

2-D ANALYSIS OF OPEN CAVITY WITH SHOCK IMPINGEMENT

Submitted in partial fulfillment of the requirements for the award of

Bachelor of Engineering degree in

Aeronautical Engineering

By

ADITHYA M (REG. NO. 38260001)

MAJELLA RIONA R (REG. NO. 38260022)

ROOPA B J (REG. NO. 38260035)



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

MAY 2022



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
ADITHYA M (38260001), MAJELLA RIONA R (38260022), ROOPA B J
(38260035) who carried out the project entitled **2-D ANALYSIS OF OPEN CAVITY**
WITH SHOCK IMPINGEMENT under our supervision from November 2021 to May
2022.

Internal Guide

Dr. P. BOOMADEVI, ME., PhD.,

Head of the Department

Dr. P. BOOMADEVI, ME., PhD.,

Submitted for Viva voce Examination held on

Internal Examiner

External Examiner

DECLARATION

We **ADITHYA M (38260022), MAJELLA RIONA R (38260022) AND ROOOA B J (38260035)** hereby declare that the Project Report entitled **2- D ANALYSIS OF OPEN CAVITY WITH SHOCK IMPINGEMENT** done by us under the guidance of **DR. P. BOOMADEVI M.E.,PhD.**, at Sathyabama Institute of Science and Technology is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.

DATE: 05-05-2022

PLACE: CHENNAI

SIGNATURE OF THE CANDIDATE

1. NAME: ADITHYA M

SIGNATURE:

2. NAME: MAJELLA RIONA R

SIGNATURE:

3. NAME: ROOPA BJ

SIGNATURE:

ACKNOWLEDGEMENT

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

We convey our 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.

We would like to express our sincere and deep sense of gratitude to our **Project Guide Dr. P. Boomadevi M.E., Ph.D.**, for her valuable guidance, suggestions and constant encouragement paved way for the successful completion of our project work.

We wish to express our 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

The effort to interpret the characteristics of the cavity began in the 1950s. Throughout the years the comprehensive understanding of cavities was not fully achieved due to the intricateness and the varying range of flows. The project mainly studies the outcome of the interaction between the shock wave, which is generated with the aid of the wedge, and the shear layer over the cavity for different cases. For all cases, the cavity model remains unchanged with horizontal span to profoundness ratio (L/D) 3, offset ratio (OR) 1 and hind wall ramp 90 degrees. The initial work is done with 15 degrees half wedge angle and at Mach 3. Here, the shock is allowed to intersect with the shear layer flowing over the open cavity. The Mach contour demonstrates the emergence of a weak shock wave at the cavity frontal edge and the vertical rise of the shear layer due to collision with shock. The result displays a significant increase in the static pressure along the walls of the cavity. The further analysis is done at Mach 3 with varying wedge half angles 5, 10, 15 and 20 respectively. Each of the cases is compared in terms of pressure. As expected the outcome shows increase in the pressure with increasing angles. There exists some influence over the extent of the recirculation zones. The trend wherein the pressure reaches a local peak at the bottom edge of the aft wall of the cavity and reaches the highest value of the pressure at the upper edge of the aft wall is seen in all the cases. The last part of the project is with varying mach numbers. Starting from mach 2 Mach number, Mach numbers are varied with 0.5 increments up to mach 4. The main trend noticed is that a peak of 0.8 is seen at mach 3 and after that a decrease in the value of peak is seen with increasing Mach number.

Keywords: Open Cavity, Shock Impingement, Wedge, Coefficient of Pressure

TABLE OF CONTENTS

ABSTRACT	v
LIST OF FIGURES	viii
LIST OF TABLES	xi
LIST OF SYMBOLS AND ABBREVIATION	xii

CHAPTER NO	TITLE	PAGE NO.
1.	INTRODUCTION	1
	1.1. BACKGROUND	1
	1.2. TYPES OF CAVITY	2
	1.2.1. Open Cavity	3
	1.2.2. Transitional Cavity	4
	1.2.3. Closed Cavity	4
	1.3. APPLICATION - CAVITY BASED FLAMEHOLDER	4
2.	LITERATURE REVIEW	6
	2.1. UNSTEADY SUPERSONIC CAVITY FLOWS	6
	2.2. SUPERSONIC OPEN-CAVITY FLOW PHYSICS	6

	2.3. CAVITY-BASED FLAMEHOLDER FOR SUPERSONIC COMBUSTORS	6
	2.4. VELOCITY AND SURFACE PRESSURE MEASUREMENTS	7
	2.5. CAVITY-BASED FLAMEHOLDER DESIGNS	8
	2.6. FUELED SUPERSONIC CAVITY FLAME HOLDER	8
	2.7. HYPERSONIC FLOWS OVER 3D CAVITIES	9
	2.8. SUB CAVITY IN SUPERSONIC CAVITY FLOW	9
	2.9. UPSTREAM CAVITY IN SCRAMJET COMBUSTORS	10
	2.10. CONTOURED UPSTREAM WALL	11
	2.11. OPEN STRUT CAVITY	11
3.	AIM AND SCOPE	13
	3.1. AIM	13
	3.2. SCOPE	13

4.	METHODOLOGY	14
	4.1. COMPUTATIONAL ANALYSIS	14
	4.1.1. Computational Domain	14
	4.1.2. Shock Generator Or Wedge	15
	4.1.3. Meshed Domain	18
	4.1.4. Solver Setup	20
	<i>4.1.4.1. Solver Type - Density Based</i>	20
	<i>4.1.4.2. Solver Time – Steady</i>	21
	<i>4.1.4.3. Models</i>	21
	<i>4.1.4.3.1. Energy</i>	21
	<i>4.1.4.3.2. Viscous - k-ω Model</i>	22
	<i>4.1.4.4. Materials - Density - Ideal Gas</i>	22
	<i>4.1.4.5. Spatial Discretization – Green Guass</i> <i>Cell Based</i>	23
	4.1.5. Validation	23
	4.1.6. Grid Independence Test	24

4.2. EXPERIMENTAL SETUP	25
4.2.1 Supersonic Wind Tunnel	25
4.2.2. Cavity And Wedge Model	26
4.2.3. Material Used	29
 5. RESULTS AND DISCUSSIONS	 30
5.1. COMPUTATIONAL RESULTS	30
5.1.1. Initial Analysis (With And Without Wedge)	30
<i>5.1.1.1. Mach Contour</i>	<i>30</i>
<i>5.1.1.2. Recirculation Zone</i>	<i>31</i>
<i>5.1.1.3. Pressure Plot</i>	<i>32</i>
5.1.2. Varying Mach Number Analysis	33
<i>5.1.2.1. Mach Contour</i>	<i>33</i>
<i>5.1.2.2. Recirculation Zone</i>	<i>35</i>
<i>5.1.2.3. Pressure Plot</i>	<i>37</i>

5.1.3. Varying Wedge Angle	38
5.1.3.1. <i>Mach Contour</i>	38
5.1.3.2. <i>Recirculation Zone</i>	40
5.1.3.3. <i>Pressure Plot</i>	42
5.2. EXPERIMENTAL ANALYSIS	43
6. CONCLUSION	45
REFERENCES	47

