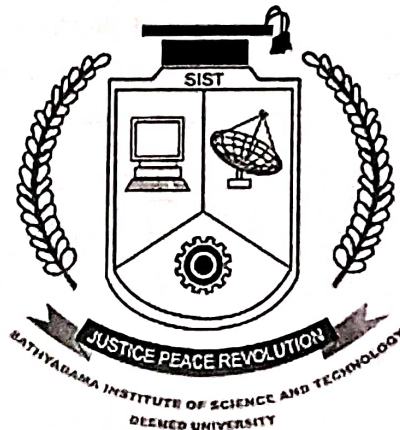


EXPERIMENTAL AND NUMERICAL SIMULATION OF BLUFF BODIES UTILISED IN SUPERSONIC STEP COMBUSTER

Submitted in partial fulfillment of the requirements for the award of Bachelor's
Engineering Degree in Aeronautical Engineering

By

AMARAPALLI DEEPAK (39260003)
BANDUGULA SAI THARUN (39260005)



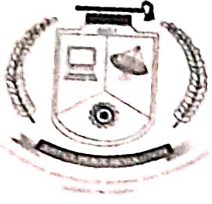
DEPARTMENT OF AERONAUTICAL ENGINEERING
SCHOOL OF MECHANICAL ENGINEERING

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)

Accredited with Grade "A" by NAAC 12B status by UGC | Approved by AICTE
JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600119

APRIL – 2023



SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)

Accredited with Grade "A" by NAAC 12B status by UGC I Approved by AICTE
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai – 600119
www.sathyabama.ac.in

DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of **DEEPAK. A (39260003)** and **SAI THARUN. B (39260005)** who carried out the project entitled **"EXPERIMENTAL AND NUMERICAL SIMULATION OF BLUFF BODIES UTILISED IN SUPERSONIC STEP COMBUSTER"** under my supervision from November 2022 to April 2023

Internal Guide

Mr. P. GUNA SEKAR, M.E, (PhD).

Head of the Department

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

Submitted for Viva-voce Examination held on 24th April 2023

Internal Examiner

Dr.A. ANDERSON, M.E, Ph.D.

External Examiner

Dr.G. SENTHILKUMAR, M.E, Ph.D.

DECLARATION

I SAI THARUN.B (39260005) on behalf of my team mate hereby declare that the Project Report entitled "EXPERIMENTAL AND NUMERICAL SIMULATION OF BLUFF BODIES UTILISED IN SUPERSONIC STEP COMBUSTER" done by me under the guidance of Mr. P. Guna Sekar, M.E, (Ph.D.), Assistant professor, Dept. of Aeronautical Engineering, SATHYABAMA INSTITUTE OF SCIENCE AND TECHNOLOGY, CHENNAI is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.

DATE: 24-04-2023

PLACE: CHENNAI



SIGNATURE OF 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 thanks to **Dr. Prakash, M.E., Ph.D., Dean, School of Mechanical and Dr. P. Boomadevi M.E., Ph.D., Head of the Department, Dept. of Aeronautical Engineering** for providing us 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 our **Project Guide Mr. P. Guna Sekar M.E., (Ph.D.)**, for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of our project work.

I wish to 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.

ABSTRACT:

Scramjet engines are propulsion systems that operate in the supersonic flight regime, where the flow of air into the engine is supersonic. In a scramjet engine, the fuel is mixed with air in the supersonic combustion process, and the resulting exhaust gases provide the thrust for the engine. One of the critical challenges in designing scramjet engines is achieving efficient fuel-air mixing, which is crucial for achieving high combustion efficiency and generating high thrust. To address this challenge, an injection strategy with high penetration depth has been developed for the short residence time of fuel in the supersonic step combustor. The step combustor is a component of the scramjet engine that is responsible for slowing down the incoming air to subsonic speeds before mixing it with fuel. By using an injection strategy with high penetration depth, the fuel can mix more efficiently with the air, resulting in improved combustion efficiency. The study focuses on analyzing the supersonic flow over various bluff body models placed in the step combustor. Bluff bodies are objects that are placed in the flow path of air to create turbulence and mixing. The study observes the shock wave interactions between the bluff body models and differentiates them based on pressure, velocity, temperature, and recirculation area. The goal is to determine the optimal fuel-air mixing ratio for efficient combustion by selecting the best-performing bluff body model. The study aims to provide insights into designing more efficient scramjet engines for supersonic flight. By improving fuel-air mixing efficiency, the study could contribute to the development of more efficient and cost-effective scramjet engines, which could have significant applications in supersonic transportation, hypersonic flight, and space exploration.

KEYWORDS: Bluff bodies, Step combustors, Supersonic combustors, Streamlined bodies, Aero foils, Recirculation area, Disturbed flow Wake, Near-wake Flow instabilities, Penetration depth, Turbulence , Reynolds number.

TABLE OF CONTENTS

ABSTRACT	v
LIST OF FIGURES	ix
LIST OF TABLES	xi
LIST OF ABBREVIATIONS	xii

CHAPTER	TITLE	PAGE NO
1	INTRODUCTION	1
1.1	BLUFF BODY	1
1.2	FLOW OVER DIFFERENT TYPES OF BLUFF BODY	1
1.3	UNDERSTANDING THE DRAG COEFFICIENTS ON DIFFERENT BLUFF BODIES	2
1.4	WAKE CHARACTERISTICS	3
1.5	UNDERSTANDING THE SHOCK INTERACTIONS ON BLUFF BODIES	4
1.6	COMBUSTER	5
1.7	RECIRCULATION ZONE	6
1.8	APPLICATION CAVITY-BASED FLAME HOLDER	7
2	LITERATURE SURVEY	8
2.1	BLUFF BODY FLAME STABILIZATION	8
2.2	INNER HEATING OF THE FUEL-AIR MIXTURE DURING COMBUSTION BY BLUFF BODY SHAPES	9
2.3	PRESSURE WAVE IN SUPERSONIC COMBUSTION ENGINES	11
2.4	THE IMPACT OF A BLUFF BODY WITH A HOLE IN A MICRO-CHANNELS REACTION CHAMBER	12
2.5	CONTOURED UPSTREAM WALL	12

2.6	UPSTREAM CAVITY IN SCRAMJET COMBUSTORS	13
3	AIM AND SCOPE	15
3.1	AIM	15
3.2	SCOPE	15
4	COMPUTATIONAL AND EXPERIMENTAL ANALYSIS	16
4.1	METHODOLOGY	16
	4.1.1 COMPUTATIONAL ANALYSIS	16
	4.1.2 SHOCK GENERATOR OR WEDGE	18
	4.1.2.1 EQUATIONS FOR SHOCK WAVE RELATION	19
	4.1.3 MESHED DOMAIN	21
	4.1.4 SOLVER SETUP	22
	4.1.4.1 SOLVER TYPE-DENSITY BASED	22
	4.1.4.2 SOLVER TIME-STEADY	23
	4.1.4.3 MODELS	23
	4.1.4.3.1 ENERGY	23
	4.1.4.3.2 VISCOUS-K- ω MODEL	23
	4.1.4.4 MATERIALS -DENSITY – IDEAL GAS	24
	4.1.4.5 SPATIAL DISCRETIZATION	24
	4.1.5. VALIDATION	25
	4.1.6. GRID INDEPENDENCE TEST	26
4.2	EXPERIMENTAL SETUP	27
	4.2.1. SUPERSONIC WIND TUNNEL	27
	4.2.2. CAVITY AND WEDGE MODEL	28

	4.2.3. MATERIAL USED	31
5	RESULTS AND DISCUSSION	32
5.1	COMPUTATIONAL RESULTS	32
	5.1.1. RELATION BETWEEN CD, RMS, AND RECIRCULATION AREA	37
	5.1.2 CALCULATION OF CD VALUES FOR ALL THE WEDGE BODIES USED FOR THE EXPERIMENT	43
	5.1.2.1 TRIANGULAR BODY CD VALUE	42
	5.1.3 RECIRCULATION ZONE	45
	5.1.3.1 RECIRCULATION REGION WITH RESPECTIVE TO MACH	45
	5.1.3.2 RECIRCULATION ZONE WITH RESPECTIVE TO THE SHAPE OF THE WEDGE BODY	46
5.2	EXPERIMENTAL RESULTS	49
	5.2.1 THE EXPERIMENTAL VALUES FOR THE DIFFERENT WEDGE BODIES	49
	5.2.2. SHOCK OCCURRENCE DURING EXPERIMENTAL WORK	50
6	SUMMARY AND CONCLUSION	51
6.1	SUMMARY	51
6.2	CONCLUSION	51
	REFERENCES	52

LIST OF FIGURES

FIGURE NO	TITLE	PAGE NO
1.1	FLOW OVER STREAMLINED AND WEDGE BODIES	3
1.2	TYPES OF DIFFERENT BLUFF BODY SHAPES	4
1.3	DIFFERENT REGIMES OF FLOW	5
1.4	FLOW ATTRIBUTES OF CAVITY-BASED FLAME-HOLDER	7
2.1	FLAME STABILIZATION USING A BLUFF BODY WITH A HOLE IN A MICRO-CHANNELS REACTION CHAMBER	8
2.2	LOW-VELOCITY LIMIT	10
	HIGH-VELOCITY LIMIT	10
2.3	THE ENGINE LAYOUT	11
2.4	RJTF WITH 6/6-HEIGHT STRUCT	11
2.5	THE IMPACT OF A BLUFF BODY	12
4.1	DIMENSIONS OF THE COMPUTATIONAL DOMAIN,	16
4.2	DISPLAYS THE 2D SKETCHING OF THE COMPUTATIONAL DOMAIN	17
4.3	FLOW PAST A WEDGE.WHERE θ -HALF WEDGE ANGLE AND IS THE SHOCK ANGLE	18
4.4	EQUATIONS FOR SHOCK WAVE RELATIONS	19
4.5	THE BELOW PICTURES THE DIMENSIONS OF EACH BLUFF BODY UTILIZED IN THE COMPUTATIONAL DOMAIN	20
4.6	TYPES OF MESH ELEMENTS	21
4.7	MESHED DOMAIN FOR THE CURRENT STUDY	22
4.8	NORMALISED PRESSURE VS EFFECTIVE LENGTH IN INCHES	25
4.9	GRID INDEPENDENCE TEST	26
4.10	PHOTOGRAPH OF WIND TUNNEL FACILITY	27
4.11	WEDGE HOLDER	28
4.12	TEST SECTION	29
4.13	CLOSE-UP OF THE WEDGE AND CAVITY MODEL MOUNTED IN THE TEST SECTION	29
4.14	THE ABOVE PICTURE DISPLAYS DIFFERENT FABRICATED WEDGE BODIES FROM ATO E	30

4.15	PICTURE F AND G SHOWCASE THE THREADED HOLE AND SCREW UTILISED FOR HOLDING THE BLUFF BODY IN THE TEST SECTION AND CHANGING THE POSITIONS	31
5.1	THE EDDY VISCOSITY FLUID CONDITION RESULTS AT DIFFERENT OF THE WEDGE BODY	34
5.2	THE TURBULENCE KINETIC ENERGY FLUID CONDITION RESULTS AT DIFFERENT POSITIONS OF THE BODY	35
5.3	THE VISCOCITY U FLUID CONDITION RESULTS AT DIFFERENT POSITIONS OF THE WEDGE BODY	36
5.4	VECTOR PLOTS FOR (A) MACH 1 (B) MACH 1.5 (C) MACH 2 (D) MACH 2.5 (E) MACH 3	45
5.5	RECIRCULATION ZONE WITH RESPECTIVE TO THE SHAPE OF THE WEDGE BODY	48
5.6	VISUALS OF SHOCK WAVE OCCURRENCE DURING EXPERIMENTAL WORK	50

LIST OF TABLES

TABLE NO	TITLE	PAGE NO
2.1	SUMMARY OF FLAMMABILITY LIMITS DURING SHORT AND LARGE LEAN LIMITING CONDITIONS	10
4.1	DIFFERENCE BETWEEN QUADRILATERAL AND TRIANGULAR MESH ELEMENT	21
5.1	THE PLOTS BELOW IS CONSIDERED FOR POSITION OF WEDGE VS RMS	37
5.2	THE PLOTS BELOW IS CONSIDERED FOR POSITION OF WEDGE WITH SQUARE ENDING VS RMS	38
5.3	THE GRAPH PLOTTED BETWEEN THE POSITION OF WEDGE WITH SQUARE ENDING VS RMS	39
5.4	THE GRAPH PLOTTED BETWEEN THE POSITION OF DIAMOND SHAPED BODY VS RMS	40
5.5	THE GRAPH PLOTTED BETWEEN THE POSITION OF DIAMOND SHAPED BODY VS RMS	41
5.6	COEFFICIENT OF DRAG FOR ALL WEDGE BODIES UTILISED IN THE PROJECT	43

LIST OF SYMBOLS AND ABBREVIATIONS

SYMBOLS AND ABBREVIATIONS	DESCRIPTION
L/D	Length to Depth ratio
OR	Offset Ratio
PIV	Particle Image Velocimetry
I/L	Length of sub-cavity to length of main cavity ratio
k- ϵ	K – Epsilon
GPa	Giga Pascal
MPa	Mega Pascal
g	Grams
W	Watts
m	Meter
°C	Degree Centigrade
K	Kelvin

CHAPTER 1

INTRODUCTION

1.1 BLUFF BODY:

In aerodynamics, a bluff body is a shape whose length in one direction is comparable to or greater than the length in the perpendicular direction. This results in a greater influence of pressure on the force acting on the body, compared to skin friction. Even streamlined shapes like airfoils can exhibit bluff body behavior when at large angles. Circular cylinders are often used as a standard to look at the properties of bluff body flows, which are characterized by a separation of the flow that creates a disturbed flow region behind the body. This region can trigger and amplify different types of convective and absolute flow instabilities, leading to the formation of two- or three-dimensional unsteady flow structures. As the Reynolds number increases, these instabilities eventually give rise to turbulence.

1.2 FLOW OVER DIFFERENT TYPES OF BLUFF BODIES:

Viscosity affects bluff solids with rounded-end rear faces and streamlined shapes with huge angles of attack, causing significant forces and moments. The magnitude of the wake is proportional to the incoming flow's body form and orientation, with both the near wake having low velocities and a roughly stable pressure reduced than that of the undisturbed pressure. Karman Street is characterized by identical vortices of similar strength oriented across two parallel paths and displaced half a wavelength in the distant wake. Vortex shedding causes uneven flow and vibration issues.

a) Circular Bluff Body

Circular bluff bodies both have real-world and academic applications in fluid dynamics. At low flow rates, fluid movement around circular cylinders shows a variety of key physical processes like as splitting, vortexes, and instability in the wake. The existence of many other things in the flow generates flow interruption when circular cylinders are clustered together, which is critical in aerodynamics and hydrodynamics.

b) Rectangular Bluff Body

Rectangular bluff bodies describe the flow field's causes of unsteadiness. Various causes of flow-field unsteadiness, such as approaching instability, vortices (the re-attachment), and vortex-induced- induced procedures, display unique frequency ranges. The flexible system's dynamics can cause interactions among these several kinds of unsteadiness.

C) Triangular Bluff Body

The triangle bluff body is important in engine combustion, notably in combustors, where it anchors a turbulent flame. A hot gas recirculating area burns the reactive streams, which is formed by the back into the past stair or a bluffing object. For the creation of prospective combustion chambers and passive as well as active combustion control mechanisms, a greater knowledge of flame holding, turbulent kinematics interactions, vortices movements, combustion destabilization, sparking, and recrystallization is required.

d) D-shaped bluff body

D-shaped short convex bodies have a round nose as well as a squared tail afterbody and are used in vehicles, subways, blunt airfoils, and airship gondolas. Flows through D-shaped cylinders at different C/D payout ratios reveal that the C/D ratios have a considerable influence on the variations in the wake region at a constant freestream velocity. Vortex-induced vibration is just a significant issue, and studies reveal that the afterbody has a significant impact on dispersion patterns and may modify the VIV nature of a structure. The frequency responses of a D-shaped body are influenced by Vertical motion and bounding.

1.3 UNDERSTANDING THE DRAG COEFFICIENTS ON DIFFERENT BLUFF BODIES

Aerospace engineers are primarily interested in the aerodynamics of streamlined shapes that gradually taper to a sharp point at their trailing edges, such as airfoils and wings. However, in various engineering applications, blunt bodies with non-streamlined

or streamlined shapes with blunt front and/or rear faces, also known as bluff bodies, are encountered. The total drag on a body is made up of two main components: skin friction drag and pressure drag. In the case of bluff bodies, pressure drag typically dominates the total drag due to the large low-pressure zone created in the wake at the rear of the body, as shown in the accompanying figure.

