

NON-LINEAR DYNAMIC FLUID INTERACTION BETWEEN STAGED LIQUID PROPELLANT ROCKET ENGINE

Submitted in partial fulfilment of the requirements for the award of Bachelor of
Engineering degree in Aeronautical

By

K. NISHWIN ABISHEK (Reg. No. 39260021)

R. YUGENDRAN (Reg. No. 39260046)



**DEPARTMENT OF AERONAUTICAL ENGINEERING
SCHOOL OF MECHANICAL ENGINEERING**

SATHYABAMA

**INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)**

Accredited with grade "A" by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600119

APRIL 2023



SATHYABAMA

**INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)**

Accredited with "A" grade by NAAC

Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai - 600 119

www.sathyabama.ac.in

DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of **K. NISHWIN ABISHEK (Reg. No. 39260021), R. YUGENDRAN (Reg. No. 39260046)**, who carried out the Project entitled **"NON-LINEAR DYNAMIC FLUID INTERACTION BETWEEN STAGED LIQUID PROPELLANT ROCKET ENGINE"** under our supervision from June 2022 to April 2023.

Internal Guide

Mr. R. GOKULNATH, M.Tech., Ph.D.,

Head of the Department

Dr. P. BOOMA DEVI, M.E., Ph.D.,

Submitted for Viva voce Examination held on APRIL 24th 2023

Internal Examiner

External Examiner

DECLARATION

I, K. NISHWIN ABISHAN (Reg.No. 30300221) hereby declare that the Project Report entitled "NON-LINEAR DYNAMIC FLUID INTERACTION BETWEEN STAGED LIQUID PROPELLANT ROCKET ENGINE" done by me under the guidance of Mr. R. GOWDARATHI, M.Tech., Ph.D., and is submitted in partial fulfillment of the requirements for the completion of Project.

DATE: 24.04.2023

PLACE: CHENNAI


SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my sincere thanks to **Dr. S. PRAKASH, M.E., Ph.D.**, Dean, School of Mechanical Engineering and **Dr. P. BOOMA DEVI , M.E., Ph.D.**, Head of the Department, Department of Aeronautical Engineering for providing me necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Internal Guide **Mr. R. GOKULNATH, M.Tech., Ph.D.**, for her valuable guidance, suggestions and constant encouragement paved way for the successful completion of my project work.

I also express my thanks to all Teaching and Non-teaching staff members of the Department of Aeronautical Engineering who were helpful in many ways for the completion of the project work.

ABSTRACT

The dynamics of cavitation in cryogenic fluids are significantly influenced by thermal effects. This study aims to develop an effective computational approach to simulate a cryogenic cavitation model for the application of liquid rocket combustion cavity modeling. Previously developed cavitating models are utilized and their physical implications are analyzed in relation to cryogenic fluids.

Steady state computations are carried out on three cavity models of combustion using real fluid properties of liquid nitrogen and hydrogen. The thermodynamic effects are demonstrated under conditions by considering the cavity model as the reference temperature towards the critical point.

In addition, time-dependent computations are performed for various cases of cryogenic cavitation using the algorithm. The results indicate reasonable agreement with experimental data. The simulated flow fields and the reduced order strategy of Proper Orthogonal Decomposition (POD) are employed to explain the impact of the cryogenic environment and inflow perturbations on the flow structure and instabilities.

TABLE OF CONTENTS

CHAPTER No.	TITLE	PAGE No.
	ABSTRACT	v
	LIST OF FIGURES	viii
	LIST OF TABLES	x
1	INTRODUCTION	1
	1.1 INTRODUCTION TO CAVITY WALL	1
	1.2 TYPES OF CAVITY WALL	1
2	FLUID FLOW PROPERTIES	3
	2.1 TYPES OF FLOW	3
	2.1.1 Internal Flow	3
	2.1.2 External Flow	3
	2.1.3 Laminar Flow	3
	2.1.4 Turbulent Flow	4
	2.1.5 Steady Flow	5
	2.1.6 Flow Viscous	6
	2.1.7 Inviscid Flow	6
	2.1.8 Incompressible Flow	6
	2.2 TYPES OF FLUIDS	7
	2.2.1 Compressible Flow	7
	2.2.2 Newtonian Fluid	7
	2.2.3 Non-Newtonian Fluid	7
	2.3 FLUID MODELS	8
	2.3.1 k-Epsilon Models	8
	2.3.1.1 Standard k-Epsilon Model	9
	2.3.1.2 Realisable k-Epsilon Model	9
	2.3.1.3 RNG k-Epsilon Model	10
	2.3.2 k-Omega model	11
3	AIM AND SCOPE	14
4	INTRODUCTION TO CFD	16
	4.1 CFD WORKING PRINCIPLE	16
	4.2 DISCRETIZATION	17

CHAPTER No.	TITLE	PAGE No.
	4.3 GRID GENERATION	18
	4.3.1 Types Of Grid Generation	19
	4.4 APPLICATIONS IN CFD	20
	4.5 ADVANTAGES OF CFD	22
	4.6 LIMITATIONS OF CFD	23
5	DESIGN AND ANALYSIS	25
	5.1 DESIGN AND MESHING	25
	5.2 ANALYSIS	26
	5.3 EXPERIMENTAL RESULT	36
	5.4 COMPARISON TABLE	38
6	SUMMARY AND CONCLUSION	39
	6.1 SUMMARY	39
	6.2 CONCLUSION	39
	REFERENCES	40

LIST OF FIGURES

FIGURE NO	TITLE	PAGE NO
1.1	L/D<10,open cavity wall	2
1.2	L/D<10, closed cavity wall	2
1.3	Various flow fields over cavity for different length to depth ratio,L=D	2
2.1	Laminar Flow	4
2.2	Turbulent Flow	5
5.1	meshing of 0° closed cavity wall	25
5.2	meshing of 60° open cavity wall	26
5.3	pressure contour of 0° closed cavity wall epsilon model	27
5.4	velocity streamline contour of 0° closed cavity wall epsilon model	28
5.5	turbulence eddy dissipation contour of 0° closed cavity wall Epsilon model	28
5.6	velocity contour of 0° closed cavity wall epsilon model	29
5.7	pressure contour of 0° open cavity wall epsilon model	29
5.8	turbulence eddy dissipation contour of 0° open cavity wall epsilon model	30
5.9	pressure contour of 30° open cavity wall epsilon model	31
5.10	velocity streamline contour of 30° open cavity wall epsilon model	31
5.11	velocity contour of 30° open cavity wall epsilon model	32
5.12	velocity contour of 30° open cavity wall omega model	32

5.13	The experimental rig 2D sketch	33
5.14	filling level of 30% LH2 without baffle	33
5.15	50% Filling Level Pressure Distribution	34
5.16	70% Filling Level Pressure Level	34
5.17	Baffle Geometry	35
5.18	30% Filling Level	35
5.19	Experimental Result Of Case 1	36
	Filling Level 30%	
	Excitation Period 1.7 secs	
5.20	Comparison of the Numerical and Experimental Results 30% and Excitation Period of 1.7 secs	37
5.21	Experimental and Numerical Patterns	37

LIST OF TABLES

TABLE NO	TITLE	PAGE NO
5.1	<i>Temperature Difference</i>	33
5.2	Comparison between 0°, 30°, 45°, 60°	38

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION TO CAVITY WALL:

The fuel injection and flame holding aspects are crucial in the design of a supersonic combustion ramjet (scramjet) engine. Achieving molecular-level mixing of fuel and air in the near-field of the fuel injection is essential for optimal performance. This is often accomplished by employing physical obstructions, such as a backward facing step, in the combustor to create stream-wise vortices that promote fuel-air mixing. The recirculation zone generated behind the step acts as a continuous ignition source, containing hot gases that aid in sustaining combustion. The cavity flame holder, which integrates fuel injection and flame holding, has been proposed as a new concept for flame stabilization in supersonic combustors. Developed by CIAM (Central Institution of Aviation Motors) in Moscow, this approach was used in a joint Russian/French dual-mode scramjet flight test. Experimental results have shown that using a cavity after the ramp injector significantly improves hydrocarbon combustion efficiency in supersonic flows. Similar flame stabilization methods, as demonstrated by Ben Yakar et al. in a solid-fuel supersonic combustor using poly methyl meth acrylate (PMMA), showed self-ignition and sustained combustion. The presence of a cavity on an aerodynamic surface can have a significant impact on the surrounding flow, and a rectangular cavity driven by a free shear layer provides a well-defined configuration for studying flow separation and reattachment.

1.2 TYPES OF CAVITY WALL:

There are two main types of cavity flow: open and closed. The flow fields for these types of cavities are illustrated. Open cavity flow typically occurs when the length-to-depth ratio (L/D) is less than 10.

Trans-verse mechanism to longitudinal Mechanism

$M_{\infty}=1.5$ at $l/d=2$

$M_{\infty}=2.5$ at $l/d=2\sim 3$

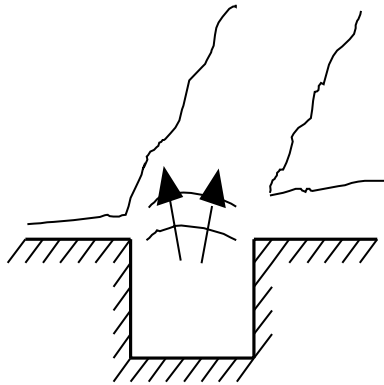


Fig: 1.1: $L/D < 10$, open cavity wall wall

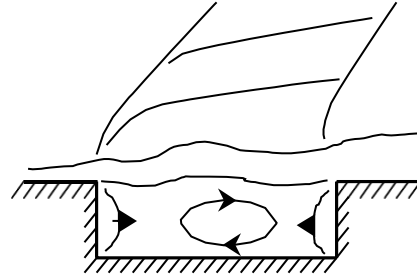


Fig: 1.2: $L/D > 10$, closed cavity

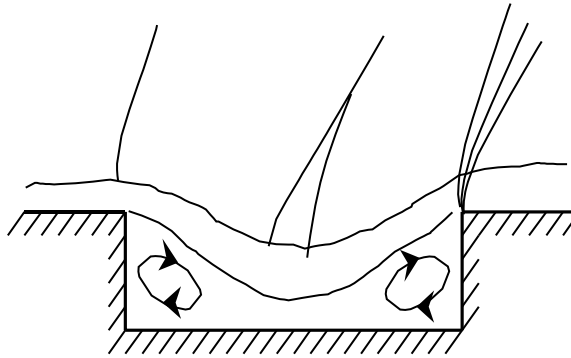


Fig: 1.3: Various flow fields over cavity for different length to depth ratio, $L=D$.

CHAPTER- 2

FLUID FLOW PROPERTIES

2.1 TYPES OF FLOW:

2.1.1 Internal Flow:

Internal flow refers to the flow of fluid within the boundaries of enclosed conduits such as pipes, channels, or ducts. It is typically controlled using valves, fans, pumps, or other mechanical devices. Examples of internal flow include pipe flow, where fluid flows through pipes of varying diameters and lengths, channel flow, where fluid flows between parallel walls, and airflow in ducts, which is commonly used in ventilation and air conditioning systems. Internal flow is an important aspect of many engineering applications and is studied extensively in fields such as fluid mechanics, heat transfer, and HVAC (Heating, Ventilation, and Air Conditioning) systems design.

2.1.2 External Flow:

External flow refers to the flow of fluid outside of a boundary or body, such as the flow over aircraft wings, flow around immersed bodies like ships or submarines, air flow around buildings, and airflow around cars. External flow is an important phenomenon in various engineering and physics applications, such as aerodynamics, hydrodynamics, and environmental modelling. Understanding and predicting external flow behaviour is crucial in designing efficient and effective systems, such as aircraft wings for optimal lift and drag, streamlined shapes for reduced drag on vehicles, and optimized building designs for energy efficiency. External flow is typically influenced by factors such as velocity, pressure, temperature, and turbulence, and is often studied using experimental, computational, and theoretical methods.

2.1.3 Laminar Flow:

The velocity of the flowing fluid is an important factor in classifying different types of flow. Laminar flow, also known as streamline flow occurs when fluid particles move in smooth, parallel layers without much mixing. In laminar flow, fluid

particles move in straight lines along the direction of flow, and there is minimal random motion or turbulence. Laminar flow is characterized by a well-defined flow pattern and is typically observed at low velocities or in small-scale systems.

On the other hand, turbulent flow is characterized by irregular, chaotic motion of fluid particles in all directions, with no uniform motion at the local level but an average velocity in the direction of flow. Turbulent flow is typically observed at higher velocities or in large-scale systems where there are disturbances or obstacles that disrupt the smooth flow of the fluid. Turbulent flow is characterized by fluctuations in velocity, pressure, and other fluid properties, and it often results in increased mixing, heat transfer, and pressure drop compared to laminar flow.

