Relaxation factors: Default values for pressure, momentum, and volume fraction equations to balance convergence rate and stability.

SIMPLE is widely used for multiphase, steady, and transient incompressible flows, providing reliable convergence behavior.

4.6.3. Spatial Discretization Schemes

To improve numerical accuracy while controlling numerical diffusion, the following discretization schemes in **Table 4.3** are used:

Table 4.3: Spatial Discretization.

Quantity	Discretization Scheme
Pressure	PRESTO (Pressure Staggering Option)
Momentum	Second-order upwind
Volume Fraction (VOF)	Geo-Reconstruct
Turbulent Kinetic Energy (k)	Second-order upwind
Turbulent Dissipation (ϵ)	Second-order upwind

- PRESTO scheme is chosen for pressure discretization to better resolve pressure gradients near free surfaces and walls.
- Geo-Reconstruct method provides a sharp, accurate capture of the free surface between phases.

4.7. Baffle Configurations and Comparison Cases

Internal baffles are widely employed to mitigate the adverse effects of liquid sloshing by disrupting the wave motion and redistributing fluid momentum. In this study, several baffle designs and placements are investigated to assess their effectiveness

under dynamic excitation conditions [10]. These configurations are given in **Figure 4.4**, **Figure 4.5**, **Figure 4.6**, **Figure 4.7**. Perforation (See **Figure 4.8**), fill level and baffle height combinations provided in **Table 4.4**.

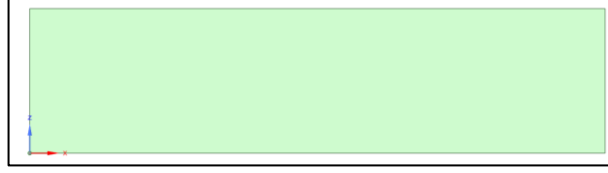


Figure 4.4: Configuration 0.

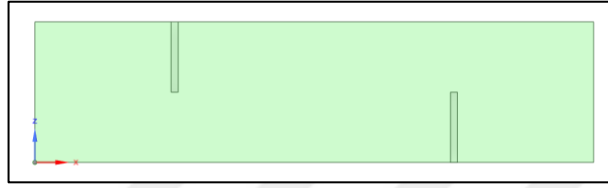


Figure 4.5: Configuration 1.

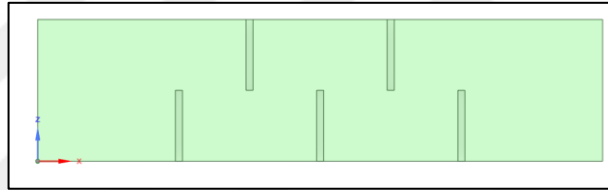


Figure 4.6: Configuration 2.

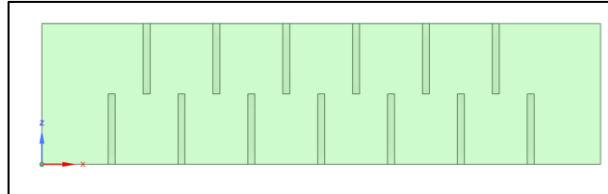


Figure 4.7: Configuration 3.

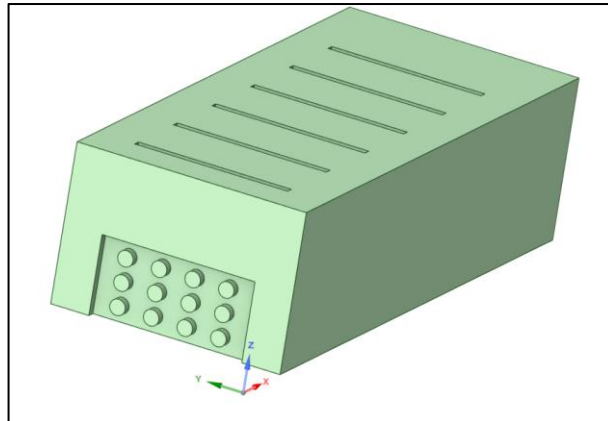


Figure 4.8: Perforated Baffle (Solution Domain).

Table 4.4: Comparison Cases.

Baffle Place	Baffle Type	Baffle Height	Fill Level	Case 1	Case 2	Case 3	Case 4	Code		
Conf. 0	N/A	N/A	33%	Fill				C0-F33		
Conf. 0	N/A	N/A	66%	Level				C0-F66		
Conf.1	Solid	Full	33%	Fill Level	Baffle Quant.			C1-S-H3-F33		
Conf. 1	Solid	Full	66%					C1-S-H3-F66		
Conf. 2	Solid	Full	33%	Fill Level				Perf.	Baffle Height	C2-S-H3-F33
Conf. 2	Solid	Full	66%							C2-S-H3-F66
Conf. 3	Solid	Full	33%	Fill Level						C3-S-H3-F33
Conf. 3	Solid	Full	66%							C3-S-H3-F66
Conf. 3	Perf.	Full	33%	Fill Level						C3-P-H3-F33
Conf. 3	Perf.	Full	66%							C3-P-H3-F66
Conf. 3	Perf.	2/3	33%	Fill Level				C3-P-H2-F33		
Conf. 3	Perf.	2/3	66%						C3-P-H2-F66	
Conf. 3	Perf.	1/3	33%	Fill Level					C3-P-H1-F33	
Conf. 3	Perf.	1/3	66%						C3-P-H1-F66	

By systematically varying baffle type and positioning, the study identifies which configurations best suppress sloshing-induced free surface elevations and wall pressures for each tank type.

4.8. Named Selections

The named selections that will be mentioned in the rest of the document are provided in **Figure 4.9** and **Figure 4.10**.

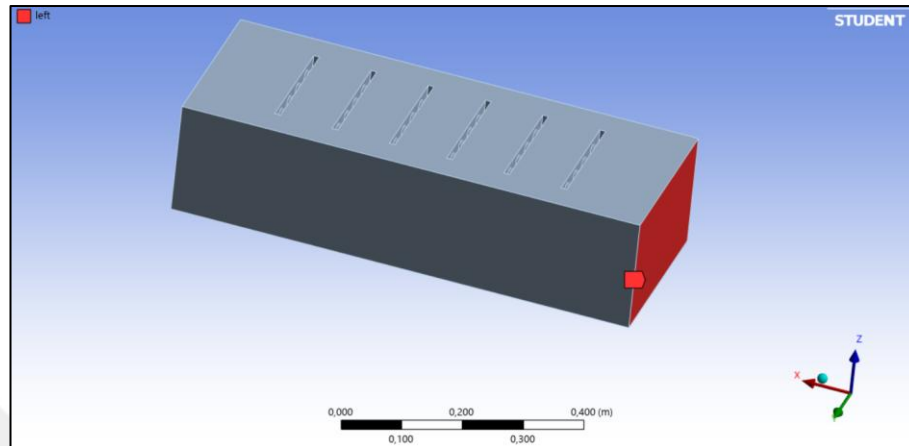


Figure 4.9: Left Wall Selection.

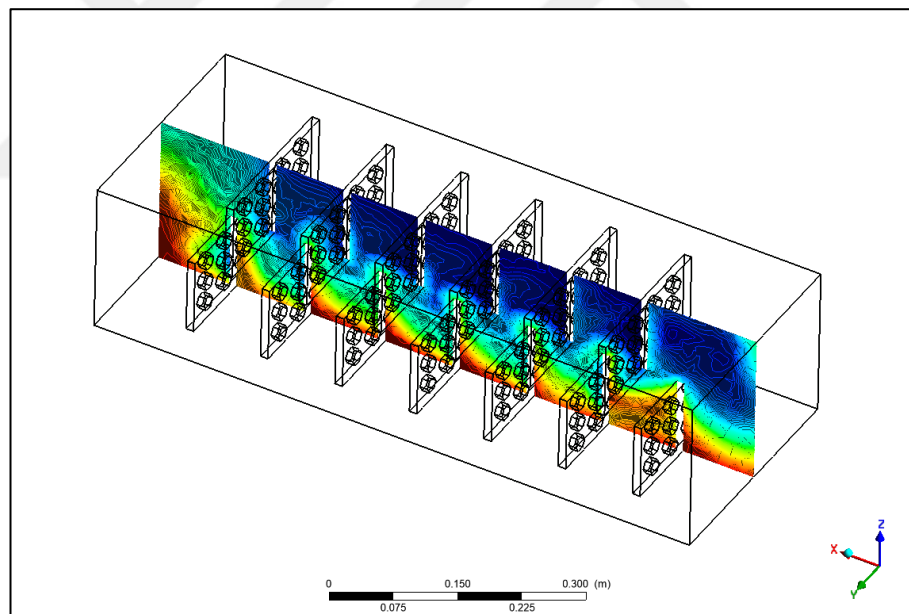


Figure 4.10: Symmetry Plane.

5. RESULTS AND DISCUSSIONS

Volume fraction results are provided in **Table 5.1**.

Table 5.1: Volume Fraction Results.

Configuration	Max. Peak Volume Frac. Diff. on Sym Wall	Last Peak Volume Frac. Diff. on Sym Plane	Difference	Mean
C0-F33	31%	4%	27%	21%
C0-F66	20%	5%	15%	
C1-S-H3-F33	24%	19%	5%	11%
C1-S-H3-F66	22%	6%	16%	
C2-S-H3-F33	17%	2%	15%	13%
C2-S-H3-F66	16%	6%	10%	
C3-S-H3-F33	17%	11%	6%	3%
C3-S-H3-F66	7%	7%	0%	
C3-P-H3-F33	7%	7%	0%	1%
C3-P-H3-F66	5%	3%	2%	
C3-P-H2-F33	7%	2%	5%	3%
C3-P-H2-F66	10%	10%	0%	
C3-P-H1-F33	6%	2%	4%	2%
C3-P-H1-F66	6%	6%	0%	

Wall pressure and shear results are provided in **Table 5.2**.

Table 5.2: Wall Pressure and Shear Results.

Configuration	Max. Pressure on Left Wall (Pa)	Mean (Pa)	Max. Shear Stress on Left Wall (Pa)	Mean (Pa)
C0-F33	738.2	784.3	10.3	16.0
C0-F66	830.4		21.6	
C1-S-H3-F33	860.4	573.8	11.7	11.4
C1-S-H3-F66	287.1		11.0	
C2-S-H3-F33	1035.9	679.7	14.3	12.8
C2-S-H3-F66	323.5		11.3	
C3-S-H3-F33	374.7	252.9	11.4	9.1
C3-S-H3-F66	131.0		6.7	
C3-P-H3-F33	367.2	233.9	10.2	8.9
C3-P-H3-F66	100.6		7.6	
C3-P-H2-F33	438.6	292.2	10.4	9.6
C3-P-H2-F66	145.7		8.7	
C3-P-H1-F33	392.2	309.1	8.9	9.3
C3-P-H1-F66	225.9		9.6	

In detail, following plots and contours are provided in APPENDICES:

- PLOT OF VOLUME FRACTION ON LEFT WALL
- PLOT OF MAXIMUM SHEAR STRESS ON LEFT WALL
- PLOT OF MAXIMUM PRESSURE ON LEFT WALL
- PLOT OF VOLUME FRACTION ON SYMMETRY PLANE
- CONTOUR OF VOLUME FRACTION ON SYMMETRY PLANE
- CONTOUR OF VOLUME FRACTION ON LEFT WALL

5.1. Effect of Fill Level

Effect of fill level are listed in **Table 5.3** and **Table 5.4**.

Table 5.3: Pressure and Shear vs. Fill Level.

Conf.	Max. Pressure on Left Wall (Pa)	Max. Shear Stress on Left Wall (Pa)	Conf.	Max. Pressure on Left Wall (Pa)	Max. Shear Stress on Left Wall (Pa)
C0-F33	738.2	10.3	C0-F66	830.4	21.6
C1-S-H3-F33	860.4	11.7	C1-S-H3-F66	287.1	11.0
C2-S-H3-F33	1035.9	14.3	C2-S-H3-F66	323.5	11.3
C3-S-H3-F33	374.7	11.4	C3-S-H3-F66	131.0	6.7
C3-P-H3-F33	367.2	10.2	C3-P-H3-F66	100.6	7.6
C3-P-H2-F33	438.6	10.4	C3-P-H2-F66	145.7	8.7
C3-P-H1-F33	392.2	8.9	C3-P-H1-F66	225.9	9.6
Mean	601.3	11.0	Mean	292.0	10.9

Table 5.4: Volume Fraction Difference vs. Fill Level.

Configuration	Difference	Configuration	Difference
C0-F33	27%	C0-F66	15%
C1-S-H3-F33	5%	C1-S-H3-F66	16%
C2-S-H3-F33	15%	C2-S-H3-F66	10%
C3-S-H3-F33	6%	C3-S-H3-F66	0%
C3-P-H3-F33	0%	C3-P-H3-F66	2%
C3-P-H2-F33	5%	C3-P-H2-F66	0%
C3-P-H1-F33	4%	C3-P-H1-F66	0%
Mean	9%	Mean	6%

The effect of fill level on sloshing-induced loads and free surface dynamics was investigated by comparing each configuration under 33% and 66% tank fill ratios. As shown in **Table 5.3**, a general trend indicates that higher fill levels result in increased fluid momentum, which often translates into higher pressure peaks and shear stresses acting on the tank walls. However, the introduction of baffles, particularly perforated ones, considerably alters this trend.

The average maximum pressure across all configurations increased from 292.0 Pa (F66) to 601.3 Pa (F33) as the fill level decreased. Similarly, the average wall shear stress was found to be nearly identical (11.0 Pa for F33 and 10.9 Pa for F66), indicating that fill level alone does not significantly influence tangential fluid forces when baffles are present. The C0-F66 (no baffle, high fill level) configuration produced the highest shear stress (21.6 Pa) and a notable pressure peak of 830.4 Pa, clearly emphasizing the amplification of dynamic loads due to the absence of internal damping structures at higher fill levels.

Table 5.4 shows that volume fraction differences, an indicator of free surface fluctuation and sloshing stability, are on average higher for F33 (9%) compared to F66 (6%). This is expected, as lower fill levels provide greater vertical room for free surface oscillations, especially in the absence of flow control elements. Notably, the configuration C3-P-H3-F33 exhibited a 0% volume fraction difference, indicating that full-height perforated baffles effectively suppress wave motion regardless of fill ratio. A similar stabilization effect was observed for C3-P-H3-F66, which yielded only 2% difference.

In literature, it has been shown that increase in fill level reduce slosh frequency. Therefore, free surface motion decreases [15]. This result matches the outcome of this study.

5.2. Effect of Baffle Quantity

Effect of baffle quantity is listed in **Table 5.5** and **Table 5.6**.

Table 5.5: Pressure and Shear vs. Baffle Quantity.

Conf.	Max. Pressure on Left Wall (Pa)	Mean (Pa)	Max. Shear Stress on Left Wall (Pa)	Mean (Pa)
C0-F33	738.2	784.3	10.3	16.0
C0-F66	830.4		21.6	
C1-S-H3-F33	860.4	573.8	11.7	11.4
C1-S-H3-F66	287.1		11.0	
C2-S-H3-F33	1035.9	679.7	14.3	12.8
C2-S-H3-F66	323.5		11.3	
C3-S-H3-F33	374.7	252.9	11.4	9.1
C3-S-H3-F66	131.0		6.7	

Table 5.6: Volume Fraction vs. Baffle Quantity.

Conf.	Max. Peak Volume Frac. Diff. on Sym Wall	Last Peak Volume Frac. Diff. on Sym Plane	Difference	Mean
C0-F33	31%	4%	27%	21%
C0-F66	20%	5%	15%	
C1-S-H3-F33	24%	19%	5%	11%
C1-S-H3-F66	22%	6%	16%	
C2-S-H3-F33	17%	2%	15%	13%
C2-S-H3-F66	16%	6%	10%	
C3-S-H3-F33	17%	11%	6%	3%
C3-S-H3-F66	7%	7%	0%	

The influence of baffle quantity on sloshing-induced pressure, shear stress, and free surface stability was systematically evaluated by comparing configurations with no baffle (C0), two baffle (C1), five baffle (C2), and thirteen baffle (C3) layouts. As seen in **Table 5.5**, a significant reduction in wall loads is evident with increasing baffle quantity.

The mean pressure decreased progressively from 784.3 Pa (no baffle) to 573.8 Pa (C1), 679.7 Pa (C2), and reached a minimum of 252.9 Pa with C3. Similarly, the mean wall shear stress dropped from 16.0 Pa to 9.1 Pa in the same progression. These findings demonstrate that adding multiple baffles improves flow dissipation and reduces the dynamic loads transmitted to tank walls.

Table 5.6 supports this conclusion from the perspective of free surface motion. The average volume fraction difference decreased from 21% (no baffle) to 11% with a single baffle, 13% with two baffles, and 3% in the case of triple baffles. The configuration C3-S-H3-F66 exhibited only 7% peak deviation and 0% final difference, indicating excellent wave suppression at both transient and steady-state conditions.

The superior performance of multiple baffles stems from their ability to fragment and redirect the sloshing flow, which dissipates kinetic energy and interrupts wave propagation across the tank. However, it is also noted that the second baffle configuration (C2) did not outperform the third baffle configuration (C3), implying a non-linear relationship between baffle quantity and effectiveness.

Previous studies proves that number of baffles significantly reduce sloshing induced forces and stabilize fluid inside the container [15]. This thesis also indicate that C3 (13 baffles) performs better than C2(5 baffles) and C1(2 baffles).

5.3. Effect of Perforation

Effect of perforation is listed in **Table 5.7** and **Table 5.8**.

Table 5.7: Pressure and Shear vs. Perforation.

Conf.	Max. Pressure on Left Wall (Pa)	Mean (Pa)	Max. Shear Stress on Left Wall (Pa)	Mean (Pa)
C3-S-H3-F33	374.7	252.9	11.4	9.1
C3-S-H3-F66	131.0		6.7	
C3-P-H3-F33	367.2	233.9	10.2	8.9
C3-P-H3-F66	100.6		7.6	

Table 5.8: Volume Fraction Difference vs Perforation.

Conf.	Max. Peak Volume Frac. Diff. on Sym Wall	Last Peak Volume Frac. Diff. on Sym Plane	Difference	Mean
C3-S-H3-F33	17%	11%	6%	3%
C3-S-H3-F66	7%	7%	0%	
C3-P-H3-F33	7%	7%	0%	1%
C3-P-H3-F66	5%	3%	2%	

The influence of baffle perforation on sloshing mitigation was assessed by comparing solid and perforated triple-baffle configurations under both 33% and 66% fill levels. The results, summarized in **Table 5.7** and **Table 5.8**, indicate that perforated baffles yield slightly better overall performance in reducing sloshing induced wall loads and stabilizing the free surface motion.

From a structural standpoint, the mean wall pressure decreased from 252.9 Pa (solid baffles) to 233.9 Pa (perforated baffles), and the mean shear stress decreased from 9.1 Pa to 8.9 Pa. Although the reduction is modest, it is consistent across both fill levels. The configuration C3-P-H3-F66 yielded the lowest pressure (100.6 Pa) and shear (7.6 Pa) values among all, highlighting the effectiveness of perforated baffles in dynamic load suppression.

More notably, volume fraction differences used as a proxy for wave stabilization improved with perforation. As shown in **Table 5.8**, the mean volume fraction difference decreased from 3% (solid) to 1% (perforated), with C3-P-H3-F66 achieving a near steady state condition (only 2% difference between max and last peak values). This indicates that perforated baffles provide superior damping of free surface fluctuations compared to their solid counterparts.

The enhanced performance of perforated baffles is attributed to their ability to dissipate energy by allowing partial flow-through, which reduces reflection and vortex formation behind the baffle. This leads to a distributed energy dissipation mechanism, unlike solid baffles that tend to reflect sloshing waves more directly, sometimes causing secondary disturbances.

It has been shown that porosity on baffle topography effectively reduces the energy levels and enables fluid movement inside the baffles, resulting in lowering sloshing induced effects [16]. This thesis concludes the same outcome as the perforation favors low sloshing.

5.4. Effect of Baffle Height

Effect of baffle height is listed in **Table 5.9** and **Table 5.10**.

Table 5.9: Pressure and Shear vs. Baffle Height.

Conf.	Max. Pressure on Left Wall (Pa)	Mean (Pa)	Max. Shear Stress on Left Wall (Pa)	Mean (Pa)
C3-P-H3-F33	367.2	233.9	10.2	8.9
C3-P-H3-F66	100.6		7.6	
C3-P-H2-F33	438.6	292.2	10.4	9.6
C3-P-H2-F66	145.7		8.7	
C3-P-H1-F33	392.2	309.1	8.9	9.3
C3-P-H1-F66	225.9		9.6	

Table 5.10: Volume Fraction Difference vs. Baffle Height.

Conf.	Max. Peak Volume Frac. Diff. on Sym Wall	Last Peak Volume Frac. Diff. on Sym Plane	Difference	Mean
C3-P-H3-F33	7%	7%	0%	1%
C3-P-H3-F66	5%	3%	2%	
C3-P-H2-F33	7%	2%	5%	3%
C3-P-H2-F66	10%	10%	0%	
C3-P-H1-F33	6%	2%	4%	2%
C3-P-H1-F66	6%	6%	0%	

The effect of baffle height on sloshing-induced loads and free surface stability was investigated by evaluating full-height (H3), two-thirds height (H2), and one-third height (H1) perforated baffle configurations under both 33% and 66% fill levels. Results presented in **Table 5.9** and **Table 5.10** reveal that baffle height plays a critical role in energy dissipation and flow control within the tank.

In terms of structural loading, the lowest mean wall pressure was observed in the full-height baffle configuration (C3-P-H3, 233.9 Pa), followed by H1 (309.1 Pa) and H2 (292.2 Pa). Shear stress values followed a similar pattern, with H3 yielding the lowest mean shear at 8.9 Pa, compared to 9.6 Pa (H1) and 9.6 Pa (H2). These results indicate that full-height baffles offer superior suppression of wall pressure and tangential fluid forces by presenting a more continuous barrier to fluid motion.

Volume fraction fluctuations, a proxy for sloshing severity, were also minimized with full-height baffles.

Table 5.10 shows that the mean volume fraction difference was 1% for H3, compared to 3% for H2 and 2% for H1. In particular, C3-P-H3-F33 showed complete

stabilization (0% difference between peak and final values), confirming that a taller baffle helps maintain sloshing control over time.

The superior performance of full-height baffles can be explained by their ability to intercept the entire vertical range of fluid motion, thus breaking up wave energy more effectively and preventing re-circulation above or below the baffle body. In contrast, shorter baffles (H1 and H2) allow portions of the fluid to bypass the structure, leading to residual wave activity and increased wall loads. However, it is noteworthy that the difference between H2 and H1 is marginal, suggesting diminishing returns below a critical height threshold.

In literature it is shown that baffle height plays crucial role in reducing rolling motion and act as blockage in front of the fluid [9]. This thesis also proves that baffle height reduces sloshing induced effects.

6. CONCLUSION AND RECOMMENDATIONS

6.1. Summary of Findings

Pressure, shear and volume fraction difference plots are provided in **Figure 6.1**, **Figure 6.2** and **Figure 6.3**.

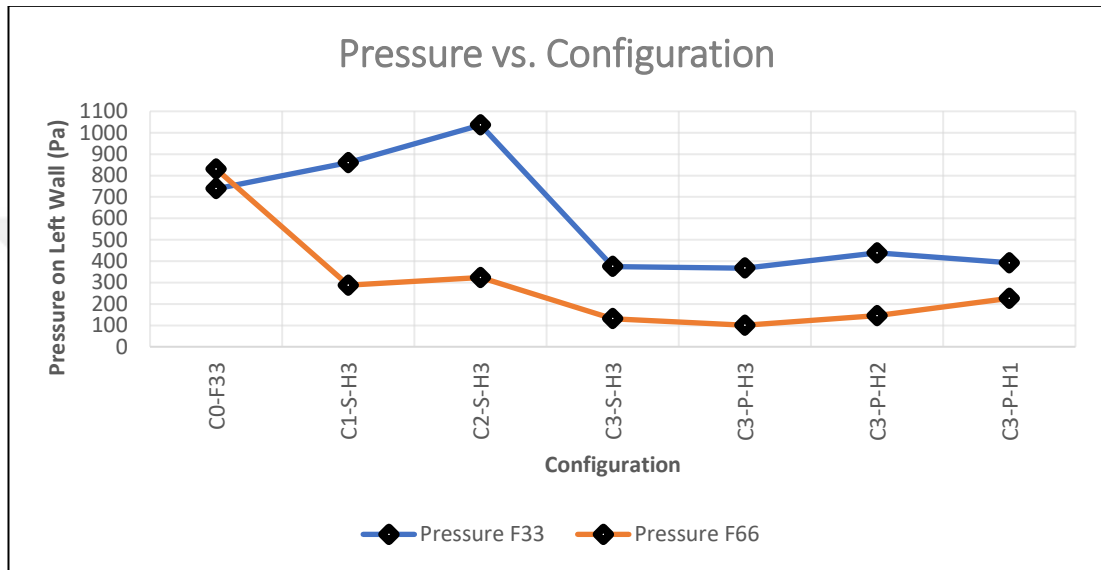


Figure 6.1: Pressure vs. Configuration.