LIST OF FIGURES

FIG. NO.	TITLE	PAGE NO
1.1.	Cavity flow	2
1.2.	Types of cavity flow	3
1.3.	Flow attributes of cavity-based flame-holder	5
4.1.	Dimensions of the computational domain, where d is the depth of the cavity	14
4.2.	Flow past a wedge. Where θ - half wedge angle and β is the shock angle	16
4.3.	Dimensions of wedges of varying angles. All the dimensions are in mm	18
4.4.	Types of mesh elements	18
4.5.	Meshed domain for the current study	20
4.6.	Normalized pressure vs. Effective length in inches	23
4.7.	Grid independence test	24
4.8.	Photograph of wind tunnel facility	25

4.9.	Wedge holder	26
4.10.	Test section	27
4.11.	Close up of the wedge and cavity model mounted in the test section	27
4.12.	(A) wedge model - 15 degrees half wedge angle (b) wedge mode -I 20 degrees half wedge angle (c) threaded hole in the wedge	28
5.1.	Mach contour of open cavity without wedge	30
5.2.	Mach contour with wedge	31
5.3.	Vector plot in the case of absence of wedge	31
5.4.	Vector plot in the case of presence of the wedge	32
5.5.	Length of the cavity wall vs. Value of c_p	33
5.6.	(A) mach 2 (b) mach 2.5 (c) mach 3 (d) mach 3.5 (e) mach 4 (f) color map for mach contour	34
5.7.	Vector plots for (a) mach 2 (b) mach 2.5 (c) mach 3 (d) mach 3.5 (e) mach 4	36-37
5.8.	Horizontal span vs. C_p graph	38

5.9.	Mach contours for (a) 5 deg (b) 10 deg (c) 15 deg (d) 20 deg	38-39
5.10.	Vectors plots of half wedge angle of (a) 5 deg (b) 10 deg (c) 15 deg (d) 20 deg	40-41
5.11.	Effective length in inches vs. C_p graph	42
5.12	Geo tag photo of the cavity and wedge model fixed to the test section	43
5.13	Visualization using shadowgraph	44

LIST OF TABLES

TABLE NO	TITLE	PAGE NO.
4.1.	Differences between quadrilateral and triangular mesh element	19
4.2.	Properties of stainless steel	29
5.1.	Values of shock angle and strength of the shock wave for different mach number	35
5.2.	Shock angle vs. Shock strength of different half wedge angle	40

LIST OF SYMBOLS AND ABBREVIATIONS

Cp	Coefficient of Pressure
°C	Degree Centigrade
ρ	Density
GPa	Giga Pascal
g	Grams
γ	Heat Capacity Ratio
K- ϵ	K – Epsilon
K	Kelvin
I/L	Length of subcavity to length of main cavity ratio
L/D	Length to Depth ratio
M	Mach Number
MPa	Mega Pascal
m	Meter
OR	Offset Ratio
PIV	Particle Image Velocimetry
P	Pressure
T	Temperature
W	Watts

CHAPTER 1

INTRODUCTION

1.1. BACKGROUND:

The emergence of grooves on the surface of airborne vehicles becomes inevitable in multiple situations. Wheel wells, slotted wind tunnel walls, aerodynamic windows, space shuttle bay vents and bomb bays are all been reported to feature cavity structure. These circumstances can either be as a part of intended design, for instance housing for bombs or other weapons and also as lodgment for landing gear and other sensitive or precise measurement instrumentations, or as a consequence of accidental circumstances, such as fittings for pipes and flaps in deflected position. Cavities are a favorable feature for the design of aerospace vehicles owing to their characteristics of drag and heat transfer. One of the most common utilizations of a cavity is as a component of separation regulation. Although cavities present a particular interest for their employment in design, they may also become an origin of serious concerns as they may turn the flow field unnecessarily intricate. The most prevailing concerns are the adverse influences in the aerodynamic attributes, the failing of structures, induction of vibrations and the destruction of highly delicate instruments. Disregarding the purpose of their presence, the interpretation of the flow in the vicinity of the cavity is crucial to employ the cavity design for future airborne vehicle design. The intricacy and vast choice of cavity flows present a significant difficulty in gaining a complete understanding of flow physics. When it comes to high velocity flow in the vicinity of the cavity, intricate and erratic flow patterns emerge. These are revealed by the acoustic nature of the region. The acoustic resonance occurring within the cavity can impede the safety and accuracy of weapons delivery. When the Reynolds number becomes higher, the cavity regions are known to accommodate broadband scaled-down oscillations that are archetypal to turbulent shear layer and distinct resonance. The attributes like magnitude and frequency rely on the design geometry and the condition to which it is exposed. In particular circumstances the emergence of dual cavity or in other words cavity cascade can

also be noted. These cavities cause higher drag, significant pressure loss, aerodynamic noise and pressure variations, which can lead to component structural deterioration. Figure 1.1 depicts a typical flow that occurs within and in the vicinity of the cavity.

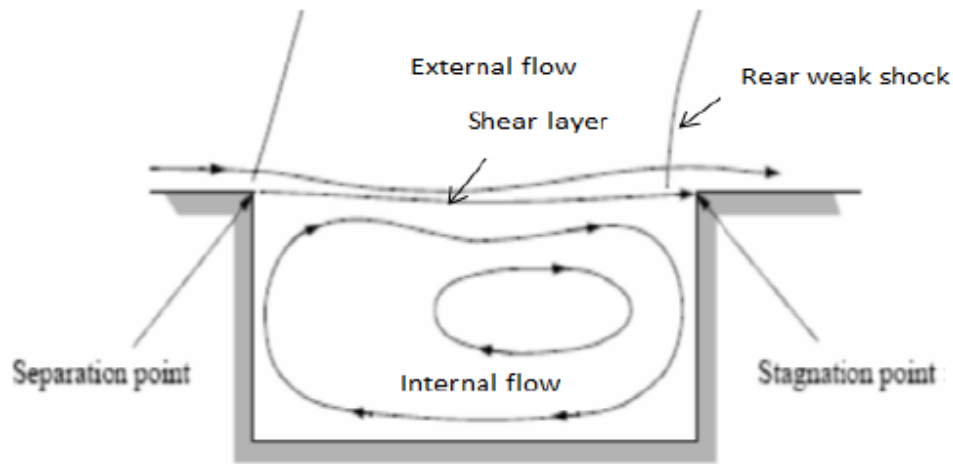


Figure 1.1: Cavity Flow

1.2. TYPES OF CAVITIES:

The cavity flow is attributed to the formation of an oscillating shear layer. High aero-acoustic pressure oscillations and high decibel noise are produced by flow across a hollow. Depending on the reattachment of the shear, cavities are said to be of three types:

- Open cavity
- Transitional cavity
- Closed cavity

These three types of flow are depicted in figure 2.