The transition between laminar and turbulent flow can occur based on various factors such as Reynolds number, which is a dimensionless parameter that describes the ratio of inertial forces to viscous forces in a flowing fluid. Understanding the type of flow, whether laminar or turbulent, is important in various engineering applications, such as designing pipes, ducts, and other fluid flow systems, as the behaviour of the fluid can significantly affect the performance and efficiency of the system.



Fig: 2.1: Laminar Flow

2.1.4 Turbulent Flow:

The Reynolds number (Re) is a dimensionless parameter that describes the ratio of inertial forces to viscous forces in a flowing fluid, and it is often used to determine the type of flow, whether it is laminar, turbulent, or in the transitional /critical region.

For pipe flow, the Reynolds number is given by the formula $Re = (v.d.\rho) / \mu$, where,

v is the mean velocity of the fluid,
 d is the diameter of the pipe,
 ρ is the density of the fluid,
 μ is the dynamic viscosity of the fluid.

According to the Reynolds number criterion you mentioned, if the Reynolds number is less than 2000, the flow is generally laminar, and if it is greater than 4000, the flow is generally turbulent. However, in the transitional or critical region between 2000 and 4000, the flow behaviour can vary and may exhibit characteristics of both laminar and turbulent flow, depending on the specific conditions and properties of the fluid and the system.

This criterion is commonly used to estimate the flow behaviour in pipes or other conduits, and it applies to both liquids and gases. It is a useful guideline in many engineering applications to predict the type of flow and design appropriate systems for optimal performance and efficiency.



Fig: 2.2: Turbulent flow

2.1.5 Steady Flow:

Steady flow refers to a type of fluid flow in which the velocity, pressure, and other flow parameters at any given point in the flow field remain constant over time. In other words, the flow does not change with time, and the flow parameters do not vary with time.

On the other hand, unsteady or non-steady flow refers to a type of fluid flow in which the velocity, pressure, and other flow parameters change with time. This can occur due to various factors such as changes in boundary conditions, time-varying forces, or fluid properties.

In many engineering applications, it is common to assume steady flow conditions for simplicity and ease of analysis. However, in reality, many flows are unsteady, especially in dynamic systems or when there are time-varying factors

affecting the flow. It is important to consider the appropriate type of flow, whether it is steady or unsteady, in order to accurately predict and analyse the behaviour of the fluid in a given system or situation.

$$\frac{\partial \mathbf{u}}{\partial t} = 0.$$

2.1.6 Flow Viscous:

Viscous flows refer to fluid flows in which the flow patterns are significantly influenced by the viscous properties of the fluid. Viscosity is a measure of a fluid's resistance to shear deformation or flow, and it plays a crucial role in determining the behaviour of fluid flow, especially in regions where velocity gradients are large. In fluids with high viscosity, such as liquids or highly viscous gases, the velocity gradients near the walls of pipes or other solid boundaries can be significant. This is because the fluid in contact with the solid boundary adheres to it and moves with zero velocity, while the fluid farther away from the boundary moves at higher velocities. This velocity difference results in large velocity gradients, and viscous effects become dominant in these regions. Viscous flows can exhibit various flow patterns, such as laminar or turbulent flows, and the behaviour of the fluid near the walls can significantly affect the overall flow behaviour and pressure drop in the system. Understanding and analysing viscous flows are important in many engineering applications, such as in pipe flow, boundary layer flows over solid surfaces, lubrication flows, and many other practical scenarios where the viscous properties of the fluid play a significant role in determining the flow behaviour.

2.1.7 Inviscid Flow:

In situations where viscosity does not play a major role, the term used to describe the flow is "inviscid flow". This type of flow is commonly observed in the central area of pipes with flowing fluids and in the case of gas flows.

2.1.8 Incompressible Flow

In most operational conditions, liquids are considered to be nearly incompressible, whether it's in open channels or high-pressure hydraulic systems.

Hence, flows involving liquids are commonly referred to as "incompressible flows". However, there are rare instances, such as during water hammer events, when liquids do not behave as incompressible fluids.

A fluid is incompressible if the divergence of $\mathbf{u}$ is zero. That is, if $\mathbf{u}$ is a solenoidal vector field.

$$\nabla \cdot \mathbf{u} = 0.$$

2.2 TYPES OF FLUIDS:

2.2.1 Compressible fluid:

Gases, being compressible fluids, typically exhibit compressible flows. When dealing with gas flows, it's important to take into account the effects of density variation along the flow paths, as this can impact operational parameters like head loss. In cases where gas velocities and pressure drops are relatively low, they can still be considered as compressible flows, requiring special consideration for accurate analysis and prediction of flow behavior.

2.2.2 Newtonian Fluid:

A Newtonian fluid, named after the renowned physicist Isaac Newton, is characterized by a linear stress versus strain rate curve that passes through the origin. The viscosity of the fluid is represented by the constant of proportionality in this linear relationship.

A simple equation to describe Newtonian fluid behavior is

$$\tau = \mu \frac{du}{dy}$$

2.2.3 Non-Newtonian Fluid:

Non-Newtonian fluids are fluids whose flow properties deviate from those of Newtonian fluids. Unlike Newtonian fluids, the viscosity of non-Newtonian fluids can depend on factors such as shear rate or shear rate history. While most non-Newtonian fluids exhibit viscosity that varies with shear rate, there are also non-Newtonian fluids that have shear-independent viscosity, but may still display other

non-Newtonian behaviours, such as normal stress-differences or other unique characteristics.

2.3 FLUID MODELS:

2.3.1 *K-Epsilon Models:*

The K-epsilon model is a commonly used turbulence model, but it may not perform well in situations with large adverse pressure gradients (as referenced in Source 4). It is a two-equation model, meaning it includes two additional transport equations to represent the turbulent characteristics of the flow. This allows the model to account for history effects such as convection and diffusion of turbulent energy.

The first transported variable in the K-epsilon model is the turbulent kinetic energy, denoted as "k". The second transported variable is the turbulent dissipation, denoted as "epsilon". The turbulent dissipation determines the scale of the turbulence, while the turbulent kinetic energy represents the energy present in the turbulence.

There are several variations of the K-epsilon turbulence model commonly used in computational fluid dynamics (CFD) simulations Including:

Standard k-epsilon model: This is the original and most basic form of the K-epsilon model, widely used in many applications. It assumes isotropic turbulence and uses empirical constants to model the turbulence production, dissipation, and transport.

Realizable k-epsilon model: This is an improved version of the standard K-epsilon model that addresses some of its limitations. It incorporates additional constraints to ensure that the modelled turbulence is physically realizable, i.e., it does not produce unphysical or unrealistic results.

RNG k-epsilon model: RNG stands for "Reynolds Number Gradient" and is a modification of the standard K-epsilon model that uses a different method for modelling the turbulence transport. It is known to provide better accuracy and stability in some cases, particularly for flows with adverse pressure gradients.

These are just a few examples of the various K-epsilon turbulence models available in CFD simulations, each with its own strengths and limitations. The choice of which model to use depends on the specific flow conditions and accuracy requirements of the simulation.

2.3.1.1 Standard k-epsilon model:

Transport equations for standard k-epsilon model

For turbulent kinetic energy k ,

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (P_k + C_{3\epsilon} P_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon$$

For dissipation ϵ ,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

2.3.1.2 Realisable k-epsilon model:

Transport Equations

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_j}(\rho\epsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu\epsilon}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} P_b + S_\epsilon$$

Where,

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{k}{\epsilon}, \quad S = \sqrt{2S_{ij}S_{ij}}$$

In the equations mentioned, "generation of turbulence kinetic energy due to mean velocity gradients" is denoted by a term, usually represented as G-k or G in the K-epsilon turbulence models. It accounts for the production or generation of turbulence kinetic energy in the flow due to the velocity gradients of the mean flow. This term is calculated using similar methods as the standard K-epsilon model, which employs empirical correlations or closure coefficients to estimate the

generation of turbulence kinetic energy.

Similarly, "generation of turbulence kinetic energy due to buoyancy" is denoted by a term, often represented as G_b or B in the K-epsilon turbulence models. It accounts for the production or generation of turbulence kinetic energy in the flow due to buoyancy effects, typically arising from temperature gradients. This term is also calculated using methods similar to the standard K-epsilon model, which use empirical correlations or closure coefficients to estimate the impact of buoyancy on turbulence kinetic energy generation.

Both of these terms, G_k and G_b , contribute to the overall turbulence kinetic energy budget in the K-epsilon turbulence models and are important for accurately predicting turbulence behaviour in flows with velocity gradients and buoyancy effects.

2.3.1.3 RNG k-epsilon model:

The RNG (Re-Normalisation Group) k-epsilon turbulence model was developed by Yakar et al. as an improvement over the standard k-epsilon model. The standard k-epsilon model calculates the eddy viscosity based on a single turbulence length scale, which means that only the turbulence at that specific scale is considered in the calculation of turbulent diffusion. However, in reality, turbulence occurs at various scales of motion, and all scales contribute to the overall turbulent diffusion.

The RNG approach uses mathematical techniques from the field of Re-Normalisation Group to derive a modified form of the epsilon equation in the k-epsilon model. This modified equation aims to account for the effects of smaller scales of motion on the turbulence behaviour. The production term in the epsilon equation is specifically modified in the RNG model to better represent the different scales of motion, and it attempts to capture the impact of these scales on the turbulence diffusion.

In summary, the RNG k-epsilon model is a modification of the standard k-epsilon model that uses Re-Normalisation Group methods to better account for the effects of different scales of motion on turbulence behaviour, particularly in the calculation of turbulent diffusion. It provides a more accurate representation of the turbulence characteristics in flows with varying scales of motion.

Transport Equations

There are many number of ways to write the transport equations for k and simple interpretation where buoyancy is neglected is

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} P_k - C_{2\epsilon}^* \rho \frac{\epsilon^2}{k}$$

Where,

$$C_{2\epsilon}^* = C_{2\epsilon} + \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3}$$

In the RNG k-epsilon model, the calculation of the turbulent viscosity is done in a similar manner as the standard k-epsilon model. The eddy viscosity is typically calculated using the approximation, which assumes that the Reynolds stress is proportional to the mean velocity gradients, and it is determined based on the turbulence lengthscale and turbulence kinetic energy.

The main difference between the standard k-epsilon model and the RNG k-epsilon model lies in the modified form of the epsilon equation, which accounts for the effects of smaller scales of motion through changes to the production term. The turbulent viscosity, however, is still calculated in a similar manner using the turbulence length scale and turbulence kinetic energy, as in the standard k-epsilon model. It's worth noting that the RNG k-epsilon model is considered to be an improvement over the standard k-epsilon model in capturing the effects of smaller scales of motion on turbulence behavior, particularly in flows with adverse pressure gradients or complex geometries. However, the specific implementation and performance of turbulence models can vary depending on the application and flow conditions, and it's always important to validate the model against experimental or numerical data to ensure accurate results.

2.3.2 k-Omega Models:

The K-omega model is a widely used turbulence model that belongs to the family of two-equation turbulence models. It includes two additional transport equations to represent the turbulent properties of the flow, allowing it to account for history effects such as convection and diffusion of turbulent energy. The first

transported variable in the K-omega model is the turbulent kinetic energy (k), which represents the energy associated with turbulence in the flow. The second transported variable is the specific dissipation (ω), which determines the scale of the turbulence. The turbulent dissipation represents the rate at which turbulence is being destroyed or dissipated in the flow. The K-omega model is known for its ability to handle a wide range of flows, including those with adverse pressure gradients, separation, and complex geometries. It has been extensively used in various engineering applications, such as aerodynamics, heat transfer, and combustion simulations. It's important to note that different turbulence models, including the K-omega model, have their own strengths and weaknesses, and the choice of the appropriate turbulence model depends on the specific flow conditions and the accuracy required for the simulation. It's always essential to validate the model against experimental or numerical data to ensure reliable results.

Wilcox's k-omega model, also known as the Wilcox's turbulence model, is a popular variation of the K-omega turbulence model. It is a two-equation model that includes two additional transport equations to represent the turbulent properties of the flow. The variables used in Wilcox's k-omega model are:

1. Kinematic Eddy Viscosity (ν_t): It represents the eddy viscosity, which is a measure of the turbulent mixing in the flow. It is calculated based on the turbulence kinetic energy (k) and the specific dissipation rate (ω) in the model.
2. Turbulence Kinetic Energy (k): It represents the energy associated with turbulence in the flow. It is calculated based on the velocity gradients and the dissipation of turbulence.
3. Specific Dissipation Rate (ω): It represents the rate at which turbulence is being destroyed or dissipated in the flow. It is calculated based on the velocity gradients and the dissipation of turbulence.