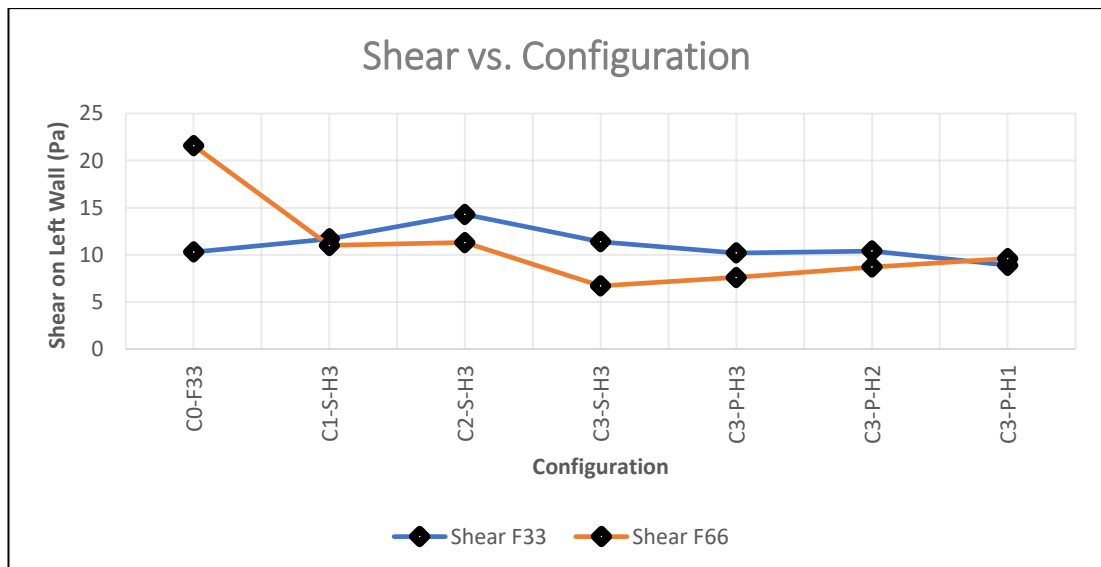


Figure 6.2: Shear vs. Configuration.

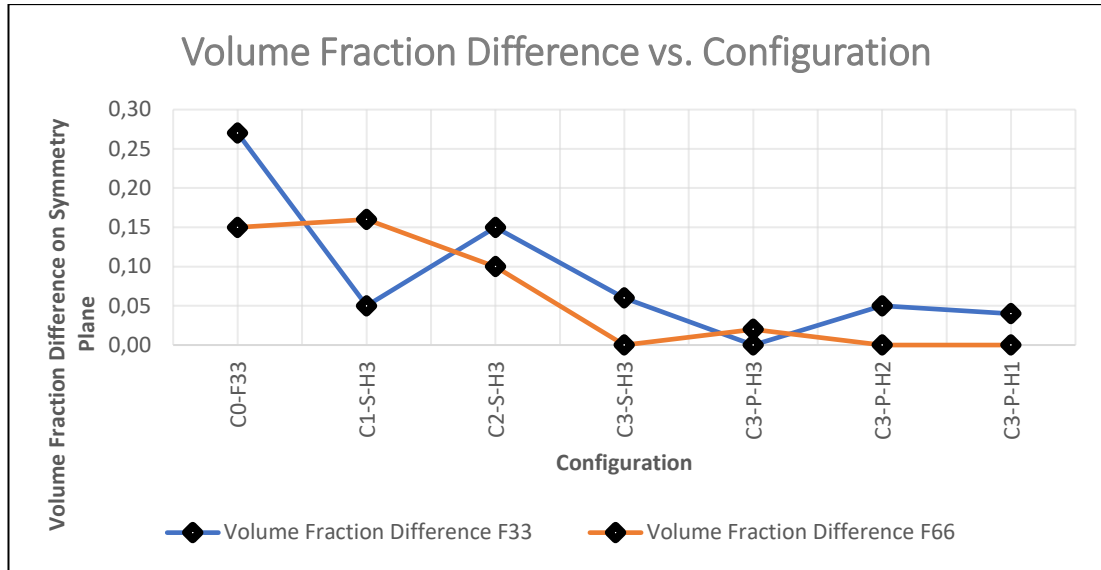


Figure 6.3: VOF Difference vs. Configuration.

The computational study conducted in this thesis provided an assessment of the effects of baffle design parameters, namely quantity, placement, height, and perforation on sloshing induced pressure, shear stress, and free surface motion within an aircraft type rectangular fuel tank under varying fill levels. The findings, summarized through graphical representations in **Figure 6.1**, **Figure 6.2** and **Figure 6.3**, highlight key trends and outcomes regarding dynamic load reduction and fluid stabilization.

As illustrated in **Figure 6.1**, baffle implementation significantly reduced wall pressure values for both 33% and 66% fill levels. The maximum pressure in the no baffle configuration (C0-F33) reached 738.2 Pa, whereas configurations with triple perforated baffles (C3-P-H3-F66) yielded values as low as 100.6 Pa, representing a substantial decrease of nearly 86%. This confirms earlier numerical studies which emphasize the role of internal structures in disrupting wave propagation and redistributing energy.

Figure 6.2 illustrates that wall shear stress followed a similar pattern. The no baffle F66 case recorded the maximum value of 21.6 Pa, while shear stress decreased to 7.6 Pa in the C3-P-H3-F66 configuration. These reductions support findings in prior research that associate sloshing suppression with energy dissipation across internal baffle surfaces.

The effectiveness of the baffle design was most apparent in the volume fraction data shown in **Figure 6.3**. Volume fraction differences dropped from 0.27 in C0-F33 to

0.02 in C3-P-H3-F66, confirming that baffles not only reduce pressure but also stabilize the free surface oscillations.

Collectively, these findings validate that baffle optimization, particularly full-height perforated designs, can drastically suppress sloshing intensity and mechanical loads. The C3-P-H3-F66 configuration provided the best overall performance, effectively balancing structural mitigation and flow control across both low and high fill scenarios.

6.2. Contributions to Engineering Practice

The findings of this thesis offer several valuable contributions to the engineering practice of fuel tank design, particularly within the aerospace and defense sectors where safety, weight optimization, and structural integrity are critical. By systematically evaluating baffle design variations through CFD simulations, the study provides a quantitative basis for design decisions involving sloshing suppression strategies under varying operational fill levels.

One of the most significant engineering contributions is the confirmation that perforated full height baffles (C3-P-H3) not only reduce dynamic wall loads but also maintain flow stability, enabling a more predictable tank behavior during acceleration phases of flight. These insights directly support tank layout optimizations that reduce the necessity for overdesign, potentially allowing for lighter and more structurally efficient tank walls. This is particularly beneficial in aviation, where weight reduction translates into fuel economy and extended operational range.

Furthermore, the study reinforces that sloshing behavior can be actively controlled by interior geometry modifications, rather than relying solely on passive damping or structural reinforcements. This principle can be extended to the design of modular baffle systems, particularly in unmanned aerial vehicles (UAVs) and rotary wing platforms, where rapid design iterations and confined volumes make integrated mitigation strategies more desirable.

From a simulation perspective, the adoption of the VOF method within ANSYS Fluent, combined with pressure and volume fraction-based performance metrics, demonstrates a repeatable and accurate methodology for evaluating sloshing scenarios

prior to physical prototyping. This supports cost and time effective digital validation workflows, aligning with modern engineering and virtual testing environments.

Ultimately, this study bridges the gap between theoretical understanding of sloshing dynamics and practical application in fuel tank design by delivering simulation backed design insights that can be readily adapted to real world engineering programs.

6.3. Design Recommendations

Based on the comprehensive CFD analyses and comparative performance evaluation of baffle configurations, several design recommendations can be proposed for future fuel tank development, particularly in aerospace applications where dynamic stability, structural integrity, and volume efficiency must be balanced.

- Utilize Full-Height Perforated Baffles for Maximum Effectiveness

Configurations with full-height perforated baffles (C3-P-H3) consistently yielded the lowest pressure and shear loads, while also minimizing volume fraction differences. These structures intercept the entire vertical sloshing profile and allow partial flow-through, which enhances energy dissipation without causing strong backflow or vortex formation.

- Adopt More Baffle Layout in High Sloshing Environments:

Designs employing thirteen baffles (C3) outperformed two (C1) or five (C2) baffle configurations in all performance metrics. Adding more baffle increase stability, however, the excessive usage may add weight and decrease space in fuel tanks.

- Avoid Short Baffles in Critical Zones:

While one-third height baffles (H1) offered some damping benefits, they were less effective in reducing both pressure and free surface movement, especially under full tank conditions. Thus, partial height baffles should be used only in low risk or volume constrained regions of the tank

- Favor Perforation Over Solid Baffles When Structural Loads Allow:

Although solid baffles provide physical separation, perforated baffles offer superior stabilization with less added weight, making them preferable in weight sensitive applications such as UAVs and helicopters.

- **Align Baffle Configuration with Operational Fill Level Ranges:**

Since sloshing behavior varies with fill ratio, configurations should be optimized for the most critical operating condition. For example, full baffles are especially crucial when tanks are only partially filled, where slosh amplitudes are greatest.

- **Validate Configurations Using CFD Prior to Physical Prototyping:**

The demonstrated use of the VOF method in transient simulations allows for preliminary virtual screening of baffle designs, reducing the need for costly physical iterations and accelerating development timelines.

In conclusion, a C3-P-H3 configuration offers the most robust solution across operational scenarios, and it is recommended as a baseline architecture for aircraft fuel tank design in this dimensions, whenever feasible within structural and volumetric constraints. Nevertheless, each baffle increase weight and reduce space in tanks. During design milestones, this aspects should be also taken into consideration.

6.4. Suggestions for Future Work

While the present study provides a thorough assessment of baffle design effects on sloshing dynamics using CFD-based simulations, several aspects remain open for further investigation and development. These areas are proposed as directions for future research and practical application enhancements:

- **Inclusion of Flexible Tank Wall Behavior:**

The current simulations assume rigid wall boundaries. However, fuel tanks especially in rotary-wing aircraft and UAVs often include flexible components or composite structures that interact dynamically with the fluid. Incorporating fluid structure interaction (FSI) models would offer a more realistic representation of sloshing-induced stresses and possible fatigue issues over time.

- **Experimental Validation of Numerical Results:**

Although the VOF method provided reliable insights into sloshing suppression trends, physical experiments in scaled tank setups under controlled excitation could strengthen confidence in the CFD results. High-speed imaging and pressure sensors may be used to validate key parameters such as wall impact forces and free surface evolution.

- Parametric Optimization Through Design of Experiments (DoE):

This study focused on discrete configurations. A broader exploration using DoE or metaheuristic optimization algorithms (e.g., genetic algorithms) could identify optimal baffle geometries and perforation patterns across a wider design space with improved efficiency and precision.

- Three Dimensional and Asymmetric Tank Geometries:

The rectangular tank geometry used here simplifies flow behavior. Future work should consider more realistic, three-dimensional tank shapes with internal obstructions or asymmetries typical of operational aircraft to better generalize design guidelines.

- Flight Scenario Based Sloshing Conditions:

Instead of harmonic excitation, future analyses may incorporate realistic acceleration profiles, such as take-off, banking maneuvers, or turbulent conditions extracted from flight data recorders, to assess the robustness of baffle configurations under mission specific loading.

- Assessment of Thermal and Fuel System Integration:

Since baffles may interfere with thermal management or fuel extraction systems, future designs should include multi-disciplinary evaluations considering thermal gradients, sensor placement, and suction port access within the baffle-integrated tank.

By addressing these areas, future research can further bridge the gap between numerical sloshing studies and fully integrated aerospace fuel system design, enhancing both safety and performance.

REFERENCES

- [1] Ibrahim, R. A., (2005), “Liquid Sloshing Dynamics: Theory and Applications”, Cambridge University Press.
- [2] Chen, M., Wang, Y., Liu, D., Zhang, L., (2021), “Investigation on the effects of vertical baffles on liquid sloshing based on a particle method”, Proc. J. Phys.: Conf. Ser., 2083, 022097.
- [3] Guan, Y., Yang, C., Chen, P., Zhou, L., (2020), “Numerical investigation on the effect of baffles on liquid sloshing in 3D rectangular tanks based on nonlinear boundary element method”, International Journal of Naval Architecture and Ocean Engineering, 12, 399–413.
- [4] Arora, S., Vasudevan, S., (2017), “Analysis of sloshing-induced loads on the fuel tank structure”, Master’s Thesis, Dept. of Mechanics and Maritime Sciences, Chalmers University of Technology, Gothenburg, Sweden.
- [5] Singal, V., Bajaj, J., Awalgaonkar, N., Tibdewal, S., (2014), “CFD analysis of a kerosene fuel tank to reduce liquid sloshing”, Proc. 24th DAAAM Int. Symp. on Intelligent Manufacturing and Automation, Procedia Engineering, 69, 1365–1371.
- [6] White, F. M., (2016), “Fluid Mechanics”, 8th ed., McGraw-Hill.
- [7] Hirt, M., Nichols, B., (1981), “Volume of Fluid (VOF) method for the dynamics of free boundaries”, Journal of Computational Physics, 39 (1), 201–225.
- [8] ANSYS Inc., (2023), ANSYS Fluent Theory Guide, Release 2023 R1, Canonsburg, PA, USA.
- [9] Akyıldız, R., (2012), “A numerical study of the effects of the vertical baffle on liquid sloshing in two-dimensional rectangular tanks”, Ocean Systems Engineering, 2 (3), 175–188.
- [10] Deshmukh, Y., Savat, S., Prakash, A., Patil, K., Patil, R., Perumal, E., (2022), “Optimization and reduction of sloshing in the fuel tank”, International Journal for Research in Applied Science and Engineering Technology (IJRASET), 10 (4).
- [11] Li, L., Shen, B., Zhu, Z., Ren, Q., Zhang, J., (2024), “Investigation of liquid sloshing characteristics in tanks using an OpenFOAM solver with arbitrary excitation”, AIP Advances, 14 (11), 115306.
- [12] Bayramoğlu, T., Bayramoğlu, K., Korkmaz, F. C., (2021), “The effect of baffle location on sloshing in the rectangular tank”, Proc. Int. Conf. May 2021, <https://www.researchgate.net/publication/351885391>.
- [13] Frosina, E., Senatore, A., Andreozzi, A., Fortunato, F., Giliberti, P., (2018), “Experimental and numerical analyses of the sloshing in a fuel tank”, Energies, 11 (5), 1–20.

- [14] Tamer, A., Muscarello, V., Quaranta, G., Masarati, P., (2020), “Cabin layout optimization for vibration hazard reduction in helicopter emergency medical service”, *Aerospace*, 7 (5), 1–18.
- [15] Rajagounder, R., Mohanasundaram, G. V., Kalakkath, P., (2016), “A study of liquid sloshing in an automotive fuel tank under uniform acceleration”, *Engineering Journal*, 20 (1), 71–85.
- [16] Yıldız, Y., Kibar, A., Yiğit, K. S., (2024), “Numerical investigation of the fuel tank sloshing condition of a commercial vehicle”, *International Journal of Automotive Science and Technology*, 8 (3), 303–311.



BIOGRAPHY

Mert AKKUŞ completed his secondary education at Ankara Atatürk Anatolian High School, graduating in 2014. He subsequently earned his Bachelor of Science degree in Mechanical Engineering from Middle East Technical University (METU).

In 2022, he commenced his Master of Science studies in the field of Applied Propulsion Systems Engineering within the Aerospace Technologies program at the Energy Technologies Institute. His graduate research has focused primarily on fluid dynamics, computational fluid dynamics, and numerical simulations, particularly within the context of aerospace applications.

Professionally, he is employed in the Turkish defense industry at Turkish Aerospace Industries, where he is actively involved in the design, analysis, testing, and certification of air vehicle subsystems. His role encompasses both theoretical and practical aspects of system-level development in aerospace engineering.

His main research interest lies in the modeling and simulation of sloshing phenomena in aircraft fuel tanks using the Volume of Fluid method and advanced CFD techniques within ANSYS Fluent. Throughout his graduate studies, he has gained significant expertise in fluid mechanics, CFD analysis, air vehicle subsystem performance, and aerospace system integration.

He intends to further pursue his career in the areas of aerospace subsystem design, simulation-driven development, experimental validation, and certification processes, contributing to the advancement of national aviation technologies.

PUBLICATIONS AND PRESENTATIONS FROM THE THESIS

Akkuş, M., Ünverdi, S. Ö., (2025), “Modeling and simulation of the fuel sloshing phenomenon in fuel tanks and design optimization,” Proceedings of the 11th International Azerbaijan Congress on Life, Engineering, Mathematical, and Applied Sciences, Baku, Azerbaijan, pp. 40, doi: 10.30546/19023.978-9952-8573-5-12025.8988.



APPENDICES

A. PLOT OF VOLUME FRACTION ON LEFT WALL

Plots of volume fraction are provided in **Figure A.1**, **Figure A.2**, **Figure A.3**, **Figure A.4**, **Figure A.5**, **Figure A.6**, **Figure A.7**, **Figure A.8**, **Figure A.9**, **Figure A.10**, **Figure A.11**, **Figure A.12**, **Figure A.13** and **Figure A.14**.

A-1) C0-F33

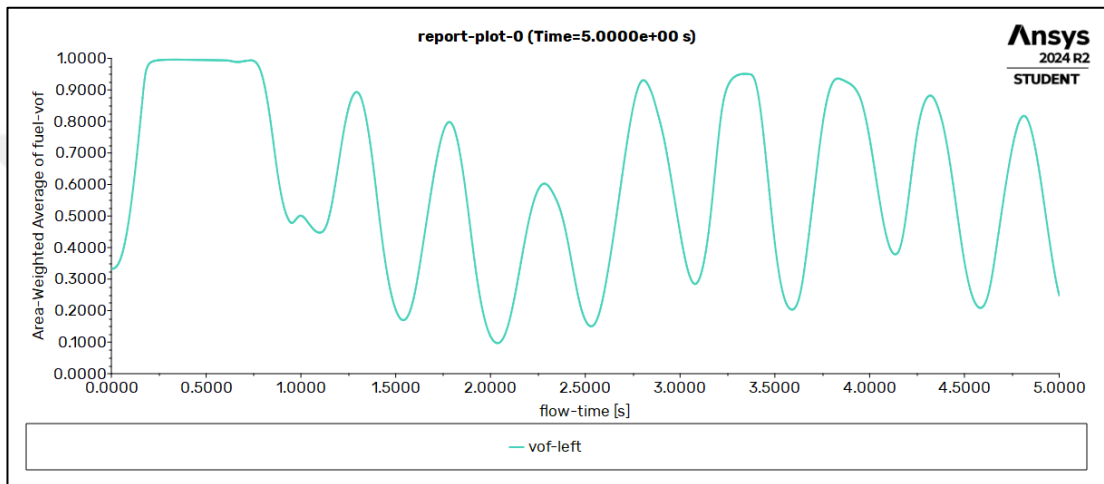


Figure A.1: C0-F33 Plot of Volume Fraction on Left Wall.

A-2) C0-F66

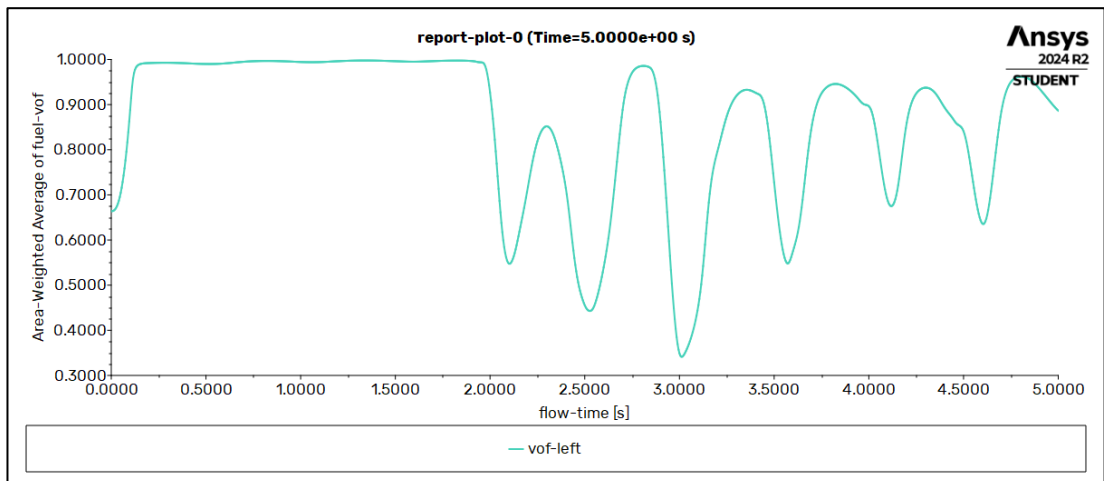


Figure A.2: C0-F66 Plot of Volume Fraction on Left Wall.

A-3) C1-S-H3-F33

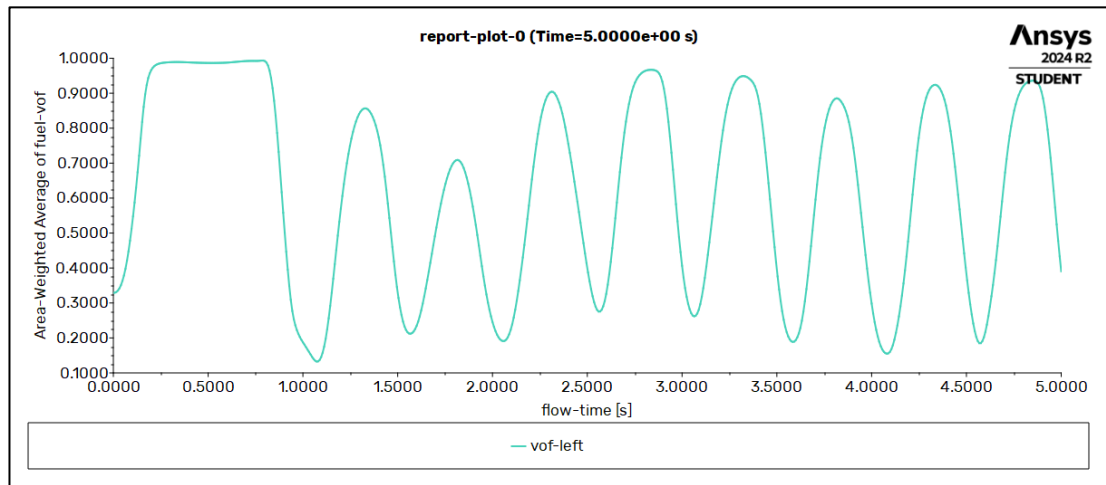


Figure A.3: C1-S-H3-F33 Plot of Volume Fraction on Left Wall.

A-4) C1-S-H3-F66

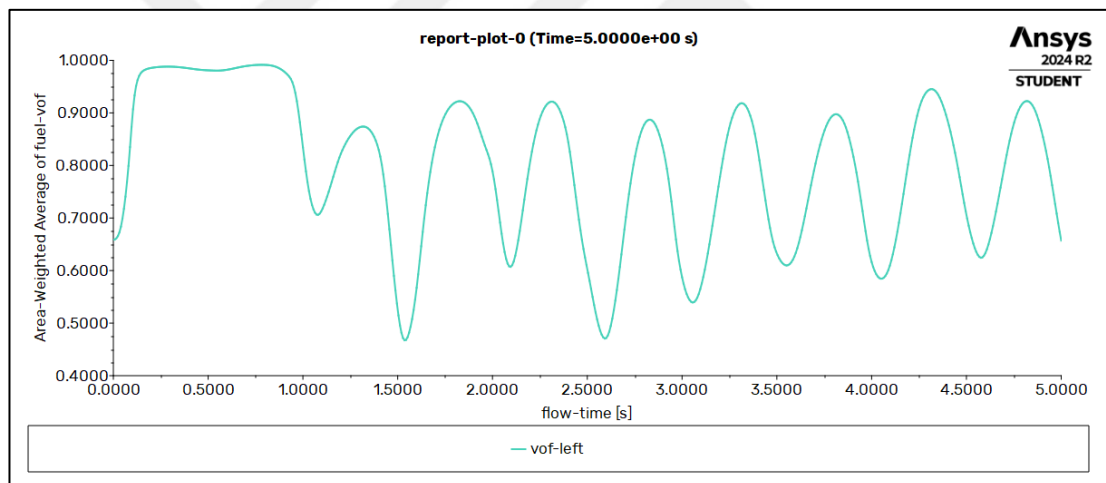


Figure A.4: C1-S-H3-F66 Plot of Volume Fraction on Left Wall.