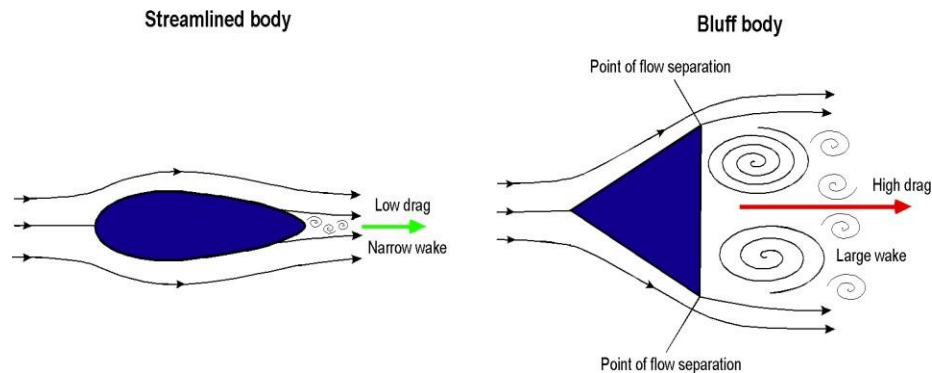


Fig1.1: Flow over streamlined and wedge bodies

A bluff body provides huge flow separation which develops at the body's extremes or corners, as well as a broad wake with low pressures, resulting in significant ultimate drag.

1.4 WAKE CHARACTERISTICS

The Wake characteristics are the features of the recirculating airflow after a static or moving blunt body which is caused by friction and also can create disturbance and fluid divergence. Moreover, changes in the density of the fluid both above and beneath the boundary, in addition to gravitation or interfacial tension, influence the wave patterns upon that sea surface created by only a moving vehicle like a boat.

As that the Reynolds number varies owing to flow regime changes, the sophistication of vortex dynamics rises. The current work does not intend to evaluate all convex bodies, but rather to investigate the features and dynamics of wake wakes under diverse fluidic circumstances and flow regimes. To that end, the study investigates the effect of four essential factors upon bluffs body reawakening for just a broad variety of Reynolds numbers: Strouhal number (St), bluff body cross-section, vorticity, and

circulation in vortices are all factors to consider.

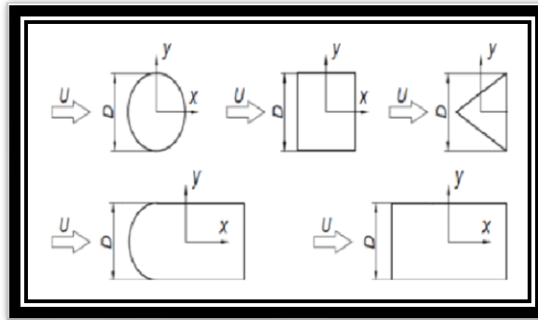


Fig 1.2: Types of different bluff body shapes

1.5 UNDERSTANDING THE SHOCK INTERACTIONS ON BLUFF BODIES

Shock waves are extremely narrow regions in which flow properties change abruptly. These waves are commonly known as oblique shock waves, as they usually form at an oblique inclination relative to the incoming flow. A regular seismic wave is an exception to the oblique shock, occurring when the wave angle is 90 degrees. In addition to oblique shock waves, supersonic flows are characterized by oblique. Prandtl-Meyer waveforms are examples of expansion waves in which the pressure lowers consistently over the wave.

These waves are created due to the propagation of information through molecular collisions, which cannot penetrate certain regions of the supersonic flow. Fig.1.3 illustrates the key parameters across oblique shock and expansion waves. As the flow crosses an oblique shock wave, the Mach number abruptly decreases while the pressure, density, and temperature experience a sudden increase. On the other hand, across an expansion wave, the Mach number increases while the pressure, density, and temperature decrease.

The curved bow shock is depicted in Fig.1.3, standing at a distance δ (shock detachment distance) in front of a blunt body in a supersonic flow. At point a, the shock wave is normal to the upstream flow, corresponding to a normal shock wave. However, the shock wave gradually becomes weaker and curved as it moves away

from point a, eventually transforming into a Mach wave (pressure wave traveling with the speed of sound) at larger distances from the body.

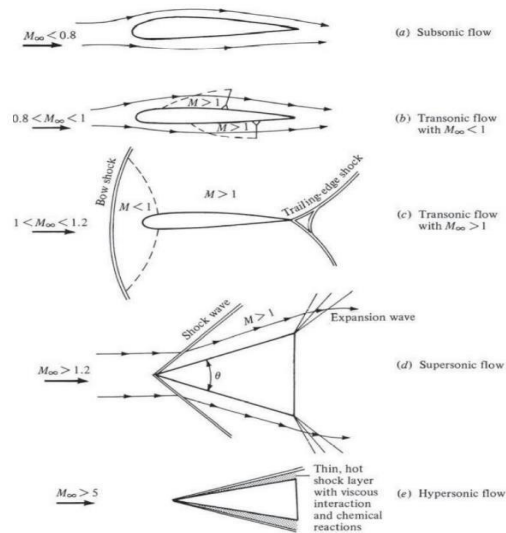


Fig 1.3: Different regimes of flow.

1.6 COMBUSTOR

A combustor also commonly referred to as a burner, combustion process, or flame catcher, is a crucial component of a ramjet or higher-efficiency engine in which combustion takes place. In a gas turbine engine, the combustion chamber or combustor is responsible for heating the high-pressure air supplied by the compression system at constant pressure as the fuel/air mix burns. The resulting burned mix is then exhausted from the combustor through the nozzle guide vanes to the turbine. In contrast, in a ramjet or scramjet engine, the exhaust is directly fed out through the nozzle. The design of the combustor is critical as it must Despite the extremely high airflow rates, maintain steady combustion. Combustors are meticulously constructed to combine & spark the fuel and the air beforehand mixing in additional air to finish the burning process. While early gas turbine engines was using singular chambers known as a can-type combustor, current combustors come in three basic shapes: could be, annulus, as well as cannular (also known as can-annular tubo-annular). Moreover, afterburners are classified as a sort of combustor. For engines such as ramjets and scramjets, the combustion process is incredibly

intense, and the use of bodies can significantly impact the engine's efficiency. These bodies play a crucial role in making the process more efficient.

1.7 RECIRCULATION ZONE

In a supersonic combustor, the recirculation zone is a crucial element that helps to achieve efficient combustion and stable operation. The recirculation zone is a region of the combustor where the fuel and air mixture recirculate back on itself. This region is typically located at the end of the combustor where the fuel is injected into the air stream. The recirculation zone is important because it helps to mix the fuel and air thoroughly, promoting efficient combustion. In a supersonic combustor, Because of the tremendous speed of air and propellant, the recirculating region is especially significant. can cause them to separate and reduce combustion efficiency. By creating a recirculation zone, the fuel, and air are forced to mix, which increases the likelihood of complete combustion.

Additionally, the recirculation zone can help to stabilize the combustion process by creating a more uniform flow of air and fuel. This helps to prevent combustion instability and flame blowout, which can be particularly problematic in supersonic combustion systems. Overall, the recirculation zone plays a critical role in the operation of a supersonic combustor, helping to achieve efficient combustion and stable operation. By ensuring that the fuel and air mix thoroughly and creating a uniform flow, the recirculation zone can help to optimize combustion performance and minimize the risk of combustion instability.

1.8. APPLICATION – CAVITY-BASED FLAME HOLDER

In the cases of combustion under supersonic conditions, the utilization of flame holder is essential. The purpose of the flame holder is to inject recirculation zones which are similar in aspect to eddy and move much slower than the free stream. The goal of the introduction of recirculation zones is to promote the effective fusion of fuel and air that will continuously sustain combustion. Attributes that determines a cavity to be an

effective flame-holder are:

- High static pressure
- Low overall temperature
- Large recirculation zones
- Minimum drag

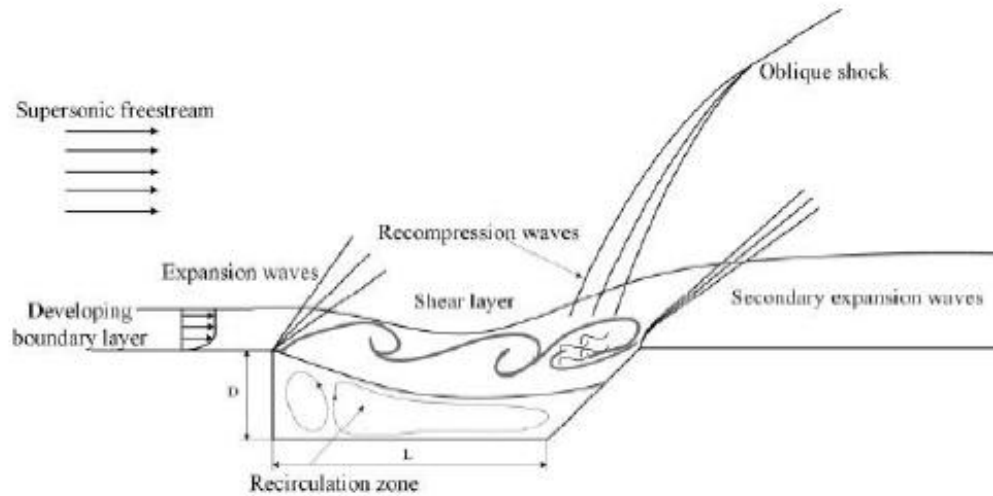


Fig 1.4: Flow attributes of cavity-based flame-holder.

CHAPTER 2

LITERATURE SURVEY

2.1 BLUFF BODY FLAME STABILIZATION

Ahmed A. Abdel – Rehim al(2013) study on Flame stabilization is a crucial process in afterburner systems that involves establishing a continuous ignition source in a gasoline-air combination with a high velocity compared to the turbulent flame speed. This is accomplished by enabling strong combustion products to mingle with the unburned fuel-air combination. A circulating flow of burnt material is often used to provide a consistent supply of high flame for this procedure. This recycling of combustion byproducts in the wake for a bluff-body ignition holder creates high - pressure steam that supply the energy and quantity of combustion products necessary to spark the incomplete combustion substance moving through the wake. Comparable processes can occur after a stage or in a wall break time.

However, heat exchange out from gas to the external wall might lower the combustion temperatures inside the recirculation region, making stability more difficult. Injecting a supplementary flue gas into the unburned flow is an alternate way for producing the requisite wake region with high recirculation. This relationship between the two streams creates a wake zone with intense recirculation, which helps to stabilize the flame.

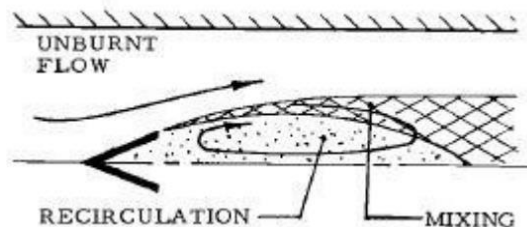


Fig 2.1 :Flame Stabilization using the bluff body.

2.2 INNER HEATING OF THE FUEL-AIR MIXTURE DURING COMBUSTION BY BLUFF BODY SHAPES

Mohammad Abdelhameed al (2021) have been conducted on the use of bluff body shapes over a broad range of equivalency proportions and shutoff valve velocity profiles for flame stabilization. This approach creates a circulating zone with better combustion byproducts and low-speed active molecules that serve as a continual igniting source for new reactants. The velocity profile created by a convex body's blocking effect drives a part of the elevated reactants into the recirculation region and ignites, leading to a stable flame.

Investigators also looked at the effect of reheating leftovers the fuel- air mixture even before entering the burning zone, which was discovered to increase flammability limits. However, care must be taken to prevent excessive heat loads, early initiation of chemical reactions upstream, and flame flashback. Most research on preheated fuel-air mixtures has been focused on miniaturized combustors, while studies related to trapped recirculation zones have concentrated on aero-engines.

This study aims to explore the effects of fuel mixture preheating aided by a trapped recirculation zone on lean flammability limits of high-velocity premixed reactants. Prior research on bluff bodies used for preheating fuel-air mixing has demonstrated that an optimal cavity height exists at which equivalence ratios are minimum. Furthermore, the low and high The maximum flow rate of air with a coinciding fuel flow rate was used to estimate the lean limits connected with each bluff body. Any rise in flow of air or reduction in fuel flow that exceeds these limitations leads in an abrupt shutoff valve.

The study placed the cavity at different blowoff limits, including low entrance velocity limits and high entrance velocity limits. Table 2 summarizes the flammability limits during low and high lean operating conditions for different types of bluff bodies, including their lower and upper diameters, as well as the corresponding blowoff

velocities.

Additionally, the study examined the impact of bluff body preheating on the blowoff velocity. The preheating effect is described by the temperature ratio, which was found to be inversely proportional to the blowoff velocity of the mixture entering the combustion zone. Therefore, it also describes the maximum allowable blowoff velocity. Based on the findings of the study, bluff body 3rd type was found to be the most efficient compared to other designs, the highest temperature throughout all hollow lengths. On the other hand, type 1 had the lowest temperature gains except at low cavity height. In this research, they placed the hollow at various blowoff limits short access velocity limitations (a) Large initial velocity limits

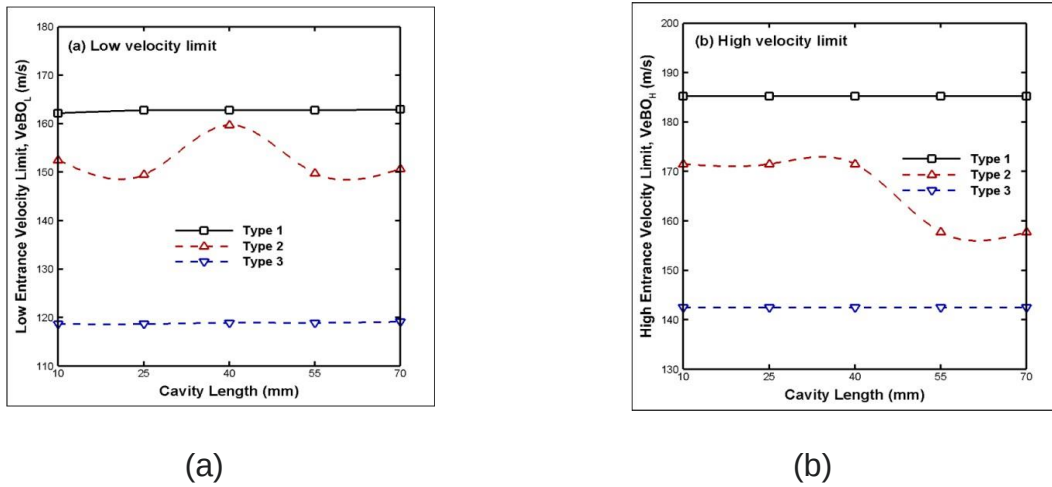


Fig 2.2: The a and b displays the plots on low and high velocity limit

Table 2.1: Summary of flammability limits during short and large lean limiting conditions

Bluff bodies	L	H	Velocity L (m/s)	Velocity H (m/s)
1	0.52	0.95	160	180
2	0.63	0.94	148	168
3	0.42	0.91	117	140

2.3 PRESSURE WAVE IN SUPERSONIC COMBUSTION ENGINES

Shijiun Diao .al (2012) According to the research, the fuel distribution in the engine is not uniform, and a considerable amount of fuel is not burned, resulting in reduced engine performance. The engine's layout, depicted in Figure 7, is rectangular with a length of 3.1m in length, 230mm in width, and 260mm in height. This consists of an intake, separator, boiler, and exhaust, with its walls drawn back at such a 45° angle to deflect leakage during the engine supersonic speeds beginning. The intake is a corner compressing type with a morphological pressure ratio of 2.76 and no strut, and the combustor entry has a back into the past step. Twelve major longitudinal gasoline H2 injection (hole diameter 2.5mm) are located 30mm downwards of the step on each side wall. The M6S42 arrangement has a 230-mm isolator and a filled (6/6-height) strut that's just as high as that of the back wall (29). The authors intend to show the outcomes of burning tasks performed on two engine

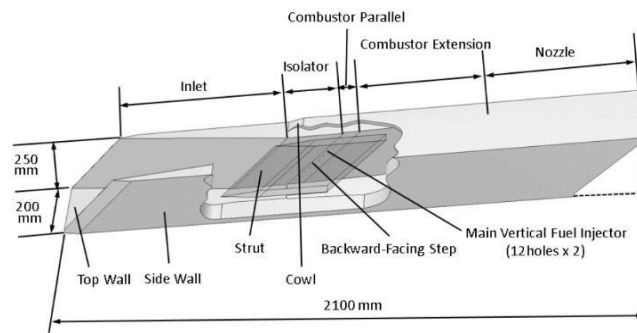


Fig 2.3: supersonic combustion engine

The investigators are interested in offering the findings of burning simulations performed on 2 stroke configurations: the RJTF with the 6/6-Height Strut as well as the suggested upgraded arrangement with the tug boat Tilt.