Wilcox's k-omega model has been widely used in various engineering simulations, particularly in aerodynamics and heat transfer applications. It is known for its ability to accurately predict turbulence behaviour in flows with adverse pressure gradients and separation, making it suitable for a wide range of practical applications. As with any turbulence model, it is important to validate Wilcox's k-

omega model against experimental or numerical data to ensure its reliability and accuracy in a specific flow scenario.

Kinetic Eddy Viscosity

$$\nu_T = \frac{k}{\omega}$$

Turbulence Kinetic Energy

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma^* \nu_T) \frac{\partial k}{\partial x_j} \right]$$

Specific Dissipation Rate

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma \nu_T) \frac{\partial \omega}{\partial x_j} \right]$$

CHAPTER 3

AIM AND SCOPE

3.1 INTRODUCTUON TO COMBUSTION CHAMBER DESIGN:

The intricate flow field inside the scramjet engine's combustor presents a significant challenge in designing and developing an optimized supersonic combustor geometry. The desired outcome is to ensure sufficient mixing of fuel and air, enabling the desired chemical reactions and heat release to occur within the residence time of the fuel-air mixture. Achieving this goal requires a thorough understanding of fuel injection processes, as well as a comprehensive knowledge of the supersonic mixing and combustion processes, and the factors that impact combustor losses.

The designer must prioritize three key objectives:

- (i) achieving effective and rapid fuel-air mixing
- (ii) minimizing total pressure losses
- (iii) maximizing combustion efficiency.

The intricate process of supersonic combustion encompasses turbulent mixing, shock interaction, and heat release in a supersonic flow. Recent experiments conducted in the T4 shock tunnel have successfully demonstrated the concept of injecting and mixing in the scramjet inlet, rather than the combustion chamber. This approach utilizes localized shock structures in the combustor to achieve ignition, even when the mean flow conditions are relatively mild. The potential outcome of this technique is the development of shorter and lighter scramjets with improved efficiency.

The supersonic inflow is compressed to achieve a desired higher pressure and temperature at the exit of the inlet, which triggers combustion in the downstream chamber. The resulting reacted gas then expands in the nozzle to generate thrust. The inlet and combustor are crucial components in the mechanism of a scramjet engine, as the combustion process is highly dependent on the temperature of the airflow compressed by the inlet. It is essential for the combustion to occur efficiently within the limited time and space provided by the combustor, as this directly affects the weight of the structure and the overall drag due to skin friction on the combustor's surface. Efficient combustion in the

combustor is crucial for optimal performance of the scramjet engine.

The advancement of scramjet technology has reached a stage where exploring sophisticated geometries is essential to achieve increased performance, particularly in terms of net positive thrust, which is critical for access-to-space systems. However, designing a scramjet engine presents significant challenges to conventional approaches due to the intricate and interrelated nature of the flow fields involved. These flow fields encompass a wide range of aerodynamic and aero-thermal phenomena, such as shock wave/boundary layer interactions, flow separation, and chemical reactions. The high degree of coupling among these phenomena necessitates an integrated approach to design the components of the scramjet engine, rather than considering them in isolation. This integrated design approach is crucial to tackle the complexity and optimize the performance of scramjet engines. When operating at supersonic flight speeds, it is crucial to maintain the flow entering the combustor at a supersonic velocity to prevent excessive heating and dissociation of air. In hypersonic engines, the residence time of air in the combustor is typically on the order of 1 millisecond under typical flight conditions. Within this extremely short timeframe, the fuel must be efficiently injected, mixed with air, and burned completely. To address this challenge, numerous studies have been conducted globally, leading to various suggested concepts for scramjet combustor configurations that can overcome the limitations imposed by the brief flow residence time. These efforts aim to optimize the fuel-air mixing and combustion processes within the combustor to achieve efficient and effective operation of hypersonic engines.

This paper presents comprehensive numerical studies that investigate the flow field characteristics of different cavity wall configurations, including open and closed cavity walls, with varying angles of 0° , 30° , 45° , and 60° . Computational simulations were conducted to analyze the flow behavior and characteristics associated with these cavity wall geometries.

CHAPTER 4

INTRODUCTION TO CFD

Computational fluid dynamics (CFD) is a scientific field that involves predicting various phenomena associated with fluid flow, such as heat transfer, mass transfer, chemical reactions, etc., by solving mathematical equations governing these processes using numerical methods. The results of CFD analyses provide relevant engineering data, which is used in various stages of product development, including conceptual studies of new designs, detailed product development, troubleshooting, and redesign.

CFD analysis is complementary to laboratory testing and experimentation, as it helps reduce the total effort required in the laboratory by providing valuable insights into the behavior of fluid flow under different conditions.

4.1 CFD WORKING PRINCIPLE:

The process of CFD analysis typically involves the following steps:

1. **Mathematical model:** Analysis begins with the formulation of a mathematical model that represents the physical problem of interest. This model includes the conservation equations for mass, momentum, and energy that must be satisfied throughout the region of interest.
2. **Empirical modeling:** Fluid properties, such as viscosity and thermal conductivity, are modeled empirically based on experimental data or established correlations.
3. **Simplifying assumptions:** Simplifying assumptions are made to make the problem tractable, such as assuming steady-state, incompressible, inviscid, or two-dimensional flow, depending on the specific problem being analyzed.
4. **Initial and boundary conditions:** Appropriate initial and boundary conditions are specified to define the starting conditions and constraints for the problem.

5. Numerical methods: CFD applies numerical methods, known as discretization, to approximate the governing equations of fluid mechanics in the fluid region of interest. This involves dividing the region into a collection of cells or grid, and solving the resulting set of algebraic equations numerically on a computer.

6. Solution of equations: The set of algebraic equations, resulting from the discretization, are solved numerically at each node or cell in the grid to obtain the values of flow field variables, such as velocity, pressure, and temperature, in the region of interest.

The results of the CFD analysis provide insights into the behavior of the fluid flow and help in understanding the physical phenomena associated with the problem under investigation.

After solving the system of algebraic equations, the resulting solution is post-processed to extract quantities of interest. This typically involves analyzing the computed flow field variables, such as velocity, pressure, and temperature, to obtain relevant engineering data. Examples of quantities of interest in CFD analysis include lift and drag forces on an aerodynamic body, torque on a rotating component, heat transfer rates, flow separation points, pressure losses, and other performance metrics.

Post-processing in CFD analysis may involve visualization techniques, such as contour plots, vector plots, and streamline plots, to visually represent the flow field and understand the flow behavior. It may also involve computation of derived quantities, such as vorticity, turbulence statistics, and other relevant parameters, to gain insights into the flow characteristics.

The extracted quantities of interest are then analyzed and compared with design specifications or experimental data to assess the performance of the system or component being analyzed. This information can be used for conceptual studies of new designs, detailed product development, troubleshooting, and redesign, among other engineering applications.

4.2 DISCRETIZATION:

In CFD analysis, the domain of interest, which could be a fluid region, is discretized into a finite set of control volumes or cells. This discretized domain is

commonly referred to as the "grid" or "mesh". The grid can be structured (such as Cartesian or cylindrical grids) or unstructured (such as triangular or tetrahedral grids), depending on the complexity of the geometry and flow field being modeled.

The governing conservation equations, such as those for mass, momentum, energy, and other relevant quantities, are discretized into algebraic equations using numerical methods. This typically involves approximating the derivatives in the partial differential equations with finite difference, finite volume, or finite element methods, among others, to obtain a set of algebraic equations that represent the behavior of the flow within each cell or control volume.

Once the algebraic equations are formulated, they are solved numerically, typically using iterative methods, to obtain the flow field variables (such as velocity, pressure, and temperature) at each node or cell in the grid. The solution process may involve multiple iterations until a converged solution is obtained, where the residuals of the equations are minimized to a desired tolerance.

The resulting flow field solution provides insights into the behavior of the fluid in the domain of interest, including the distribution of various flow variables and their interactions, and is used for further analysis, post-processing, and interpretation of the results.

4.3 GRID GENERATION:

In the discretization process of CFD analysis, the continuous system is converted into a discrete one by constructing a grid or mesh, where the approximate solution is sought at discrete grid points. The type of mesh used can have a significant impact on the accuracy and efficiency of the numerical solution.

There are generally two types of meshes used in CFD: structured and unstructured meshes.

Structured Mesh: In a structured mesh, the grid points have a regular connectivity, meaning that each grid point has the same number of neighbors. Structured meshes are typically used for simple geometries, such as Cartesian or cylindrical grids, where the geometry can be represented by a regular arrangement of grid points. Structured meshes are known for their regularity, which can result in

a higher degree of numerical accuracy and efficiency, especially for cases with uniform flow or regular geometries.

Unstructured Mesh: In an unstructured mesh, the grid points have an irregular connectivity, meaning that each grid point can have a different number of neighbors. Unstructured meshes are typically used for complex geometries, where the geometry cannot be represented by a regular arrangement of grid points. Unstructured meshes offer more flexibility in capturing complex geometries and can be more computationally efficient for such cases compared to structured meshes. They are commonly used in simulations involving complex geometries, moving boundaries, or adaptive mesh refinement.

Both structured and unstructured meshes have their advantages and limitations, and the choice of mesh type depends on the specific problem being solved, the geometry of the domain of interest, and the accuracy and efficiency requirements of the simulation.

4.3.1 Types Of Grid Generation:

Those are some common methods used for generating structured and unstructured meshes in CFD simulations.

1. STRUCTURED MESH:

Boundary-fitted meshes: In a boundary-fitted mesh, the grid points are generated along the boundaries of the geometry in such a way that the grid lines are aligned with the geometry. This approach ensures that the grid points are well-distributed near the boundaries, which can be particularly useful for capturing boundary layer flow and other flow phenomena near the geometry. Common methods for generating boundary-fitted meshes include elliptic grid generation, hyperbolic grid generation, and O-grid or C-grid generation.

Algebraic grid generation: In this approach, the coordinates of the grid points are generated algebraically based on mathematical equations or functions. Algebraic grid generation methods are typically used for generating structured meshes in simpler geometries, such as Cartesian or cylindrical grids.

2. UNSTRUCTURED MESH:

Surface meshing: In unstructured meshing, the first step is typically to generate a surface mesh that represents the geometry of the object being simulated. Surface meshing methods involve discretizing the surface of the object into small triangular or quadrilateral elements.

Common methods for surface meshing include patch-based methods, advancing front methods, and octree-based methods.

Advancing front method: The advancing front method is a widely used technique for generating unstructured meshes. It involves starting with a seed point or seed edge on the geometry surface and iteratively growing the mesh by adding new nodes and elements in a front that advances away from the seed point or seed edge. This method allows for efficient meshing of complex geometries with varying element sizes and shapes.

Delaunay triangulation: Delaunay triangulation is a popular method for generating unstructured meshes, particularly for 2D and 3D problems. It is based on a set of mathematical rules that ensure the resulting mesh is of good quality, with well-shaped elements. Delaunay triangulation methods are commonly used for generating meshes for complex geometries with irregular shapes and boundaries.

The choice of mesh generation method depends on the specific geometry of the problem, the desired level of accuracy and efficiency, and the capabilities of the CFD software being used. Each mesh generation method has its advantages and limitations, and experienced CFD practitioners may choose different methods depending on the specific requirements of their simulations.

4.4 APPLICATION IN CFD:

Computational Fluid Dynamics (CFD) is widely used in various industrial applications to simulate and analyse fluid flow, heat transfer, and related phenomena. Some common applications of CFD include:

1. Flow and heat transfer in industrial processes: CFD is used to study and optimize flow and heat transfer in a wide range of industrial processes such as boilers, heat exchangers, combustion equipment, pumps, blowers, and piping systems. CFD simulations can provide insights into the performance, efficiency,

and safety of these processes, leading to improved designs and operational strategies.

2. Aerodynamics of vehicles and missiles: CFD is extensively used in the automotive, aerospace, and defence industries to analyse the aerodynamics of ground vehicles, aircraft, and missiles. CFD simulations can help in optimizing the aerodynamic performance of vehicles, reducing drag, improving fuel efficiency, and enhancing stability and handling characteristics.

3. Film coating and thermo forming in material processing: CFD is used to study and optimize processes such as film coating and thermoforming in material processing applications. CFD simulations can help in understanding the flow behaviour of liquids or gases during coating processes, predicting coating thicknesses, optimizing process parameters, and improving product quality.