A-5) C2-S-H3-F33

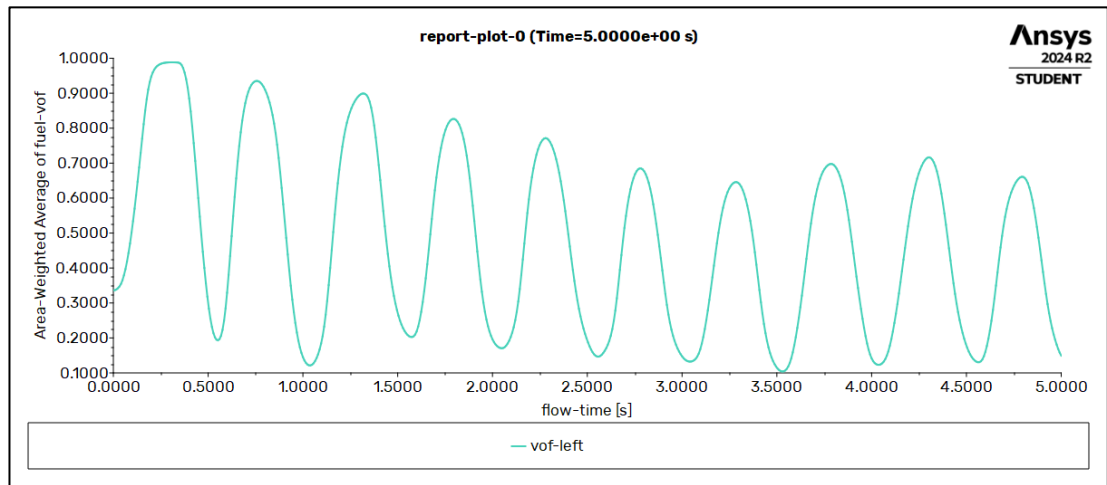


Figure A.5: C2-S-H3-F33 Plot of Volume Fraction on Left Wall.

A-6) C2-S-H3-F66

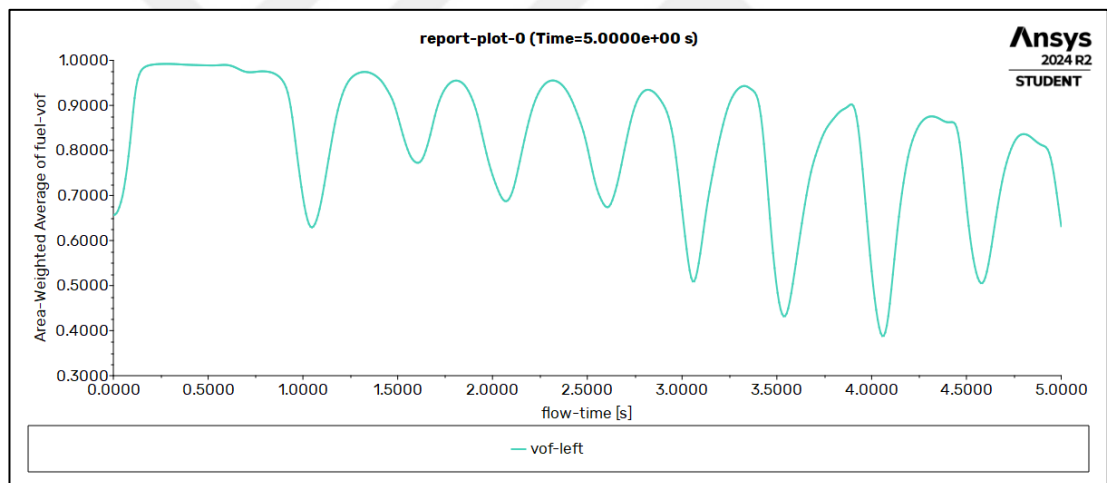


Figure A.6: C2-S-H3-F66 Plot of Volume Fraction on Left Wall.

A-7) C3-S-H3-F33

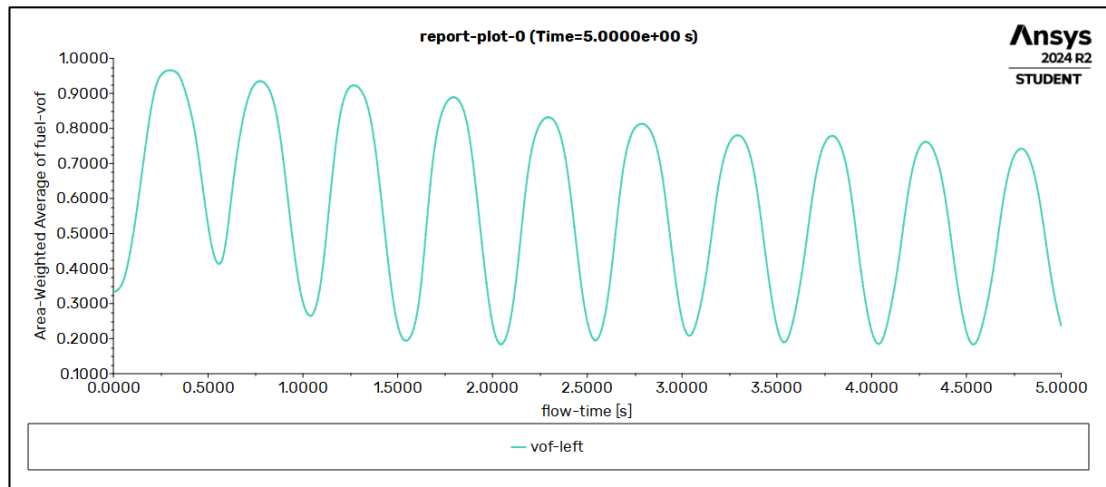


Figure A.7: C3-S-H3-F33 Plot of Volume Fraction on Left Wall.

A-8) C3-S-H3-F66

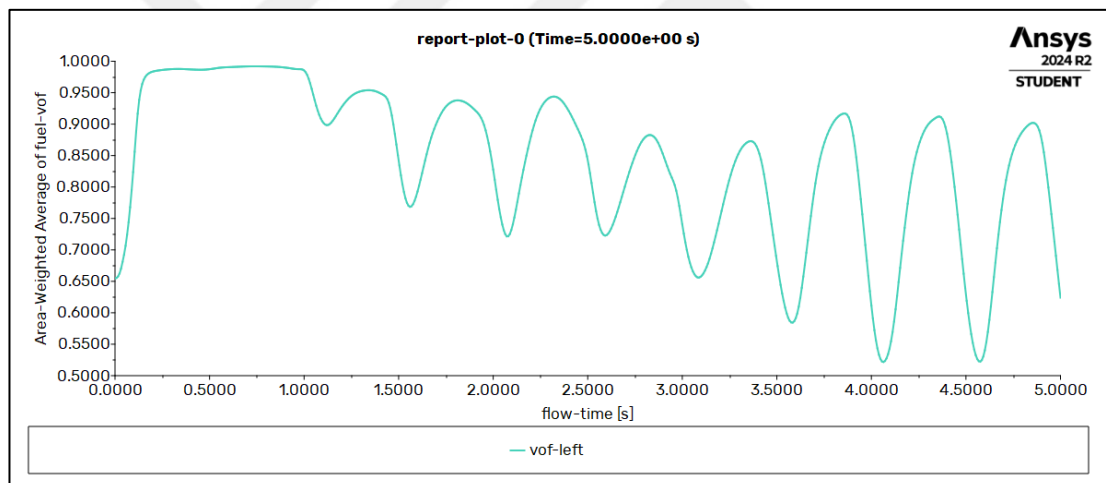


Figure A.8: C3-S-H3-F66 Plot of Volume Fraction on Left Wall.

A-9) C3-P-H3-F33

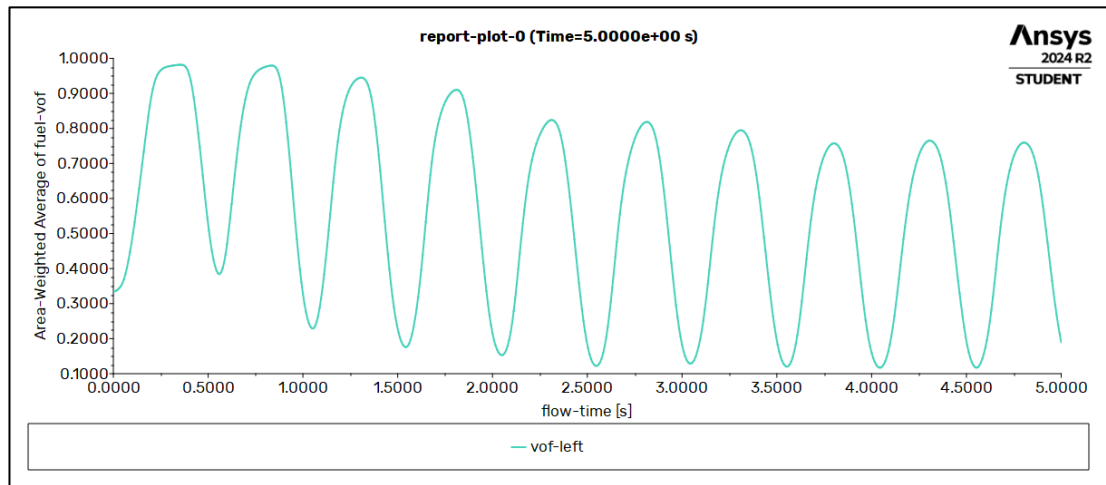


Figure A.9: C3-P-H3-F33 Plot of Volume Fraction on Left Wall.

A-10) C3-P-H3-F66

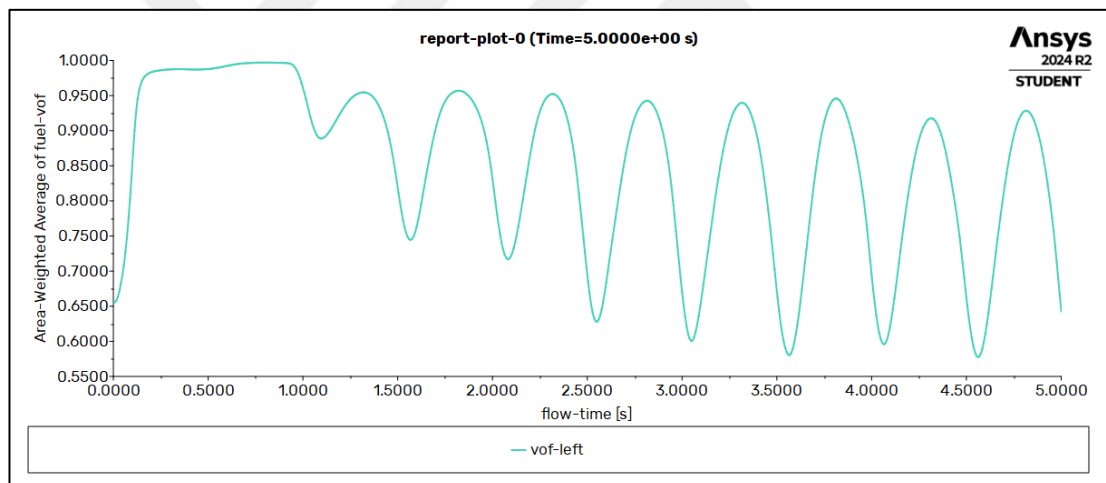


Figure A.10: C3-P-H3-F66 Plot of Volume Fraction on Left Wall.

A-11) C3-P-H2-F33

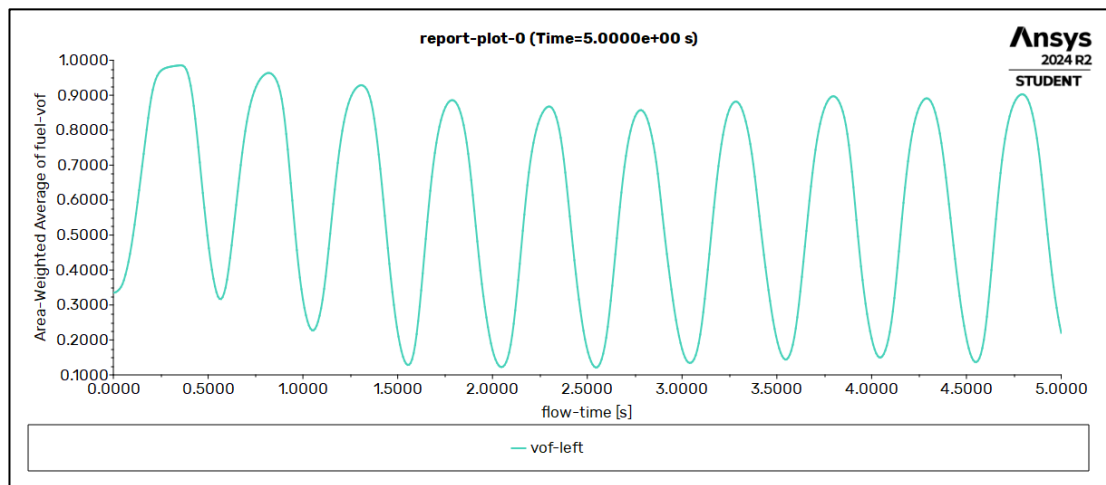


Figure A.11: C3-P-H2-F33 Plot of Volume Fraction on Left Wall.

A-12) C3-P-H2-F66

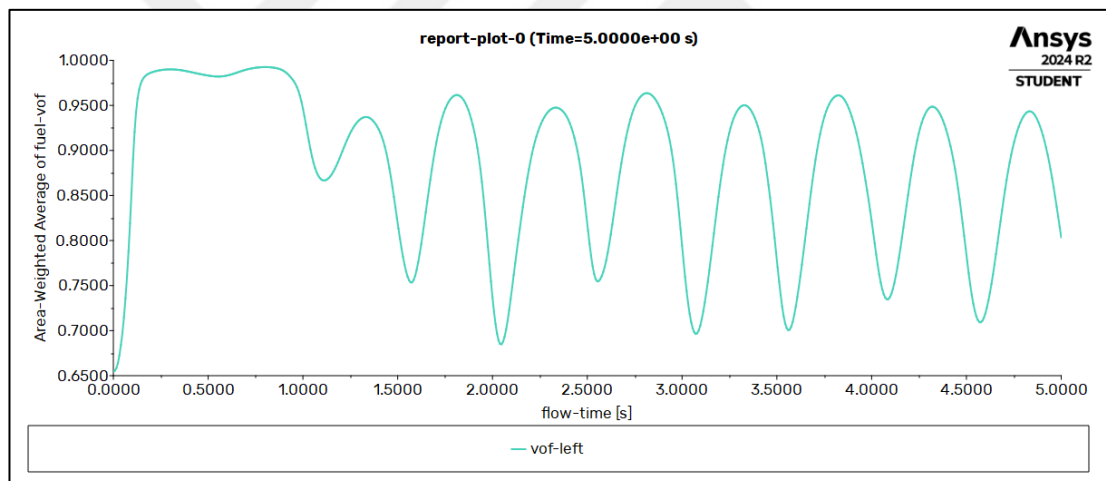


Figure A.12: C3-P-H2-F66 Plot of Volume Fraction on Left Wall.

A-13) C3-P-H1-F33

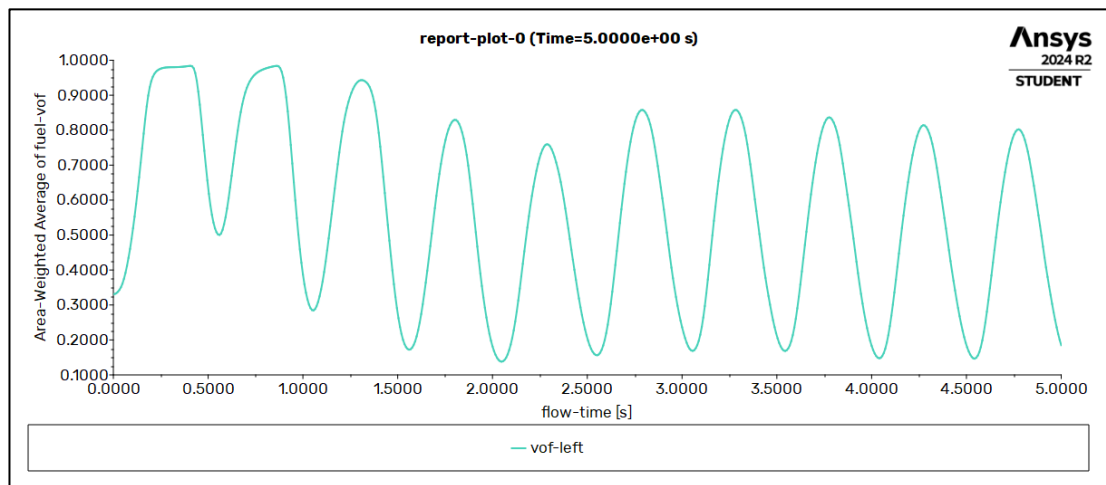


Figure A.13: C3-P-H1-F33 Plot of Volume Fraction on Left Wall.

A-14) C3-P-H1-F66

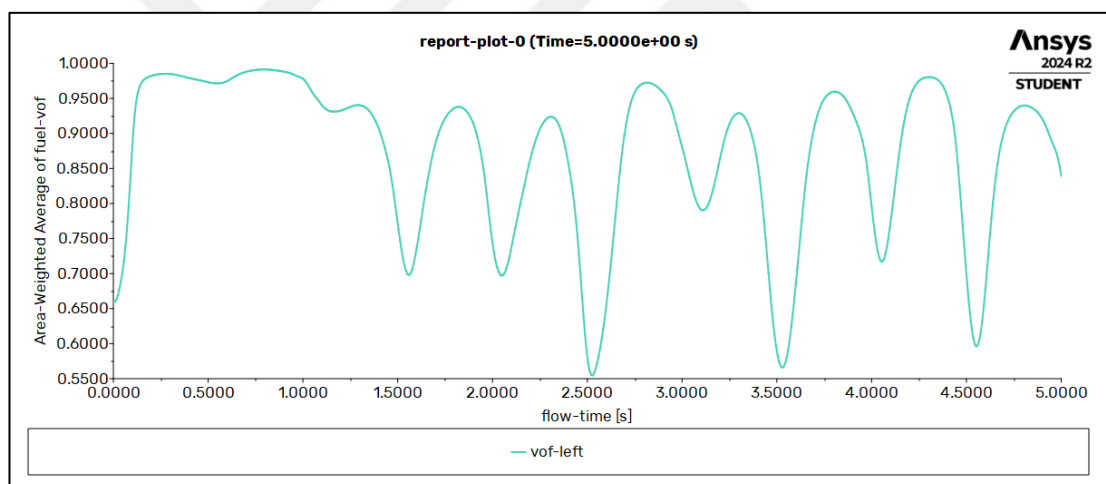


Figure A.14: C3-P-H1-F66 Plot of Volume Fraction on Left Wall.

B. PLOT OF MAXIMUM SHEAR STRESS ON LEFT WALL

Plots of maximum shear stress on left wall are provided in **Figure B.1**, **Figure B.2**, **Figure B.3**, **Figure B.4**, **Figure B.5**, **Figure B.6**, **Figure B.7**, **Figure B.8**, **Figure B.9**, **Figure B.10**, **Figure B.11**, **Figure B.12**, **Figure B.13** and **Figure B.14**.

B-1) C0-F33

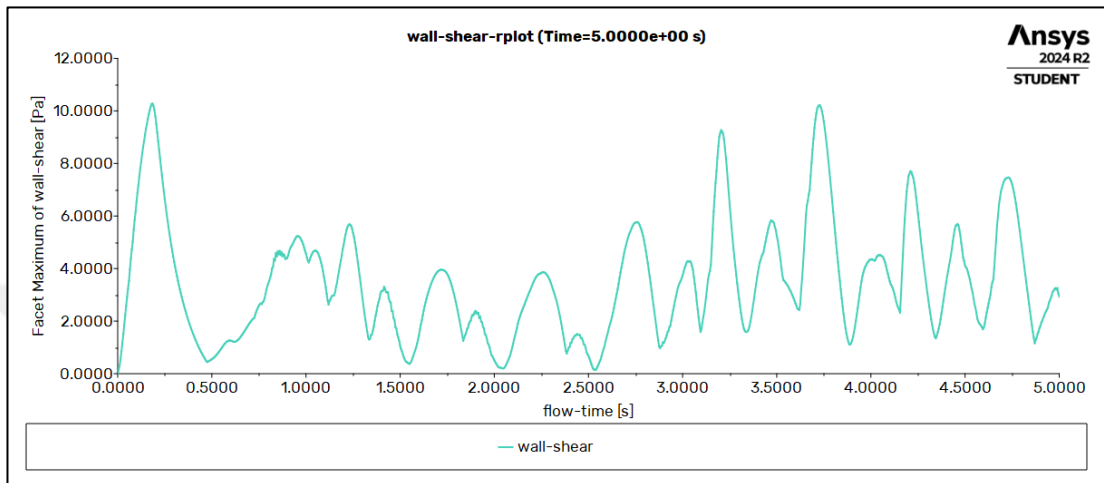


Figure B.1: C0-F33 Plot of Maximum Shear Stress on Left Wall.

B-2) C0-F66

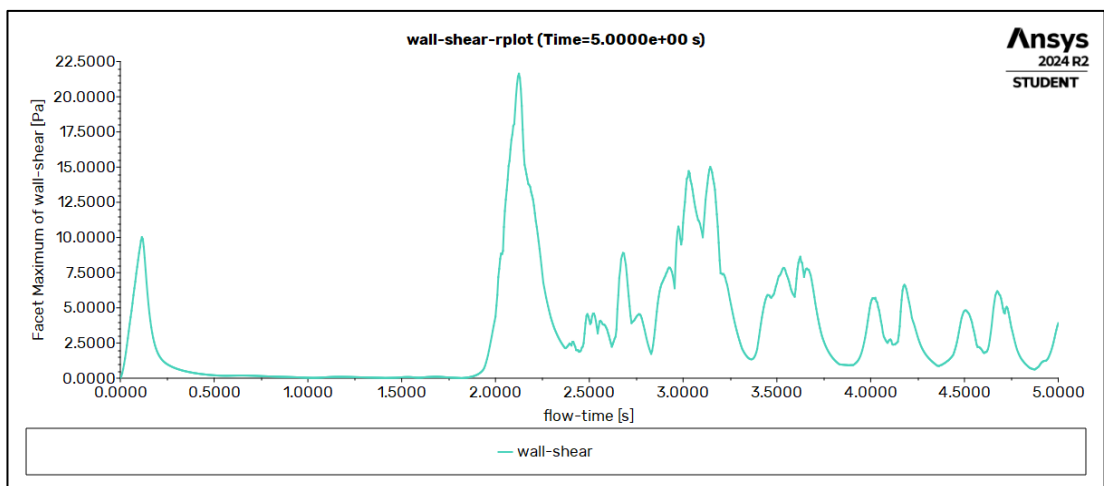


Figure B.2: C0-F66 Plot of Maximum Shear Stress on Left Wall.

B-3) C1-S-H3-F33

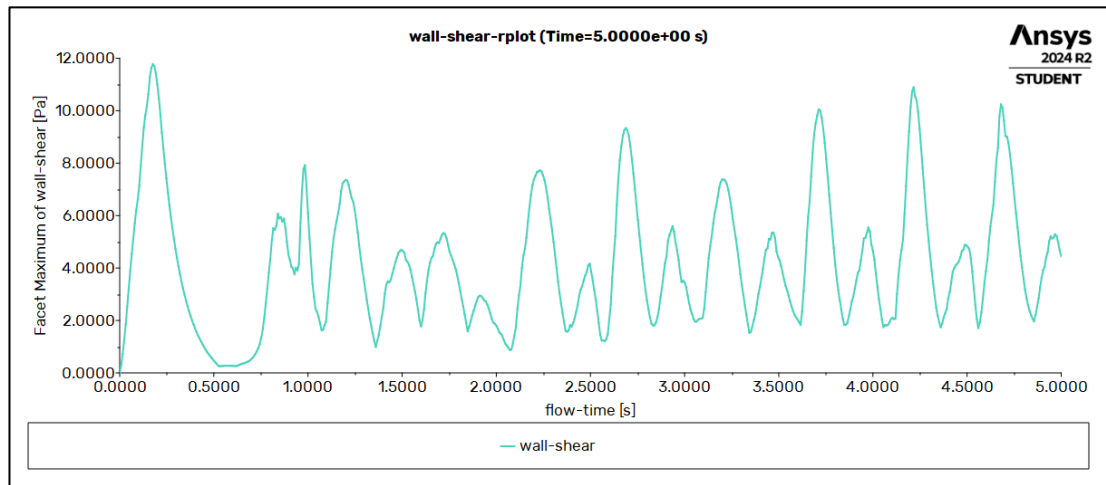


Figure B.3: C1-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.