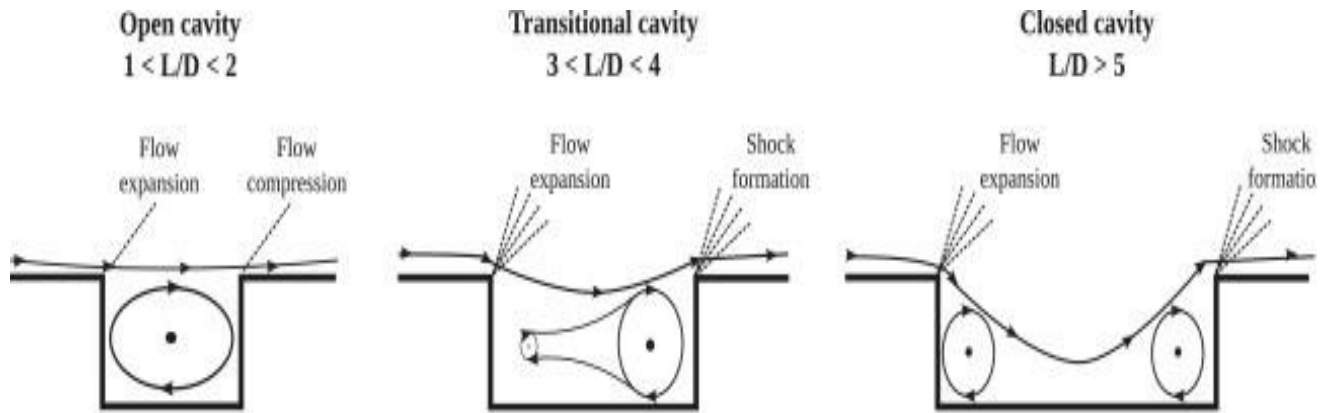


Figure 1.2: Types of cavity flow

1.2.1. OPEN CAVITY:

($1 < L/D < 10$) Upstream of the cavity, the incoming flow creates a turbulent boundary layer, which separates from the surface and causes a shear layer. The shear layer that travels with free-stream detaches from the separation point located upstream of the cavity and binds itself to the wall downstream of the cavity without coming into contact with the cavity floor. The flows in and around the cavity are split due to the creation of the shear layer, thus the external flow doesn't infiltrate the cavity. The shear layer is driven by the low pressure upstream and the higher pressure behind the hind wall of the cavity. This justifies the absence of shear layer collision with the bottom floor. The compression of flow by the shear layer results in the development of weak shock at the downstream corner of the cavity. In both stream wise and transverse directions, shear layer develops. The magnitude of pressure coefficient is retained fairly uniformly over the entirety of the span of the cavity. The peak near the hind wall is the only abnormality which disturbs the uniformity, which is interpreted to be the consequence of reattachment of the shear layer. The recirculation zone in a cavity is created by the change in pressure between the inner cavity stream and the free stream flow. The downstream wall acts as an acoustic source as a result of the pressure fluctuation. Flows in and around the cavity are split due to the creation of a shear layer. Open cavities are extremely intricate and the link for internal and external flow is contributed by self-enduring oscillation of the shear layer.

1.2.2. TRANSITIONAL CAVITY:

($10 < L/D < 14$) This type of flow is interpreted as the flow is interpreted as the flow in transition between the above mentioned types, wherein the shear layer has neither collided with the cavity floor nor directly reattached to the wall downstream. The fluctuation between closed and open type cavities characterizes this type to have unsteady flow demeanor. In this type the shear layer diverts from the point of coincidence, while both exit and impingement shock disintegrating into an unit wave. The increment in pressure across the cavity floor occurs uniformly and the reattachment region displayed non-altering pressure value.

1.2.3. CLOSED CAVITY:

($L/D > 14$) The shear layer detaches and collides on the cavity floor with the floor of the cavity and again detaches and collides with the wall downstream of the cavity. The free-stream is partitioned at the corner of the frontal wall. Two disparate separation zones are stemmed, one in the proximity of the frontal wall while the other hind wall. The trend in the altering pressure magnitudes is equivalent to that of the transitional cavity. They are akin to having higher coefficients of drag. Closed cavities have an increasing static pressure distribution over their length, although they have far lower acoustic levels or pressure oscillations than open cavities.

1.3. APPLICATION - CAVITY BASED FLAMEHOLDER

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 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

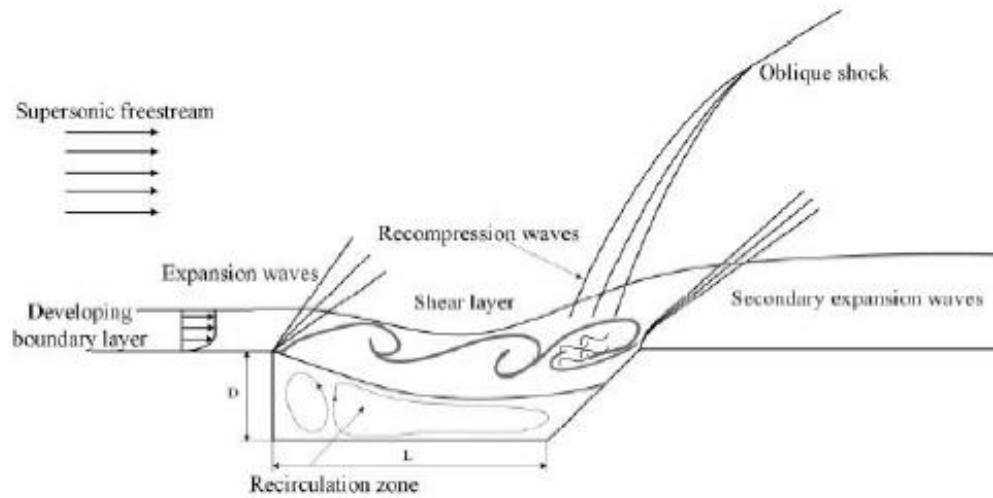


Figure 1.3: Flow attributes of cavity-based flame-holder

CHAPTER 2

LITERATURE REVIEW

2.1. UNSTEADY SUPERSONIC CAVITY FLOWS

X. Zhang et al. in 1988 solved the 2-D unsteady compressible Navier-Stokes equations with regards to mass-averaged variables to investigate cavity under supersonic conditions caused by a wide shear layer at Mach 1.5 and 2.5. The fluctuation in frequency in the three-cavity horizontal span-to-profoundness ratios are thought to be in lengthwise and regulated by alterations in flow field attributes instead of solely acoustic. The infinite fringe interferogram, in which the cavity is packed by a trailing edge vortex and downstream migrating leading edge vortices, shows lengthwise alteration in frequency in the $L/D = 3$ cavity at Mach 1.5. The shear layer is oscillatory, with a substantial shear layer incident on the cavity's trailing edge.

2.2. SUPERSONIC OPEN-CAVITY FLOW PHYSICS

Chung-Jen Tam et al. in the year 1996 described the oscillation cycle that sustains flow within and around the vicinity of the cavity. The cavity has an L/D ratio of 2 operating in an unsteady flow under supersonic conditions. The cycle starts with a shed vortex colliding on the hind wall resulting in the origin of an echoed wave off of the frontal wall. The echoed wave resulted in the emergence of a new vortex. The wave itself, however, dissipates before it could reach the hind wall. The cycle persists, with the emergence of a new wave underneath the vortex that ultimately is echoed off of the hind wall.