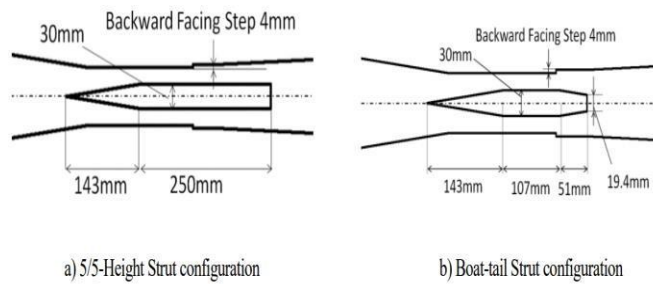


Fig 2.4: height and boat tail configuration

2.4 THE IMPACT OF A BLUFF BODY WITH A HOLE IN A MICRO - CHANNELS REACTION CHAMBER

Won Hyun Kim .al (2019) study is another useful way for stabilizing combustion as well as increasing burning rate in micro combustors is the convex body methodology. Prior research has found that introducing a convex body into to the mini reaction chamber may raise the burning keep blowing limit, as well as the length of the solid body influences the limit, according to the friction factor. Aiwu et al [30] showed that even a quasi-bluff body had a greater knock limit than a triangular bluff body due to poorer flame stretching in the sub bluff body instance.

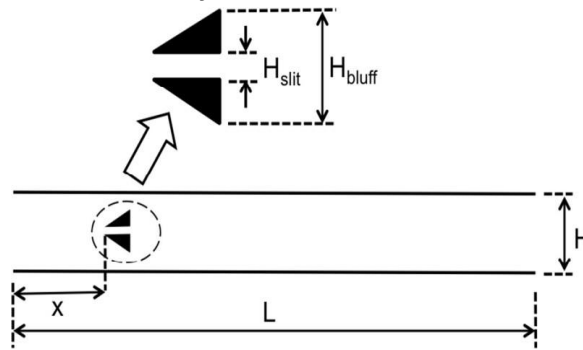


Fig 2.5: convex body in a chamber

A recent study investigated the impacts of a convex bodies with such a slot inside a segments and sub combustor were investigated, and it was discovered that perhaps the tiny hole spacing % used to have a major impact on performance and emission characteristics. The flaming region migrated downwards as the tiny hole gaps % grew likely related to the entering incomplete combustion mixed flow through into the slash gap wanting to push the flame area. Surprisingly, as even the slit space widened, the flame area traveled ahead and became linked to the bulk body border. The existence of such a supplementary swirl underneath the free surface as the slit gap expanded, thus drawing the flame upstream, explains this phenomenon.

2.5. CONTOURED UPSTREAM WALL

Will Landsberg. al (2019) research paper utilized numerical simulations to look into the impact of different hole profiles on the upstream wall of holes in scramjet combustors.

The study aimed to analyze four cavity profiles achieved by replacing the forward-facing step with ramps angled at 60 and 24.5 degrees or by stream tracing and adapt to the primary vortex aerodynamically. The results indicated that in Mach 8 combustor flows without fuel injection, the secondary vortex was eliminated completely. The initial source of hollow drag was identified as the rapid expansion of the cavity frontal wall over the rear-facing step, which led to extremely low pressure. The researchers also noted that modifying a front hollow wall to eliminate the smaller vortex could reduce the drag. For all holes, the secondary vortex was eliminated entirely in non-fueled Mach 8 scramjet combustor flows, and A ramp was installed in place of the rear-facing step. A 24.5-degree vacuum frontal ramp, on the other hand, was inadequate to generate separation among the boundary layers, but a 60-degree slope resulted in quick detachment. For fueled cases, both combined and 60-degree ramp configurations demonstrated drag reductions of 6.6 percent and 6.6 percent, respectively. Although drag reductions were observed in lower Mach number flows, aerodynamic contouring was necessary to increase flight speeds.

2.6 UPSTREAM CAVITY IN SCRAMJET COMBUSTOR

In 2019, Tim Roosa, Adrian Pudseya, Mathew Bricallia, and Hideaki Ogawa conducted a investigation on the impact of upstream voids on the performance of a generic scramjet combustor arrangement. The study focused on chemically frozen flow and considered three different wall conditions. The study's findings revealed that the forward cavity enhanced mixing efficiency by up to 8% relative to a benchmark at the domain's end ($x/d = 500$), based on the architecture and surface treatment. The hollow with the greatest volume, $L/D = 25$, was discovered to have been most effective in improving mixture effectiveness across all sidewall treatments. The cavity, on the other hand, increased overall pressure drops front of the injector. Yet, depending of the lengths of the reaction chamber, the nose shocks in instances with voids was weaker, resulting in lower total energy losses in comparison to the base behind the sprayer.. Moreover, the study observed that the introduction of cavities led to an increase in wall heat flux, especially on the cavity's aft wall. However, in cases with high wall temperatures, fuel fed into the cavity contributed to wall cooling. In conclusion, the

study highlighted the significant impact of upstream cavities on the performance of a scramjet combustor, with the largest cavity volume proving to be the most effective in enhancing mixing efficiency. The study also showed that while cavities may increase pressure drops upstream of the injector, they can lead to lower overall pressure loss depending on the combustor length.

CHAPTER 3

AIM AND SCOPE

3.1 AIM:

The aim is to investigate the use of a bluff body to generate a recirculation zone in the step-down part of a supersonic combustor to enhance combustion efficiency. The objective is to determine the optimal size, shape, and location of the wedge body to achieve the desired recirculation region while minimizing the pressure losses and flow instability.

3.2 SCOPE:

The scope of this project includes experimental and numerical simulations to optimize the design of the wedge body for a supersonic combustor. The specific research includes. The investigation of the effects of various factors on the size and position of the recirculation region is an important aspect of fluid dynamics research. To this end, several aspects of the flow will be examined. Firstly, the length and form of the wedge body will be studied to determine how they impact the size and position of the recirculation region. Secondly, research will be conducted to determine the optimal spacing between the reaction chamber and the wedge body in order to achieve the desired recirculation region. Thirdly, the impact of the inflow flow's angle of attack on the dimensions and position of the recirculation region will be analyzed. Finally, the influence of the entering flow's Drag coefficient and temperature on the size and location of the recirculation zone will be explored. These investigations will provide valuable insights into the underlying mechanisms of recirculation and help to optimize the design of fluid flow systems.

CHAPTER 4

COMPUTATIONAL AND EXPERIMENTAL ANALYSIS

4.1 METHODOLOGY

The methodology for this project will involve experimental testing in a wind tunnel to validate the numerical simulations and optimize the design of the bluff body. The expected outcomes of this project include the identification of the optimal design parameters for a wedge body in a supersonic step combustor and the development of guidelines for future research in this area. The limitations of this project include the time and resource constraints, as well as the technical challenges associated with designing and testing supersonic combustion systems. The significance of this project lies in its potential to enhance the efficiency and performance of supersonic combustion systems.

4.1.1 COMPUTATIONAL ANALYSIS

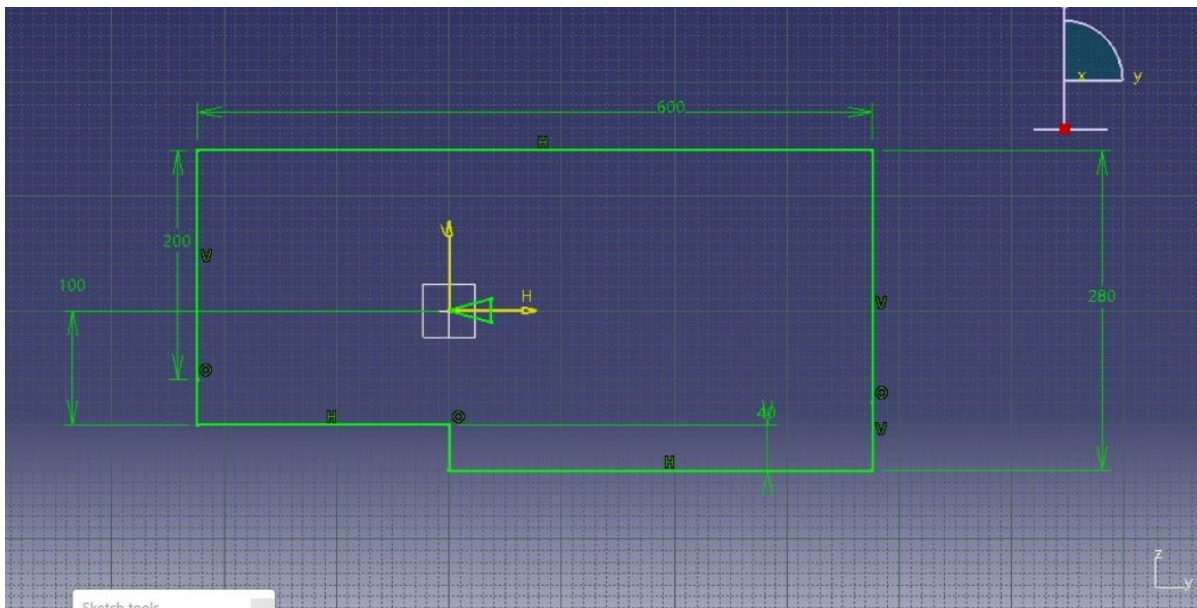


Fig 4.1: Dimensions of the computational domain, where D is the depth of the cavity

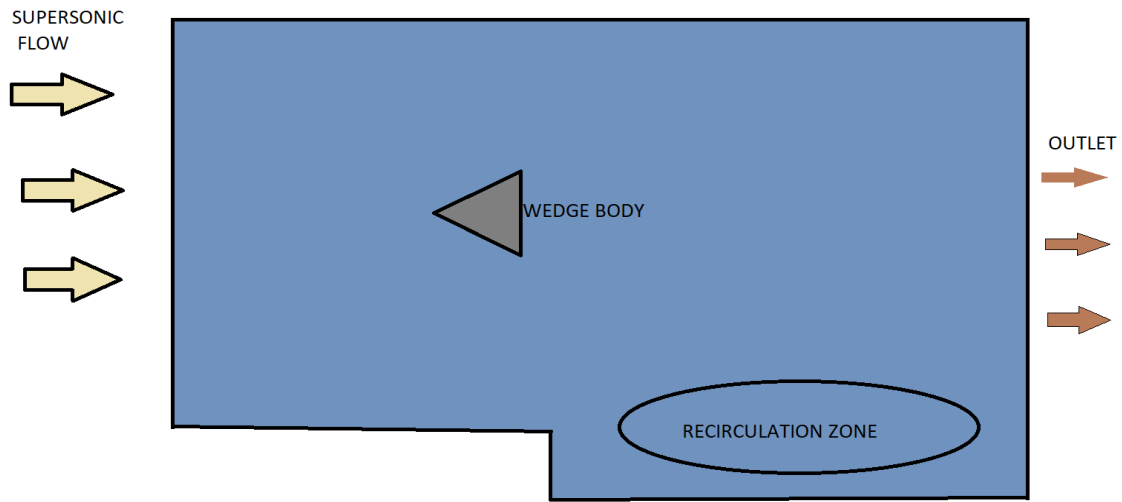


Fig 4.2: displays the 2D sketching of the computational domain.

A computational domain refers to the extension for which the governing equations must be calculated in order to obtain results concerning the flow around a geometrical object under investigation. The main objective behind the creation of the computational domain is to discretize or mesh the space within the domain. The creation of a domain must be performed after considering what type of problem is being solved, external or internal aerodynamics. The domain in figure 11 is an example of external aerodynamics since the flow, although entering the cavity, migrates over the wall which the cavity is a part of. So for these problems, the boundary of the domain must be placed adequately away from the object's surface that is in focus. In contrast an oversized domain may also pose problems, as more space means more grids or elements that might lead to more computation time. For this current project, a cavity of horizontal span to profoundness ratio of 3 is being investigated. The shape of the domain is fairly simple, which is a rectangular box with a cavity protruding at the bottom boundary of the domain. The shape is kept fairly simple for easier assigning of dimensions and boundary conditions. The dimensions of the domain is chosen in such a way that any major alterations that have been brought about to the flow, either by the presence of cavity or shock or their interaction, is well within the boundary of the domain. The dimension of the domain that was used for computational study is given in figure 1. The

entire domain was calculated with respect to the depth of the cavity. The wedge or the shock generator is kept at a height twice the depth, such that the impingement of shock occurs on the shear layer over the cavity. The nose of the wedge is placed at a height twice the depth while the other heights at which the other two edges are positioned vary with the angle of the wedge. The determination of the horizontal span and the profoundness of the cavity is determined by the base paper by Gruber et al. The cavity downstream length is twice that of the upstream span. This was done in order for the better comprehension of flow field aft of the shock and the shock generator. The pressure far-field is positioned at a height 25 times the depth of the cavity.

4.1.2. SHOCK GENERATOR OR WEDGE

An object (or in this case wedge as shown in figure 4.), when moving through air at a velocity that is greater than the sound's speed, results in compression of air flow thus resulting in the formation of a shock wave. Shock wave is an irreversible occurrence in supersonic flow properties like pressure, temperature, density etc. It is a region of discontinuity inside which the mechanisms of friction and thermal conduction take place which redeems the shock internally irreversibly giving rise to increase in entropy across it. The oblique shock wave that has thus originated is attached to the nose of the edge. The flow both upstream and down-stream of the shockwave experiences enormous alterations in the flow-field.

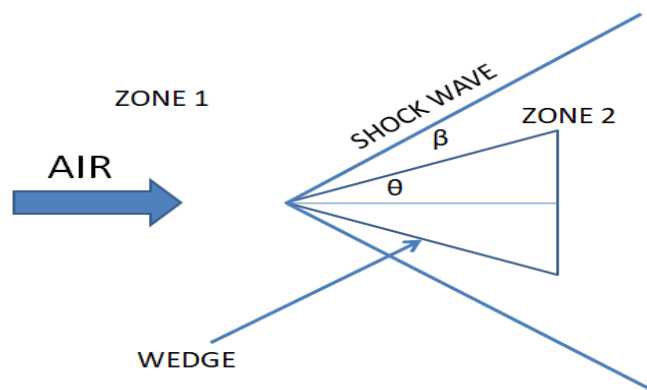


Fig 4.3 : Flow past a wedge. Where θ - half wedge angle and β is the shock angle.

4.1.2.1 Equations for shock wave relations:

(Where,

- The subscript 1 denotes the flow characteristic upstream of the shock
- The subscript 2 denotes the flow property downwards of the shock; and P denotes the air pressure at that point.
- T refers to the temperature of the fluid •
- ρ is that its density
- M is its Mach number
- γ is the heat capacity ratio

$$M_2 = \csc(\beta - \theta) \sqrt{\frac{2 + (\gamma - 1) M_1^2 (\sin \beta)^2}{2 \gamma M_1^2 (\sin \beta)^2 - (\gamma - 1)}}$$

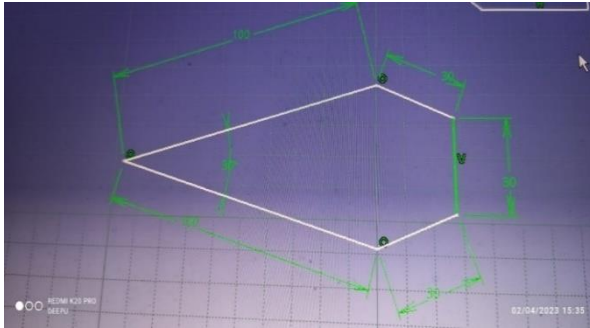
$$\frac{P_2}{P_1} = \frac{2 \gamma M_1^2 (\sin \beta)^2 - (\gamma - 1)}{\gamma + 1}$$

$$\frac{\rho_2}{\rho_1} = \frac{(\gamma + 1) M_1^2 (\sin \beta)^2}{2 + (\gamma - 1) M_1^2 (\sin \beta)^2}$$

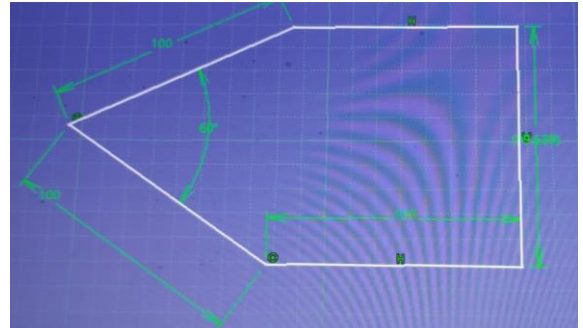
$$\frac{T_2}{T_1} = \frac{[(\gamma - 1) M_1^2 (\sin \beta)^2] [(\gamma - 1) M_1^2 (\sin \beta)^2]}{(\gamma - 1)^2 M_1^2 (\sin \beta)^2}$$