B-4) C1-S-H3-F66

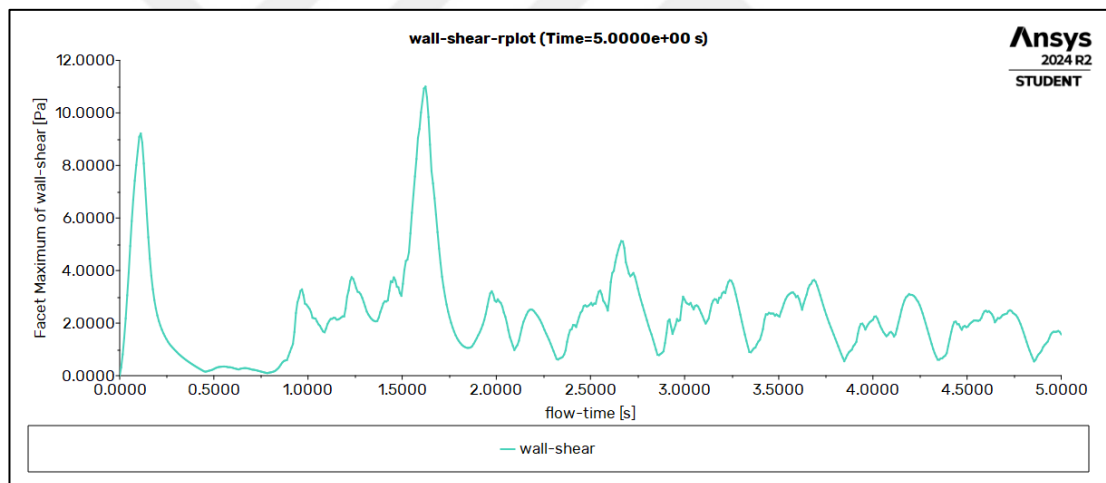


Figure B.4: C1-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.

B-5) C2-S-H3-F33

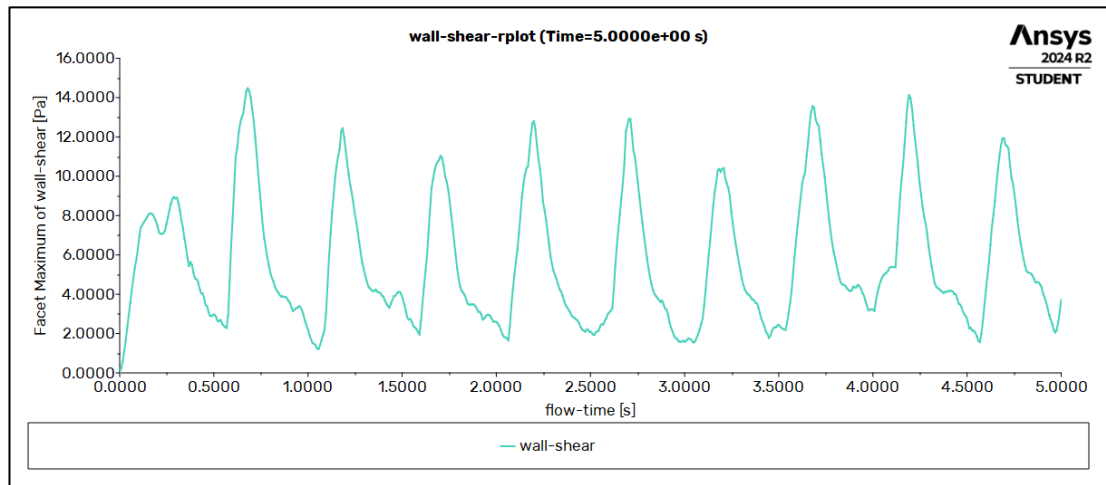


Figure B.5: C2-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.

B-6) C2-S-H3-F66

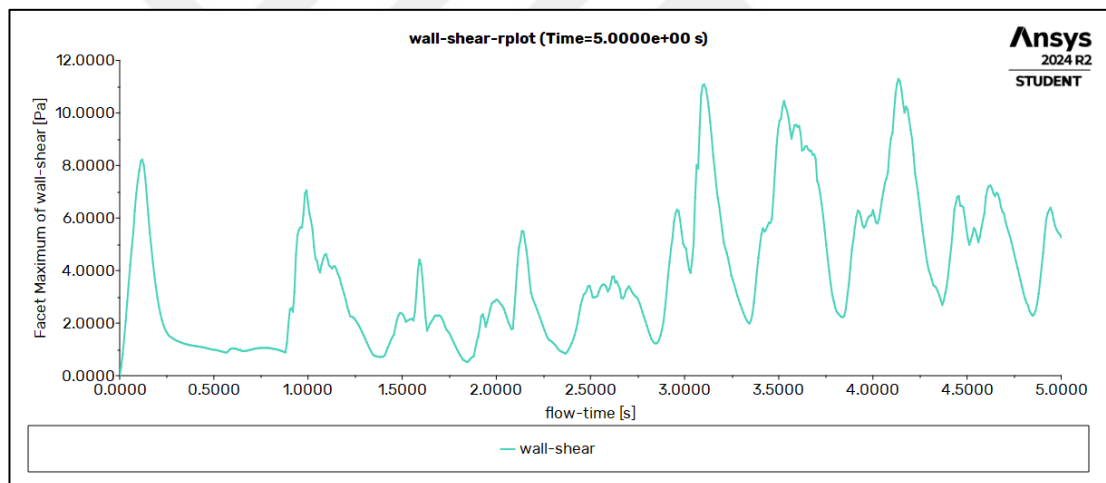


Figure B.6: C2-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.

B-7) C3-S-H3-F33

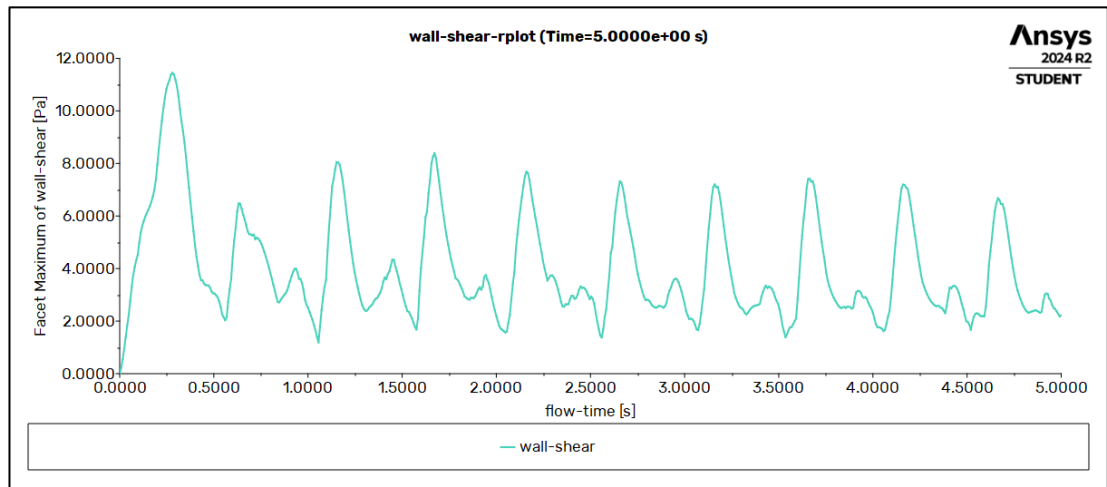


Figure B.7: C3-S-H3-F33 Plot of Maximum Shear Stress on Left Wall.

B-8) C3-S-H3-F66

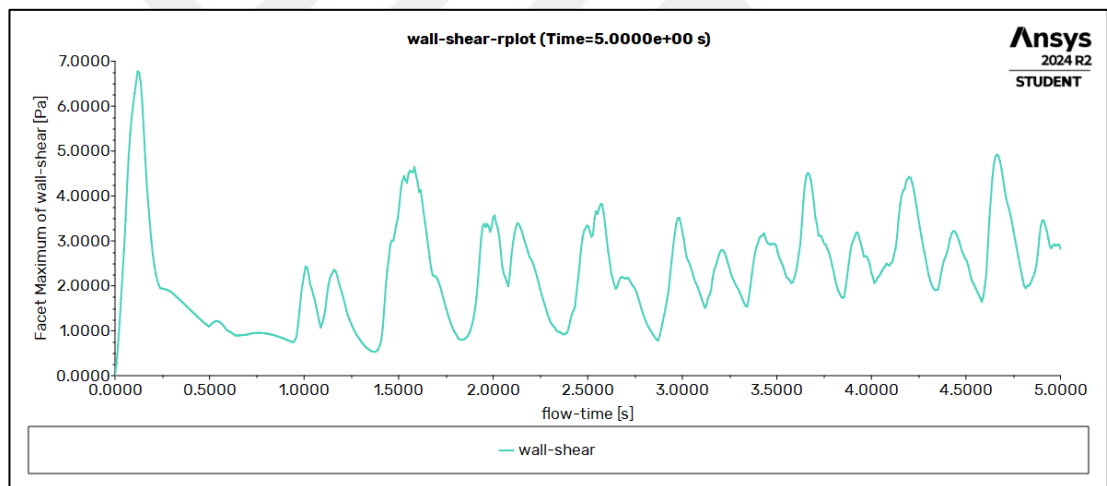


Figure B.8: C3-S-H3-F66 Plot of Maximum Shear Stress on Left Wall.

B-9) C3-P-H3-F33

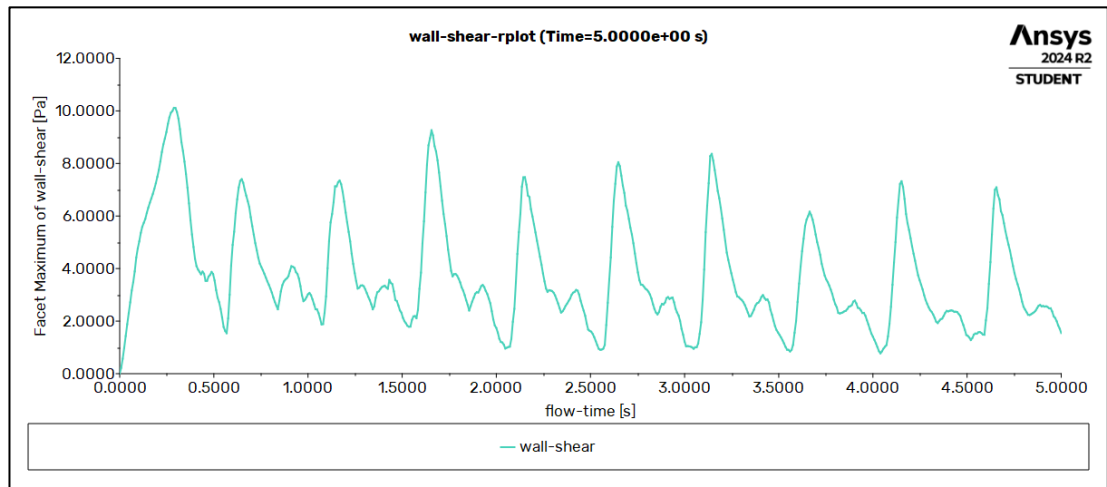


Figure B.9: C3-P-H3-F33 Plot of Maximum Shear Stress on Left Wall.

B-10) C3-P-H3-F66

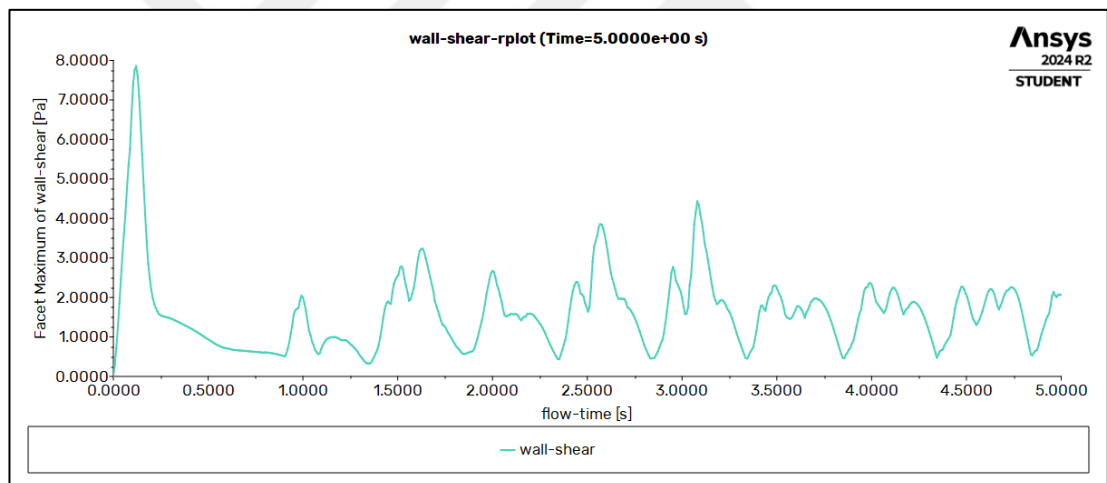


Figure B.10: C3-P-H3-F66 Plot of Maximum Shear Stress on Left Wall.

B-11) C3-P-H2-F33

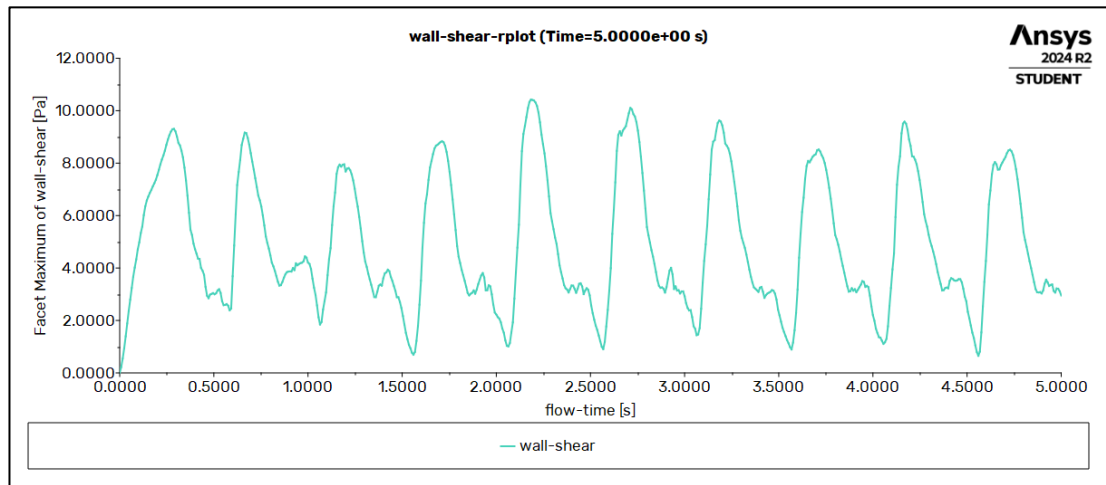


Figure B.11: C3-P-H2-F33 Plot of Maximum Shear Stress on Left Wall.

B-12) C3-P-H2-F66

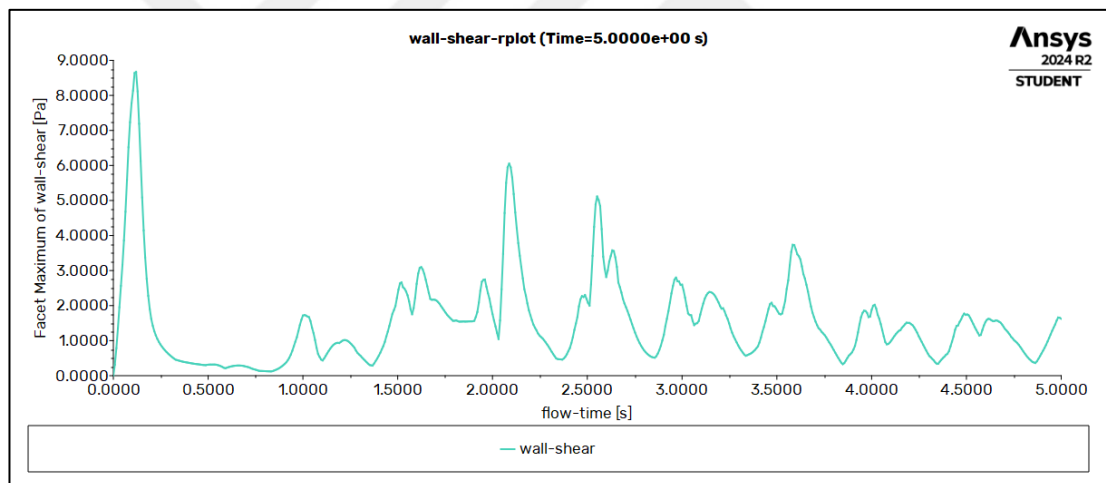


Figure B.12: C3-P-H2-F66 Plot of Maximum Shear Stress on Left Wall.

B-13) C3-P-H1-F33

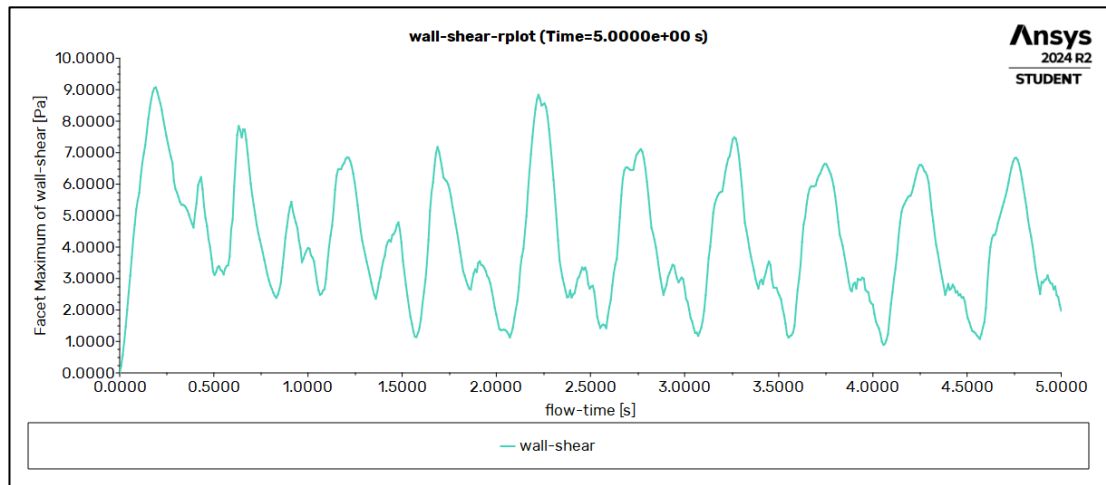


Figure B.13: C3-P-H1-F33 Plot of Maximum Shear Stress on Left Wall.

B-14) C3-P-H1-F66

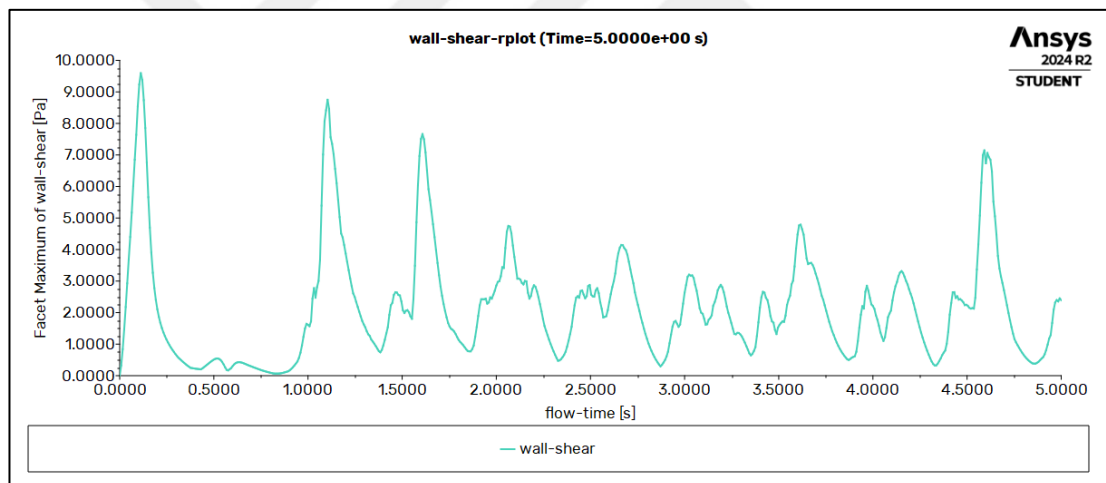


Figure B.14: C3-P-H1-F66 Plot of Maximum Shear Stress on Left Wall.

C. PLOT OF MAXIMUM PRESSURE ON LEFT WALL

Plots of maximum pressure on left wall are provided in **Figure C.1**, **Figure C.2**, **Figure C.3**, **Figure C.4**, **Figure C.5**, **Figure C.6**, **Figure C.7**, **Figure C.8**, **Figure C.9**, **Figure C.10**, **Figure C.11**, **Figure C.12**, **Figure C.13** and **Figure C.14**.

C-1) C0-F33

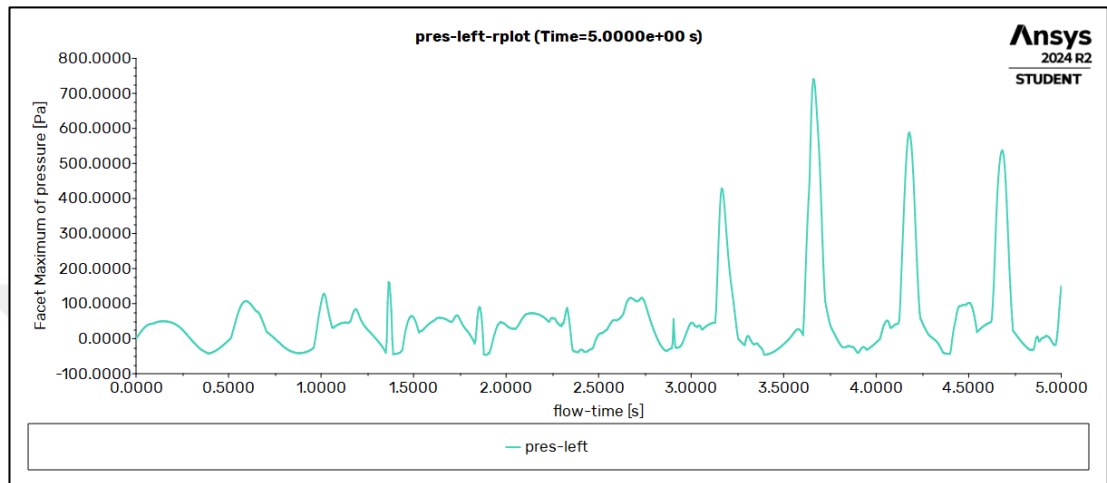


Figure C.1: C0-F33 Plot of Maximum Pressure on Left Wall.

C-2) C0-F66

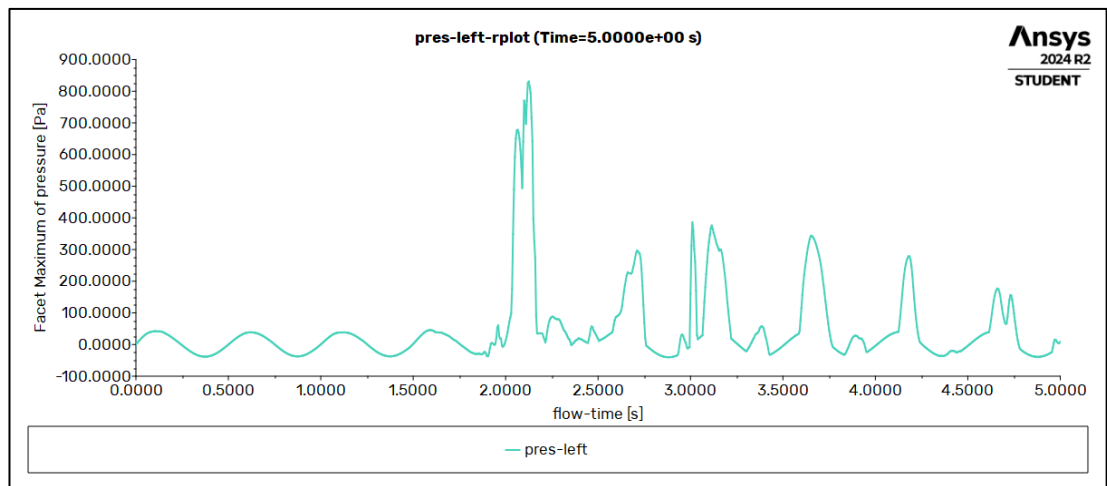


Figure C.2: C0-F66 Plot of Maximum Pressure on Left Wall.

C-3) C1-S-H3-F33

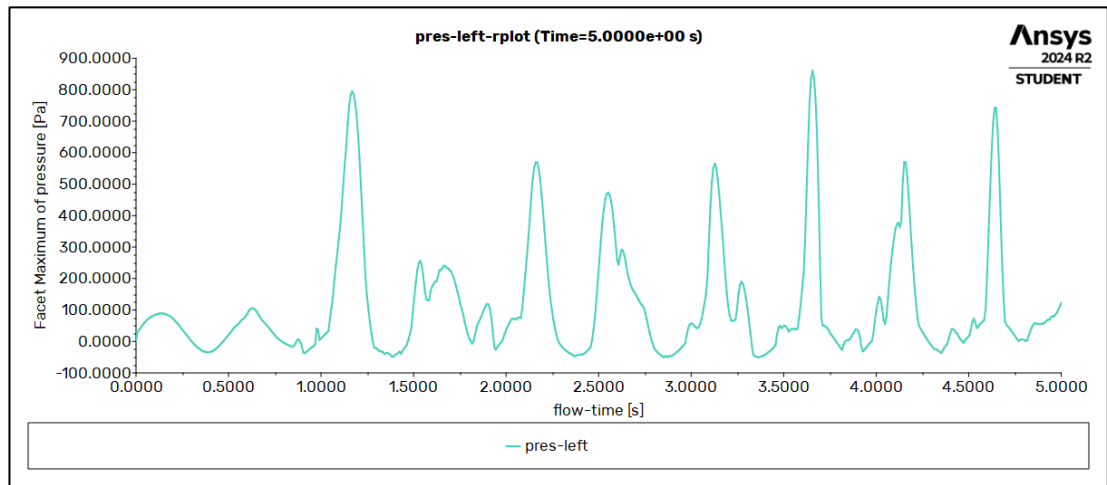


Figure C.3: C1-S-H3-F33 Plot of Maximum Pressure on Left Wall.