2.3. CAVITY-BASED FLAMEHOLDER FOR SUPERSONIC COMBUSTORS

Both empirical and computational examination of cavities with diverse hind wall angles and offset ratios were carried out by M. R. Gruber et al. Various L/D ratios and its consequences were inspected. The emergence of two counter-rotating

recirculation zones is recorded. The extent of the secondary recirculation zone adjacent to the frontal wall contracts with inflation in L/D ratio. The schlieren technique revealed the origination of shock wave at the upper left corner of the cavity. The cavity oscillations cause the originated wave to arch. The depreciating value of the hind wall angle stabilizes the flow. The conversion of wave nature from compressive to expansive is detected. The decreasing hind wall angle resulted in the expansion of the primary recirculation zone, which appears in the vicinity of the hind wall, which resulted in the breaching of external flow that caused decline in residence time. Also cavity drag inflates during the same due to the increasing exposure to higher pressure. The increment of Offset Ratio (OR) value improved the expansion wave, the pressure on the frontal wall dipped drastically promoting mixing of external flow with internal flow. Under these circumstances the impact of wall angle in determining the value of pressure is diminished, but it has an influence in the alteration of the layout of the recompression wave.

2.4. VELOCITY AND SURFACE PRESSURE MEASUREMENTS

Particle image velocimetry (PIV), hot-wire and surface pressure measurements, were used to investigate incompressible flow of about Mach 0.2 over cavities with L/D ratios of 1.49 and 5.16 in this current study carried out by Lawrence Ukeiley, Nathan Murray in 2005. In either circumstance, turbulence appears on the incoming boundary layer. Mean flow parameters, vorticity field information, and two-point statistics for the velocity field were extracted from the PIV data. To investigate the relationships between velocity and pressure variations, the hot-wire data was paired with surface pressure measurements. For the leading edge boundary layer condition and free stream speed that was investigated, the results show a discrepancy between a-periodic flow pattern and a shear layer mode flow. Many similarities and differences in how the shear layer interacts with the cavity in the two cavity aspect ratios were discovered by examining the velocity field. Despite the fact that the shear layer growth in the two cavities was different, they both plunged into the hollow and had mean streamlines that impacted on the rear wall. The turbulent velocities in the

shallower cavity had a shear layer-like structure, whereas in the deeper cavity, there were also indications of excess turbulent energy linked with the circulating flow. Furthermore, the shear component of the Reynolds' stress indicated that the shear layer was driving recirculation in the profound cavity.

2.5.CAVITY-BASED FLAMEHOLDER DESIGNS

This paper written by Sanchayata Nayal, Sampada Lamb, Devabrata Sahoo, N. V. Raghavendra, S.A.I. Bellary in 2020 investigates the alterations that had been brought about by the frontal wall inclination. Two distinct values of frontal and hind wall angles are considered. For 90 degree hind wall angle, a weak shock wave was generated in the upstream separation corner. In contrast to this the 16 degree hind wall angle configurations generated an expansion wave instead. The change in the frontal wall inclinations produced no change in the phenomenon. Of all the configurations considered 110_90 (frontal wall angle_hind wall angle) had the lowest temperature distribution and recirculation zones rotating at higher speeds. Although the 70_90 configuration exhibited lowest drag, judging by the overall performance the paper concludes that an obtuse frontal wall angle is much more efficient as a cavity-flame holder.

2.6. FUELED SUPERSONIC CAVITY FLAME HOLDER

The stream wise concentration distribution of an ethylene fuel surrogate (acetone-seeded nitrogen) directly injected into the recirculation zone of a cavity flame holder embedded in a non-reacting supersonic airflow was quantified using planar laser-induced acetone fluorescence in the current investigation done by Frank W. Barnes, Qiuya Tu, and Corin Segal in 2014. The impacts of fueling rate and injection configuration were investigated. Parallel injection from the cavity frontal wall resulted in a predominantly fuel-rich recirculation area mixture, whereas downward angled injection from the cavity floor resulted in the opposite. With rise in fueling rates, the flammable volume within the cavity recirculation area decreases. The mixing

efficiency of the front wall injection arrangement looked to decline as the fueling rate increased, whereas it remained mostly unchanged by downstream angled injection. The other injection configuration's improved mixing ability has the capability to yield increased cavity flame holder functioning by allowing for a wider range of combustion and blowout limitations, as well as more igniter location versatility than traditional direct-fueling systems.

2.7. HYPERSONIC FLOWS OVER 3D CAVITIES

Rodrigo C. Palharini Thomas, J. Scanlon, Craig White in 2018 looked into the cavities that appear on space vehicles and the conditions of the cavity during re-entry of these vehicles into the atmosphere. Naturally, the conditions of the flow are hypersonic and the air is under rarified conditions. The influence of L/D ratio of the cavity on the flow-field variables within and around the cavity is studied. For ratios 1 and 2, the cavity behaves much like an open cavity. There emerges only one recirculation zone and the detection of high density of particles inside the cavity is explained by the absence of penetration of the external flow within the cavity. For L/D ratio 3 the cavity operates much analogously to a transitional cavity where penetration of external flow, although detected, remains minimal. For the consecutive two ratios of 4 and 5, the two elongated-conjoined recirculation zones that arose in the preceding case separates with the shear layer attaching to the cavity floor at ratio 5. For a hypersonic flight, a cavity of L/D ratio 5 poses a threat of damage to the surface due to shear layer attachment to the floor.

2.8. SUB CAVITY IN SUPERSONIC CAVITY FLOW

Chintan Panigrahi, Aravind Vaidyanathan, and Manoj T. Nair in 2019 investigated experimentally and numerically the effects of the sub cavity on pressure oscillations at different points of the major cavity with L/D = 2 in supersonic flow at a Mach number of 1.71. By using experimental and numerical schlieren techniques and statistical analysis of the unsteady pressure data the flow physics is studied. The sub cavity's

location and dimensions are crucial in understanding the types of oscillations that exist inside the cavity. The sub cavity was placed at the front wall, aft wall, and both front and aft walls concurrently in cavity models with a sub cavity length to main cavity length of 0.2 ($l/L = 0.20$). It has been discovered that putting the sub cavity on the front wall or the back wall causes the shear layer to be modified in various ways for the same effective length, resulting in different patterns of oscillations. When the $l/L = 0.20$ sub cavity was placed against the aft wall, it revealed the presence of FR oscillations, however when the same sub cavity was placed against the front wall, it revealed the absence of any kinds of oscillations. The combined influence of both front wall and aft wall sub cavity flow features was observed when the sub cavity was placed at both the front and aft walls. Finally, it can be inferred that the positioning of the sub cavity in the main cavity has a significant impact on the sorts of oscillations that occur inside the cavity, whether fluid-dynamic or fluid-resonant, and thus on the suppression or enhancement of oscillations.