Fig 4.4: The equation for shock wave relation

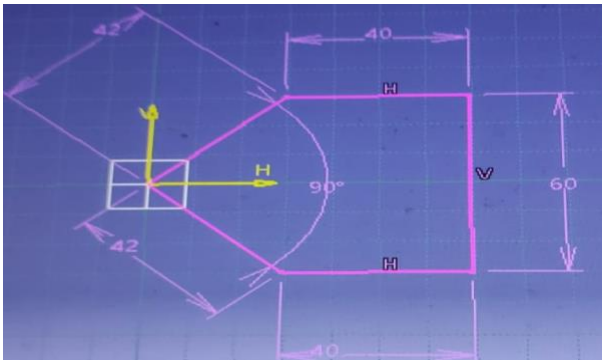
The lengths of the wedge for all the cases were kept constant which is 100mm. The constant wedge that is utilized for varying positions and the positioning distance is measured with length of the wedge body from initial position. This process is applied for all different bluff bodies with specific dimension as shown below



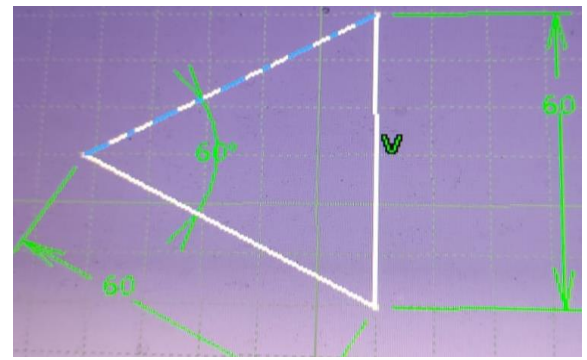
(A) PENTAGONAL WEDGE BODY



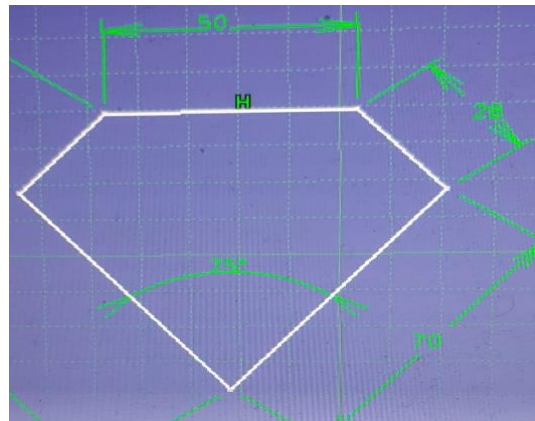
(B) WEDGE WITH SQUARE ENDING



(C) WEDGE WITH RECTANGLE ENDING



(D) TRIANGULAR WEDGE BODY



(E) DIAMOND BODY

Fig 4.5: The below pictures display the dimensions of each bluff body utilized in the computational domain

4.1.3. Meshed Domain:

There are two types of 2-D meshes:

- Quadrilateral
- Triangle

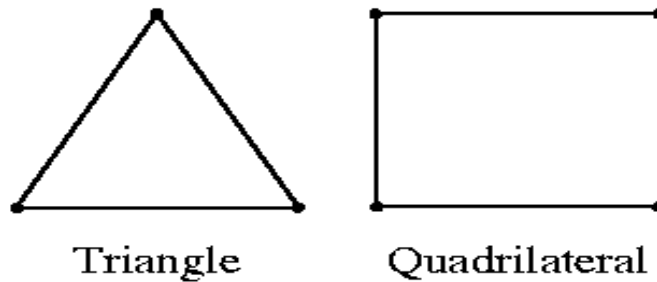


Fig 4.6: Types of mesh elements.

Table 4.1: Differences between quadrilateral and triangular mesh element

Quadrilateral	Triangle
• Low numerical diffusion. • Minimal discretization error. • For flow not perpendicular to the flow, the value of diffusion rises. • For intricate geometries, the possibility of generation of structure mesh is low. • Inadequate for industrial applications.	• Possibility of automatically producing the mesh at a large degree. • Redundant elongation not achievable. • Higher quantities of elements are required. • Adjoining nodes prevail in the same plane which complicates computation. Therefore, inferior convergence.

For the current project, the computational domain is maintained fairly uncomplicated. Therefore, the necessity for automated production of meshed elements is absent. Also, the production of automated mesh results in the requirement of more memory space as both the value and location of each individual cell are to be computed each time. So a quadrilateral mesh is chosen. This can be seen in figure 4.7. The quadrilateral mesh ensures that the error caused due to numerical diffusion remains fairly low. The use of this mesh allowed for specification of cell distance from the wall. This facilitated the close interpretation of shear layer

effects at nearer portions of the wall. The regions in the proximity of the cavity are heavily meshed so as to visualize the flow more accurately. The visualization of the interaction of the shock wave and the shear layer is critical for the current study. Hence this region contains relatively higher grids than the region close to the pressure far field, where little to no alterations occur.

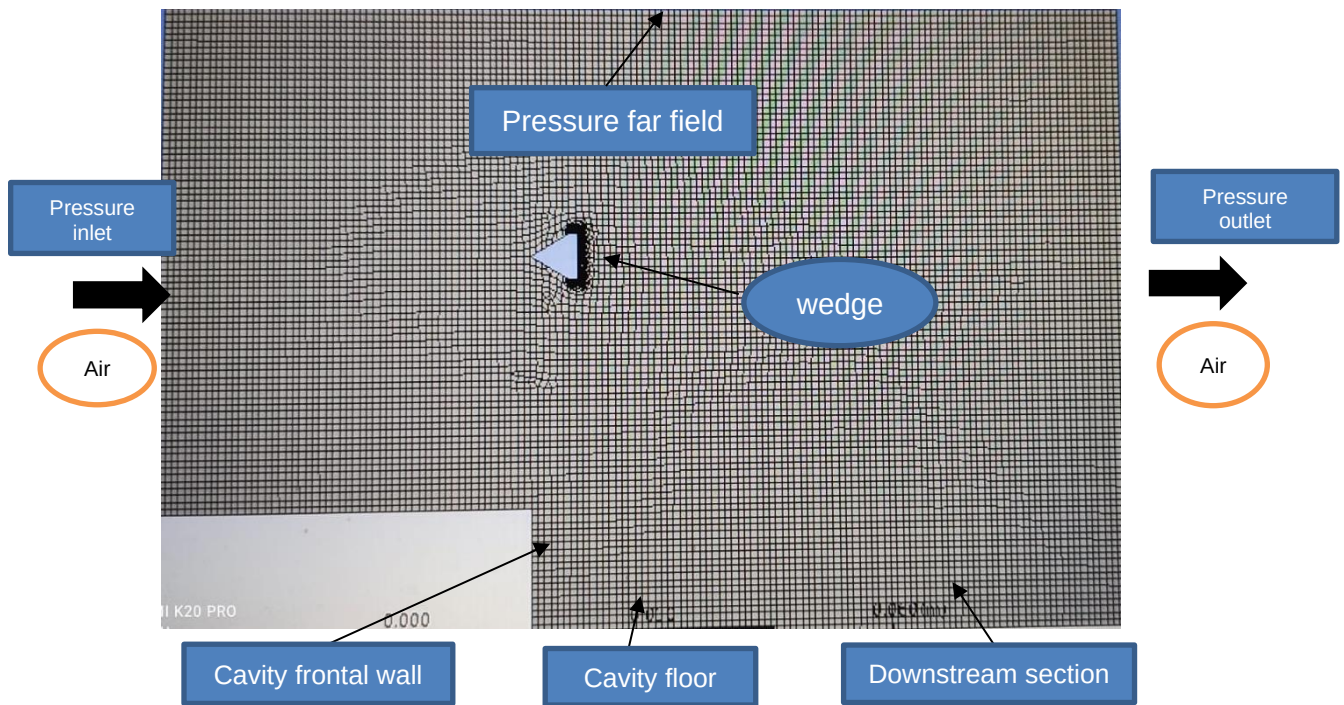


Fig 4.7: Meshed domain for the current study

4.1.4. Solver Setup:

For the current study, the commercial software Ansys Fluent © is chosen.

4.1.4.1. SOLVER TYPE - DENSITY BASED

The concurrent interpretation of the governing equations including energy equation can be achieved by utilization of density-based solvers. There exists further classification based on how the linearization of the above-mentioned equations are carried out.

Implicit - While solving a particular cell the unknown value from adjacent cells that is yet to be solved are included in the equation. The appearance of each unknown occurs in multiple equations. Therefore, there arises a requisite for concurrent solving of the entire domain.

Explicit - While solving a particular cell the known values of the adjacent cells are included in the equation that is solved in order to gain a solution. The appearance of each unknown occurs only once. Therefore, computation is done in one cell after another.

4.1.4.2. SOLVER TIME – STEADY

The objective of the project is the comprehension of the alterations brought about in the basic flow attributes. Hence as an example, steady state flow is appropriate it allows this and other complicated flow attributes which are time-dependent can be analyzed further in the future. Also, the steady-state analysis provides excellent results that can be matched with theoretical calculations. Although in order for accuracy and for matching with real-time flow alterations transient state analysis has to be employed. Since the current study is fairly new, steady-state analysis has been adopted.

4.1.4.3. MODELS

4.1.4.3.1. ENERGY

The generation of shock in the current study provides the necessity for the inclusion of the energy equation. The reason for inclusion is to gain appropriate final results and this in turn will aid in the calculation of temperature alteration in the region of the shock.

4.1.4.3.2. VISCOUS - K- ω MODEL

For the consideration of the turbulence effect of the flow under analysis, a viscous model is adopted. For the current study k- ω model is chosen. This model includes

during computation, in addition to the governing equations, two transport equations. The two transported variables are:

- k - turbulent kinetic energy, for ascertaining the energy in turbulence
- ω - Specific turbulent dissipation rate (or) scale of turbulence, for ascertaining the rate of dissipation per unit turbulent kinetic energy.

This model is suitable for the present study as the shear layer close to the wall is the topic of interest.

4.1.4.4. MATERIALS - DENSITY - IDEAL GAS

The ideal gas is selected to implement the ideal gas law, wherein the only interaction that occurs between the molecules of a gas is a collision, whose energy is conserved, while neglecting the intermolecular attraction or repulsion that is present in real gases.

The ideal gas equation:

$$PV = nRT$$

Where, P - Pressure of the gas

V - Volume occupied by the gas

n - Amount of substance

R - Ideal gas constant

T - Temperature of the gas

4.1.4.5. SPATIAL DISCRETIZATION

This type was used because of low computational cost. Low computational cost refers to low computation time and memory. However, this may result in the accuracy of the computation being compromised due to excessive diffusion.

4.1.5. VALIDATION:

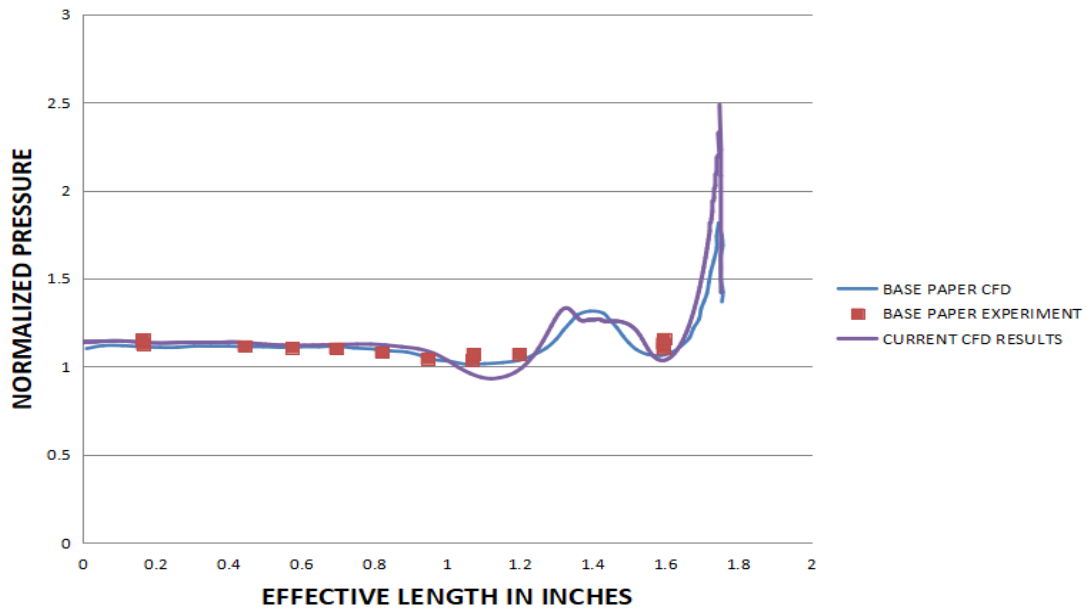


Fig 4.8: Normalized Pressure vs. Effective Length in Inches

For the validation of the solver that is going to be used for the current study, the effective length vs. normalized pressure graph is being utilized as it is in the base paper as depicted in figure 8. The validation is carried out without the presence of the wedge.

Where, Normalized Pressure = $\frac{\text{Static pressure on the wall}}{\text{Free Stream Pressure}}$

The effective length is taken in terms of inches of the cavity wall. The overall graph agrees in terms of both computational and empirical data obtained from the base paper with the computational data obtained for the current study. This allows for further proceeding of conducting the study.

4.1.6. GRID INDEPENDENCE TEST:

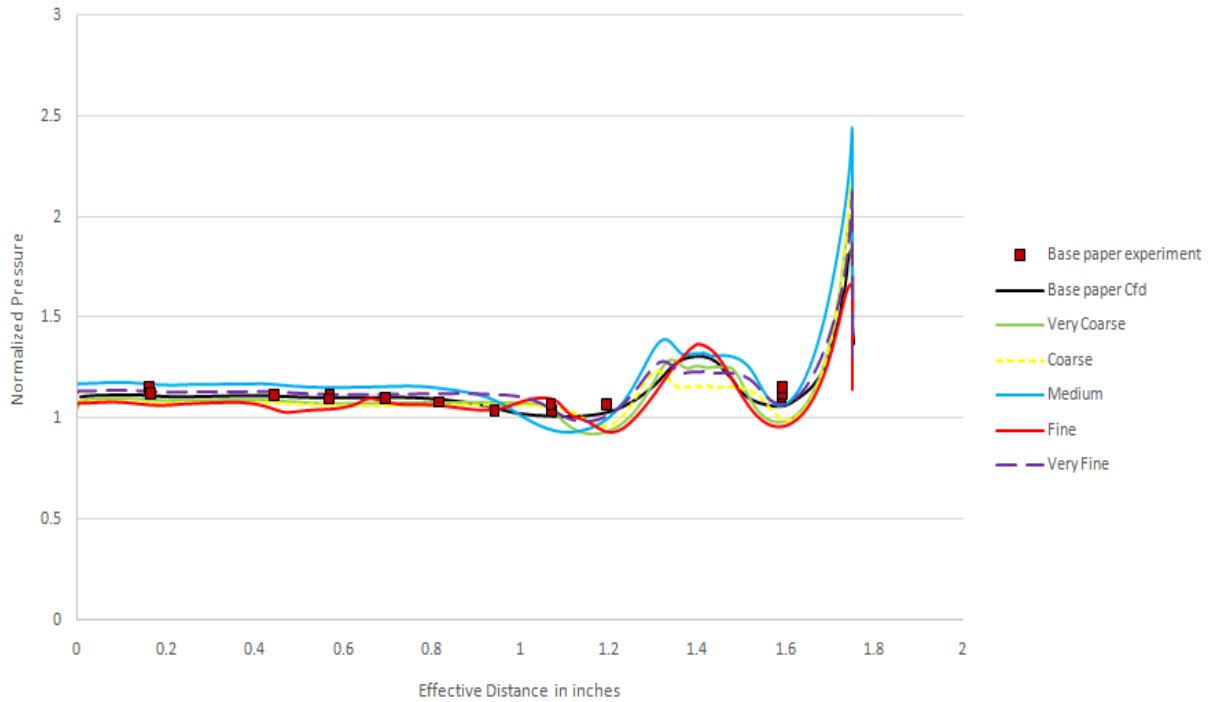


Fig 4.9: Grid Independence Test

The next step was to conduct the grid independence test. This is done to make sure the values that are obtained through the study are influenced only by the conditions of the flow that is provided and not by the refinement of the mesh. For the choice of a suitable grid for the current study, the multiple grids were compared and that resulted in the selection of the suitable grid i.e. medium.

4.2. EXPERIMENTAL SETUP:

4.2.1. SUPERSONIC WIND TUNNEL:



Fig 4.10: Photograph of wind tunnel facility.

The experimental analysis was achieved in the supersonic wind tunnel, which was a part of the facility provided in the supersonic wind tunnel laboratory. The test section housed both the cavity and the wedge model. The air in the tank is compressed with the help of a compressor. This air is then allowed to pass into the test section by opening the valve of the settling chamber. The air then passes through the test section where the model is kept. On either side of the test section, thick sheets of acrylic are attached to provide visibility of the flow. The pressure transducers are fixed to the acrylic casing. These transducers measure the value of pressure at various locations of the test section. The air is then allowed to exit the nozzle, which is attached to the

end of the test section. The cross-section of the dimension of the test section is 25mmx25mm. For the present study the chamber pressure is operated at 1-2 kg/cm². The test was carried out at Mach 1.2, which is the maximum capacity of the facility. The C-D nozzle which is a part of the test section is designed so as to achieve the required flow conditions.