C-4) C1-S-H3-F66

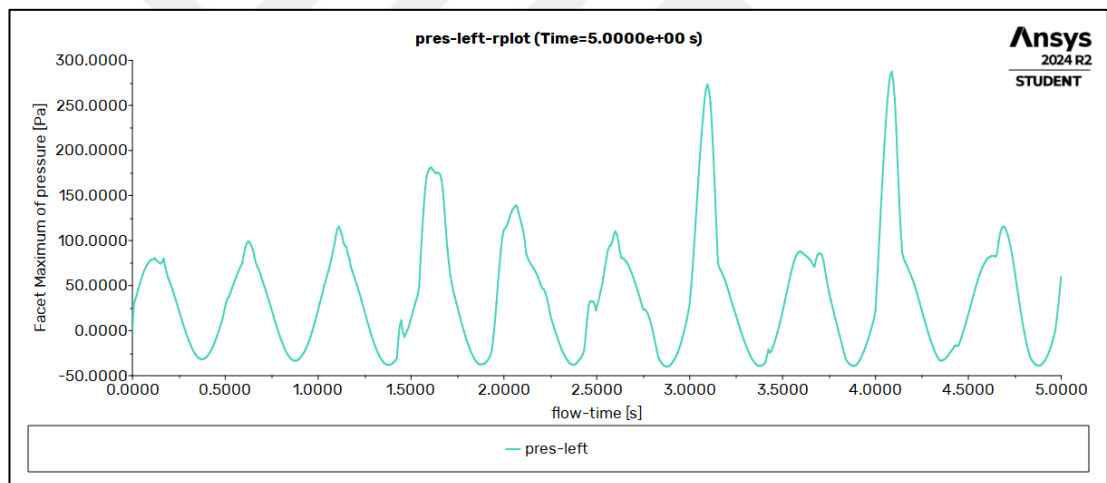


Figure C.4: C1-S-H3-F66 Plot of Maximum Pressure on Left Wall.

C-5) C2-S-H3-F33

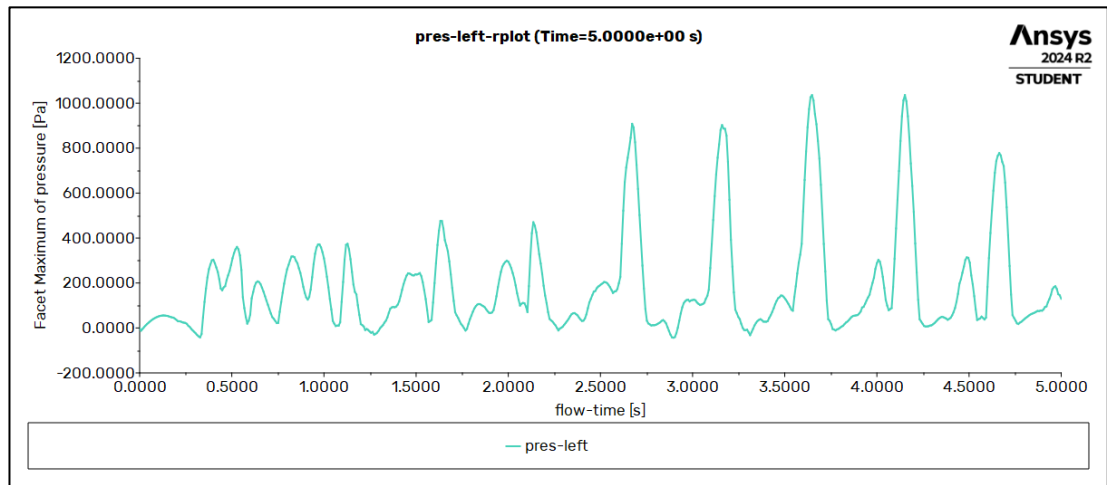


Figure C.5: C2-S-H3-F33 Plot of Maximum Pressure on Left Wall.

C-6) C2-S-H3-F66

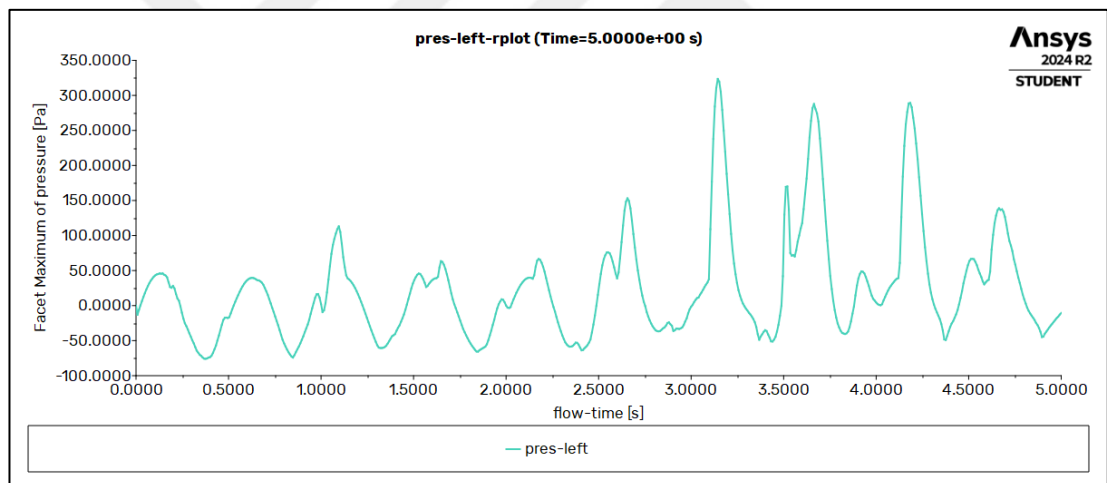


Figure C.6: C2-S-H3-F66 Plot of Maximum Pressure on Left Wall.

C-7) C3-S-H3-F33

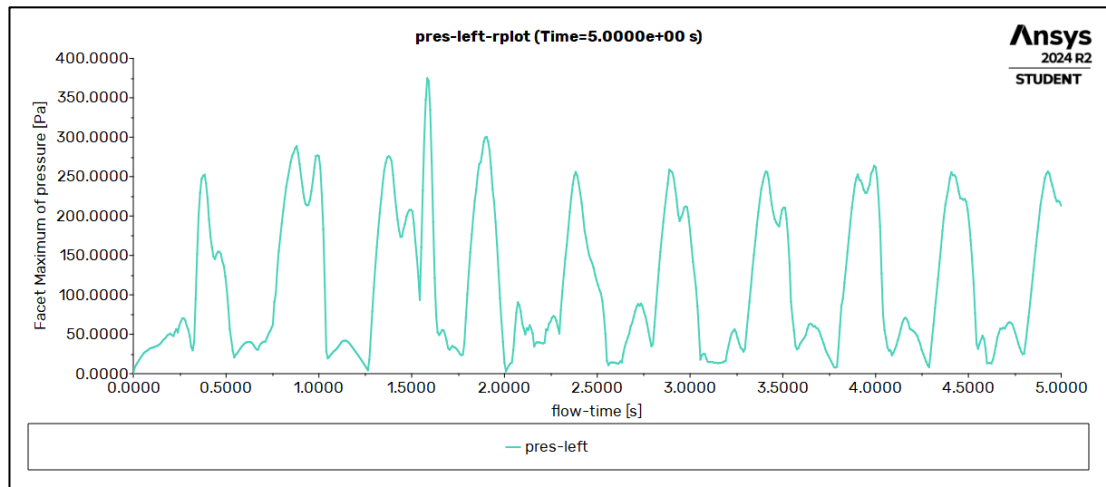


Figure C.7: C3-S-H3-F33 Plot of Maximum Pressure on Left Wall.

C-8) C3-S-H3-F66

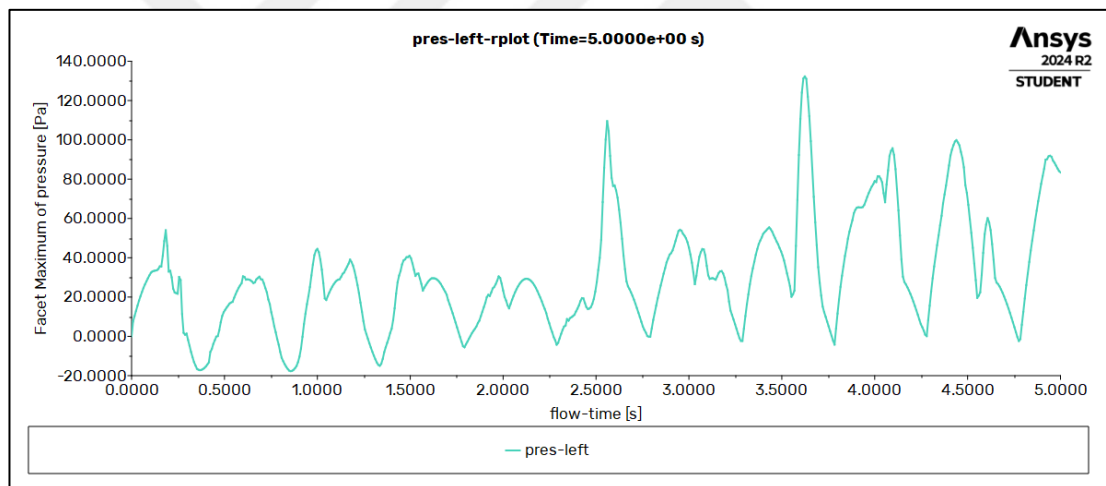


Figure C.8: C3-S-H3-F66 Plot of Maximum Pressure on Left Wall.

C-9) C3-P-H3-F33

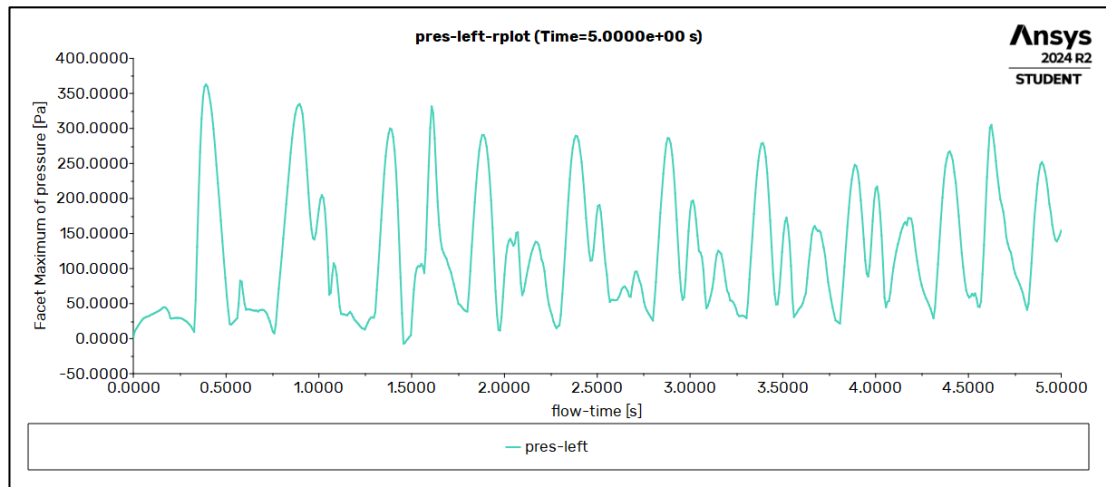


Figure C.9: C3-P-H3-F33 Plot of Maximum Pressure on Left Wall.

C-10) C3-P-H3-F66

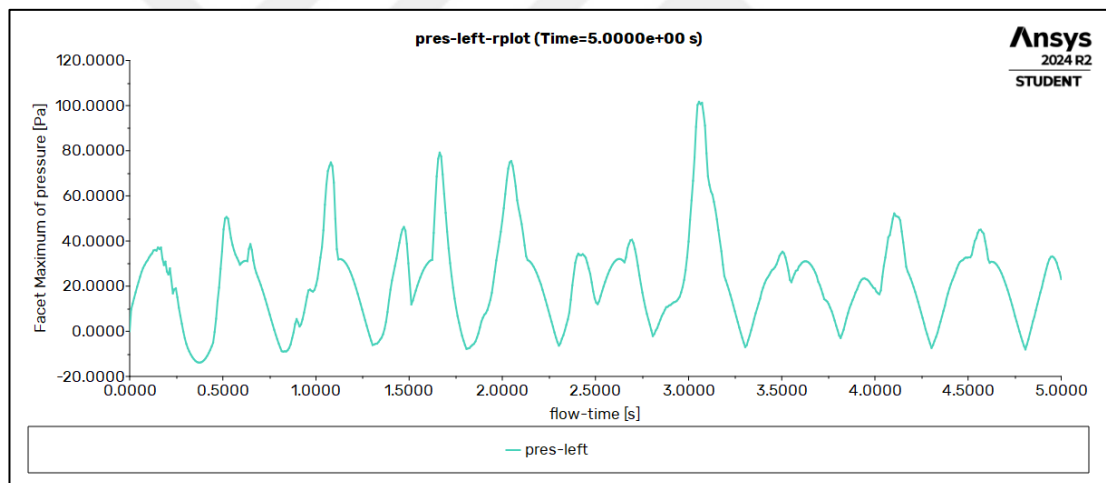


Figure C.10: C3-P-H3-F66 Plot of Maximum Pressure on Left Wall.

C-11) C3-P-H2-F33

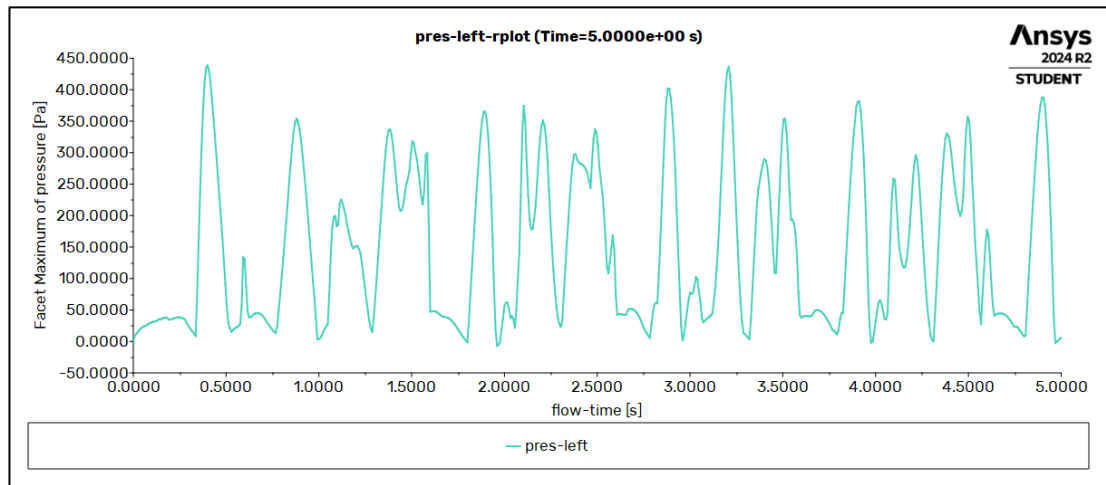


Figure C.11: C3-P-H2-F33 Plot of Maximum Pressure on Left Wall.

C-12) C3-P-H2-F66

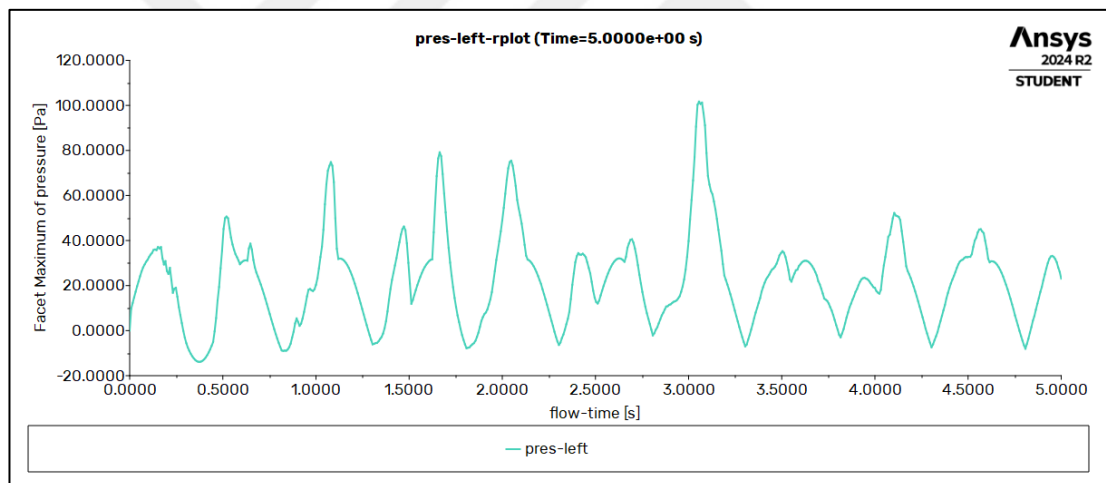


Figure C.12: C3-P-H2-F66 Plot of Maximum Pressure on Left Wall.

C-13) C3-P-H1-F33

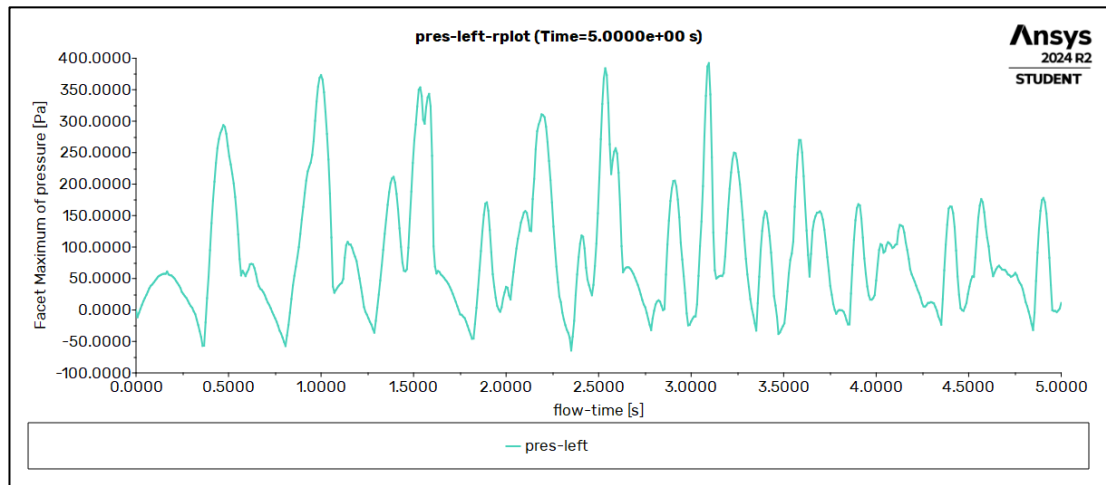


Figure C.13: C3-P-H1-F33 Plot of Maximum Pressure on Left Wall.

C-14) C3-P-H1-F66

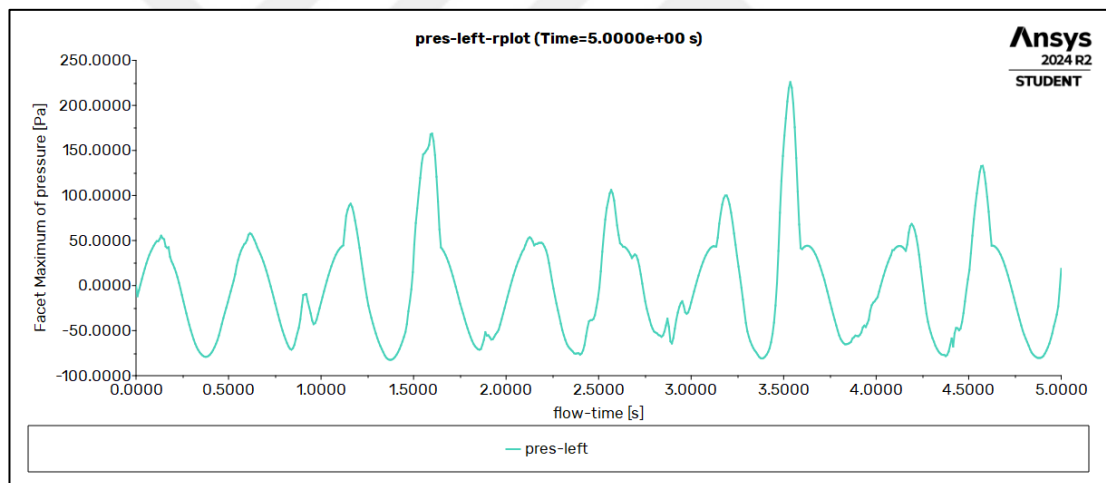


Figure C.14: C3-P-H1-F66 Plot of Maximum Pressure on Left Wall.

D. PLOT OF VOLUME FRACTION ON SYMMETRY PLANE

Plots of volume fraction on symmetry plane are provided in **Figure D.1**, **Figure D.2**, **Figure D.3**, **Figure D.4**, **Figure D.5**, **Figure D.6**, **Figure D.7**, **Figure D.8**, **Figure D.9**, **Figure D.10**, **Figure D.11**, **Figure D.12**, **Figure D.13** and **Figure D.14**.

D-1) C0-F33

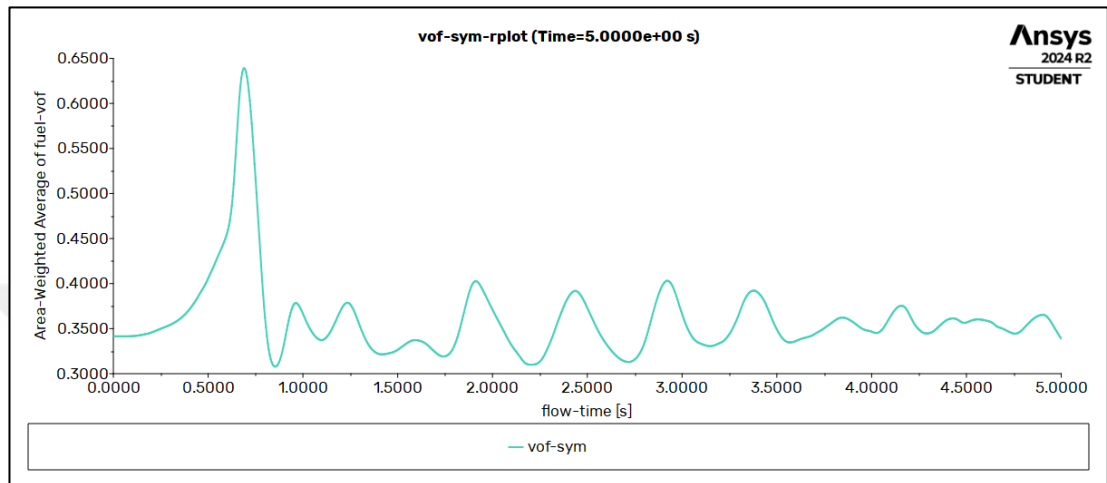


Figure D.1: C0-F33 Plot of Volume Fraction on Symmetry Plane.

D-2) C0-F66

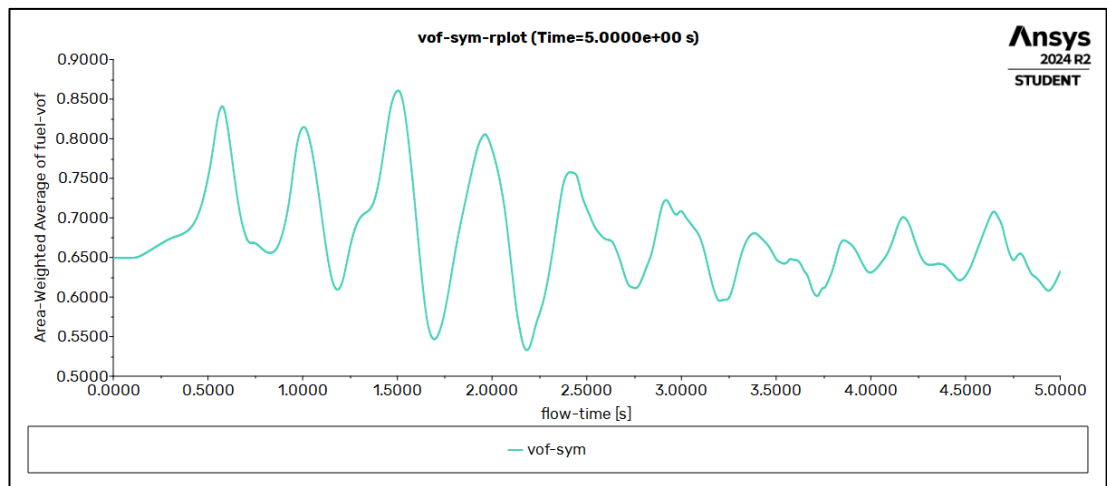


Figure D.2: C0-F66 Plot of Volume Fraction on Symmetry Plane.