2.9. UPSTREAM CAVITY IN SCRAMJET COMBUSTORS

In a generic scramjet combustor arrangement, the performance of cavities positioned upstream of a single injector was quantitatively investigated by Tim Roosa, Adrian Pudsey, Mathew Bricallia, Hideaki Ogawa in 2019. Only chemically frozen flow was considered to study the effect of wall temperature out of three different wall conditions. Depending on the layout and wall treatment, the upstream cavity was demonstrated to improve mixing by up to 9% compared to a baseline at the domain's end ($x/d = 400$). The cavity with the largest volume, $L / D = 15$, was shown to be the most successful, increasing mixing efficiency for all wall treatments. When the cavity was introduced, total pressure losses upstream of the injector were found to increase. However, the bow shock in cavity cases was shown to be weaker, resulting in lower overall pressure loss than in the baseline behind the injector, depending on combustor length. In systems with cavities, wall heat flux rises, notably on the cavity's aft wall, whereas fuel fed into the cavity is observed to contribute to wall cooling in cases of high wall temperatures.

2.10. CONTOURED UPSTREAM WALL

The upstream wall of cavities within scramjet combustors is numerically examined in this research paper written by Will O. Landsberg, Ananthanarayanan Veeraragavan in 2019. The present study aims to investigate four different cavity profiles by replacing the rear-facing step with ramps (angled at 45 and 22.5 degrees) or stream tracing it to adapt to the primary vortex aerodynamically. Non-fuelled flow fields in Mach 8 combustor flows show full elimination of the secondary vortex. The quick expansion of the cavity frontal wall over the rear-facing step causes incredibly low pressure and is the principal source of cavity drag. Furthermore, because only the larger of the two cavity vortices leads to mixing, the cavity front wall can be altered to exclude the smaller vortex and lower the drag. Non-fuelled flow fields in a Mach 8 scramjet combustor see the secondary vortex removed completely for all cavities, and the rear-facing step is substituted by a ramp. A 22.5-degree cavity front ramp was insufficient to establish boundary layer separation, while a 45-degree ramp achieved rapid separation. For the combined and 45 deg ramp examples, ethylene-fueled cases showed drag reductions of 5.6 percent and 6.6 percent, respectively. Universal drag reductions for the Non-fuelled and fueled cases were achieved in lower Mach number flows, but mixing was comparable in most circumstances. Thus while increasing flight speeds, aerodynamic contouring becomes necessary.

2.11. OPEN STRUT CAVITY

The open strut cavity in Mach 2 supersonic flow was quantitatively investigated by Ajay Dhankarghare, T. Jayachandran and T.M. Murugananda using the RNG $k-\epsilon$ turbulence model and the SLAU2 method. The cavity between two struts positioned in parallel, along the flow, is known as an open strut cavity. The goal was to identify the major oscillation modes for various aspect ratio open strut cavities. The model is conducted with a 2 inlet Mach number, 3 bar of stagnation, and 300 K of temperature. Three distinct aspect ratios are simulated: 4, 6, and 8. The cavity's flow features are found to be identical to those of the wall-mounted cavity. Pressure oscillations with a

dominant mode at one of the Rossiter mode frequencies have been detected inside the cavity. The dominant mode is found to change with the cavity's aspect ratio. With an aspect ratio of 8, an open strut cavity behaved like a closed cavity with a full bow shock in front of the leading strut. The asymmetric character of the flow in the symmetric configuration is shown by investigating flow near the front wall. It is discovered that the travelling compression wave interacts differently with the two shear layers, resulting in asymmetry at the leading wall.

CHAPTER 3

AIM AND SCOPE

3.1. AIM

The intention of this analysis is to comprehend the elemental impacts or the generic performance of the flow-field external and internal to the cavity in the existence of shock. The entire project was executed with the aid of Ansys-Fluent. Fabrication of the cavity and the wedge was done using CNC. The project was implemented in three stages.

- The influence a shock wave, stemming from the wedge, has on a cavity
- The influence of wedge angle on the attributes of the flow external and internal to the cavity
- The influence that Mach number has on the attributes of the flow external and internal to the cavity

The flow is in a steady state and hence time dependent variables weren't investigated.

3.2. SCOPE

Since the appearance of cavities on air-borne vehicles becomes inevitable, this project proves to be integral for designers as it informs them regarding the flow-field variables. This grants the designer to fashion the cavity in such a manner that is desirable for that specific mission, while evading the adverse consequences that it can engender. These adverse consequences can, in specific cases, lead to structural deterioration.

CHAPTER 4

METHODOLOGY

4.1. COMPUTATIONAL ANALYSIS:

4.1.1. COMPUTATIONAL DOMAIN:

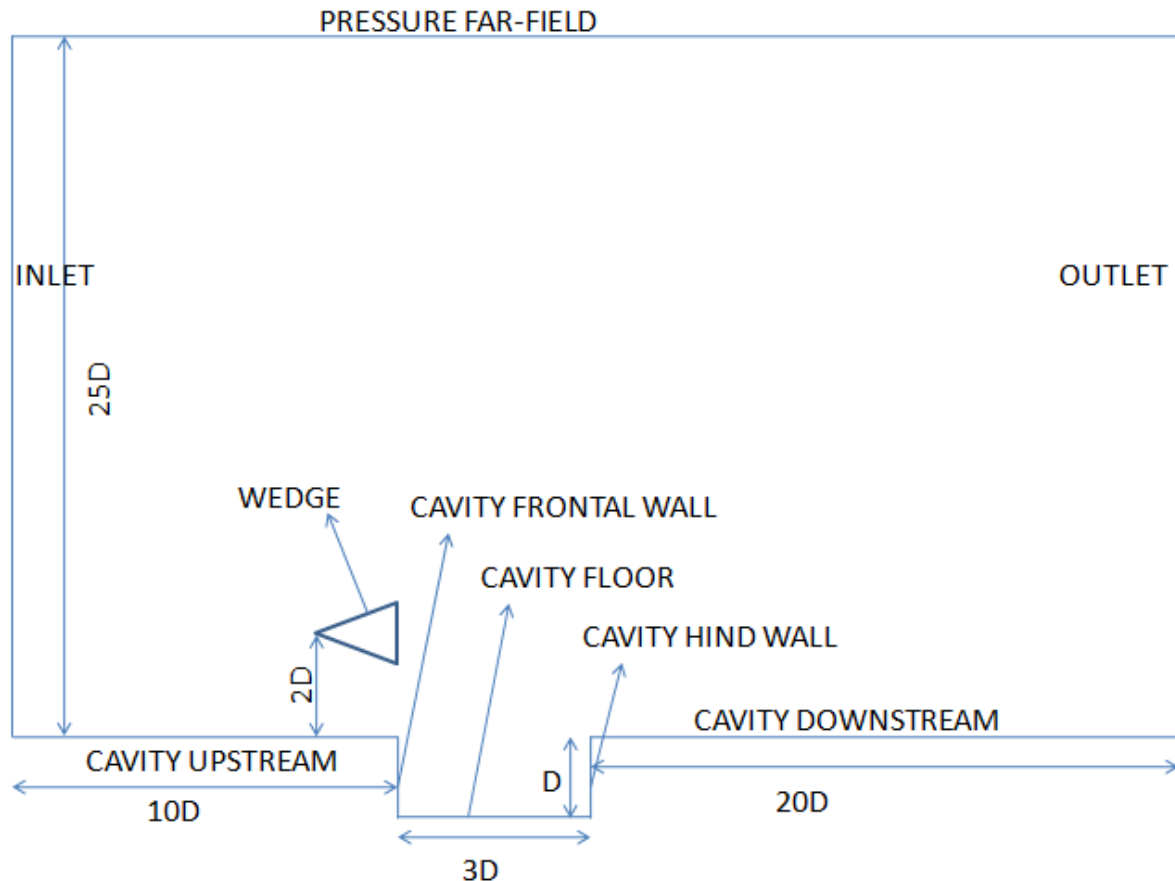


Figure 4.1: Dimensions of the computational domain, where D is the depth of the cavity.