4.2.2. CAVITY AND WEDGE MODEL:

The cavity model is designed with L/D ratio is 3. Since the position where the wedge is mounted is fixed, the depth of the cavity is taken with respect to the distance between this position and the lower half of the test section. Therefore, the indent created in the lower half of the wind tunnel for the cavity has a depth of 11.5 mm. The wedge is then modeled out of stainless steel. The wedge is fixed in the holder as shown in figure 19. The screw at the end is a standard M4 screw (outer diameter = 4 mm). The wedge model that was designed must have a threaded hole in order to fix it to the holder. The cavity model was also scaled in accordance with the cavity depth.



Fig 4.11: Wedge holder

The entire test section, portrayed in figure 20, is made of two stainless steel rod, one at the bottom and the other at the bottom. The rod is designed so as to form C-D nozzle

that controls the attributes of the flow over the model. This setup has a casing acrylic on both sides. The acrylic casing at the back has taps through which pressure transducer pipes are attached

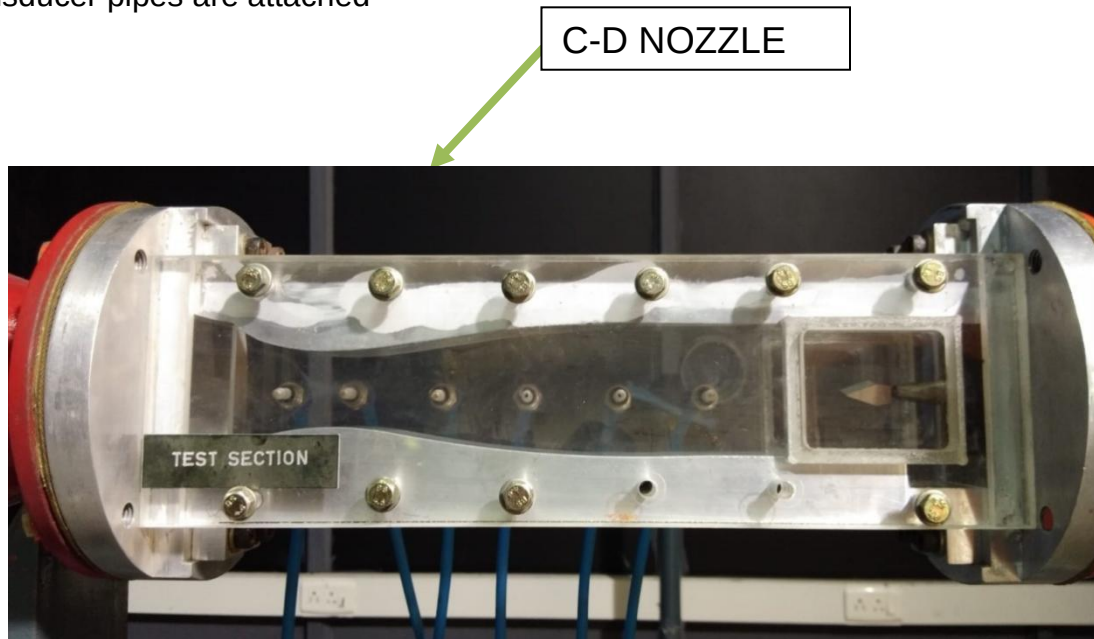


Fig 4.12: Test section

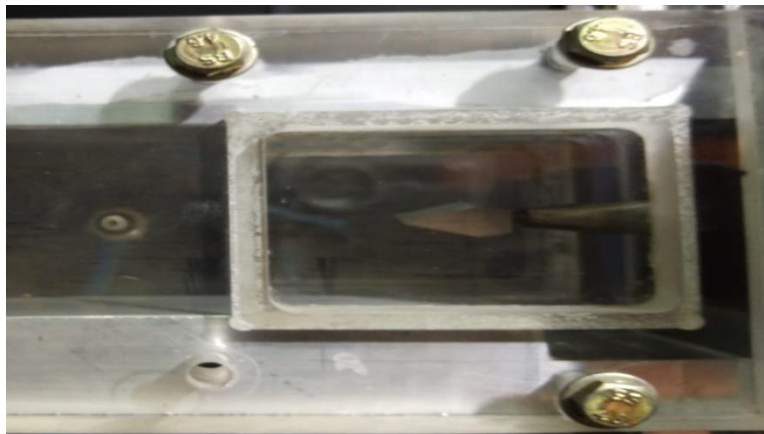
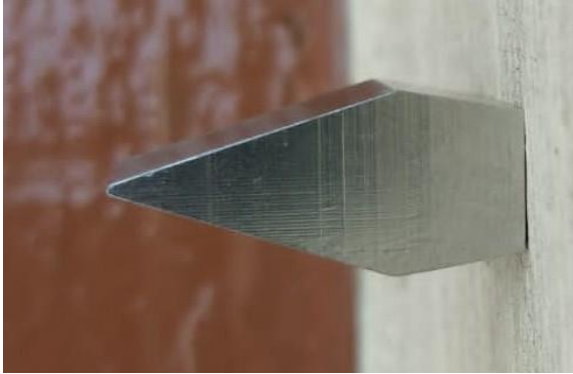


Fig 4.13: Close-up of the wedge and cavity model mounted in the test section

The wedge model is fixed into the holder as shown in Fig 4.13. For the design of the cavity model the entire stainless-steel rod is designed entirely and a groove is made according to the dimensions of the cavity. The groove thus made is L-shape. The rectangular hole through which the exits, acts as the hind wall of the cavity.

In contrast to the design used in the computational analysis, the wedge that was fabricated had to have an excess projection of the flat surface at the aft of the wedge. This projection is to accommodate the threaded hole that is made for the fixture.



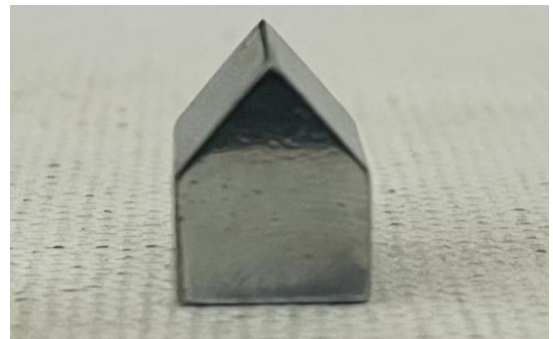
(A)



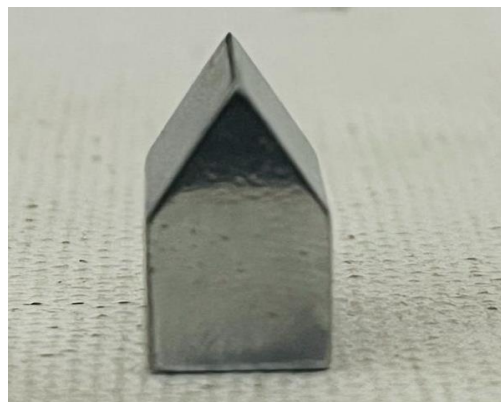
(B)



(C)



(D)



(E)

Fig 4.14: The above picture displays different fabricated wedge bodies from A to E



(F)



(G)

Fig 4.15: Picture F and G showcases the threaded hole and screw utilized for holding the bluff body in the test section and changing the positions

4.2.3. MATERIAL USED

The material used in order to fabricate the wedge bodies is Mild Steel by CNC (Computer numerical control) cutting method. Mild steel is a popular material for the fabrication of various components, including wedge bodies used in supersonic combustor test sections. Mild steel is an alloy of iron and carbon, typically containing less than 0.25% carbon by weight. It is known for its strength, durability, and ease of fabrication. Mild steel parts are substantially more resistant to scratches and impact damage than aluminum parts. Mild steel alloys are increasingly being used in aircraft components that demand high strength while also being able to withstand the increased weight.

CHAPTER 5

RESULTS AND DISCUSSION

5.1. COMPUTATIONAL RESULTS:

The computational results for the wedge bluff bodies were obtained through a 2D geometry design approach, which involved keeping all the bodies within the boundaries of the 2D Step combustor domain. This approach was chosen because it simplifies the design process and computational analysis, allowing for more efficient simulations.

To carry out the simulations, the wedge body was initially placed at the step position within the combustor domain. Then, the simulation was performed to obtain the results. The same process was repeated for all the differently shaped wedge bodies. Additionally, the wedge body was moved forward up to three positions towards the inlet of the combustor, and the simulation was again performed to obtain the corresponding results. The above-described process ensured that each of the wedge bodies was evaluated under similar conditions, which allowed for a fair comparison of their respective performances.

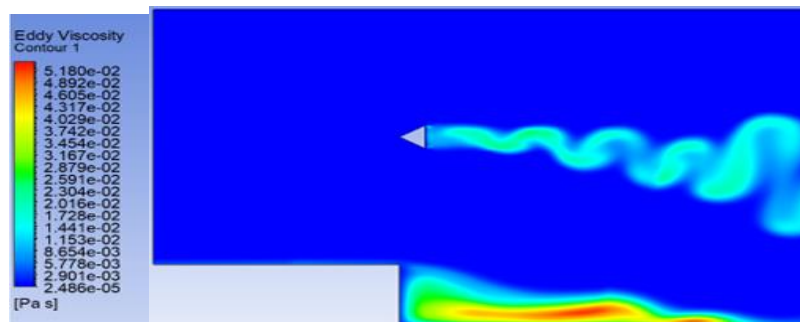
To study the flow behavior in a step combustor domain with a wedge body inside it using computational fluid dynamics (CFD), it is important to carefully consider the fluid conditions that can accurately model the complex flow behavior. In this research work, the primary fluid conditions that are considered essential to visualize the recirculation zone are eddy viscosity, turbulence kinetic energy, and velocity u .

Eddy viscosity is a measure of the turbulent viscosity in a fluid, which is critical for accurately modeling turbulent flows. Turbulent flows are characterized by chaotic, unpredictable motion and are common in high-speed or complex flow scenarios. By estimating the effects of turbulence on the flow behavior, eddy viscosity models allow for a more accurate prediction of the flow field.

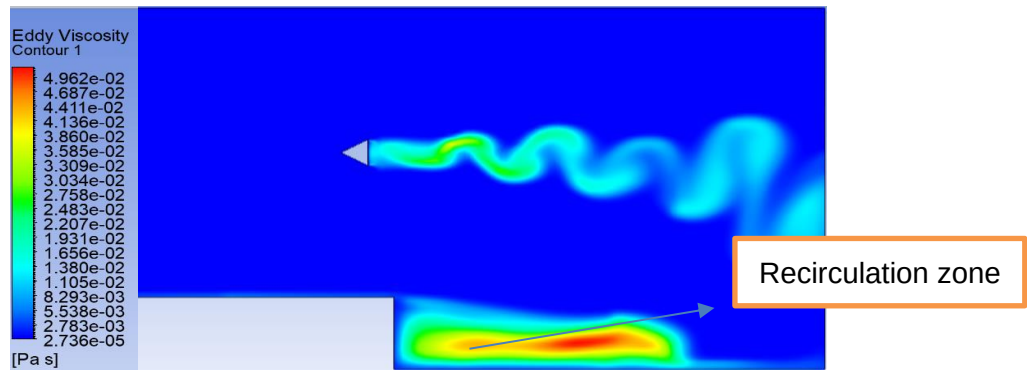
Turbulence kinetic energy (TKE) is another important fluid condition to consider in CFD simulations. TKE measures the energy associated with turbulent motion in a fluid and provides information on the size and intensity of turbulent eddies. By incorporating TKE in the simulation, it is possible to better understand the nature of turbulence in the flow and how it affects the recirculation zone.

Finally, velocity u refers to the velocity of the fluid at a given point in the domain. It is a fundamental fluid property that is necessary for any CFD simulation and provides crucial information on the direction and magnitude of the flow. In the context of this research work, velocity u is vital for visualizing the recirculation zone and understanding how it interacts with the wedge body.

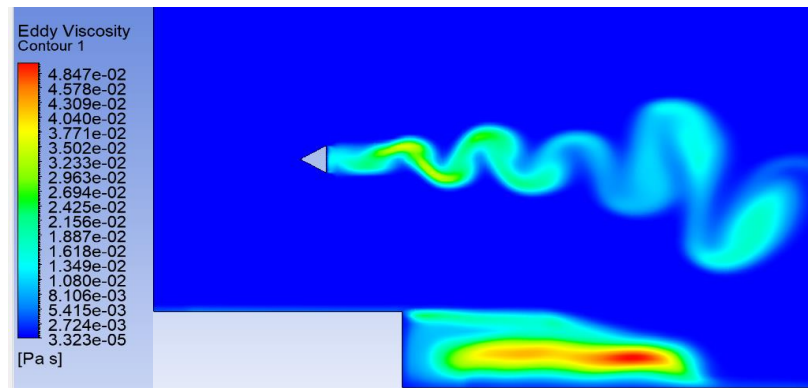
While there are many other fluid conditions that can be important in CFD simulations, these three - eddy viscosity, turbulence kinetic energy, and velocity u - are the most critical to accurately model the flow behavior in the step combustor domain with a wedge body inside it. By carefully selecting and modeling these fluid conditions, this research work aims to gain insight into the nature of the recirculation zone and how it interacts with the wedge body, ultimately contributing to a better understanding of the flow behavior in this complex geometry.



Position A

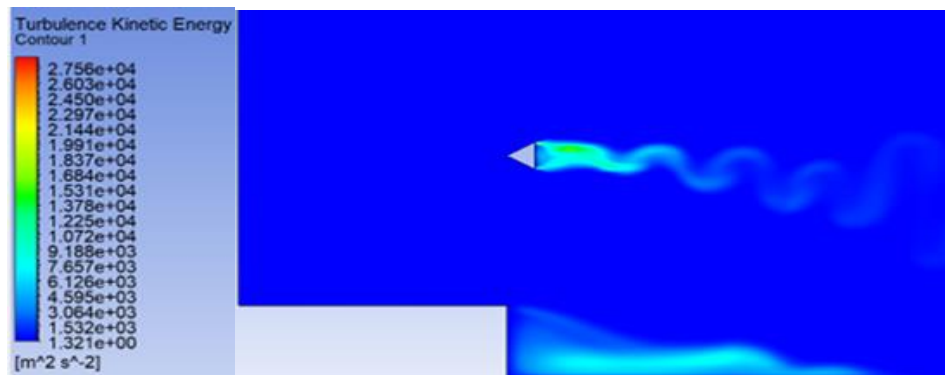


Position B

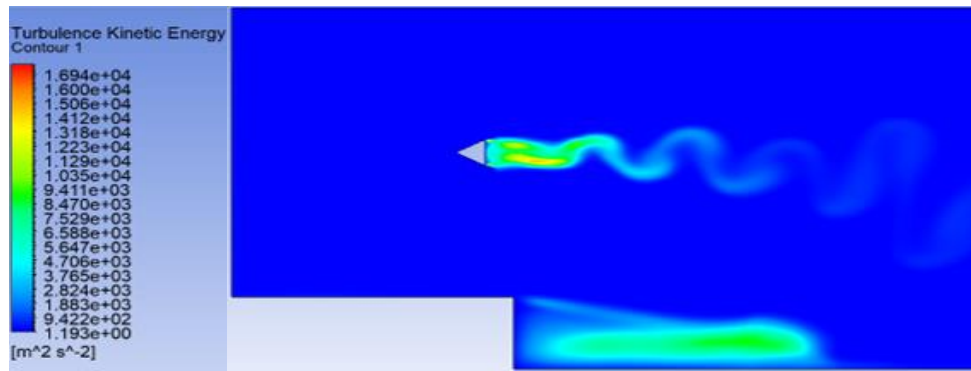


Position C

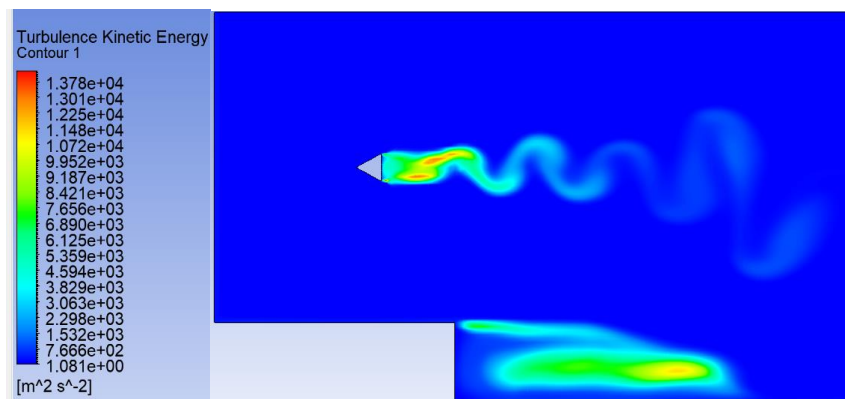
Fig 5.1: The Eddy viscosity fluid condition results at different positions of the wedge body from positions A to C