4. Flow and heat transfer in propulsion and power generation systems: CFD is used to study and optimize flow and heat transfer in propulsion systems such as engines, turbines, and rockets, as well as in power generation systems such as nuclear reactors and renewable energy systems. CFD simulations can provide insights into the performance, efficiency, and safety of these systems, leading to improved designs and operational strategies.

5. Ventilation, heating, and cooling flows in buildings: CFD is used to analyse and optimize ventilation, heating, and cooling flows in buildings to ensure comfort, energy efficiency, and indoor air quality. CFD simulations can help in designing optimal HVAC (Heating, Ventilation, and Air Conditioning) systems, studying the effectiveness of natural ventilation strategies, and predicting thermal comfort and air quality in indoor environments.

6. Chemical vapour deposition (CVD) for integrated circuit manufacturing: CFD is used to study and optimize chemical vapour deposition (CVD) processes used in the manufacturing of integrated circuits. CFD simulations can help in understanding the complex fluid flow and heat transfer phenomena during CVD processes, optimizing process parameters, and improving film quality and uniformity.

7. Heat transfer for electronics packaging applications: CFD is used to study and optimize heat transfer in electronics packaging applications, such as heat sinks, electronic enclosures, and printed circuit boards. CFD simulations can help in

designing efficient cooling solutions, predicting temperature distributions in electronic components, and optimizing thermal management strategies for electronic devices. These are just a few examples of the wide range of applications where CFD is used to analyse and optimize fluid flow, heat transfer, and related phenomena in various industries. CFD simulations can provide valuable insights and help in improving designs, processes, and operational strategies, leading to enhanced performance, efficiency, and safety in industrial applications.

4.5 ADVANTAGES OF CFD:

Computational Fluid Dynamics (CFD) offers several advantages as a numerical simulation tool for studying fluid flow, heat transfer, and related phenomena:

1. **Relatively low cost:** CFD simulations are generally more cost-effective compared to experimental methods. Conducting experiments in real-world conditions can be expensive, especially when dealing with high temperatures, high pressures, or hazardous environments. CFD simulations provide a cost-effective alternative to study and analyze complex fluid flow and heat transfer problems without the need for expensive physical experiments.

2. **Ability to simulate real conditions:** CFD allows for the simulation of real-world conditions that may be difficult or impractical to replicate in experiments. CFD simulations can accurately represent complex geometries, boundary conditions, and environmental conditions, providing insights into how fluid flows and heat transfer occur in real-world situations. This enables engineers and researchers to study and optimize systems under realistic operating conditions.

3. **Speed:** CFD simulations can be conducted relatively quickly compared to experimental methods. Once the numerical model is set up, CFD simulations can be run on powerful computers, allowing for rapid simulations and analysis of different scenarios. This makes CFD a time-efficient tool for exploring multiple design options and optimizing system performance.

4. **Ability to simulate ideal conditions:** CFD simulations allow for the study of idealized scenarios, where certain parameters can be controlled or manipulated to isolate specific effects. This provides researchers and engineers with the flexibility to study hypothetical scenarios or ideal conditions, which may not be achievable or feasible in experiments. This can help in understanding fundamental principles and

optimizing system performance.

5. Comprehensive information: CFD simulations provide detailed information about fluid flow patterns, temperature distributions, pressure gradients, and other relevant parameters. This comprehensive information can help in gaining insights into the behavior of complex systems, identifying potential issues or limitations, and optimizing system designs. CFD simulations can also provide visual representations, such as velocity vectors, contour plots, and animations, which aid in interpreting the results and communicating findings to stakeholders.

Overall, the advantages of CFD, such as its relatively low cost, ability to simulate real conditions, speed, ability to simulate ideal conditions, and comprehensive information, make it a valuable tool for studying fluid flow, heat transfer, and related phenomena in a wide range of industrial applications.

4.6 LIMITATIONS OF CFD:

Physical models, numerical models, and boundary conditions are important components in the process of conducting computational fluid dynamics (CFD) simulations.

Physical models: Physical models are mathematical representations of the real-world physical phenomena being studied. They describe the behavior of fluids, such as fluid flow, heat transfer, and other relevant physical processes, based on established physical principles, equations, and laws. Physical models provide the foundation for developing numerical models in CFD simulations and are used to represent the behavior of fluids in the virtual environment.

Numerical models: Numerical models are mathematical approximations of the physical models that are used in CFD simulations. These models are discretized representations of the physical domain, such as a volume or surface, using a grid or mesh. Numerical methods, such as finite difference, finite volume, or finite element methods, are applied to the discretized domain to obtain numerical solutions for the physical quantities of interest, such as velocity, temperature, and pressure.

Numerical models are used to numerically solve the governing equations of fluid mechanics and heat transfer to obtain predictions of fluid behavior in the virtual domain.

Boundary conditions: Boundary conditions are used to define the interactions between the fluid domain and its surroundings in CFD simulations. They specify the values of relevant physical quantities, such as velocity, temperature, and pressure, at the boundaries of the numerical domain. Boundary conditions are essential for accurately simulating the real-world conditions and behaviors of fluid flow and heat transfer. They can represent inflow or outflow of fluid, solid walls, heat sources or sinks, and other relevant physical interactions. Appropriate boundary conditions are crucial for obtaining accurate and reliable results from CFD simulations.

In summary, physical models provide the foundation for developing numerical models, which are used to discretize and solve the governing equations of fluid mechanics and heat transfer in CFD simulations. Boundary conditions are used to define the interactions between the fluid domain and its surroundings, and they play a critical role in accurately representing real-world conditions and obtaining reliable results from CFD simulations.

CHAPTER 5

DESIGN AND ANALYSIS

5.1 DESIGN AND MESHING:

Creating a computational mesh that precisely depicts the geometry of the cavity and the surrounding fluid domain is required for meshing of a zero degree closed cavity wall.

The mesh resolution should be adequate to capture the necessary flow characteristics and boundary layers. Various meshing tools and approaches were utilised by researchers to mimic natural convection in cavities, lid-driven cavity flow, and pressure instabilities in stator-rotor disc systems.

For example, with ANSYS Fluent, one may model a wall with a hollow inside and mesh the air-gap inside to estimate heat convection.

Creating a block structure of the mesh for the cavity using a CFD approach capable of identifying cavity flow instabilities may be part of the meshing process.

The meshing instrument and technique used may be determined by the application.

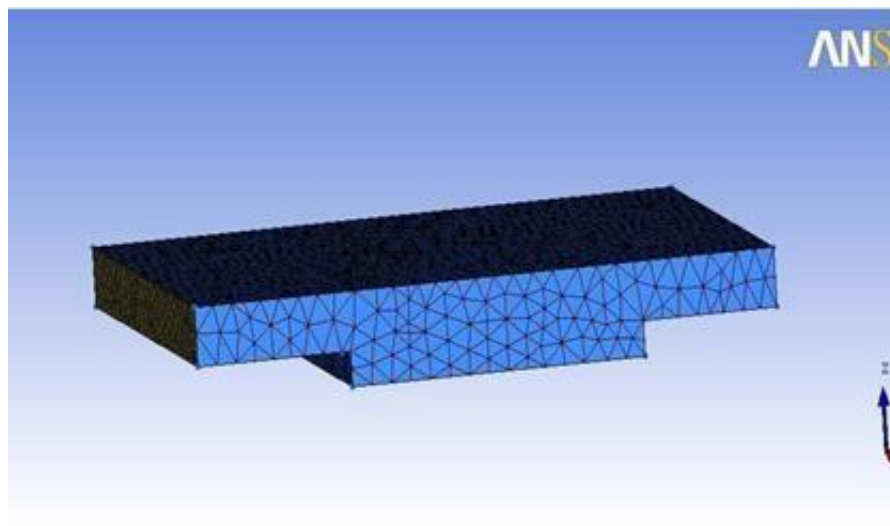


Fig: 5.1: meshing of 0° closed cavity wall

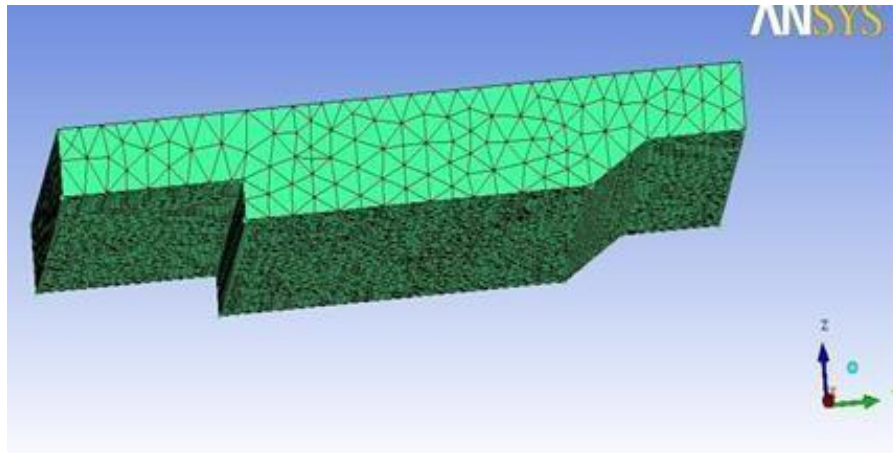


Fig: 5.2: meshing of 60° open cavity wall

5.2 ANALYSIS:

CLOSED CAVITY WALL:

The pressure contour of a 0° closed cavity wall epsilon model in CFD may vary depending on parameters such as cavity shape, flow conditions, and the individual epsilon model utilised.

Closed cavities are distinguished by an increasing static pressure distribution over their length.

For wall-bonded turbulent flows, the k-epsilon model has been suggested, and it works well for turbulent boundary layers with zero pressure gradients.

The wall static pressure contour for supersonic cold cavity flow has been displayed, demonstrating that the expected results are often grid independent.

To effectively forecast the pressure contour of a 0° closed cavity wall epsilon model in CFD, numerical simulations employing suitable models and boundary conditions are required.

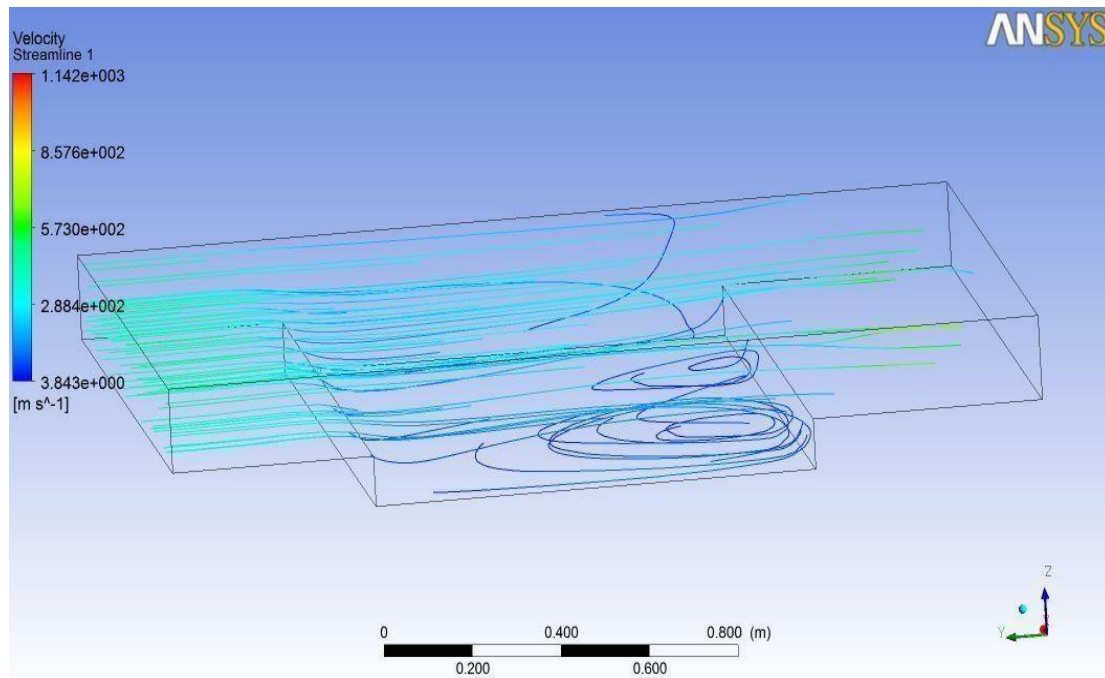


Fig: 5.3: pressure contour of 0° closed cavity wall epsilon model

The velocity streamline contour of a 0° closed cavity wall epsilon model in CFD may be affected by a number of factors, including cavity shape, flow conditions, and the individual epsilon model utilised.