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 4 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 4. 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 5), when moving through air at a velocity that is greater than the speed of sound, 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 downstream of the shock wave experiences enormous alterations in the flow-field.

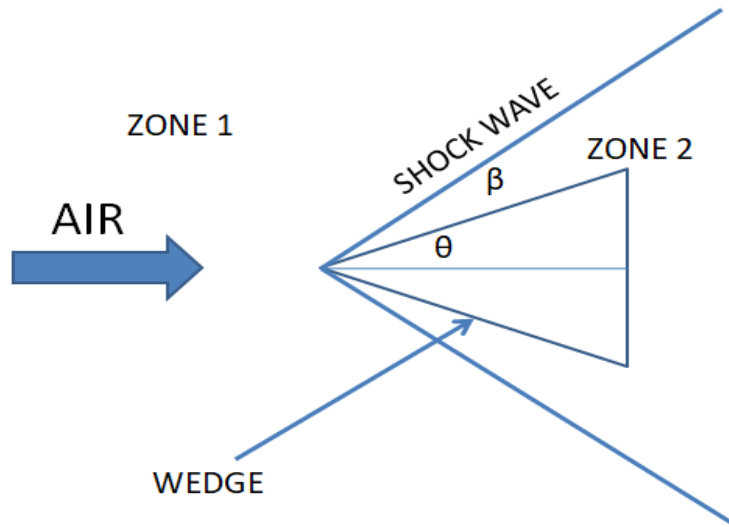


Figure 4.2: Flow past a wedge. Where θ - half wedge angle and β is the shock angle.

Equations for shock wave relations:

(Where,

- The subscript 1 indicates the property of the flow upstream of the shock
- The subscript 2 relates to the property of the flow downstream of the shock
- P is the pressure of the air
- T is the temperature of the air
- ρ is the density of the air
- M is the mach number of the air
- γ is the heat capacity ratio)

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

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

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

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

The lengths of the wedge for all the cases were kept constant which is 10mm. The constant wedge that is utilized for varying mach numbers is the wedge with 15 degree angle. For varying wedge angles, 4 different angled wedges are utilized. The base of each wedge is calculated based on the angle. The varying wedge angles not only alter the shock angle but also the shock strength. The figure below shows the dimensions of the wedge for various cases.

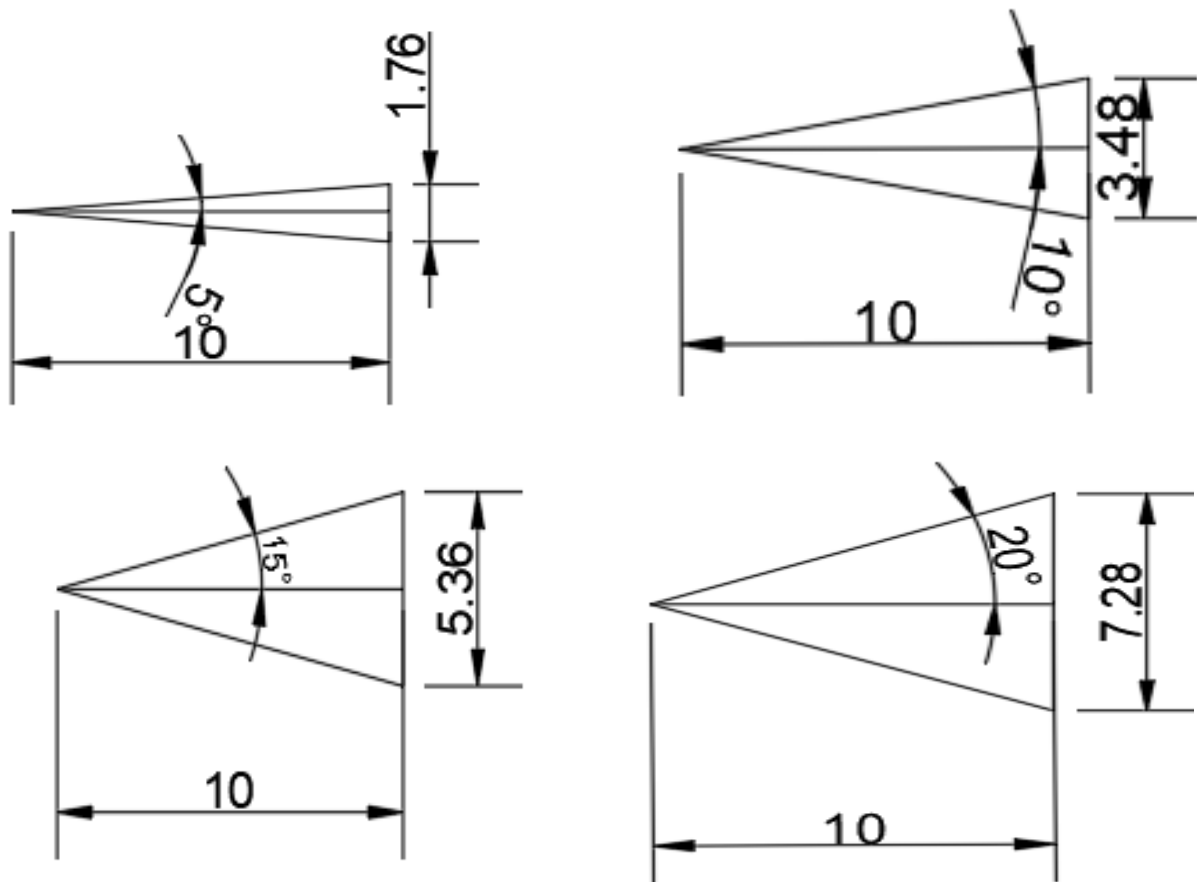


Figure 4.3: Dimensions of wedges of varying angles. All the dimensions are in mm.

4.1.3. MESHED DOMAIN:

There are two types of 2-D meshes:

- Quadrilateral
- Triangle

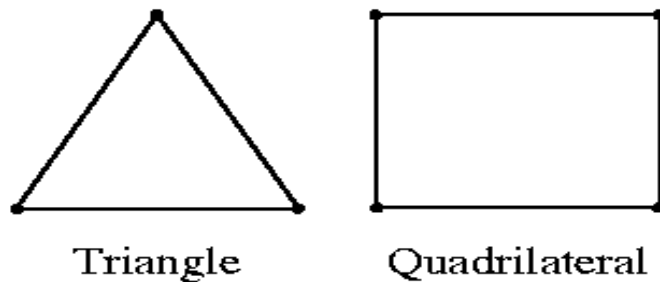


Figure 4.4: Types of mesh elements.