Position A

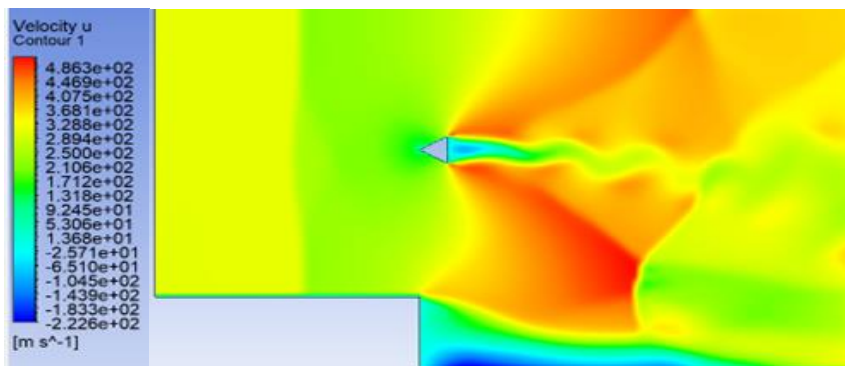


Position B

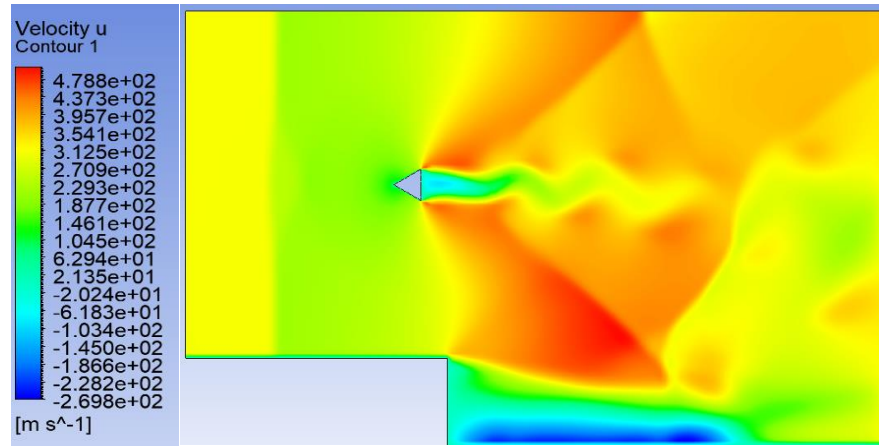


Position C

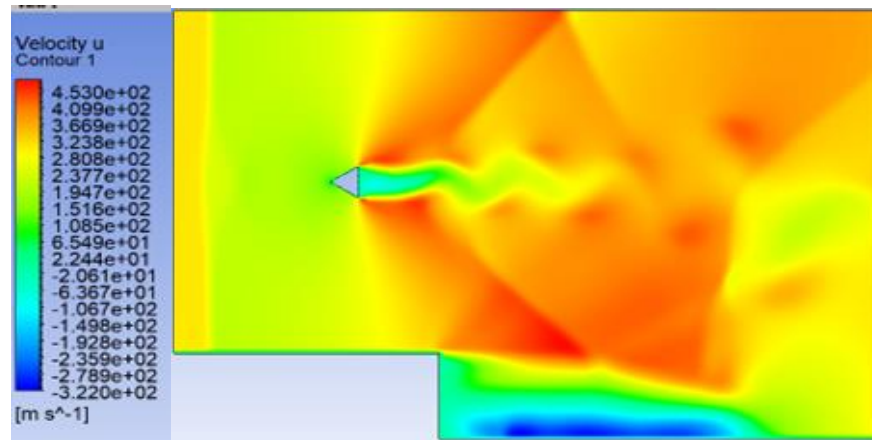
Fig 5.2: The Turbulence kinetic energy fluid condition results at different positions of the wedge body from A to C



Position A



Position B



Position C

Fig 5.3: The Velocity U fluid condition results at different positions of the wedge body from positions

From the above pictures we can visualize the recirculation zone at different fluid conditions at different positions. The drag coefficient (C_D value) of a wedge-shaped body placed in a supersonic step combustor depends on several factors, such as the shape and size of the wedge, the flow conditions, and the wedge's placement in the combustor. Generally, introducing a wedge into a supersonic flow increase drag, which can result in a higher C_D value. However, the wedge's precise impact on the C_D value relies on its placement in the combustor and the specific flow conditions. In a supersonic step combustor, the wedge may be strategically positioned to encourage

proper fuel-air mixing and combustion efficiency. If the wedge is arranged to produce a recirculation zone that enhances mixing, it could reduce the CD value by minimizing the wake size behind the wedge. Ultimately, determining how a wedge-shaped body affects the CD value in a supersonic step combustor necessitates specific information about the geometry and flow conditions.

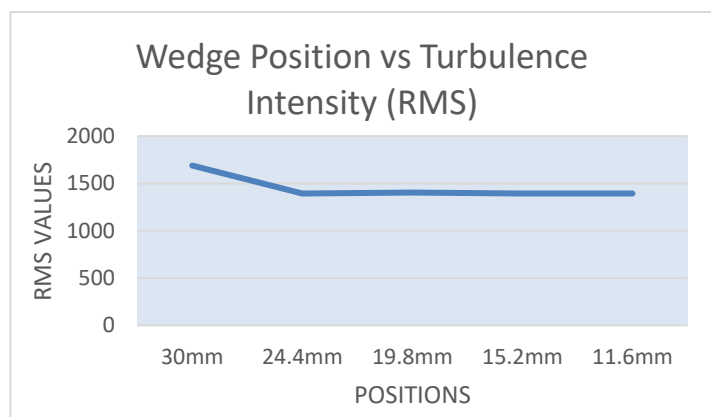
5.1.1 Relation between CD, RMS and RECIRCULATION AREA

The RMS value, Cd and recirculation zone are interconnected factors in the study of supersonic step combustors that utilize wedge bodies to increase the recirculation area. The RMS value represents the turbulence intensity of the flow, which reflects how much the velocity varies at different points in the flow. Cd, on the other hand, measures the resistance of the wedge body to the flow of air, while the recirculation zone is an area in the flow where the air flows back on itself, forming a vortex that enhances fuel-air mixing for combustion.

When the position of the wedge body is moved along the axes, it affects the flow of air around it, which in turn influences the size of the recirculation zone and the turbulence in the flow. By measuring the Cd and the RMS value at different positions of the wedge body, researchers can evaluate the impact of the wedge body position on the recirculation zone. For instance, a wedge body position that increases the recirculation zone may also increase the turbulence and the drag coefficient (Cd), resulting in a higher RMS value. Conversely, a position that decreases the recirculation zone may also reduce the turbulence and the drag coefficient, leading to a lower RMS value. Therefore, by analyzing the relationship among the RMS value, Cd, and the recirculation zone, researchers can optimize the position of the wedge body to achieve the best trade-off among these factors, resulting in optimal fuel-air mixing and combustion efficiency.

Table 5.1: The plots below is considered for position of wedge vs RMS

Positions	RMS VALUES
30mm	1688.993
24.4mm	1393.566
19.8mm	1403.567
15.2mm	1393.879
11.6mm	1393.879



In this graph, the position values are on the x-axis and the RMS values are on the y-axis. Each data point represents a specific position and its corresponding RMS value. As you can see, the RMS values appear to be relatively constant across different positions of the wedge body, with only slight variations.

The x-axis of the graph represents the different positions of the wedge body, ranging from 11.6mm to 30mm. The y-axis represents the corresponding RMS values, which range from 526.5108 to 561.5634.

Based on the data in the graph, it appears that the RMS value increases as the position of the wedge body moves from 30mm towards 11.6mm, indicating an increase in turbulence intensity as the recirculation zone increases. This information can be used to determine the optimal position of the wedge body that provides the best trade-off between the recirculation zone and turbulence intensity for optimal fuel-air mixing and combustion efficiency.

Table 5.2: The plots below is considered for position of wedge with square ending vs RMS

Positions	RMS VALUES
30mm	534.2536
24.4mm	526.5108
19.8mm	543.5191
15.2mm	553.8715
11.6mm	561.5634

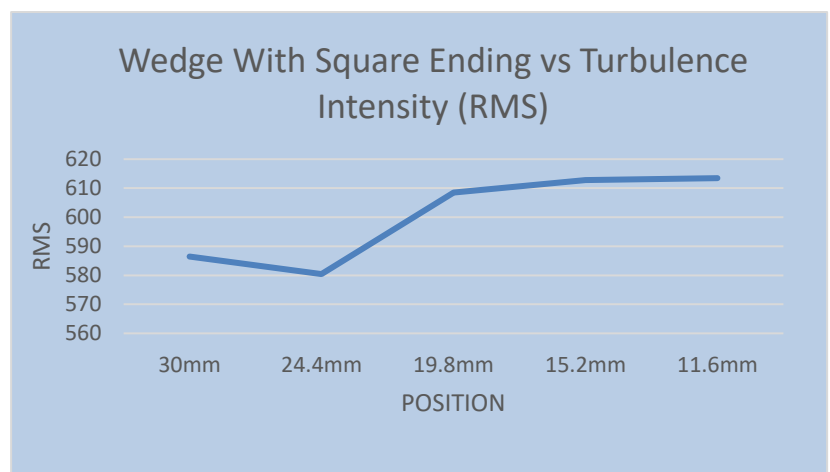
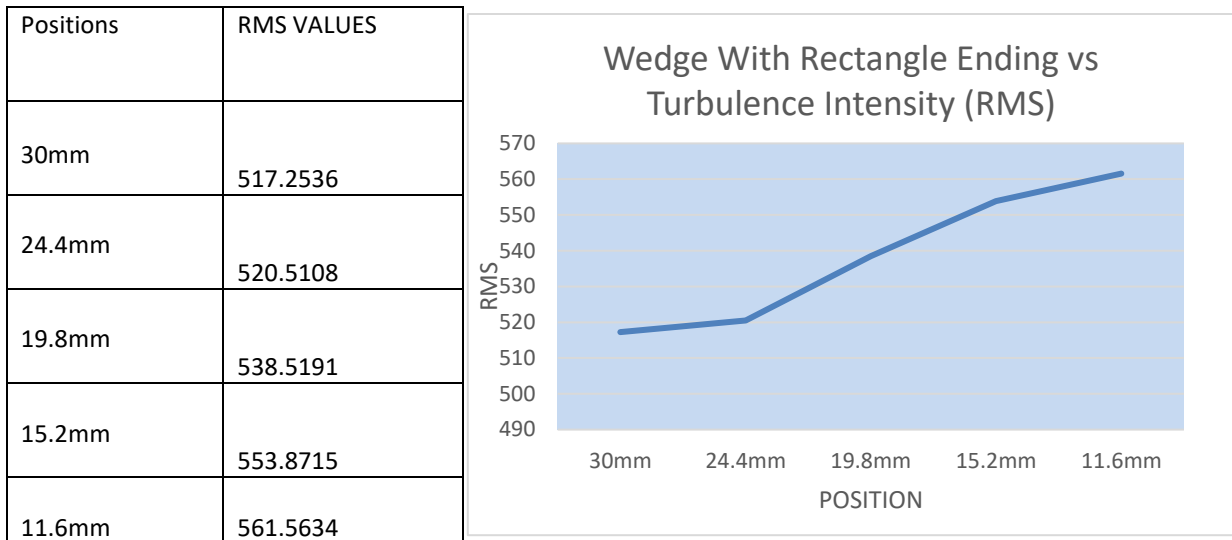


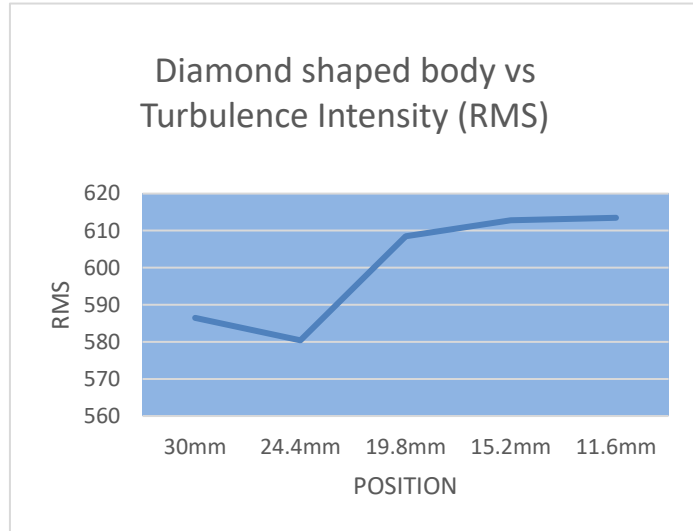
Table 5.3: The graph plotted between the position of wedge with square ending vs Rms



The x-axis of the graph represents the different positions of the wedge body, ranging from 11.6mm to 30mm. The y-axis represents the corresponding RMS values, which range from 517.2536 to 561.5634. Based on the data in the graph, it appears that the RMS value increases as the position of the wedge body moves from 30mm towards 11.6mm, indicating an increase in turbulence intensity as the recirculation zone increases. This information can be used to determine the optimal position of the wedge body that provides the best trade-off between the recirculation zone and turbulence intensity for optimal fuel-air mixing and combustion efficiency. However, it is important to note that further analysis and experimentation may be necessary to fully understand the relationship between these factors and optimize the supersonic step combustor design.

Table 5.4: The graph plotted between the position of diamond shaped body vs Rms

Positions	RMS VALUES
30mm	630.451
24.4mm	620.4218
19.8mm	648.5091
15.2mm	663.8315
11.6mm	665.4634



The x-axis of the graph represents the different positions of the wedge body, ranging from 11.6mm to 30mm. The y-axis represents the corresponding RMS values, which range from 620.4218 to 665.4634.

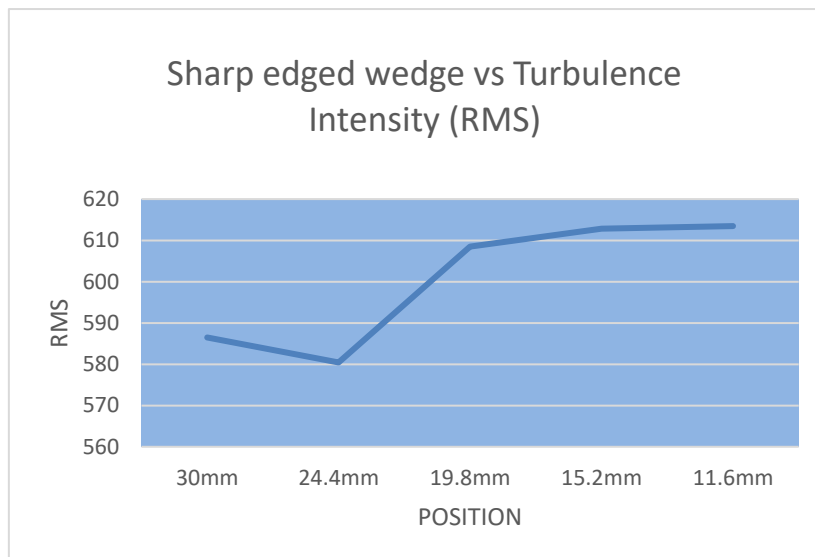
Based on the data in the graph, it appears that the RMS value increases as the position of the wedge body moves from 30mm towards 11.6mm, indicating an increase in turbulence intensity as the recirculation zone increases. However, compared to the previous graph, the RMS values are higher, indicating a higher turbulence intensity for the same positions of the wedge body. This information can be used to determine the optimal position of the wedge body that provides the best trade-off between the recirculation zone and turbulence intensity for optimal fuel-air mixing and combustion efficiency, taking into account the higher turbulence intensity values in this case.

Based on the data in the graph, it appears that the RMS value increases as the position of the wedge body moves from 30mm towards 11.6mm, indicating an increase in turbulence intensity as the recirculation zone increases. However, compared to the previous graphs, the RMS

values are lower, indicating a lower turbulence intensity for the same positions of the wedge body. This information can be used to determine the optimal position of the wedge body that provides the best trade-off between the recirculation zone and turbulence intensity for optimal fuel-air mixing and combustion efficiency, taking into account the lower turbulence intensity values in this case.

Table 5.5: The graph plotted between the position of diamond shaped body vs Rms

Positions	RMS VALUES
30mm	586.451
24.4mm	580.4218
19.8mm	608.5091
15.2mm	612.8315
11.6mm	613.4634



5.1.2 CALUCLATION OF CD VALUE FOR ALL THE WEDGE BODIES USED FOR THE EXPERIMENT

5.1.2.1 TRIANGULAR BODY

To calculate the drag coefficient for a triangular shaped body with an area of 20mm, we need to know the drag force acting on the body, the fluid density, the velocity of the fluid, and the reference area of the body.