The researchers ran numerical simulations of cavities with varying aspect ratios using the ANSYS FLUENT 14 CFD system and showed the streamline outlines.

For wall-bonded turbulent flows, the k-epsilon model has been suggested, and it works well for turbulent boundary layers with zero pressure gradients.

Closed cavities are distinguished by an increasing static pressure distribution over their length.

CFD analysis was utilised by researchers to investigate the pressure and flow properties of cavities.

A 0° closed cavity wall epsilon model's velocity streamline shape may be modified.

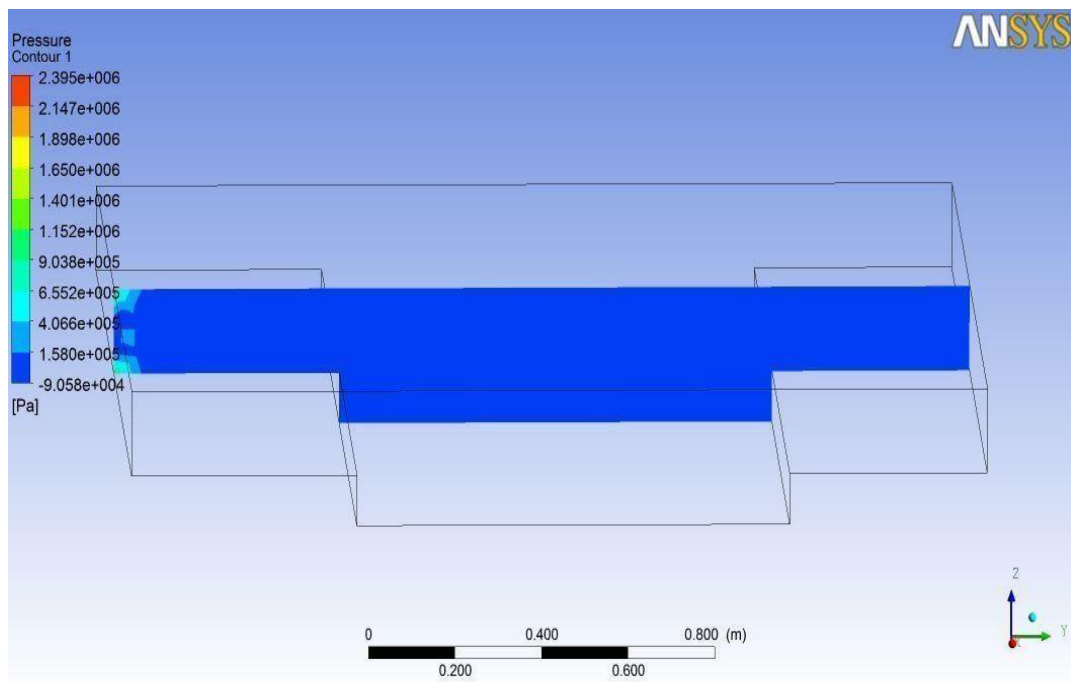


Fig: 5.4: velocity streamline contour of 0° closed cavity wall epsilon model

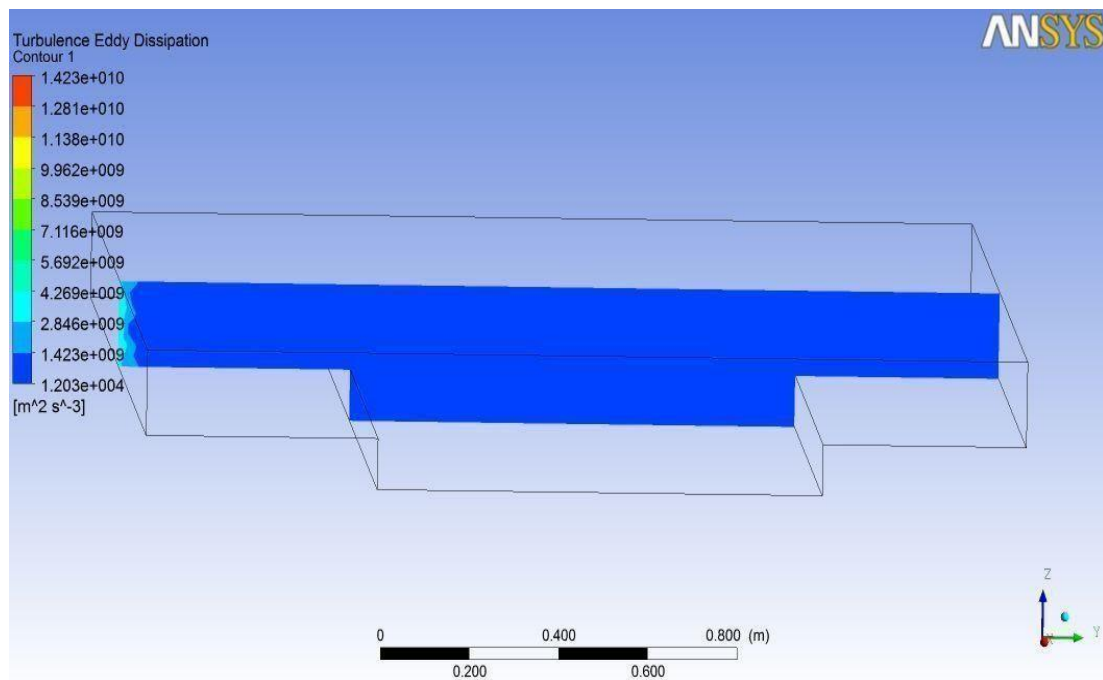


Fig: 5.5: turbulence eddy dissipation contour of 0° closed cavity wall Epsilon model

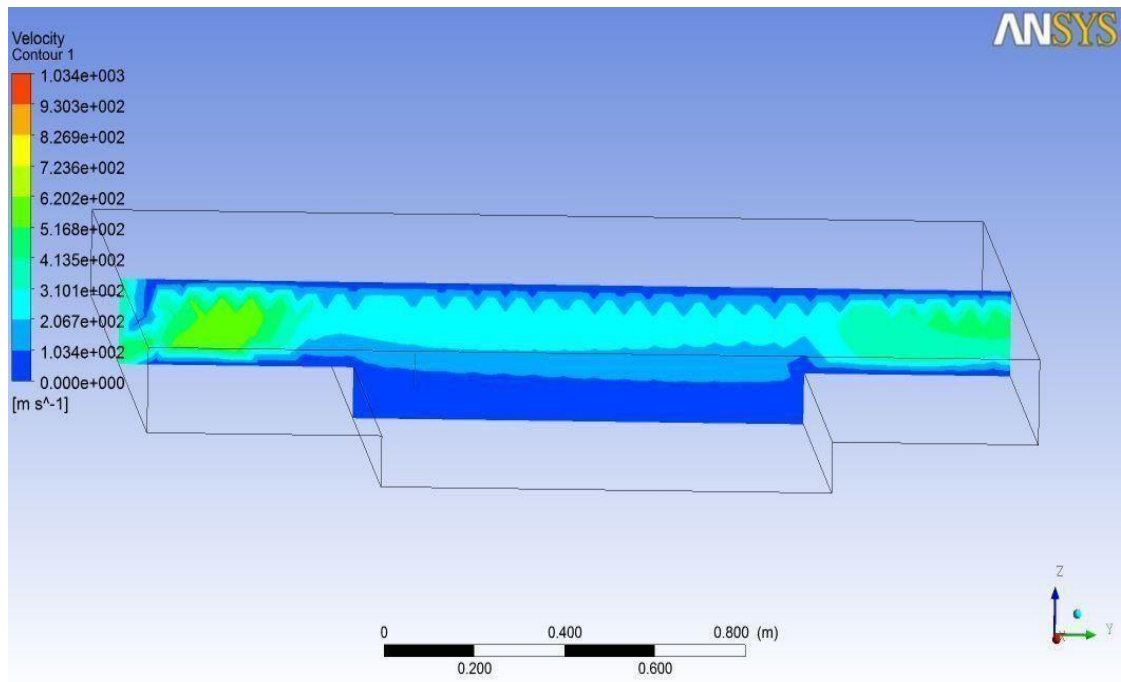


Fig: 5.6: velocity contour of 0° closed cavity wall epsilon model

CLOSED CAVITY WALL:

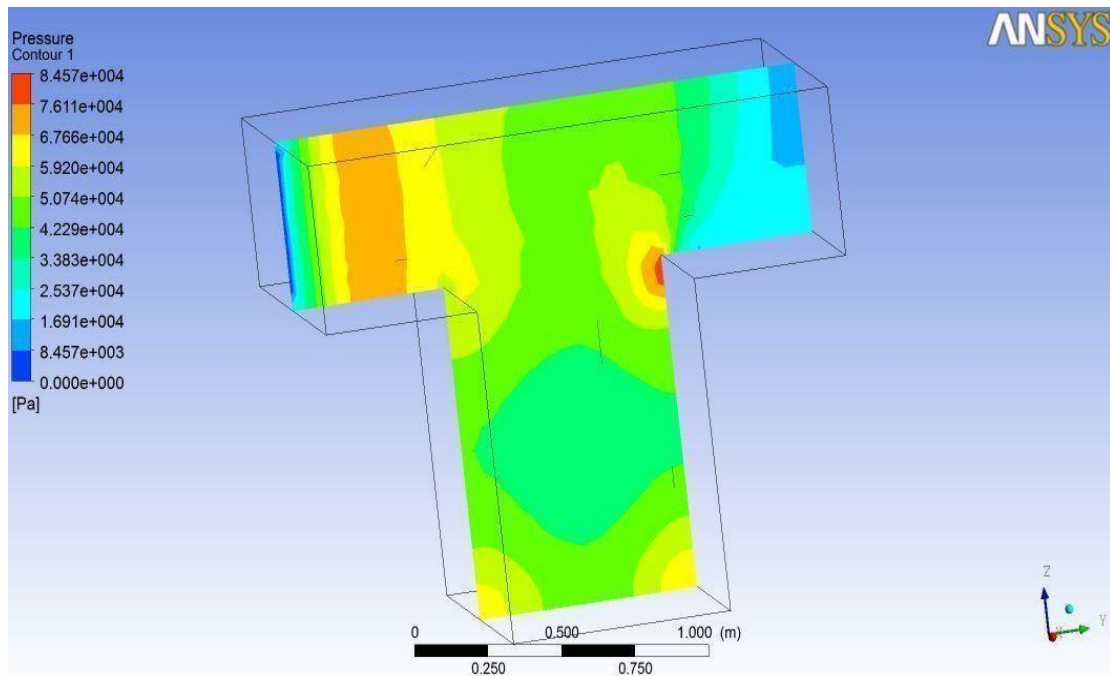


Fig: 5.7: pressure contour of 0° open cavity wall epsilon model

The turbulent eddy dissipation contour of a 0° open cavity wall epsilon model in CFD may be affected by a number of factors, including cavity shape, flow conditions, and the individual epsilon model utilised.

Researchers studied three-dimensional open cavity flow using detached eddy simulation and plotted the turbulent viscosity contour of the DES S-A model.

The k-epsilon model, which has a turbulent velocity scale and a turbulent time scale, has been proposed for wall-bounded turbulent flows.

Based on the number of partial differential equations solved in addition to the continuity and momentum equations, turbulent models are classed as zero, one, two, or stress equation models.

The choice of turbulence model and the boundary conditions employed in the CFD simulation may alter the eddy dissipation contour of a 0° open cavity wall epsilon model.

To properly forecast the turbulent eddy dissipation contour of a 0° open cavity wall epsilon model in CFD, numerical simulations employing suitable models and boundary conditions are required.