D-3) C1-S-H3-F33

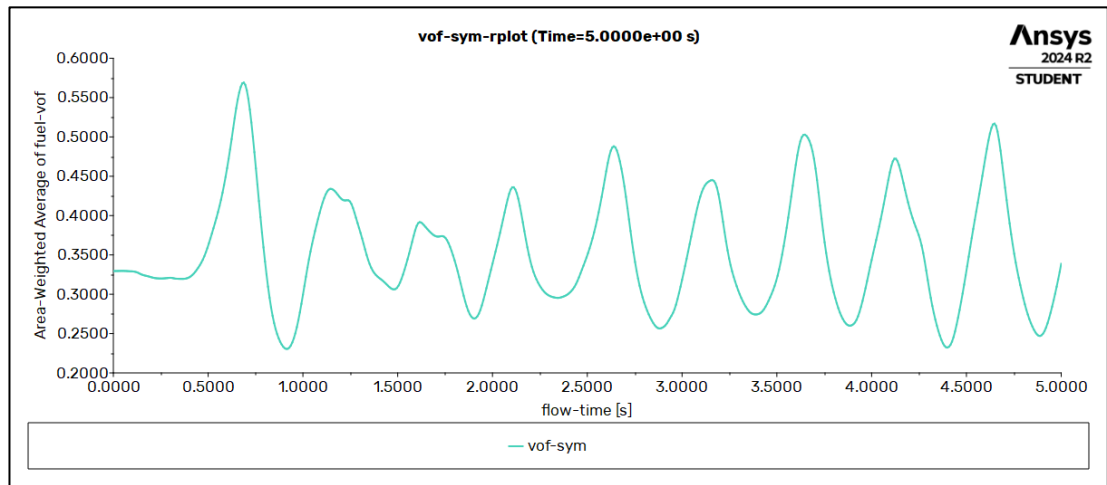


Figure D.3: C1-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.

D-4) C1-S-H3-F66

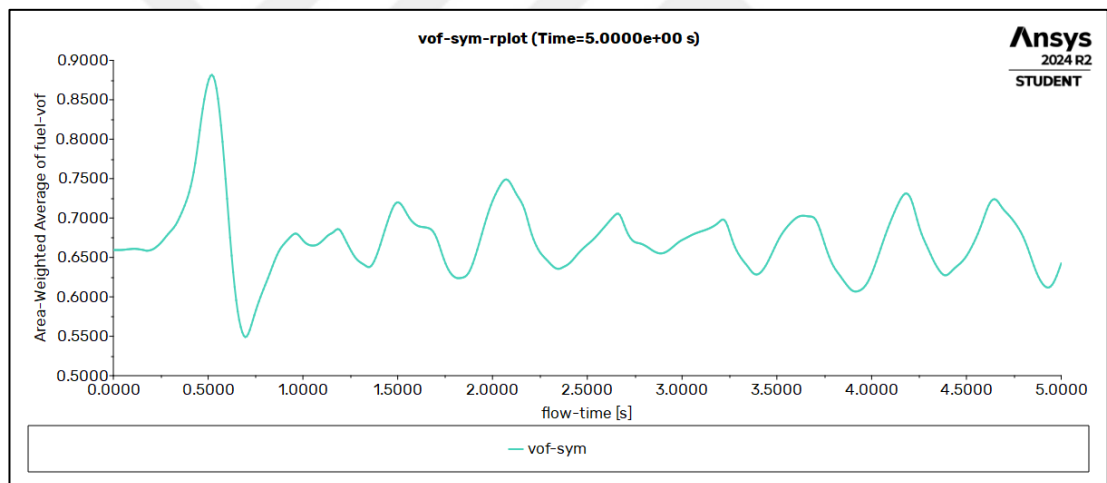


Figure D.4: C1-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.

D-5) C2-S-H3-F33

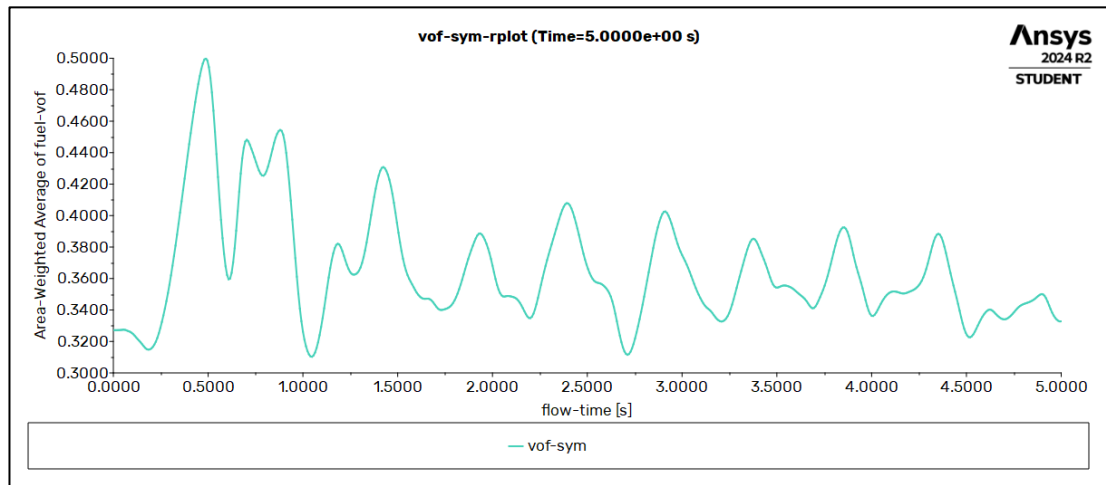


Figure D.5: C2-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.

D-6) C2-S-H3-F66

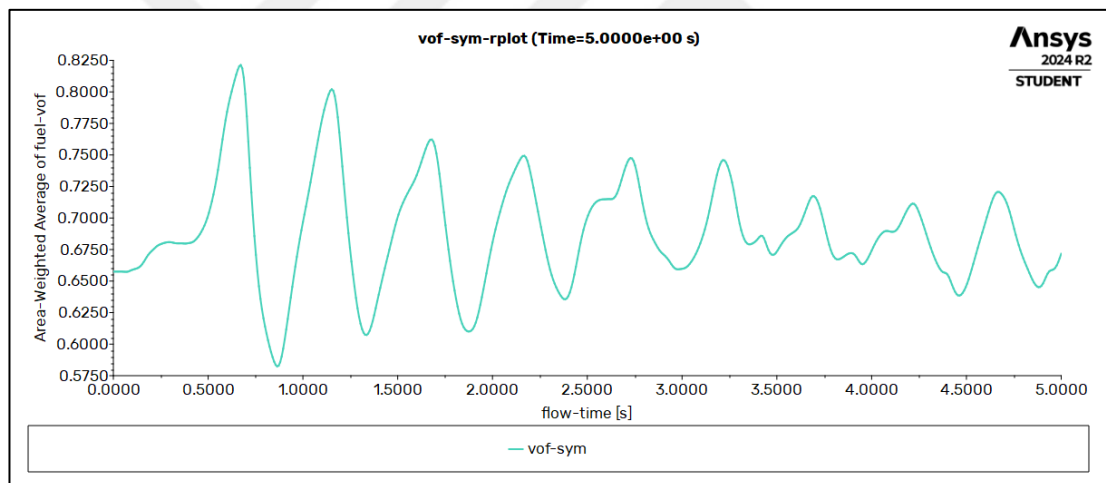


Figure D.6: C2-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.

D-7) C3-S-H3-F33

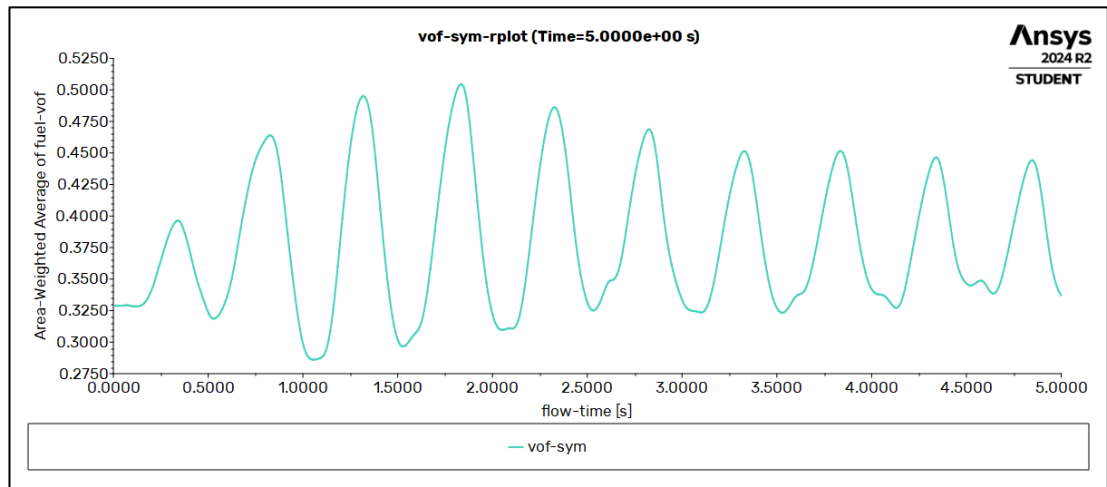


Figure D.7: C3-S-H3-F33 Plot of Volume Fraction on Symmetry Plane.

D-8) C3-S-H3-F66

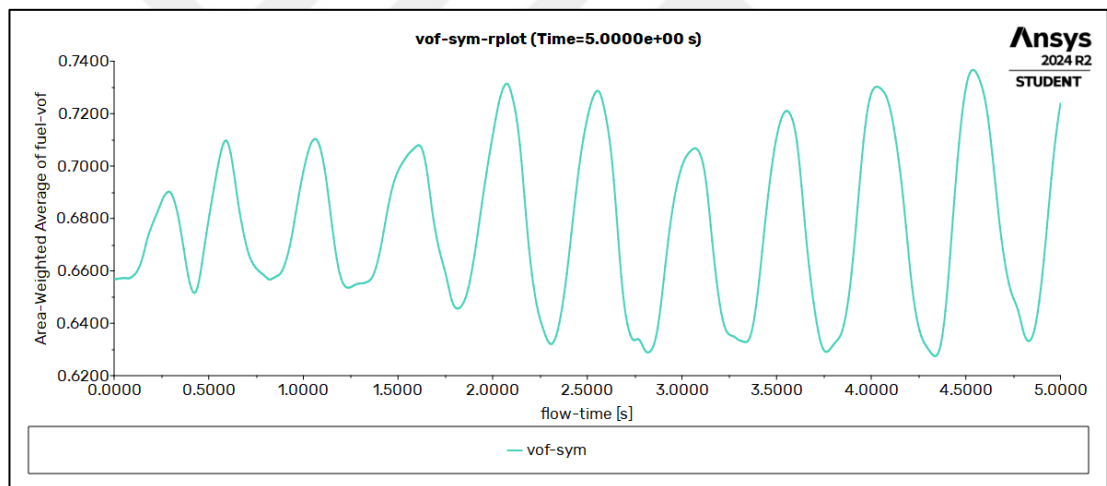


Figure D.8: C3-S-H3-F66 Plot of Volume Fraction on Symmetry Plane.

D-9) C3-P-H3-F33

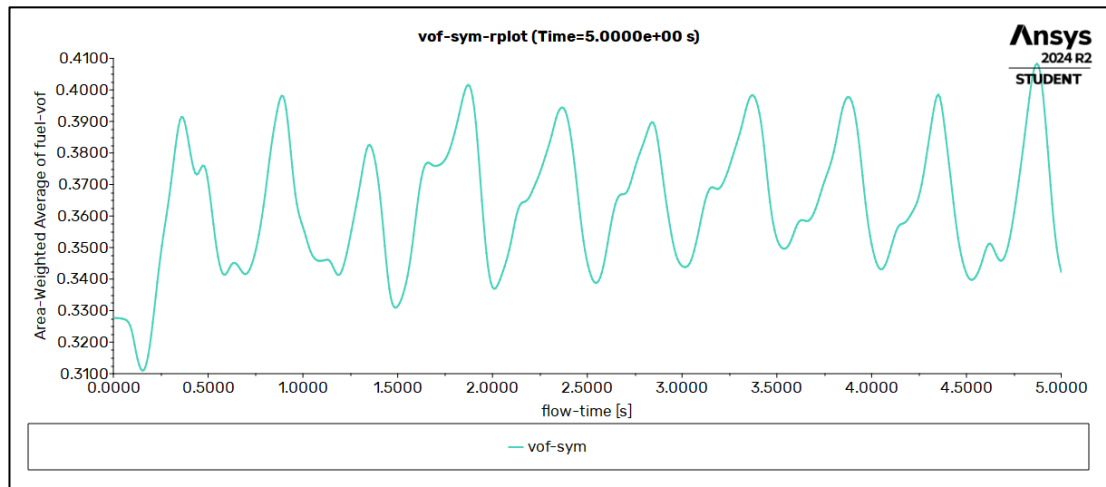


Figure D.9: C3-P-H3-F33 Plot of Volume Fraction on Symmetry Plane.

D-10) C3-P-H3-F66

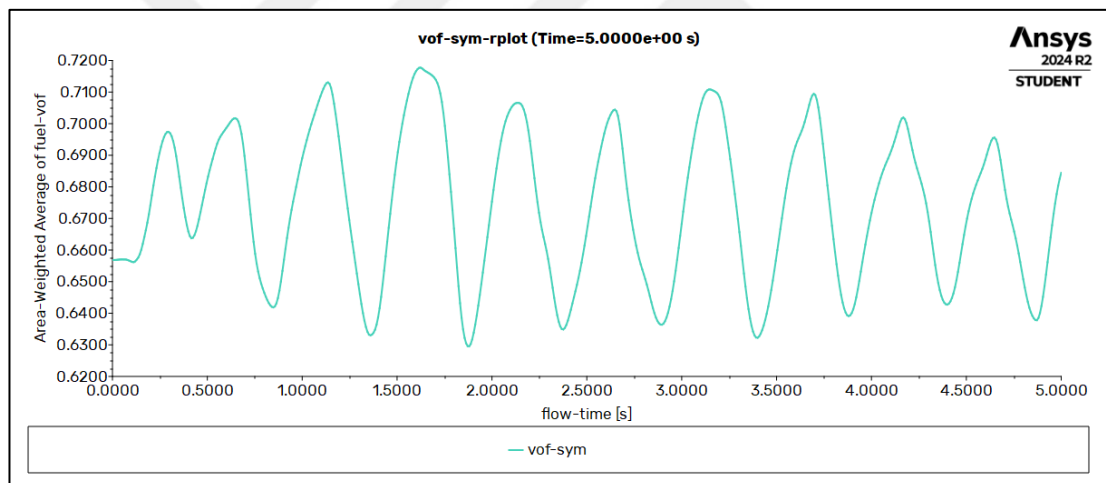


Figure D.10: C3-P-H3-F66 Plot of Volume Fraction on Symmetry Plane.

D-11) C3-P-H2-F33

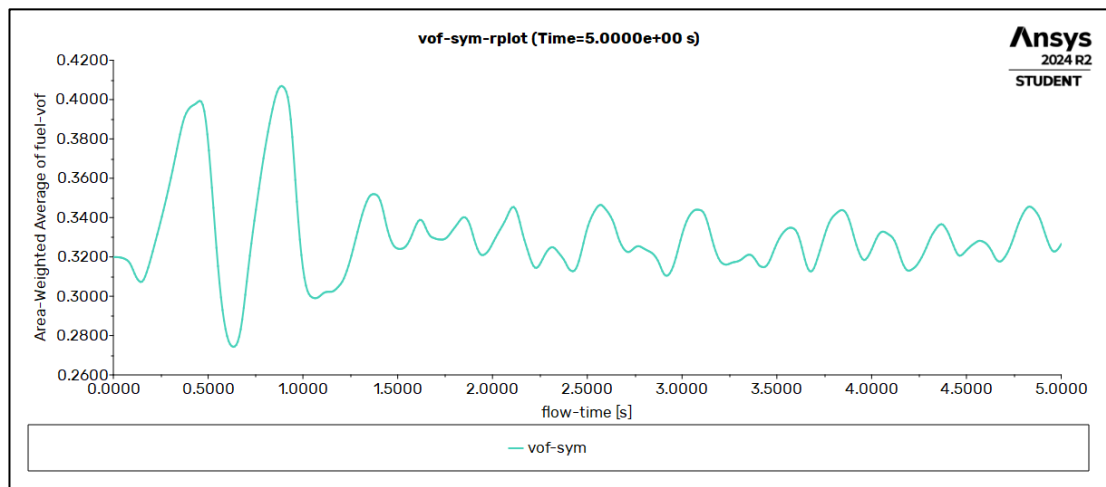


Figure D.11: C3-P-H2-F33 Plot of Volume Fraction on Symmetry Plane.

D-12) C3-P-H2-F66

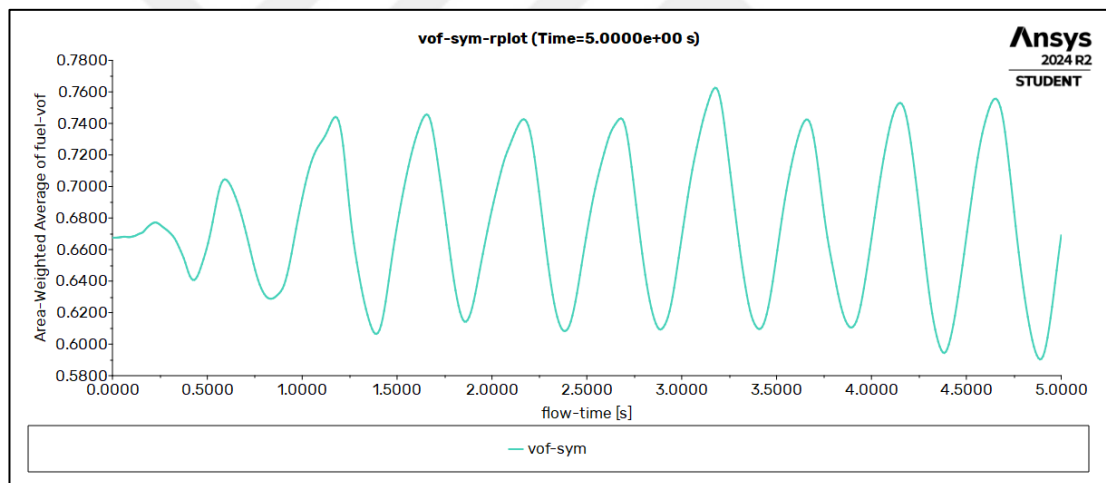


Figure D.12: C3-P-H2-F66 Plot of Volume Fraction on Symmetry Plane.

D-13) C3-P-H1-F33

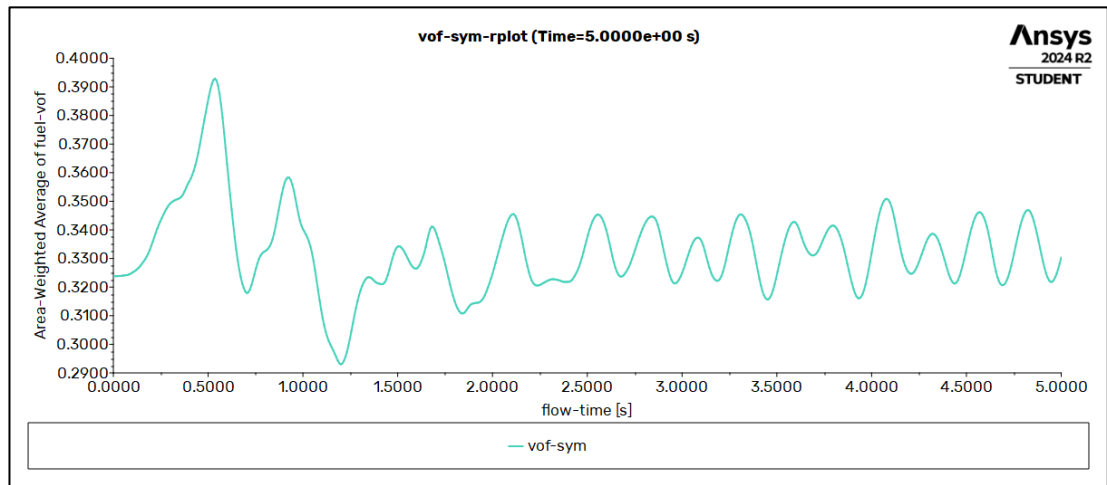


Figure D.13: C3-P-H1-F33 Plot of Volume Fraction on Symmetry Plane.

D-14) C3-P-H1-F66

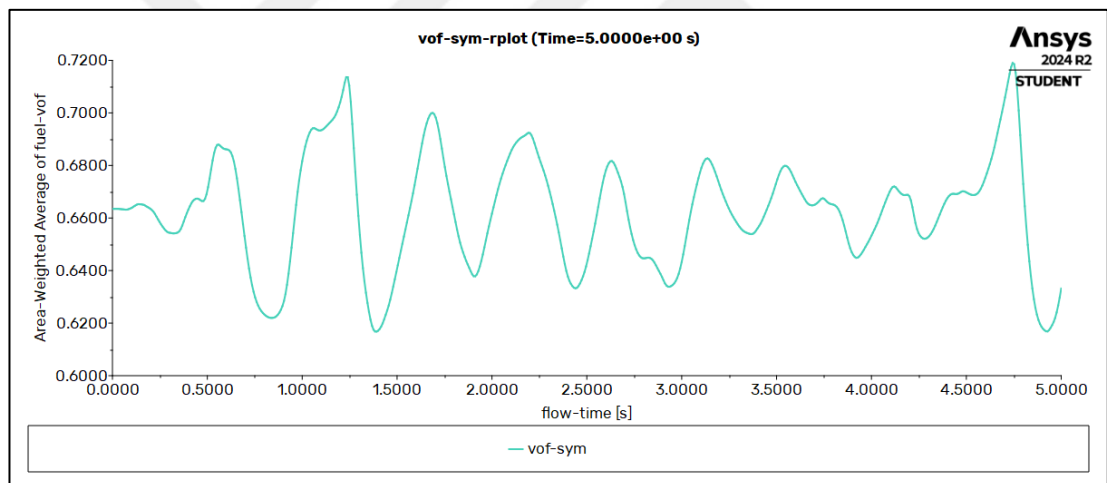


Figure D.14: C3-P-H1-F66 Plot of Volume Fraction on Symmetry Plane.

E. CONTOUR OF VOLUME FRACTION ON SYMMETRY PLANE

Contours of volume fraction on symmetry plane are provided in **Figure E.1**, **Figure E.2**, **Figure E.3**, **Figure E.4**, **Figure E.5**, **Figure E.6**, **Figure E.7**, **Figure E.8**, **Figure E.9**, **Figure E.10**, **Figure E.11**, **Figure E.12**, **Figure E.13** and **Figure E.14**.