Assuming we have the same fluid density and velocity as in the previous example (1.2 kg/m³ and 408.36 m/s, respectively), we need to determine the reference area of the triangular shaped body. Since the body is triangular, its reference area will depend on the orientation of the body relative to the flow direction. Let's assume that the triangular body is oriented such that its base is perpendicular to the flow direction, and its height is parallel to the flow direction. Then, the reference area would be:

$$\text{Reference area} = 0.5 * \text{base} * \text{height} = 0.5 * 20 \text{ mm} * 20 \text{ mm} / (1000 \text{ mm/m})^2 = 0.0002 \text{ m}^2$$

Next, we need to measure the drag force acting on the triangular body experimentally. Once we have measured the drag force, we can solve for the drag coefficient using the same formula as in the previous example:

$$\text{Drag coefficient} = \text{Drag force} / (0.5 * \text{fluid density} * \text{velocity}^2 * \text{reference area})$$

Assuming that the drag force on the triangular body is 0.1 N, we can plug in the known values to obtain:

$$\text{Drag coefficient} = 0.1 \text{ N} / (0.5 * 1.2 \text{ kg/m}^3 * (408.36 \text{ m/s})^2 * 0.0002 \text{ m}^2)$$

$$\text{Drag coefficient} = 1.05$$

So, for this specific case, the drag coefficient of the triangular-shaped body with an area

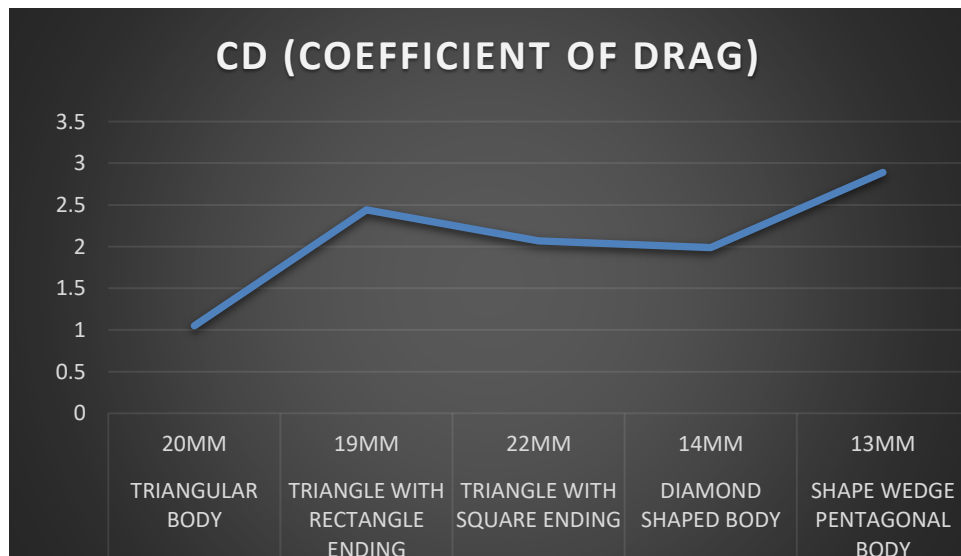
of 20mm is 1.05.

Similarly, we calculated the cd values for all the remaining wedge bodies as shown in the below table 5.6

Table 5.6: Coefficient of drag for all wedge bodies utilized in the project

WEDGE BODY SHAPE	AREA OF TFH BODY	CD (COEFFICIENT OF DRAG)
TRIANGULAR BODY	20MM	1.05
TRIANGLE WITH RECTANGLE ENDING	19MM	2.44
TRIANGLE WITH SQUARE ENDING	22MM	2.07
DIAMOND SHAPED BODY	14MM	1.99
SHAPE WEDGE PENTAGONAL BODY	13MM	2.89

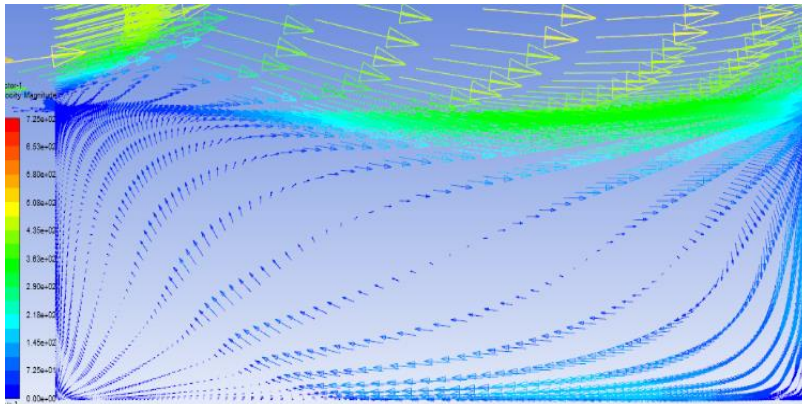
The graph plot based on the above values



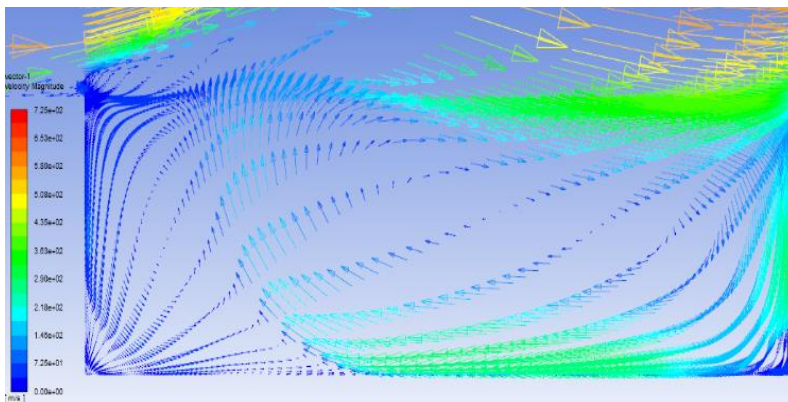
5.1.3 Recirculation zone

5.1.3.1 Recirculation region with respect to Mach

For the increasing Mach number, there is a reduction in the extent of the primary recirculation zone. At Mach 2, the counter rotating recirculation is almost negligible. The migration in the position of the shear layer lifts off dictates the extent of the primary recirculation zone. For each increasing case of Mach number, the position over the cavity where the interaction between the shocks generated from the wedge and the shear layer occur moves closer to the hind wall. The clockwise rotating zone starts from the shear layer lift up position. At Mach 4, there is a formation of small recirculation zone close to the frontal cavity wall. Except for Mach 2, all other cases exhibited the zone close to hind wall rotating at a much higher velocity than the other.



(a)



(b)

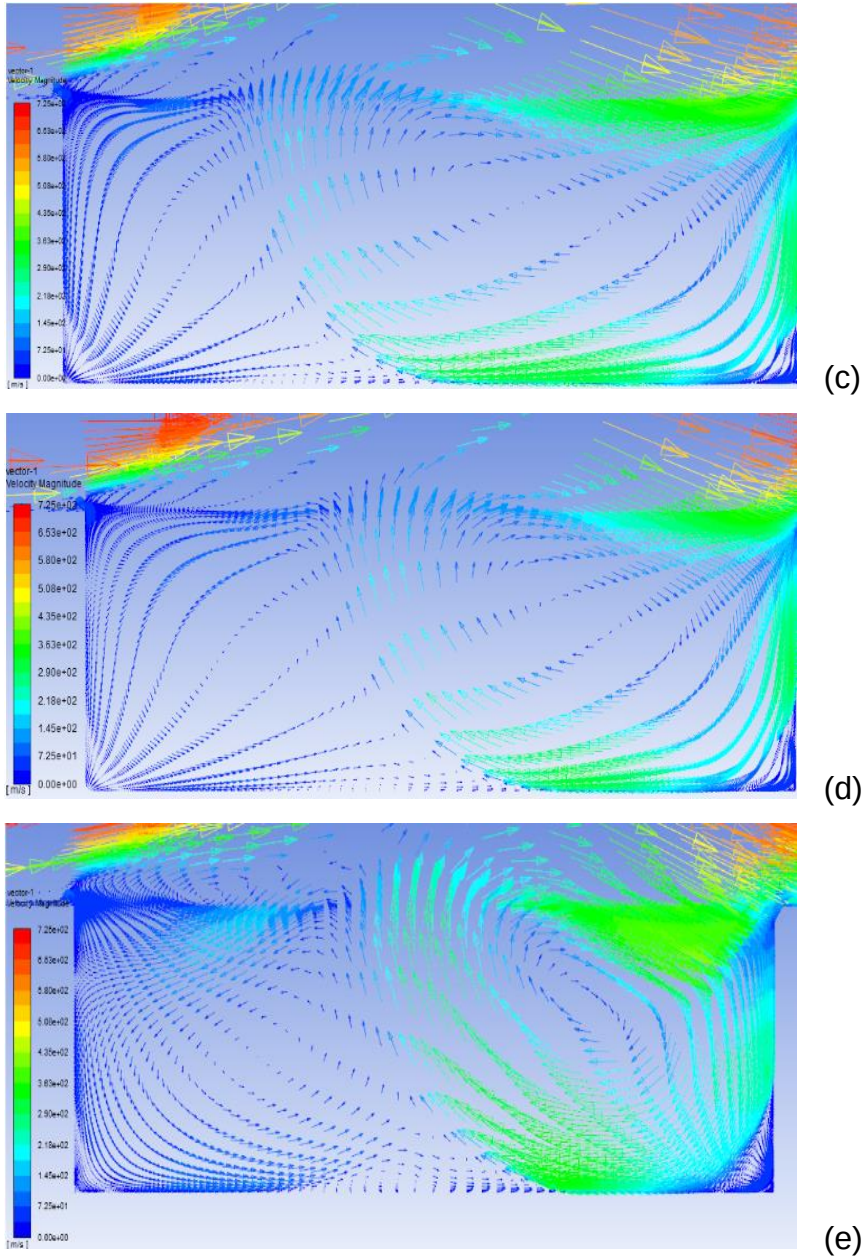


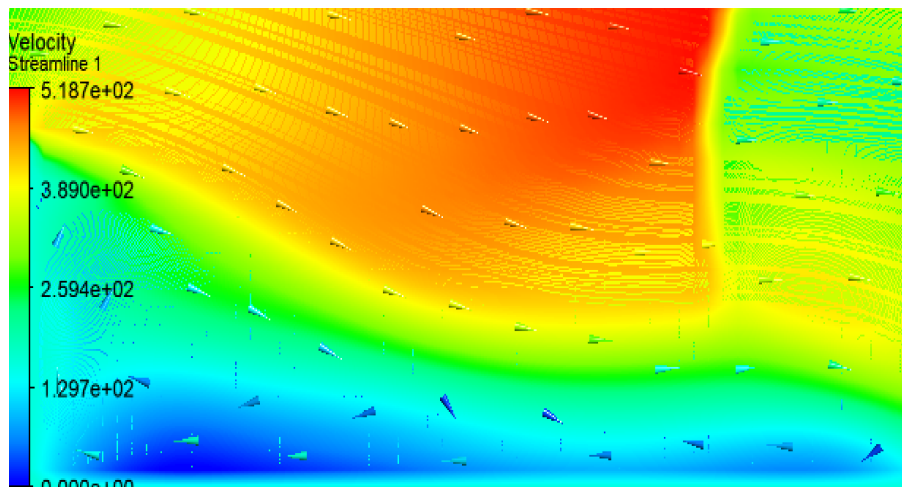
Fig 5.4: Vector plots for (A) Mach 1 (B) Mach 1.5 (C) Mach 2 (D) Mach 2.5 (E) Mach 3

5.1.3.2 Recirculation zone with respective to the shape of the wedge body

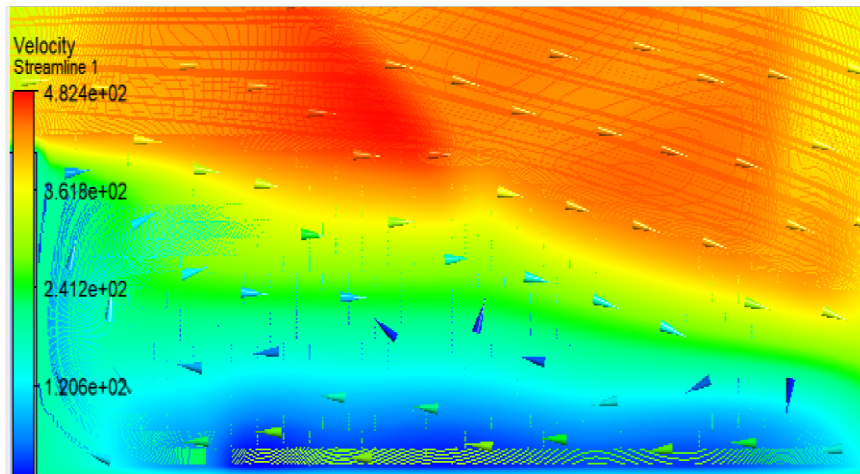
By considering the wedge bodies values from the table 9 we can conclude the efficient recirculation zone. The recirculation is a phenomenon that occurs when the flow separates from the surface of the body and forms a vortex that moves back

towards the body. This can increase the drag on the body and cause other undesirable effects. The likelihood and strength of recirculation depend on several factors, including the shape of the body, the flow conditions, and the properties of the fluid.

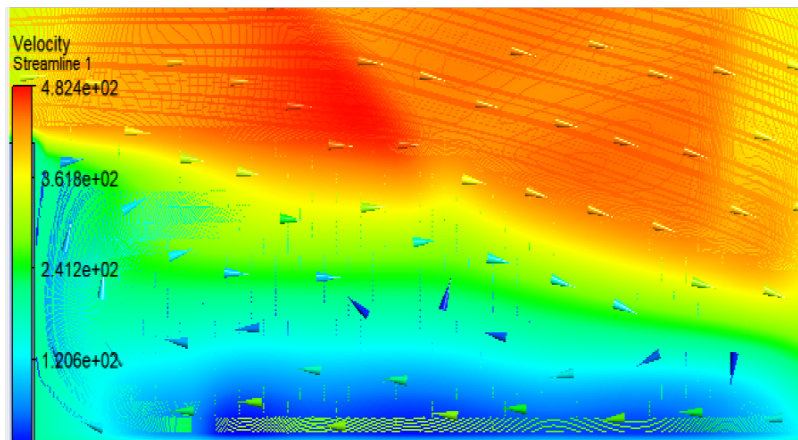
Based on the information provided, it is difficult to determine which body would have the most recirculation without additional information on the flow conditions and fluid properties. However, in general, bodies with sharp edges and corners, such as the wedge pentagonal body and the triangle with square ending, are more likely to experience recirculation than bodies with smoother, rounded shapes, such as the diamond-shaped body. Therefore, it is possible that the wedge pentagonal body or the triangle with square ending may have more recirculation compared to the other bodies listed.