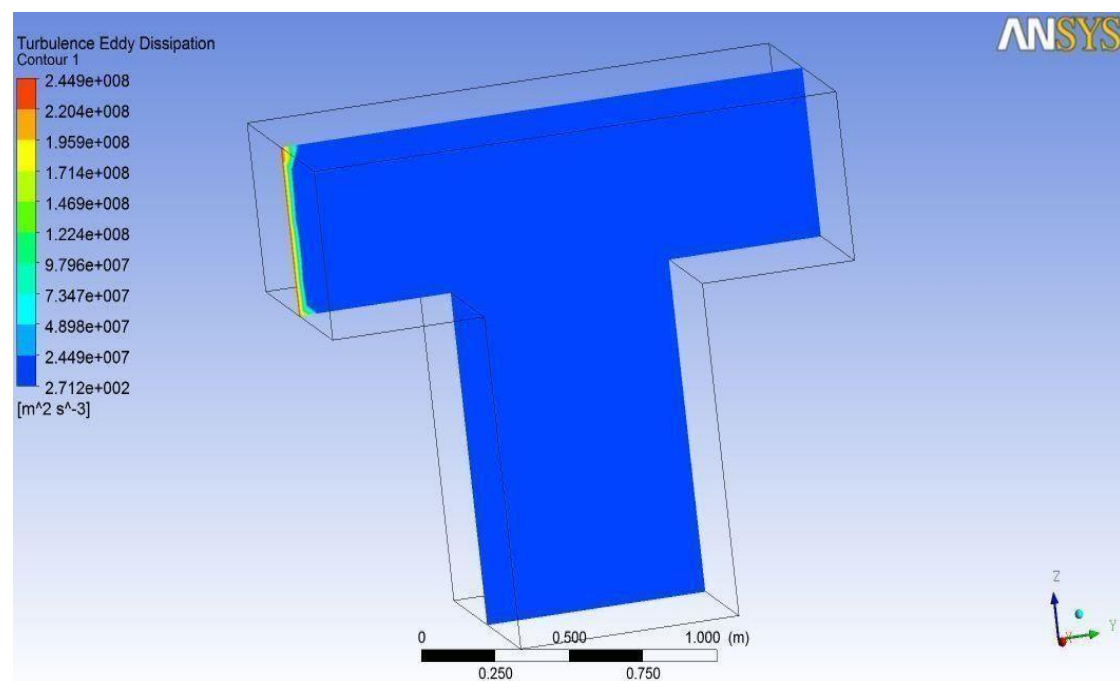


Fig: 5.8: turbulence eddy dissipation contour of 0° open cavity wall epsilon model

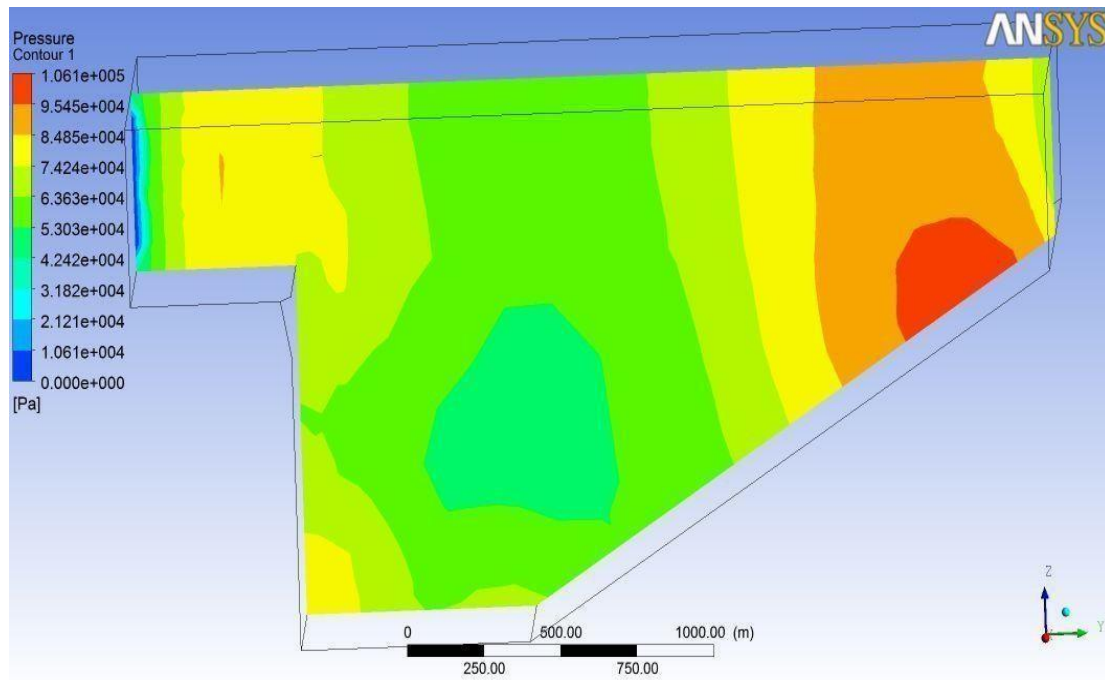


Fig: 5.9: pressure contour of 30° open cavity wall epsilon model

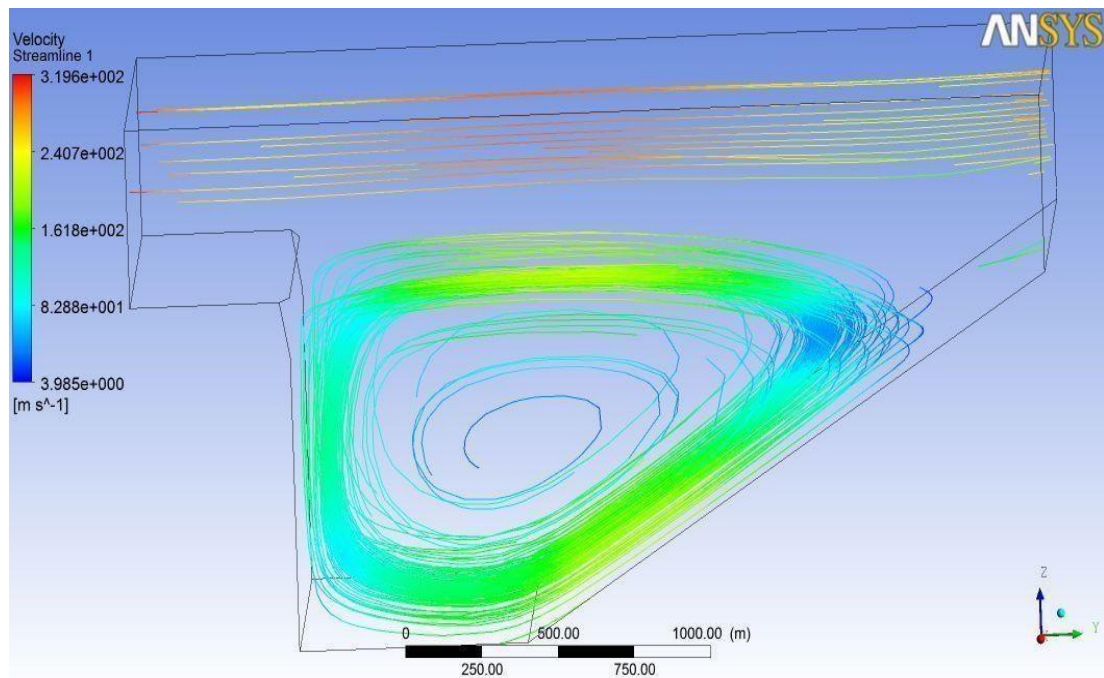


Fig: 5.10: velocity streamline contour of 30° open cavity wall epsilon model

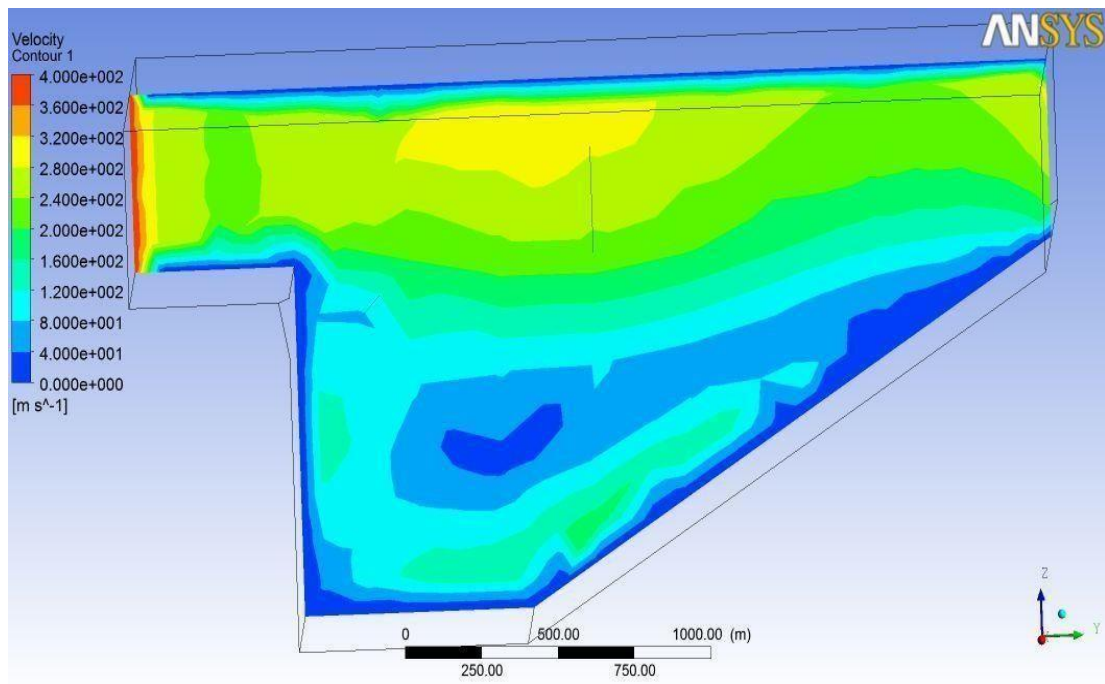


Fig: 5.11: velocity contour of 30° open cavity wall epsilon model

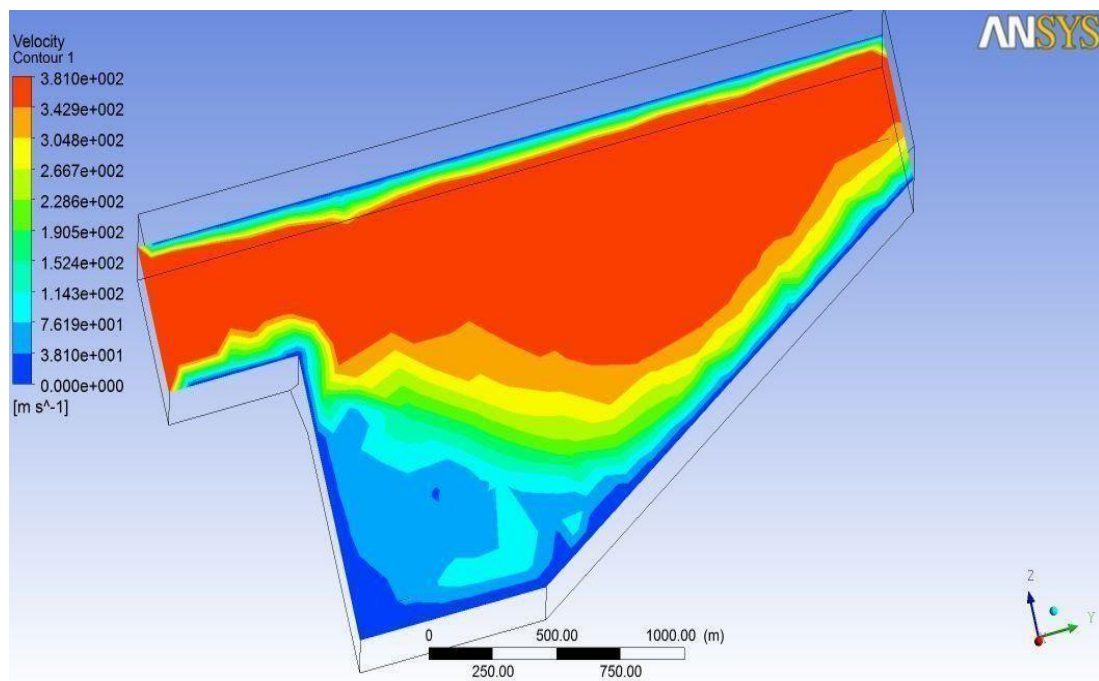


Fig: 5.12: velocity contour of 30° open cavity wall omega model

Table 5.1: Temperature Difference

Seria I	Cases	Temperature(k)
1	1	1651.94
2	2	5376.46
3	3	2912.78

Case 1 temperature is 1651.94 k which is the lowest of other other two cases.

Case 2 temperature is 5376.46 k which is the highest of other other two cases.