E-1) C0-F33

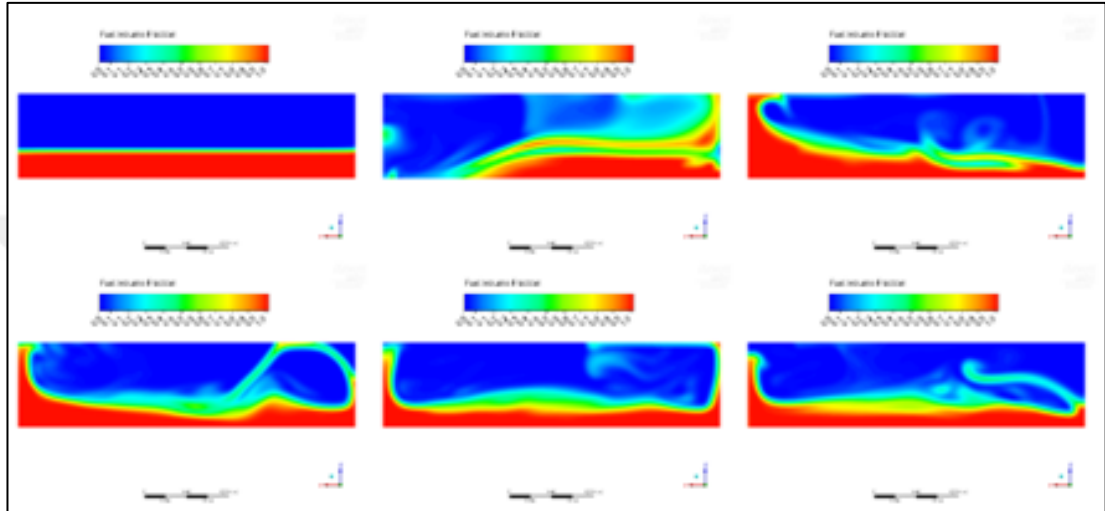


Figure E.1: C0-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-2) C0-F66

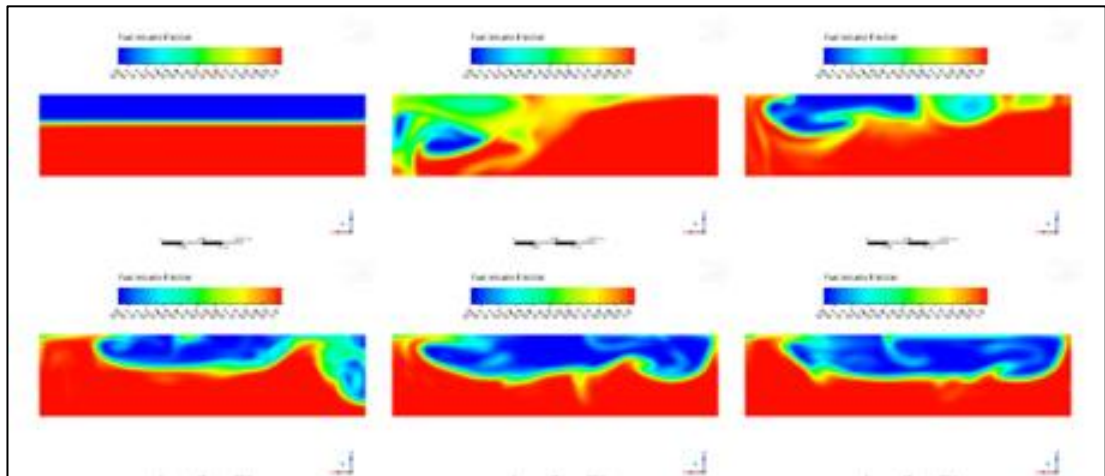


Figure E.2: C0-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-3) C1-S-H3-F33

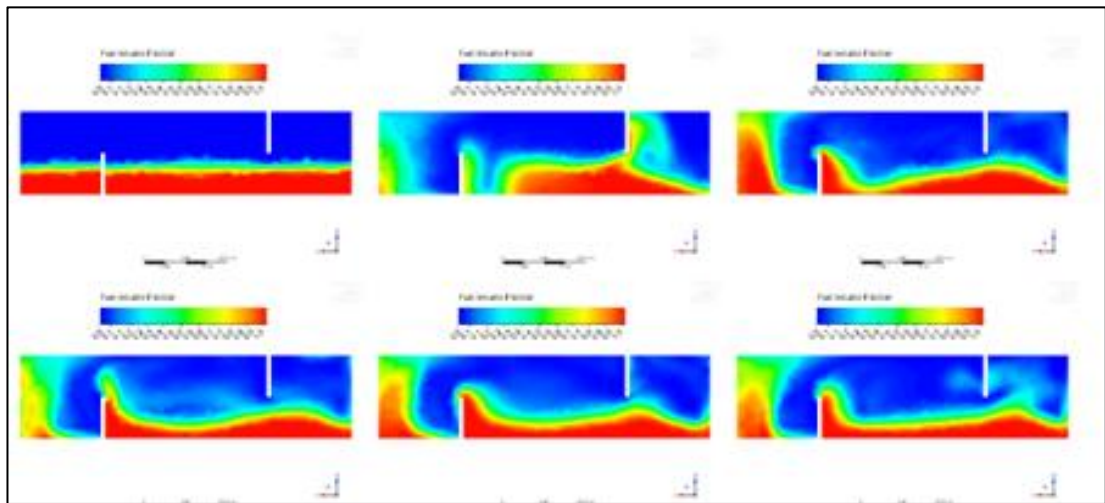


Figure E.3: C1-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-4) C1-S-H3-F66

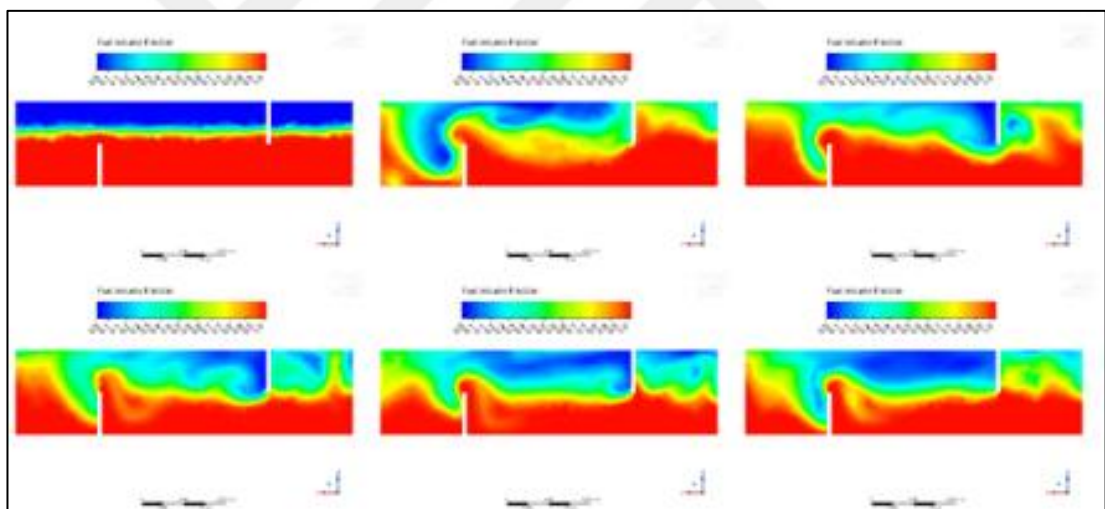


Figure E.4: C1-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-5) C2-S-H3-F33

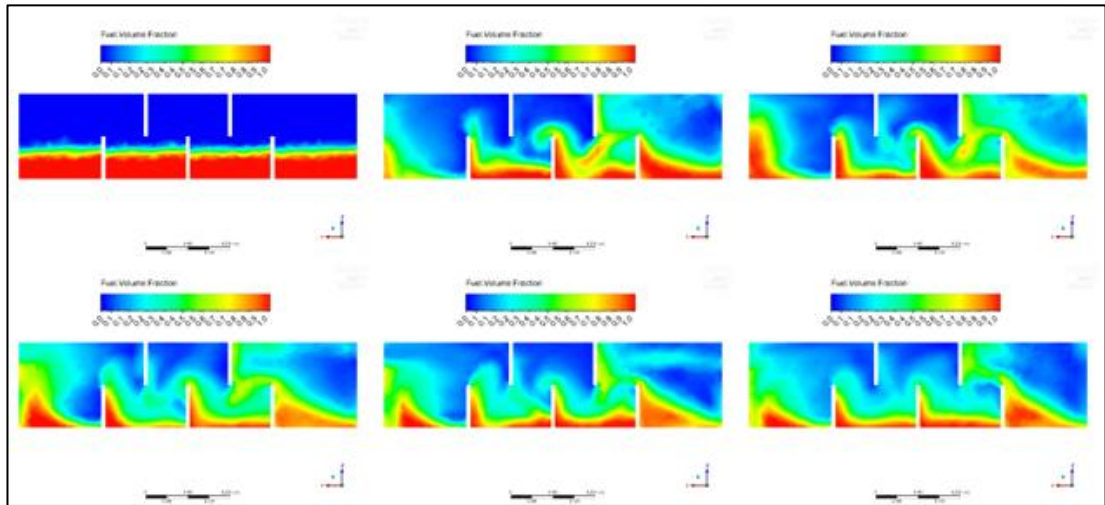


Figure E.5: C2-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-6) C2-S-H3-F66

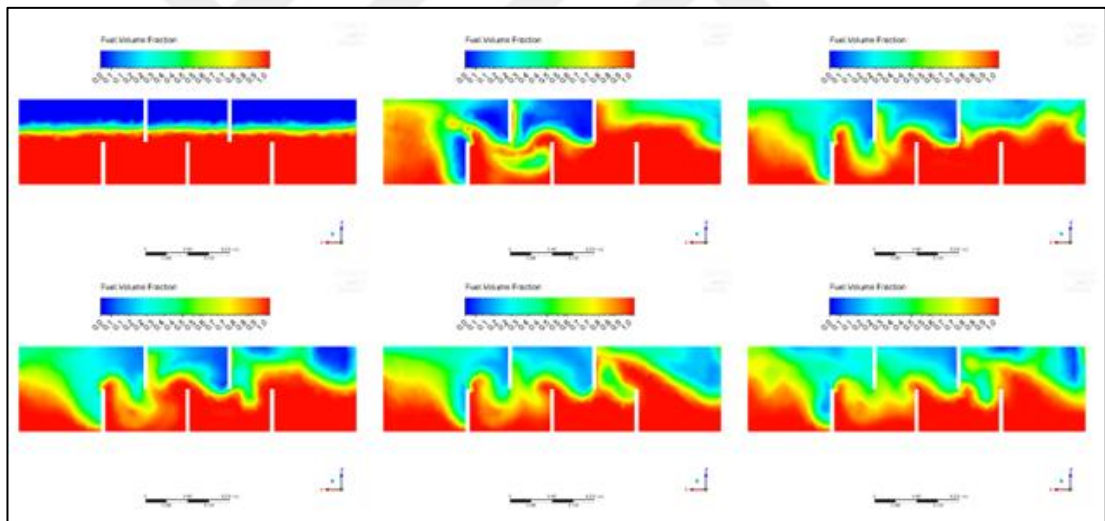


Figure E.6: C2-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-7) C3-S-H3-F33

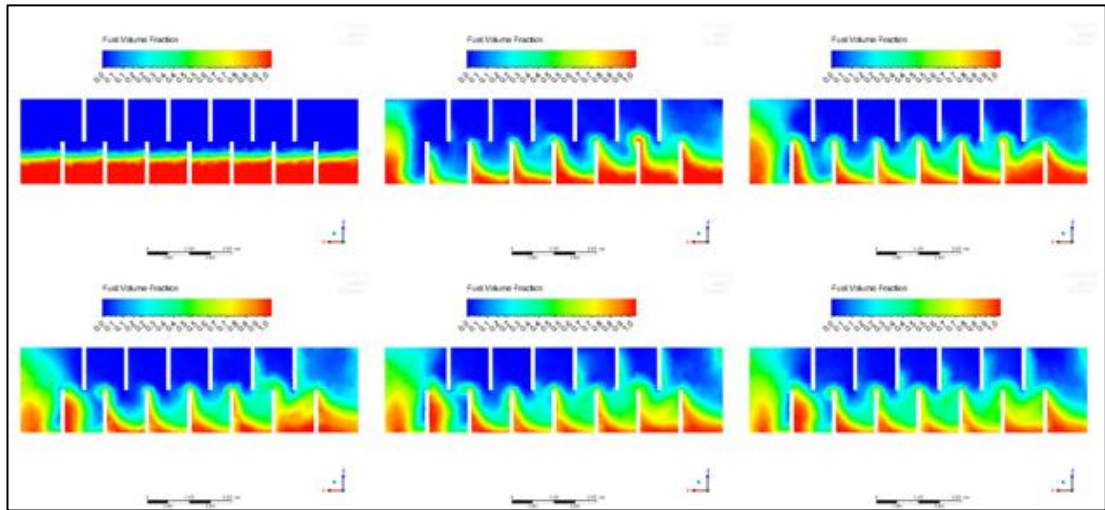


Figure E.7: C3-S-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-8) C3-S-H3-F66

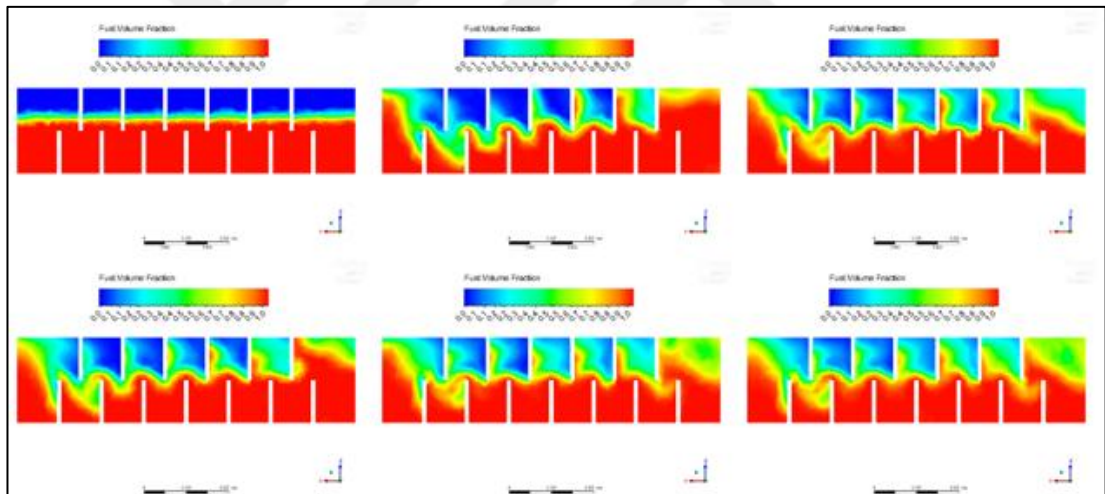


Figure E.8: C3-S-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-9) C3-P-H3-F33

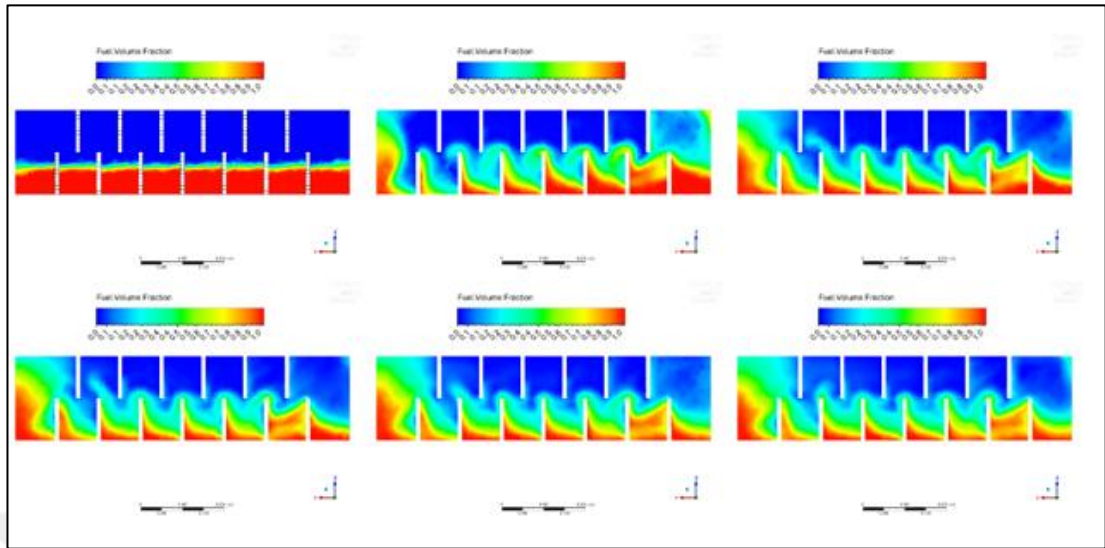


Figure E.9: C3-P-H3-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-10) C3-P-H3-F66

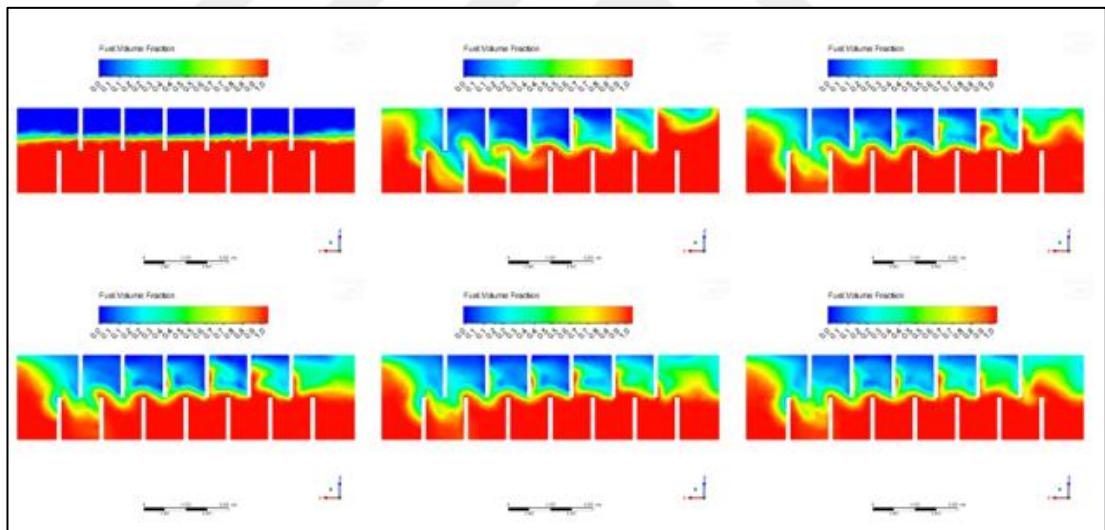


Figure E.10: C3-P-H3-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-11) C3-P-H2-F33

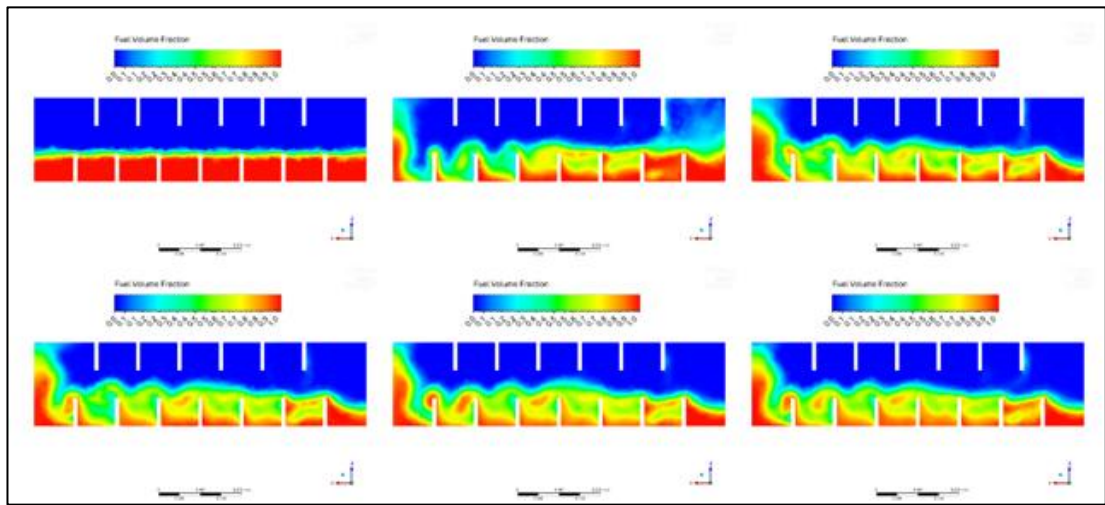


Figure E.11: C3-P-H2-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-12) C3-P-H2-F66

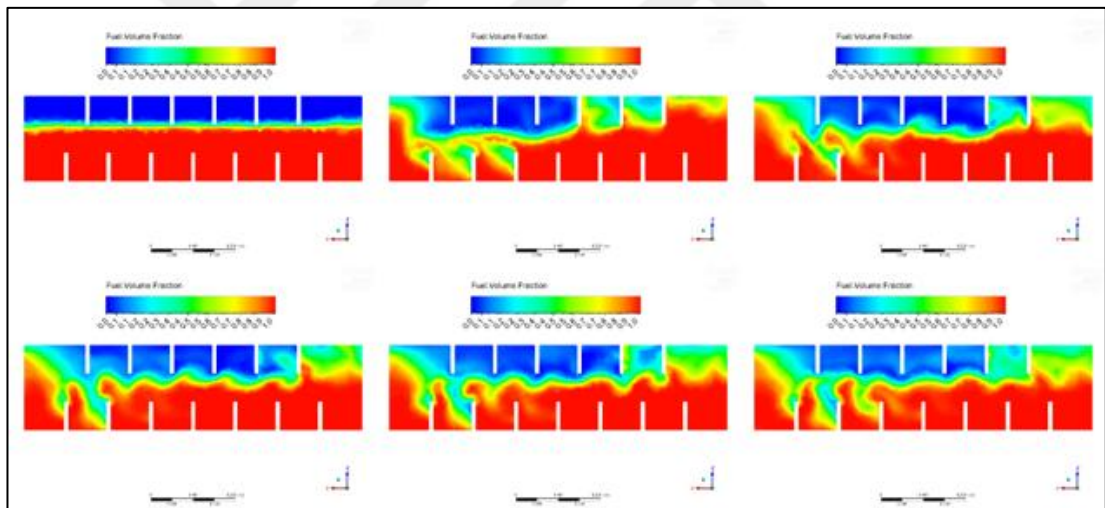


Figure E.12: C3-P-H2-F66 Contour of Volume Fraction on Symmetry (0-5s).

E-13) C3-P-H1-F33

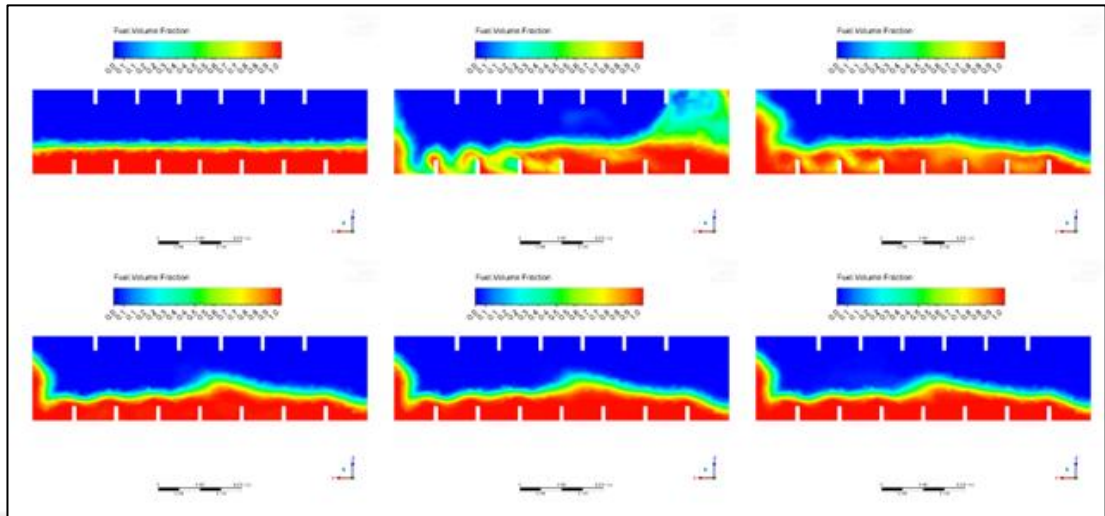


Figure E.13: C3-P-H1-F33 Contour of Volume Fraction on Symmetry (0-5s).

E-14) C3-P-H1-F66

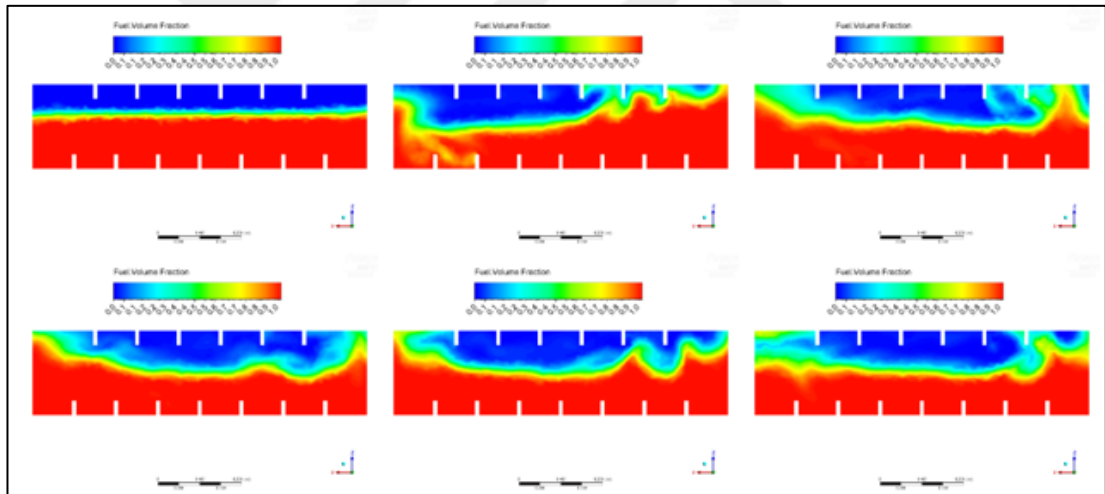


Figure E.14: C3-P-H1-F66 Contour of Volume Fraction on Symmetry (0-5s).