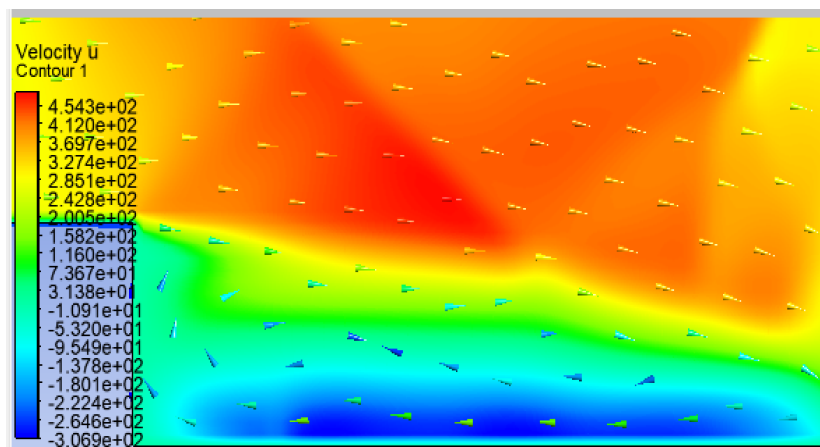
(a) Recirculation zone of Triangular body



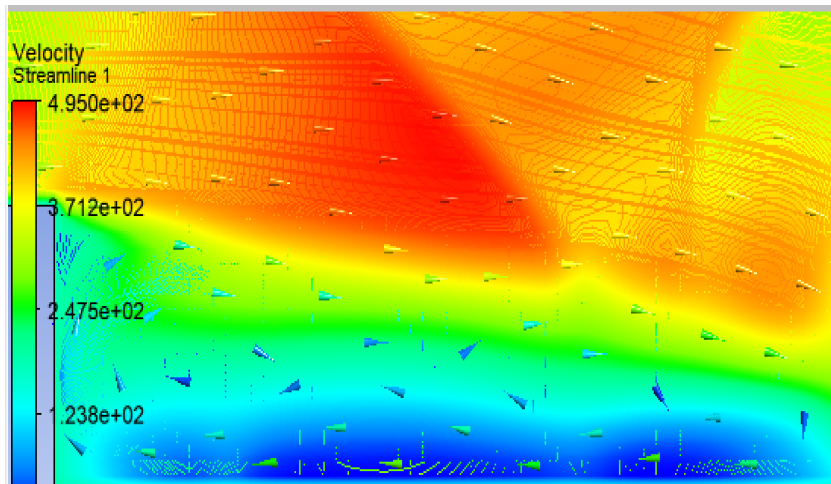
(b) Recirculation zone of triangle with rectangle ending



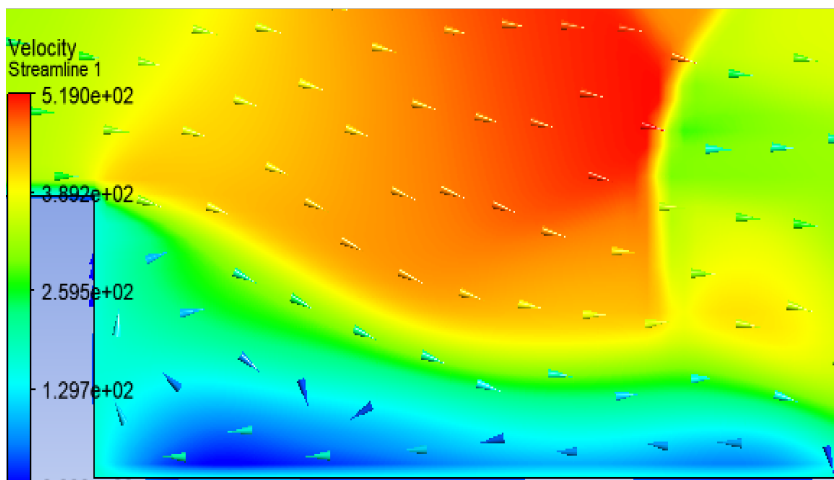
(c) Recirculation zone of triangle with rectangle ending



(d) Recirculation zone of triangle with square ending



(e) Recirculation zone for diamond body



(f) Recirculation zone for Sharp wedge pentagonal body

Fig 5.5 (a-f) Recirculation zone with respective to the shape of the wedge body

5.2 EXPERIMENTAL RESULTS:

5.2.1 The experimental values for the different wedge bodies are:

I. TRIANGLE WITH RECTANGLE ENDING:

CD = 2.44

Pressure = 53375 Pa

Temperature = 240 K

Velocity = 373 m/s

II. TRIANGLE WITH SQUARE ENDING:

CD = 2.07

Pressure = 53225 Pa

Temperature = 240 K

Velocity = 371 m/s

III. DIAMOND-SHAPED BODY:

CD = 1.99

Pressure = 53375 Pa

Temperature = 240 K

Velocity = 371 m/s

IV. WEDGE PENTAGONAL BODY:

CD = 2.89

Pressure = 53200 Pa

Temperature = 240 K

Velocity = 371 m/s

V. TRIANGULAR BODY:

CD = 1.05

Pressure = 53550 Pa

Temperature = 240 K

Velocity = 377 m/s

5.2.2 Impedance occurrence during experimental work

- From the below pictures we can conclude that during the supersonic flow over the wedge body it gets tilted due to the shock impedance occurred on it.
- By this schlieren photographic method we can able to visualize the shocks occurrence on the different wedge bodies

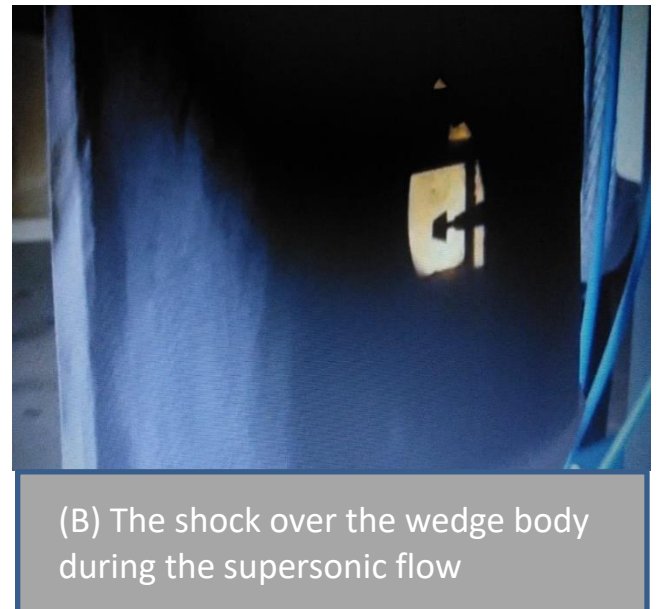
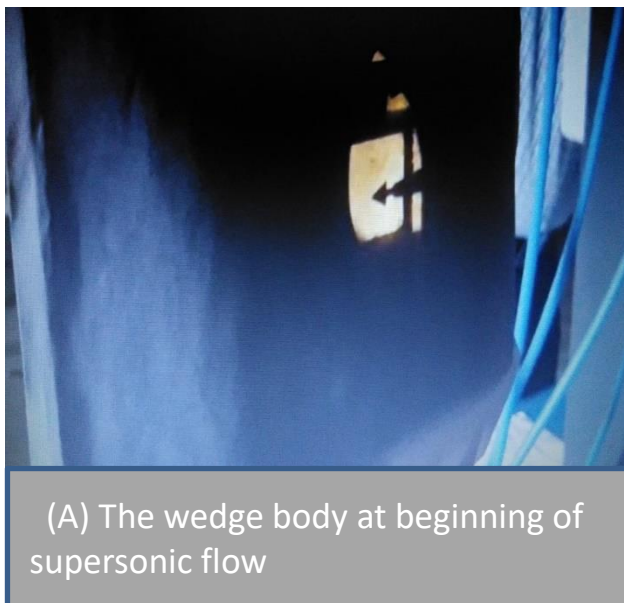


Fig 5.6: Visuals of shock wave occurrence during experimental work

CHAPTER 6

SUMMARY AND CONCLUSION

6.1 SUMMARY

The study aimed to investigate the impact of differently shaped wedge bodies on the flow behavior in a 2D step combustor domain using computational fluid dynamics (CFD) Simulations. In this we considered three primary fluid conditions – eddy viscosity, turbulence kinetic energy, and velocity u to accurately model the complex flow behavior and visualize the recirculation zone. The results showed that the position of the wedge body in the combustor domain affects the flow behavior and the size of the recirculation area. By analyzing the relationship among C_d , RMS, and the recirculation zone, researchers can optimize the position of the wedge body to achieve the best trade-off among these factors. Additionally, introducing a wedge into a supersonic flow generally increases drag, but the precise impact on the C_d value depends on the wedge's shape, size, placement, and specific flow conditions.

6.2 CONCLUSION

From serious computational analysis the two models that are triangular wedge with rectangle ending and sharp pentagonal wedge body, which are positioned at 13mm and 19mm from the origin are considered to be the efficient bodies to achieve maximum recirculation zone was concluded. There are many other bluff bodies are needed to be investigated to Capture the best recirculation zone and its effects are still under research. In conclusion, the study provides valuable insights into the flow behavior in a 2D step combustor domain with a wedge body inside it and highlights the importance of carefully selecting and modeling fluid conditions to accurately predict the flow field. The findings can aid in the design and optimization of supersonic step combustors for improved fuel-air mixing and combustion efficiency.

REFERENCES

1. Aames .C. Massey, Ivan Langella, and Nedunchezian Swaminathan, 2018 A scaling law for the recirculation zone length behind a bluff body in reacting flows, , Department of Engineering, University of Cambridge, Trumpington Street, Cambridge CB2 1PZ, UK,.On 201,DOI: <https://doi.org/10.1017/jfm.2019.475>
2. Aostiyarov .A.W, A.G. Userov, T.B. Maysutov. Analysis of studies on emissions of toxic components of gas compressor units of type GTK-10-4 under operating conditions at the Makat compressor station at Atyrau UMG. International Scientific and Practical C
3. Agarwal, 2016 J.F. Rakshanda a, Md Mahbub Allam, A review of bluff bodywakes.OceanEngineering182(2019)475-488.
<https://doi.org/10.1016/j.expthermflusci.2015.10.032>
4. Aiwu Fan, Jianlong Wan, Yi Liu, Boming Pi, Hong Yao, Wei Liu. "Effect of bluff bodyshapeontheblowofflimitofhydrogen/airflameinaplanarmicrocombustor."Applied ThermalEngineering62(2014):1319.<https://doi.org/10.1016/j.applthermaleng.2013.09.010>
5. Alan Catovic, Elvedin Kljuno, Avdo Voloder, Analysis of flow around high speed irregularly shaped bodies using numerical simulations,2018 on International JournaofAdvancedandAppliedSciences)<https://doi.org/10.21833/ijaas.2018.08.001>
.
6. Anderson JD (2017). Fundamentals of Aerodynamics. 6th Edition, McGraw-Hill Editions, New York, USA).
7. Bearman, P.W., Gartshore, I., Maull, D., Parkinson, G., 1987. Experiments on flow-induced vibration of a square-section cylin, [https://doi.org/10.1016/S0889-9746\(87\)90158-7](https://doi.org/10.1016/S0889-9746(87)90158-7)
8. Bengt Fornberg (2014), Some Observations Regarding Steady Lamina Flows Past Bluff_Bodie)DOI:[10.1016/J.OCEANENG.2012.07.011](https://doi.org/10.1016/J.OCEANENG.2012.07.011)
9. Bruno, L., Fransos, D., Coste, N., Bosco, A., 2010. 3D flow around a rectangular cylinder:computationalstudy.J.WindEng.Ind.Aerod.98,263276)<http://dx.doi.org/10.1016/j.jweia.2012.03.018>

10. Camarri, S., Salvetti, M.V., Buresti, G., 2006. Large-eddy simulation of the flow around a triangular prism with moderate aspect ratio. *J. Wind Eng. Ind. Aerod.* 94, 309–322), DOI:[10.1016/J.JWEIA.2006.01.003](https://doi.org/10.1016/J.JWEIA.2006.01.003)
11. Chattopadhyay, H., 2007. Augmentation of heat transfer in a channel using a triangular prism. *Int. J. Therm. Sci.* 46, 501–505.
<https://doi.org/10.1016/j.ijthermalsci.2006.07.003>
12. Dalal .D.E, A.K., , A., 2006. Numerical simulation of unconfined flow past a triangle cylinder. *Int. J. Numer. Methods Fluids* 52, 801–821
13. Derakhshandeh, J.F., Alam, M.M., 2017. Flow structures around a rectangular cylinder in the vicinity of a wall. *Wind Struct.* 26 (5), 293–304. ASEM.)
[10.12989/WAS.2018.26.5.293](https://doi.org/10.12989/WAS.2018.26.5.293)
14. Efstathios Konstantinidis and Demetri Bouris, (2012), p. 17). DOI: 10.5772/38684
15. Experimental Simulation of Flame Holding in Air-Breathing Engines, Shivam singh , department of mechanical engineering beans college of engineering and Technology, gurdaspur published on research gate(June 2016).
16. Hassan Zadeh. R, Sahin.B, Ozgoren. M., 2012. Large eddy simulation of free-surface effects on the wake structures downstream of a spherical body. *J. Ocean. Eng.* 54, 213–222). DOI:[10.1016/J.OCEANENG.2012.07.011](https://doi.org/10.1016/J.OCEANENG.2012.07.011)
<https://doi.org/10.1002/flid.1210>
17. Huang, R.F., Lina, B.H., Yen, S.C., 2010. Time-averaged topological flow patterns and their influence on vortex shedding of a square cylinder in cross-flow incidence. *J. Fluids Struct.* 26, 406–429).
DOI:[10.1016/J.JFLUIDSTRUCTS.2010.01.003](https://doi.org/10.1016/J.JFLUIDSTRUCTS.2010.01.003)
18. Rakshanda a .J.F, Md Mahbub Allam, A review of bluff body wakes. *Ocean Engineering* 182 (2019) 475-488. DOI:[10.1016/J.OCEANENG.2019.04.093](https://doi.org/10.1016/J.OCEANENG.2019.04.093)
19. Kim, S., Alam, M.M., Maita, D.K., 2018. Wake and suppression of flow-induced vibration of circular cylinder. *Ocean Eng.* 151, 298-307. <https://doi.org/10.1016/j.oceaneng.2018.01.043>

20. Larose, G., Auteuil, A., 2006. On the Reynolds number sensitivity of the aerodynamics of bluff bodies with sharp edges. *J. Wind Eng. Ind. Aerod.* 94, 365–376). DOI: [10.1016/j.jweia.2006.01.011](https://doi.org/10.1016/j.jweia.2006.01.011)
21. Last accessed 20th July 2013. 11. J.D. Anderson (1998). *Fundamentals of Aerodynamics*. New Delhi: Tata McGraw Hill Publications. Pg. 348–352. Lübcke, H., Schmidt, S., Rung, T., Thiele, F., 2001. Comparison of LES and RANS in bluff body flows. *J. Wind Eng. Ind. Aerod.* 89, 1471–1485), DOI: [10.1016/S0167-6105\(01\)00134-9](https://doi.org/10.1016/S0167-6105(01)00134-9)
22. M. M. Abd Elhameed, A. M. Hamed, A.E. Hussin, W. Aboelsoud, M. M. Kamal Bluff Body Assisted Combustion with Internal Preheat of Fuel-Air Mixture *N Y Sci J* 2021;14(8):1-7 | ISSN1554-0200(print); ISSN2375-723X(online) <http://www.sciencepub.net/newyor>, doi:10.7537/marsnys140821.01.
23. Mannini, C., Soda, A., Schewe, G., 2011. Numerical investigation on the three-dimensional unsteady flow past a 5:1 rectangular cylinder. *J. Wind Eng. Ind. Aerod.* 99, 469–482). DOI: [10.1016/j.jweia.2010.12.016](https://doi.org/10.1016/j.jweia.2010.12.016)
24. Mr. Bhupal Rakham, Mohd Nehal Aman, Study of Flow Behaviour over Different Bluff Body. *Volumen 3(2016), Issueno (9sep)*. DOI: [10.12989/WAS.2018.26.5.293](https://doi.org/10.12989/WAS.2018.26.5.293)
25. Raghavan, K., Bernitsas, M.M., 2011. Experimental investigation of Reynolds number effect on vortex induced vibration of rigid circular cylinder on elastic supports. *Ocean Eng.* 38 (5– 6), 719–731 , <https://doi.org/10.1002/fld.1210>
26. Taghinia, J., Rahman, Md M., Siikonen, T., 2015. Large eddy simulation of flow past a circular cylinder with a novel sub-grid scale model. *Eur. J. Mech. B Fluid* 52, 11–18). DOI: [10.1016/j.euromechflu.2015.02.001](https://doi.org/10.1016/j.euromechflu.2015.02.001)
27. Takahiro Watanabe, Masaharu Takahashi and Toshihiko Munakata Hitachi Solutions East Japan, Ltd., Shock Wave in Combustion in a Scramjet Engine, 10.2514/6.2019-0678, on 2019 by Japan Aerospace Exploration Agency. Published by the American Institute of. <https://doi.org/10.2514/6.2019-0678>
28. Taylor, Z.J., Palombi, E., Gurka, R., Kopp, G.A., 2011. Features of the turbulent flow around symmetric elongated bluff bodies. *J. Fluids Struct.* 27, 250–265). <https://doi.org/10.1016/j.jfluidstructs.2010.10.004>
29. Thuib W. M. A. W., Azam Che Idris , Mohd Rashdan Saad , Mohd Faizal Hasan ,

Mohd Rosdzimin Abdul Rahman, Study on The Effect of Bluff Body with Slit in the Micro Channel Combustor on Journal of Advanced Research in Fluid Mechanics and Thermal Scien,2018.

30. Yamagata, T., Saito, N., Fujisawa, N., 2017. Experimental and numerical studies on aeoliantone from d-shaped cylinder. Proc. Inst. Mech. Eng. C 232, 4007–4018, DOI: [10.4236/jfcmv.2016.44014](https://doi.org/10.4236/jfcmv.2016.44014)
31. Zhao, J., Hourigan, K., Thompson, M.C., 2018. Flow-induced vibration of d-section cylinders: an afterbody is not essential for vortex-induced vibration. J. Fluid Mech. 232, 317–343), DOI:[10.1017/jfm.2018.501](https://doi.org/10.1017/jfm.2018.501)
32. Ziwan Li , Yixiang Yuan , Baoting Guo , V. L. Varsegov and Jun Yao , College of Jet Engines and Power Plants, Kazan National Research Technical University. The Recirculation Zone Characteristics of the Circular Transverse Jet in Cross flow Publish). DOI: [10.1016/J.AST.2021.106599](https://doi.org/10.1016/J.AST.2021.106599)
33. Zongnan Chen, Chih-Yung Wen, Flow control of a D-shaped bluff body using different DBD plasma actuators, Journal of Fluids and Structures 103 (2021) 103292). <https://doi.org/10.1016/j.jfluidstructs.2021.103292>