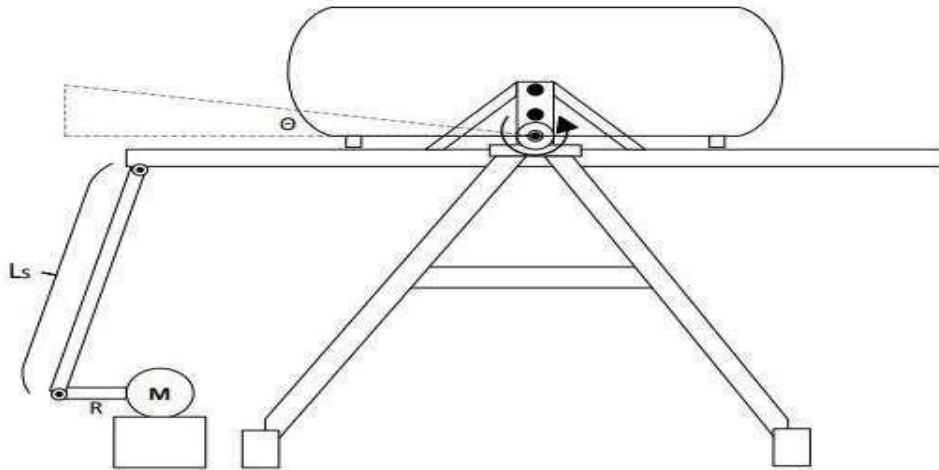


Fig: 5.13: The experimental rig 2D sketch

Simcenter STAR-CCM+

Solution Time 2.81 (s)

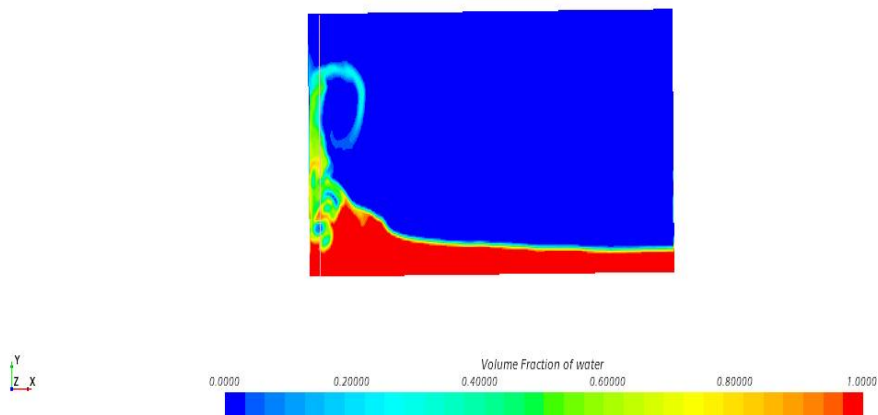


Fig: 5.14: filling level of 30% LH2 without baffle

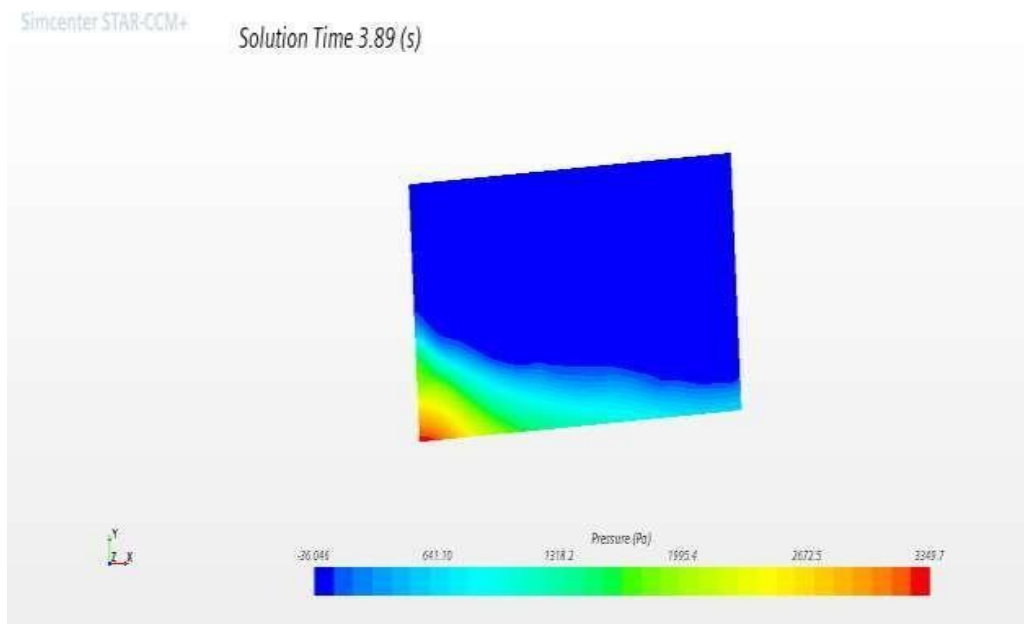


Fig: 5.15: 50% Filling Level Pressure Distribution

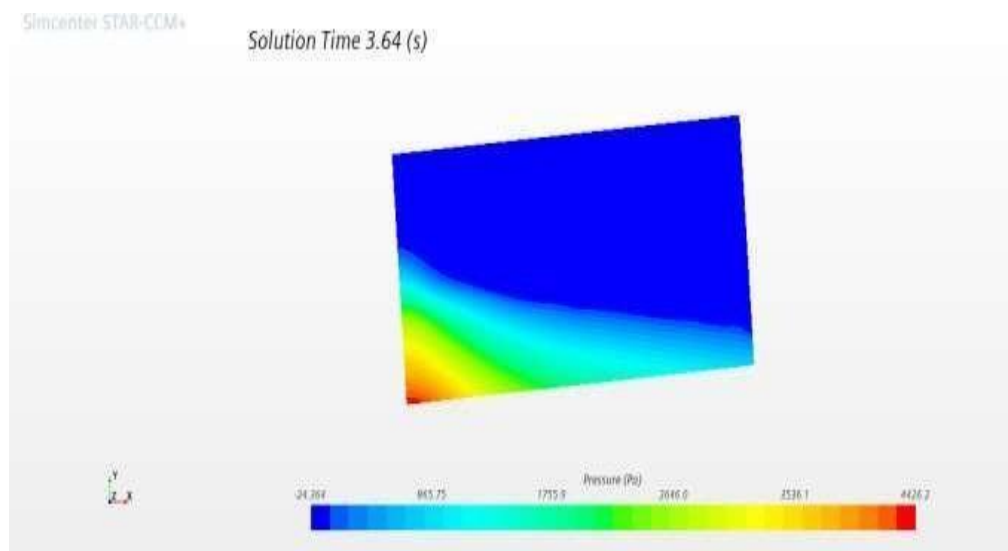


Fig: 5.16: 70% Filling Level Pressure Level

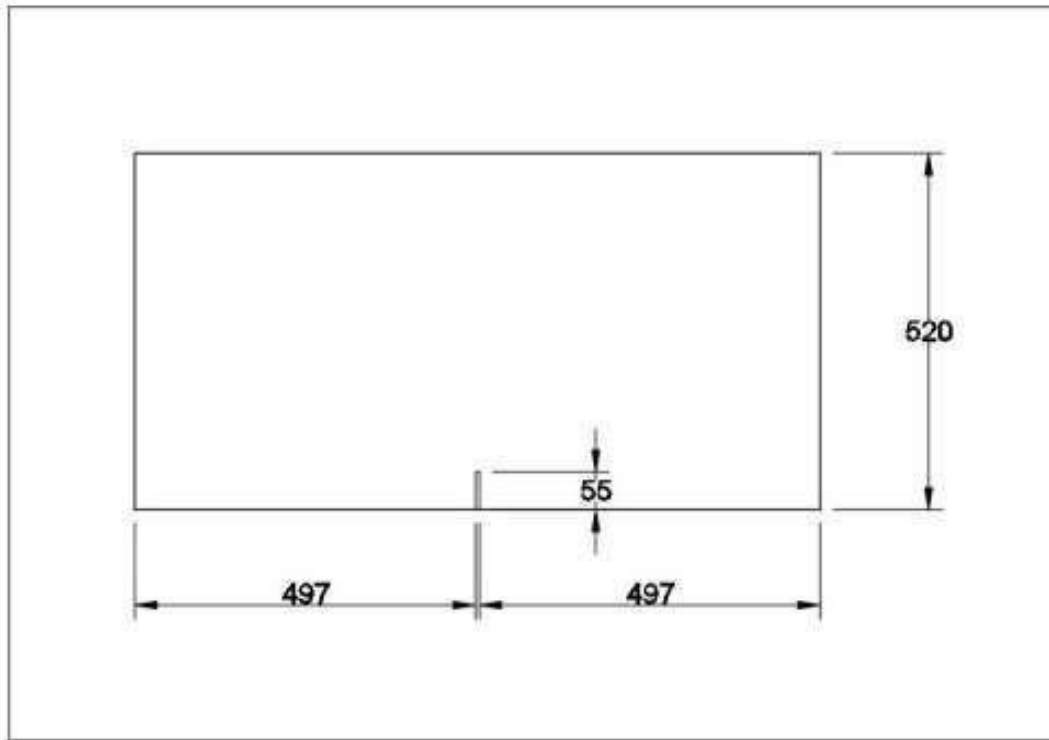


Fig: 5.17: Baffle Geometry

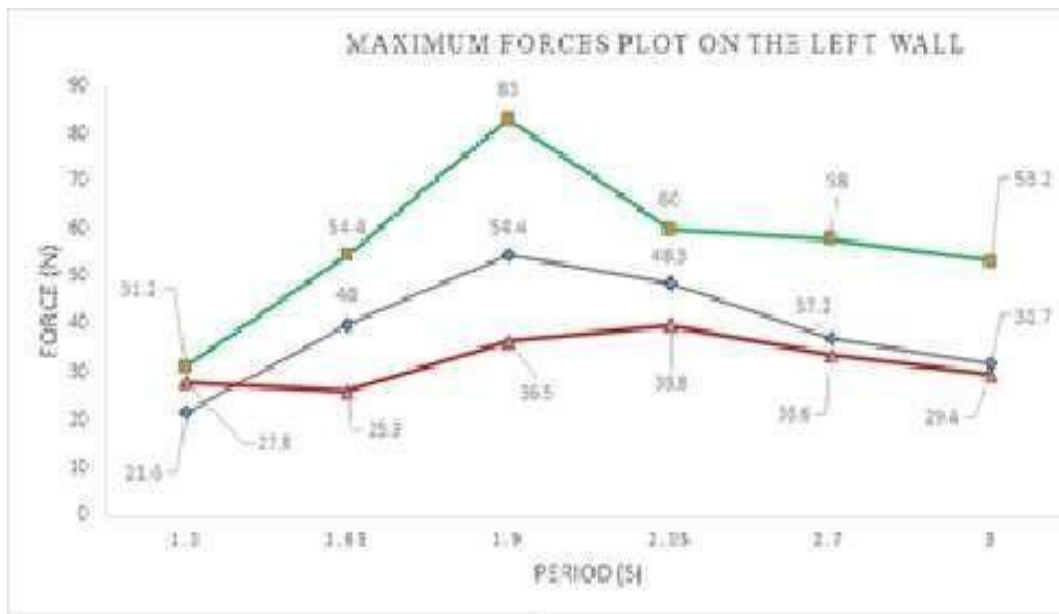


Fig: 5.18: 30% Filling Level

5.3 EXPERIMENTAL RESULT:

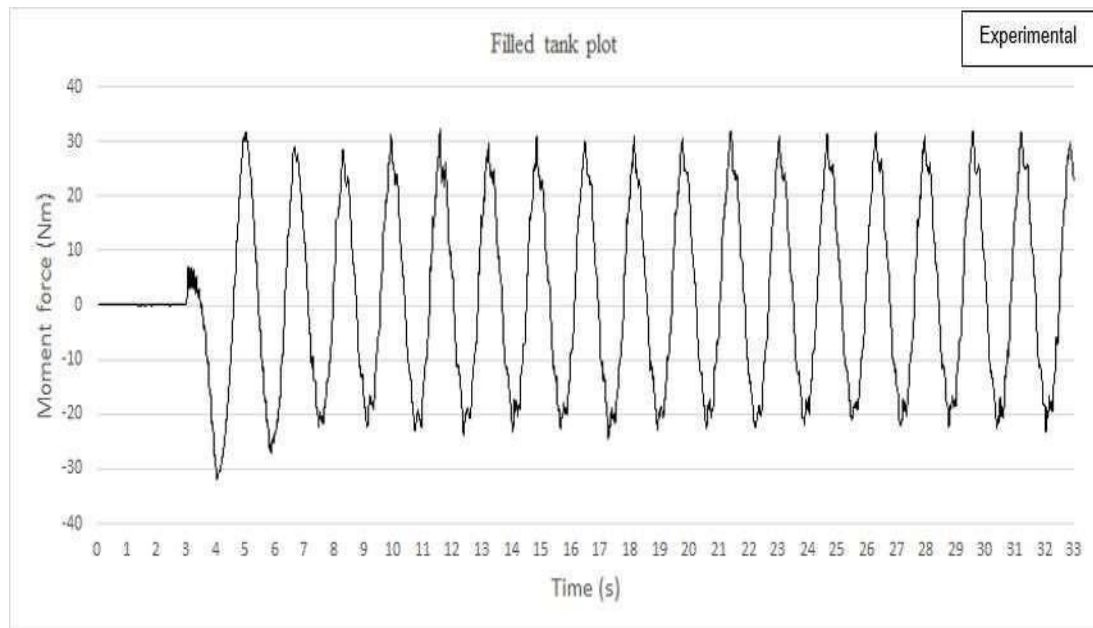


Fig: 5.19: Experimental Result Of Case 1

Filling Level 30%

Excitation Period 1.7 secs

In several research works, the comparison of numerical and experimental findings for a 30% excitation amplitude and an excitation time of 1.7 seconds in CFD has been explored.