F. CONTOUR OF VOLUME FRACTION ON LEFT WALL

Contours of volume fraction on left wall are provided in **Figure F.1**, **Figure F.2**, **Figure F.3**, **Figure F.4**, **Figure F.5**, **Figure F.6**, **Figure F.7**, **Figure F.8**, **Figure F.9**, **Figure F.10**, **Figure F.11**, **Figure F.12**, **Figure F.13** and **Figure F.14**.

F-1) C0-F33

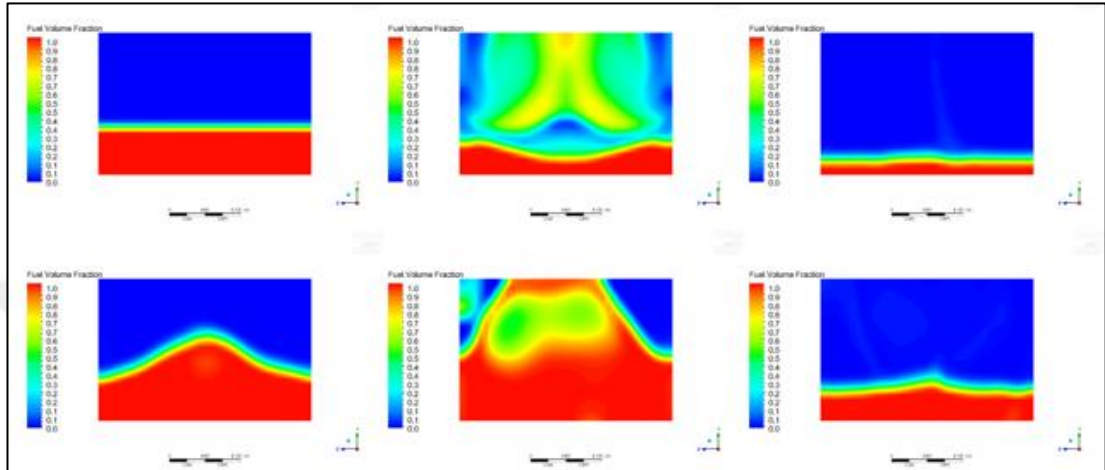


Figure F.1: C0-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-2) C0-F66

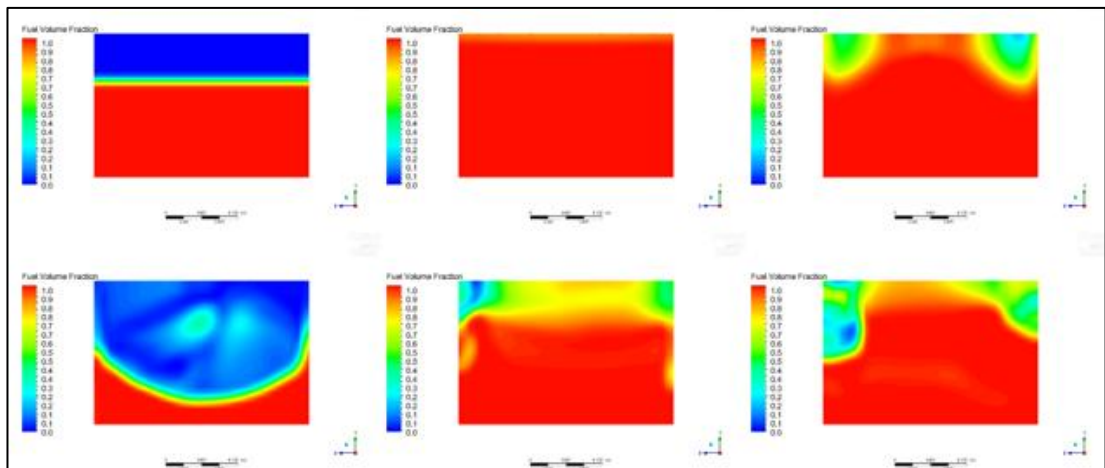


Figure F.2: C0-F66 Contour of Volume Fraction on Left Wall (0-5s).

F-3) C1-S-H3-F33

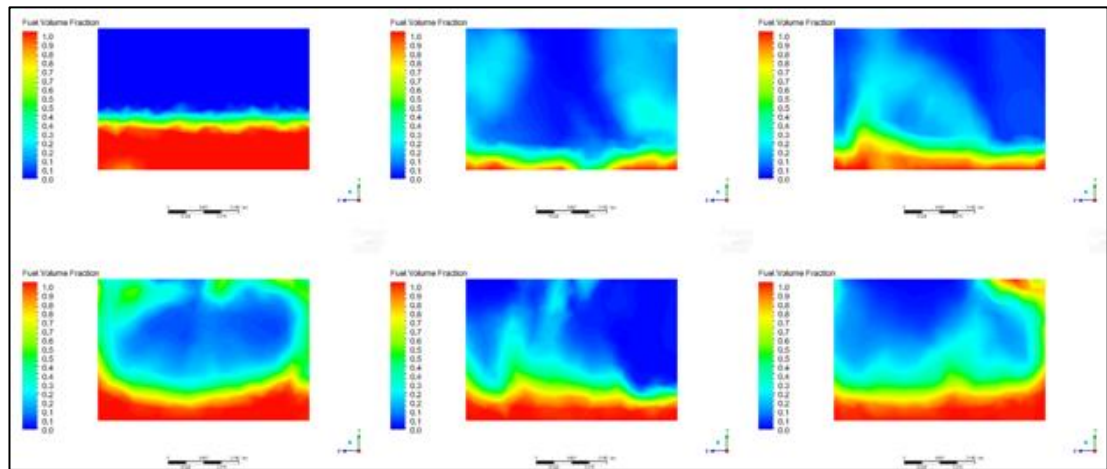


Figure F.3: C1-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-4) C1-S-H3-F66

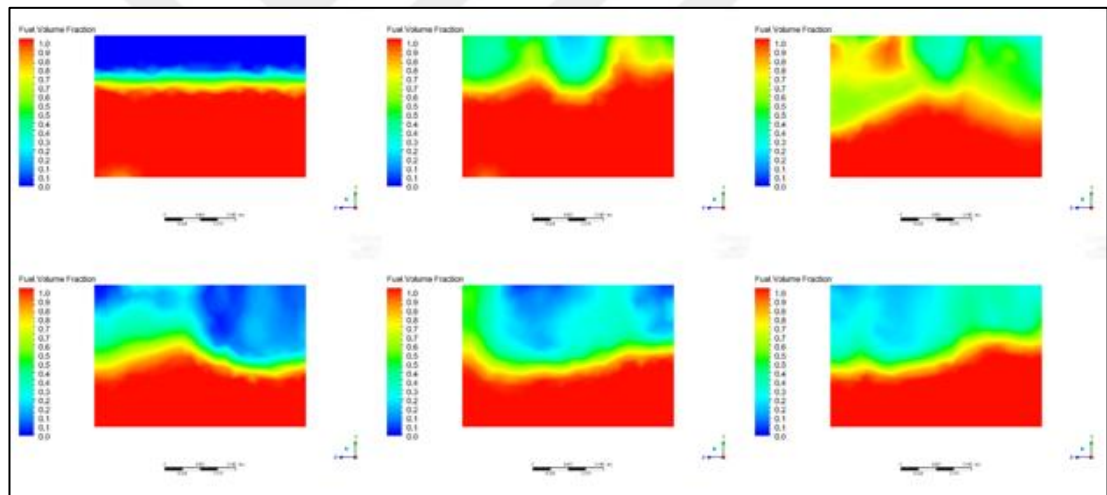


Figure F.4: C1-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).

F-5) C2-S-H3-F33

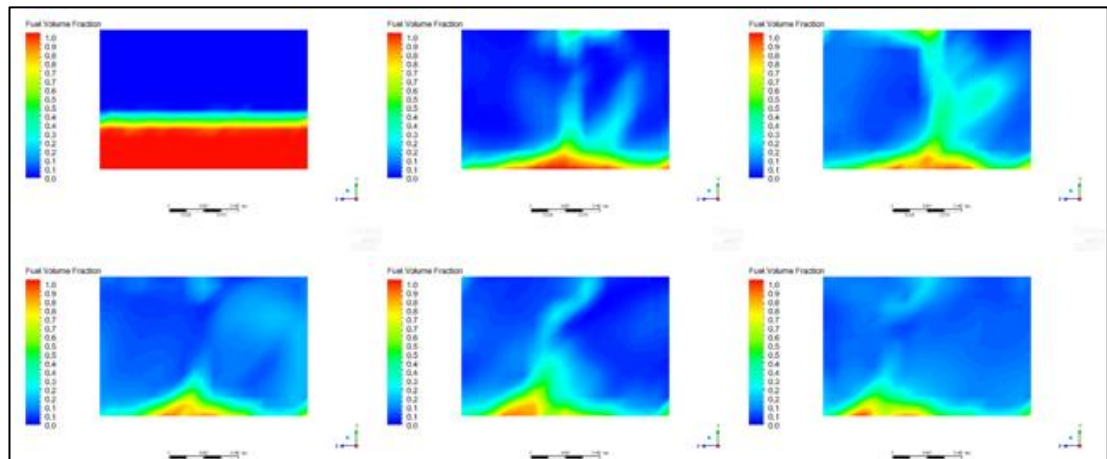


Figure F.5: C2-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-6) C2-S-H3-F66

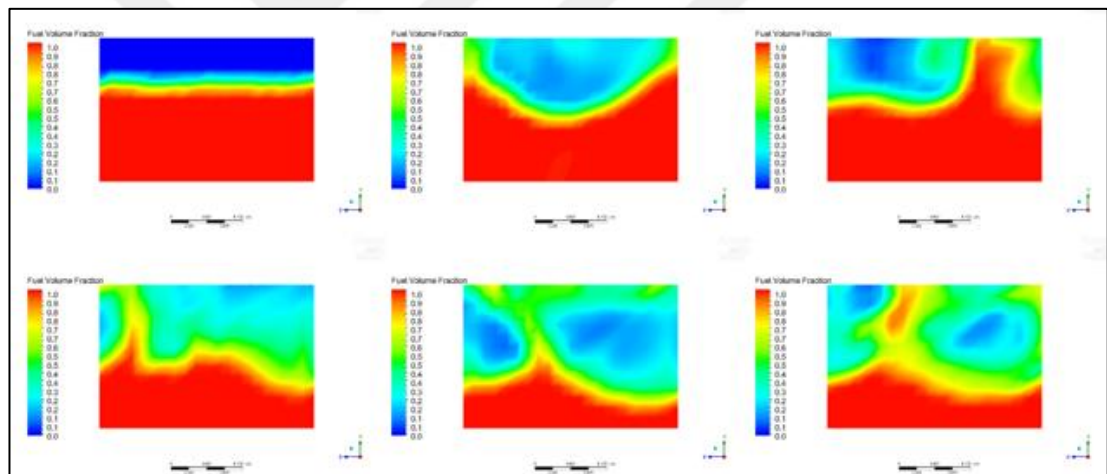


Figure F.6: C2-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s)

F-7) C3-S-H3-F33

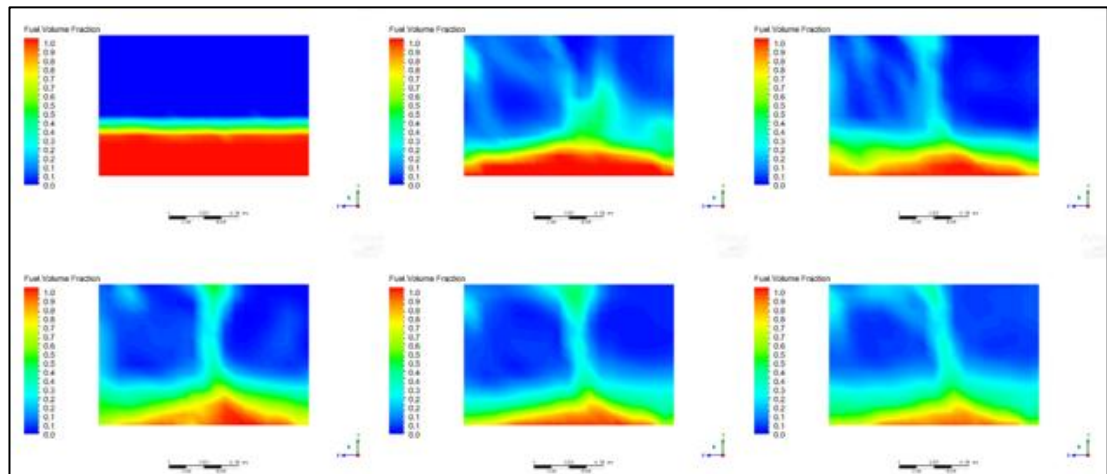


Figure F.7: C3-S-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-8) C3-S-H3-F66

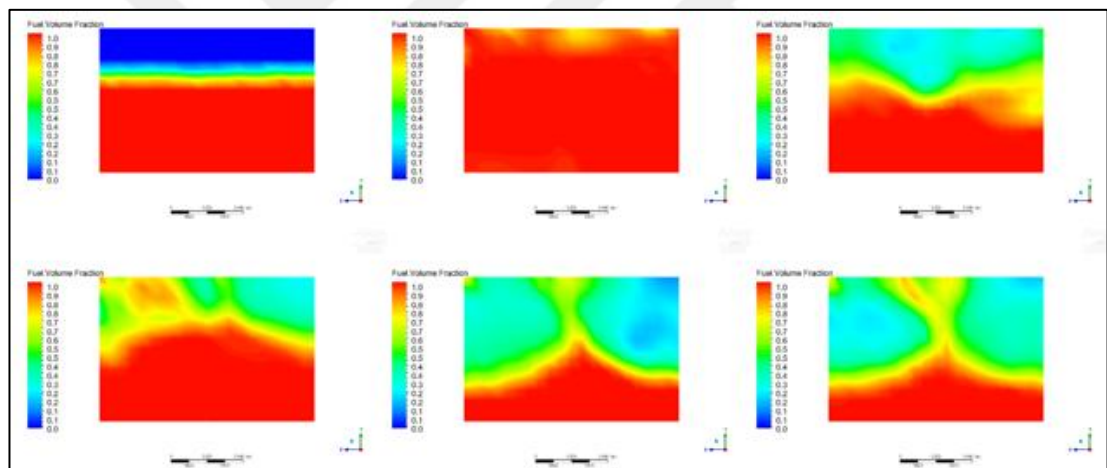


Figure F.8: C3-S-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).

F-9) C3-P-H3-F33

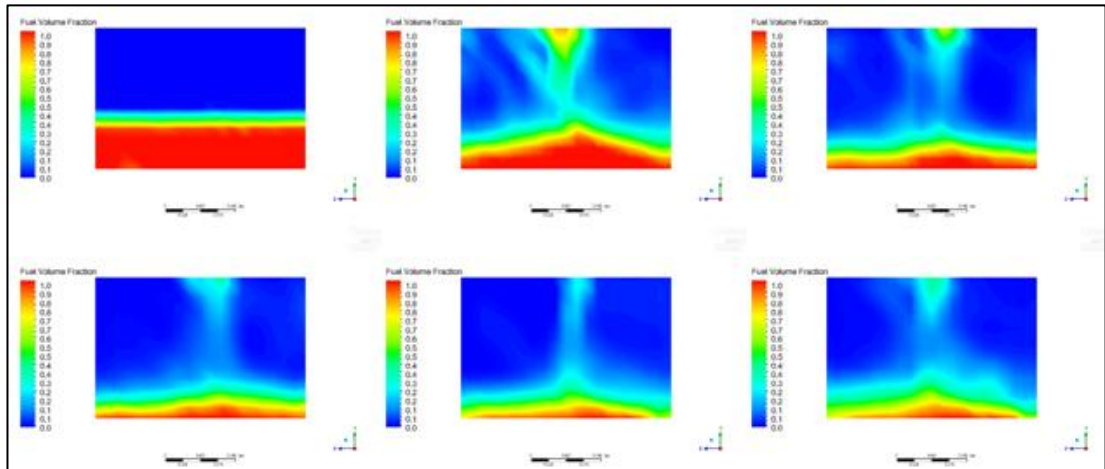


Figure F.9: C3-P-H3-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-10) C3-P-H3-F66

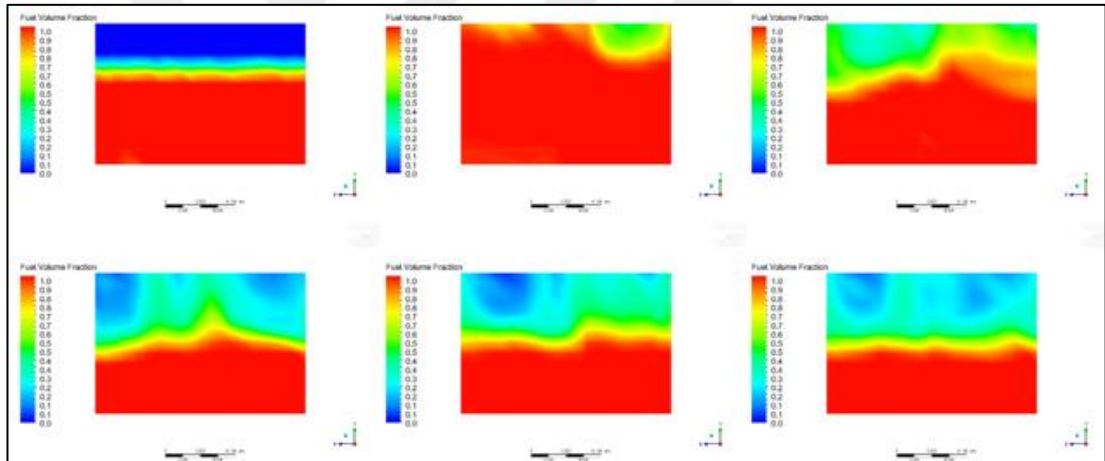


Figure F.10: C3-P-H3-F66 Contour of Volume Fraction on Left Wall (0-5s).

F-11) C3-P-H2-F33

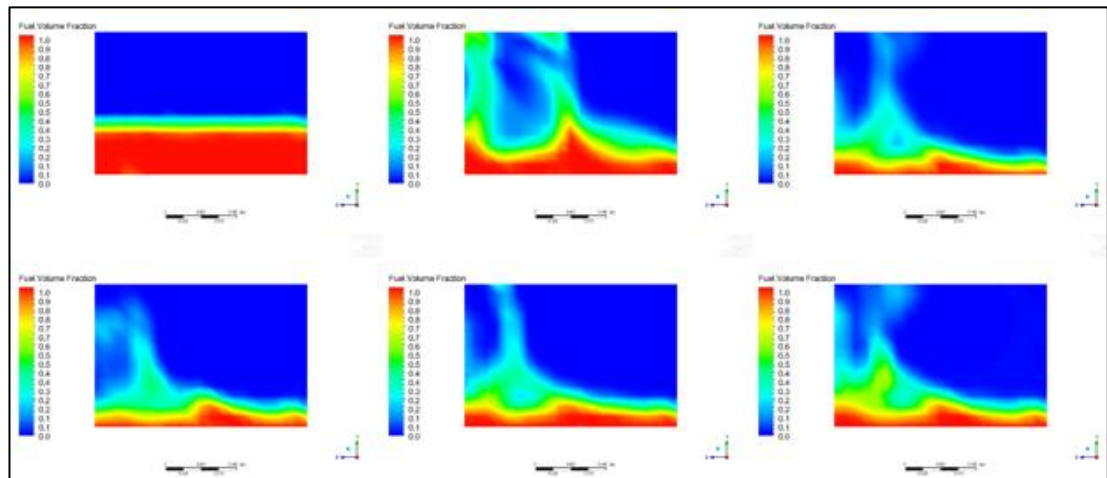


Figure F.11: C3-P-H2-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-12) C3-P-H2-F66

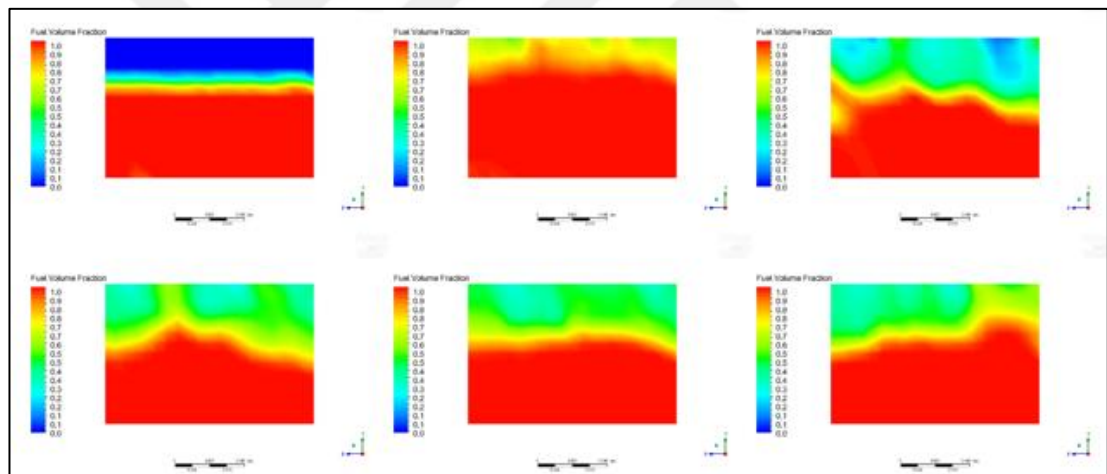


Figure F.12: C3-P-H2-F66 Contour of Volume Fraction on Left Wall (0-5s).

F-13) C3-P-H1-F33

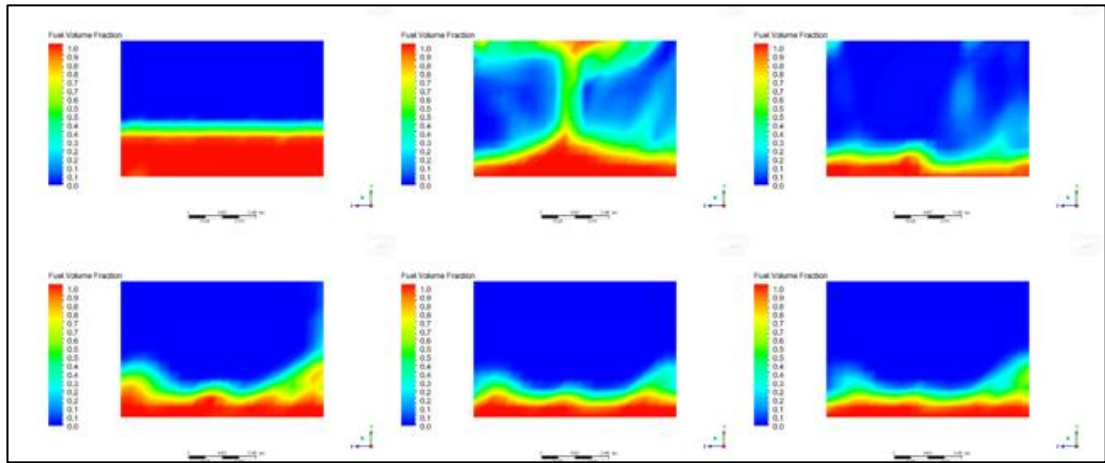


Figure F.13: C3-P-H1-F33 Contour of Volume Fraction on Left Wall (0-5s).

F-14) C3-P-H1-F66

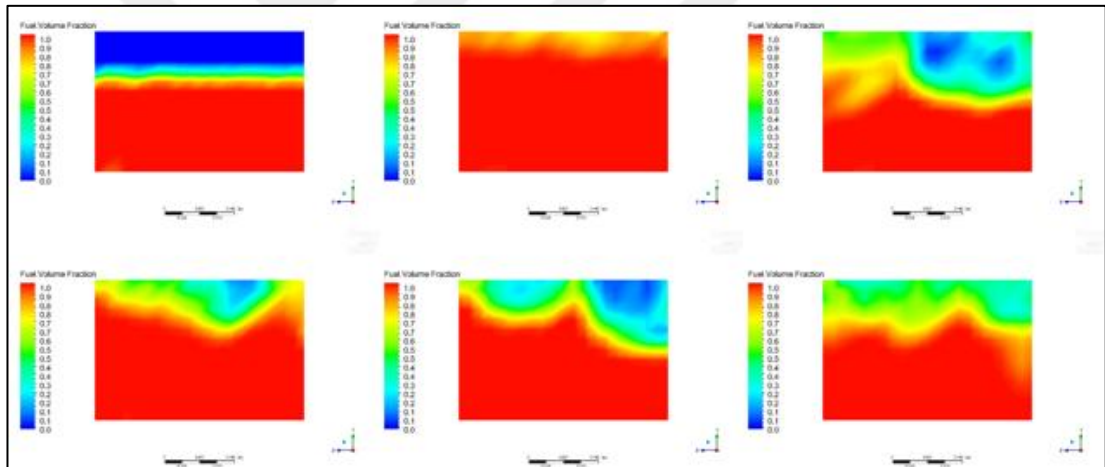


Figure F.14: C3-P-H1-F66 Contour of Volume Fraction on Left Wall (0-5s).