A novel unsteady CFD technique was employed to predict the flow-generated excitation phenomena in one research, and the findings were compared to experimental data.

Another research compared computational and experimental fluid dynamic data and found extremely high convergence for both single and two-phase flow.

In a third investigation, the numerical findings for floating structures were confirmed against experimental data using STAR-CCM+.

In general, comparing numerical and experimental findings in CFD is critical for validating numerical model correctness and improving knowledge of the underlying physics.

The numerical approach, boundary conditions, and turbulence model used can all have an impact on the accuracy of the numerical results and their comparability to experimental data.

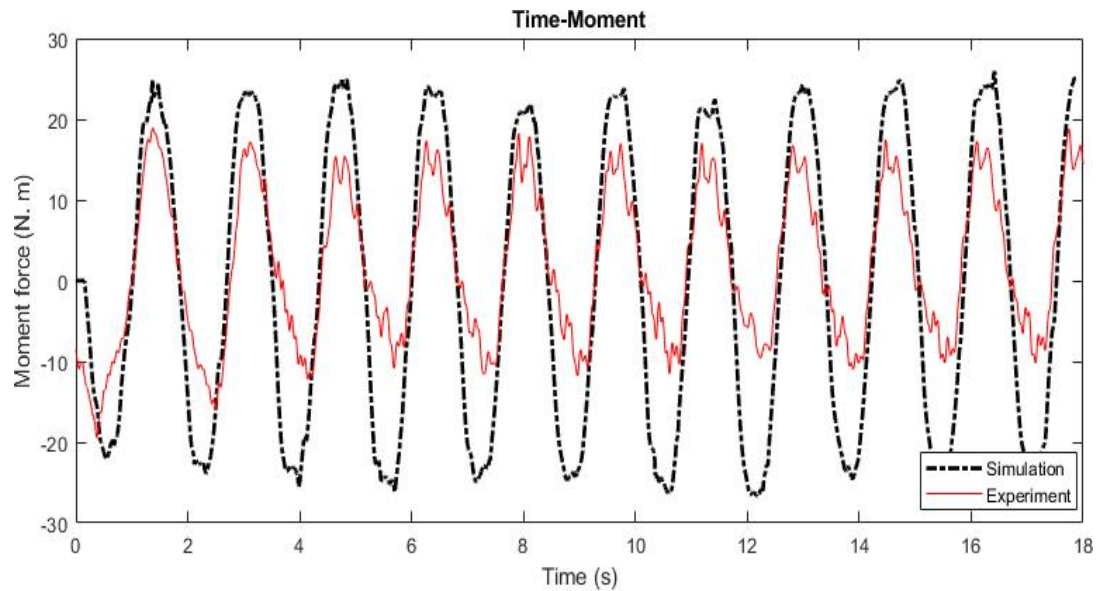


Fig. 5.20: Comparison of the Numerical and Experimental Results 30% and Excitation Period of 1.7 secs

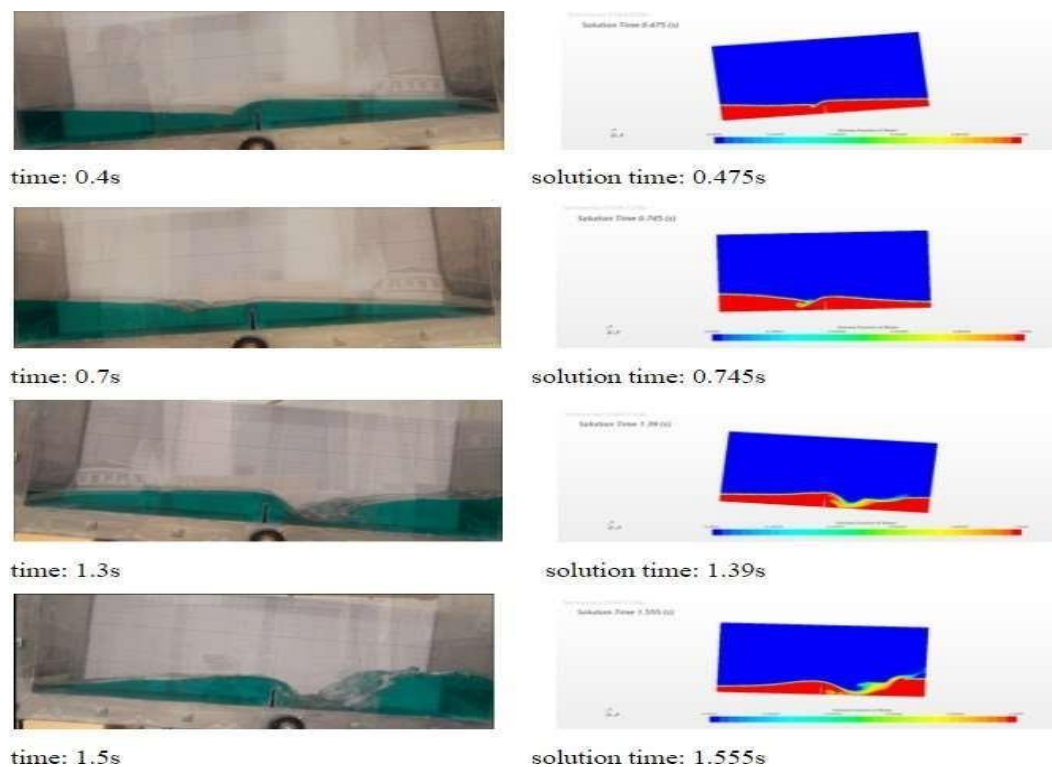


Fig. 5.21: Experimental and Numerical Patterns

5.4 COMPARISON TABLE:

The flow over closed cavities is distinguished by growing static pressure distribution over the cavity's length, whereas the flow over open cavities is distinguished by the boundary layer splitting at the upper stream corner and reattaching towards the downstream corner. Aeroacoustics analysis of a hybrid control mechanism for the cavity wall and the overall sound pressure level over the fluid was performed by researchers. The choice between open and closed cavities may be dictated by the individual application and flow characteristics sought. The flow across open and closed cavities may be simulated and analysed using CFD analysis.

Table 5.2: Comparison between 0°, 30°, 45°, 60°

		0°	30°	45°	60°
Closed Cavity Wall(k-Epsilon)	Pressure (pa)	2.395E006	2.256E006	1.771E006	2.065E006
	Velocity (ms ⁻¹)	1.034E003	6.243E002	7.514E002	7.523E002
Closed Cavity Wall(k-Omega)	Pressure (pa)	2.202E006	1.734E006	1.642E006	1.770E006
	Velocity (ms ⁻¹)	7.791E002	5.438E002	6.722E002	7.387E002
Open Cavity Wall(k-Epsilon)	Pressure (pa)	8.457E004	1.061E005	9.952E002	4.427E003
	Velocity (ms ⁻¹)	4.000E002	4.000E002	6.042E001	1.275E002
Open Cavity Wall(k-Omega)	Pressure (pa)	8.427E004	3.091E005	9.920E002	4.408E003
	Velocity (ms ⁻¹)	4.000E002	3.810E002	6.055E001	1.276E002

CHAPTER 6

SUMMARY AND CONCLUSION

6.1 SUMMARY:

The aim of this study was to analyse the impact of the cavity on the combustion chamber wall under supersonic air flow. Results indicated that the presence of a cavity typically led to an increase in the temperature of the combustor and total pressure loss, while simultaneously enhancing the combustion of fuel and oxidizer. By altering the aft wall angle, the researchers discovered that increasing the angle resulted in improved combustion efficiency, but also caused an increase in total pressure loss. This suggests that the trailing edge shock over the cavity may have a significant influence on these outcomes.

This paragraph describes a project that investigated the effects of a cavity on the combustion chamber wall in a supersonic air flow. The researchers found that the cavity increased the temperature of the combustor and total pressure loss, while also improving the combustion of fuel and oxidizer. They varied the aft wall angle and found that increasing it resulted in improved combustion efficiency but also increased total pressure loss. The paragraph suggests that the trailing edge shock over the cavity may play a key role in these outcomes.

6.2 CONCLUSION:

The nonlinear fluid interaction between staged liquid propellant rocket engines is a complicated phenomena that necessitates the use of dynamic and nonlinear modelling approaches to investigate. Because of the asynchronous engine start-ups during rocket lift-off, engine transients and clustered engine dynamics are produced. A novel dynamic and non-linear simulation approach for liquid-propellant engines has been presented, and software for a specific engine is provided.

The final amplitude of triggered instability is determined by non-linear combustion instability in liquid propellant rocket engines. During the steady running of a liquid-propellant rocket engine, the injected propellants are transformed into hot gases that are released via a nozzle to create thrust by different physical and chemical processes.

REFERENCES

- 1) Kyung Moo Kim, Seung Wook Baek, Cho Young Han, "Numerical study on supersonic combustion with cavity-based fuel injection", Korea advanced Institute of Techonology, (2003), Vol 973(7), Pg. No:305-701.
- 2) Albert Rougeux and Faure J., "Numerical Studies for Supersonic Cavity Based Flows", Malo-Molina 17th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, (Apr 2011), Pg. No: 11-14.
- 3) Cao X L, Zhang X B, Qiu L M, :Validation of full cavitation model in cryogenic fluids", Chin Sci Bull, (2009), Vol 54(10), Pg. No: 1633–1640.
- 4) Franc J P, Pellone C, "Analysis of thermal effects in a cavitating inducer using Rayleigh equation", J Fluids Eng-Trans Asme, (2007), Pg. No: 21- 80.
- 5) Goel T, Zhao J, Thakur S, "Surrogate model-based strategy for cryogenic cavitation model validation and sensitivity evaluation", Intern J Numeric Methods Fluids, (2008), Vol 58(9), Pg. No: 311-346.
- 6) Hosangadi A and Ahuja V, "Computational fluid dynamic study on cavitation in liquid nitrogen", J. Fluids Engineering, (2005), Vol 127, Pg. No: 267-281
- 7) Hosangadi A, Ahuja V, Ungewitter R J, "Numerical study of a flat plate inducer: Comparison of performance in liquid hydrogen and water", In Proceedings of the 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Sacramento, CA, United States, (2006), Vol 72, Pg. No:107-222.
- 8) Shi S G, Wang G Y, "Thermal effects on cryogenic cavitating flows around an axisymmetric olive", Int J Fluid Machinery Syst, (2010), Vol 3(4), Pg. No:324–331.
- 9) Tseng C, Wei Y J, Wang G Y, "Modeling of turbulent, isothermal and cryogenic cavitation under attached conditions", J Acta Mech Sin, (2010), Vol 26, Pg. No: 325–353.
- 10) Utturkar Y, Thakur S and Shyy W, "Computational Modeling of Thermodynamic Effects in Cryogenic", Cavitation Proc. of the 43rd AIAA Aerospace Sciences Meeting and Exhibit, (2005), Vol. 121, Pg. No:1142-2156

- 11)** Yoshida.Y, K. Kikuta, S. Hasegawa, M. Shimagaki, and T. Tokumasu, "Thermodynamic effect on a cavitating inducer in liquid nitrogen," *Journal of Fluids Engineering*, (2007), Vol. 129, no. 3, Pg. No: 273–278.
- 12)** Zhang Xiaobin, WU Zhao, XIANG ShiJun & QIU LiMin, "Modeling cavitation flow of cryogenic fluids with thermodynamic phase- change theory", *Journal of Fluids and Engineering*, (2012), Vol. 134, Pg. No: 145-180.
- 13)** Zhang X B, Qiu L M, Qi H, "Modeling liquid hydrogen cavitating flow with the full cavitation model", *Int J Hydrogen Energy*, (2008), Vol. 33, Pg. No: 7197–7206.
- 14)** Zhang X B, Qiu L M, Gao Y, "Computational fluid dynamic study on cavitation in liquid nitrogen", *Cryogenics*, (2008), Vol. 48, Pg.No: 432–438.