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 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 much closer to 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 occurs.

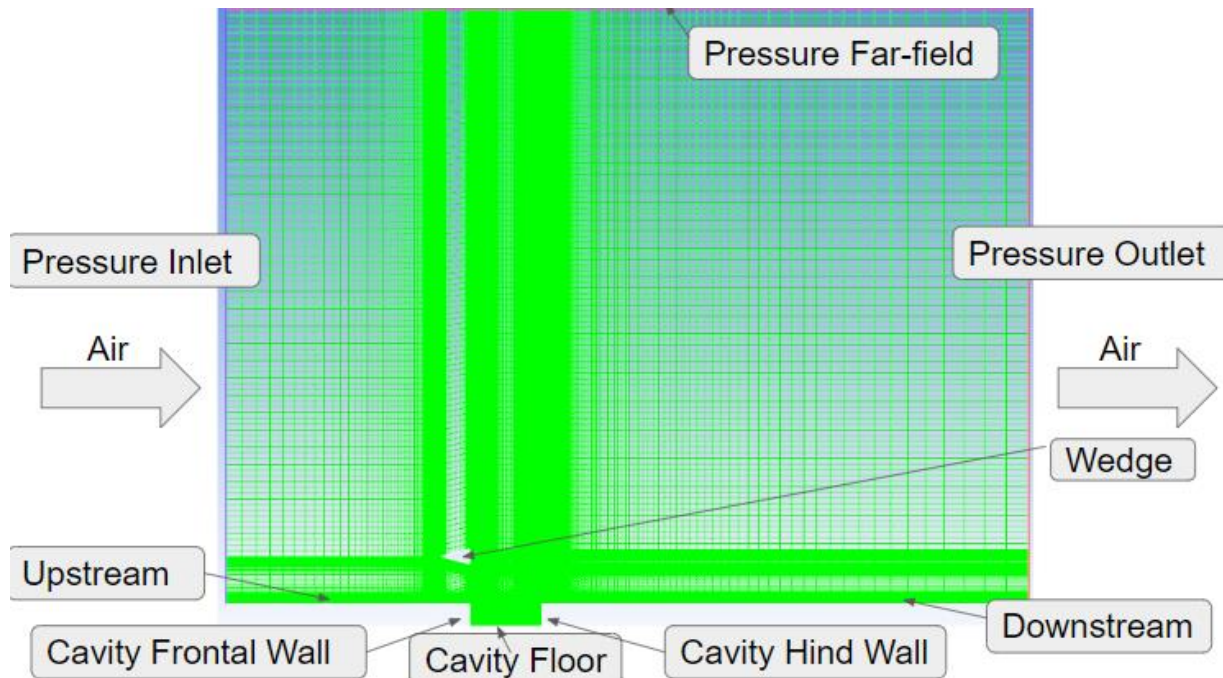


Figure 4.5: Meshed domain for the current study

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.

Since the flow under study is compressible and high speed, the consideration of influence of density in determination of pressure and temperature has to be taken into account. Although the use of pressure based solver is currently extended to compressible flow, the above mentioned consideration puts forth the necessity of using the density based solver. The additional advantage of choosing this solver is the shock resolution.

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 a steady state flow is suitable as it allows this and other complicated flow attributes which are time-dependent can be analyzed further in the future. Also 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 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 – GREEN GUASS CELL BASED

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:

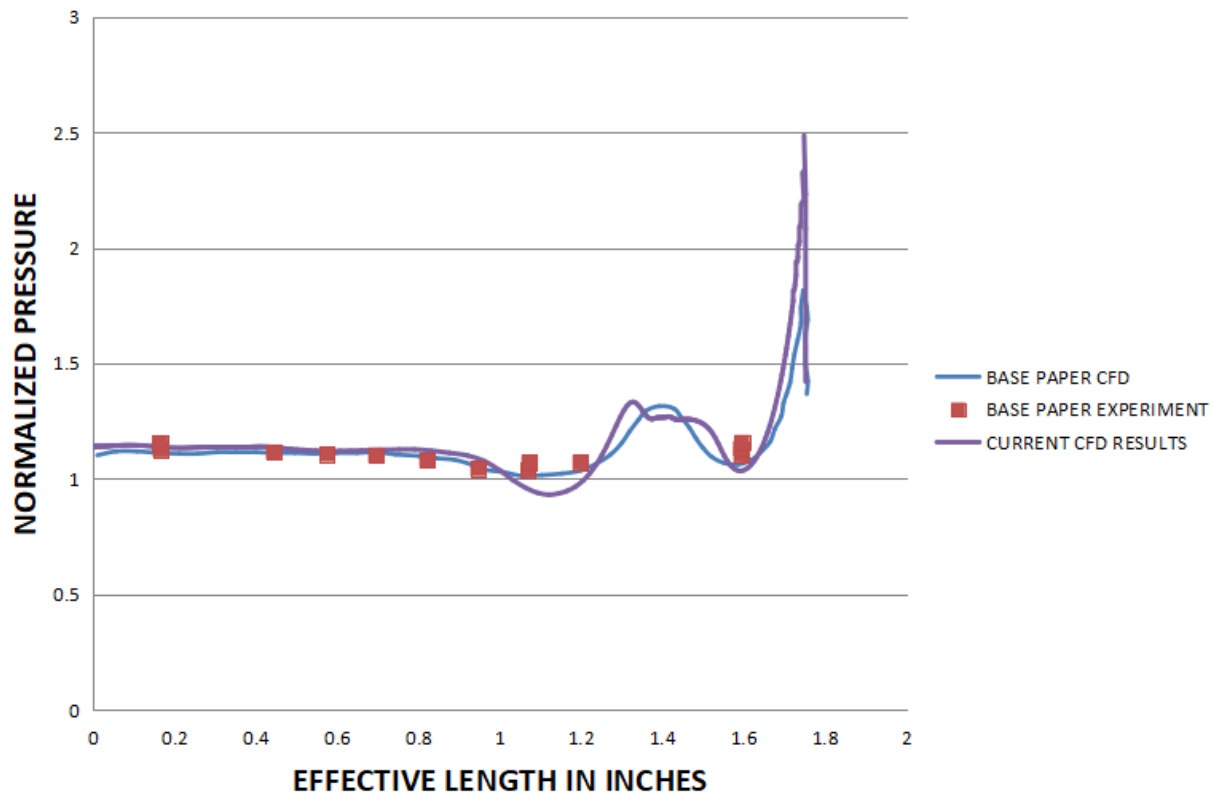


Figure 4.6: Normalized Pressure vs. Effective Length in Inches

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:

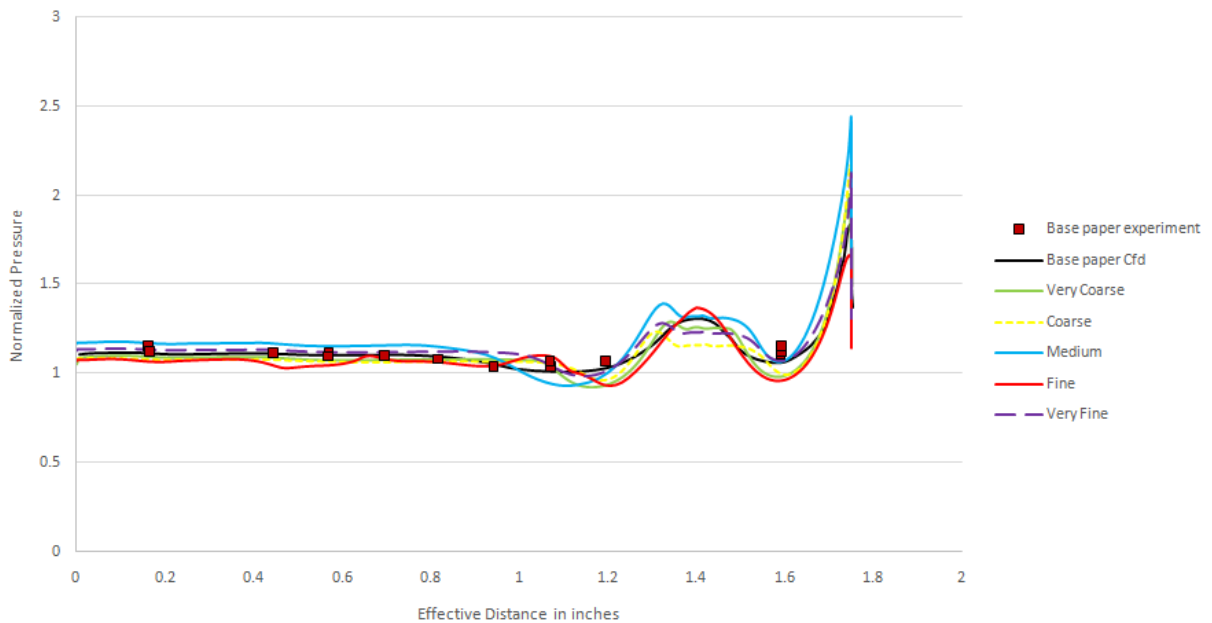


Figure 4.7: Grid Independence Test

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 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:



Figure 4.8: 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 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 12. 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.



Figure 4.9: Wedge holder

The entire test section, portrayed in figure 4.10, 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

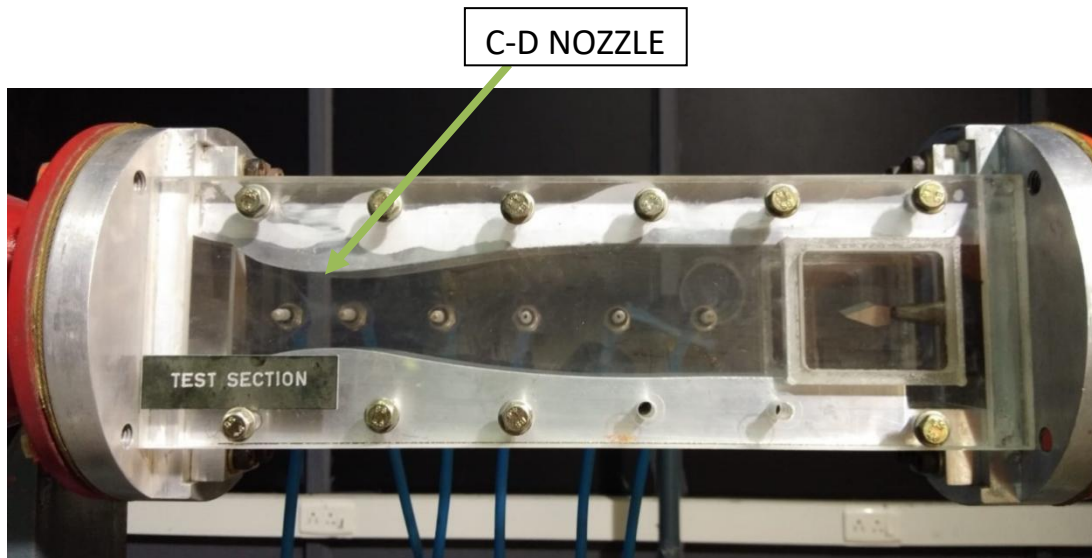


Figure 4.10: Test section

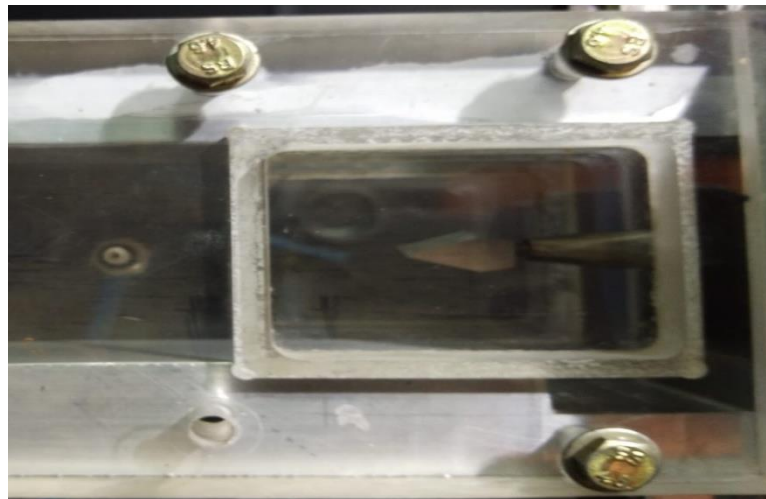
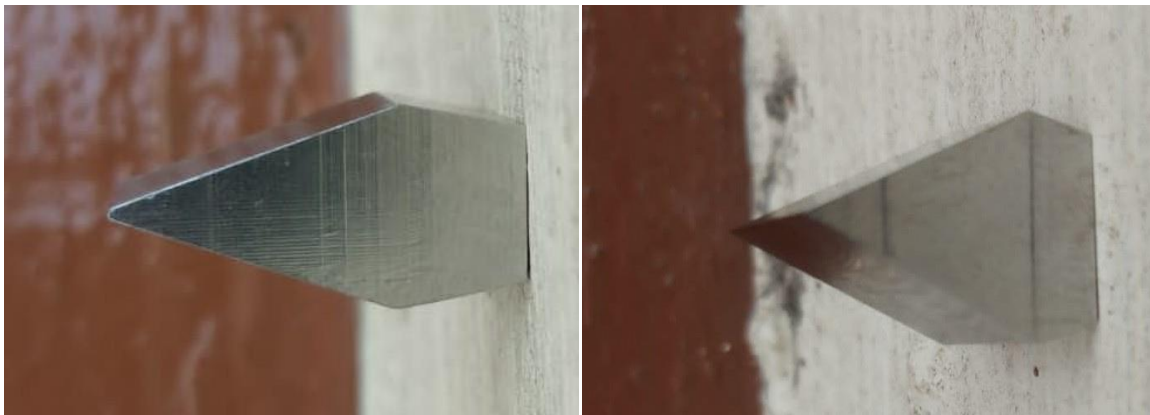


Figure 4.11: Close up of the wedge and cavity model mounted in the test section

The wedge model is fixed into the holder as shown in figure 4.11. 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 fixture.



(A)

(B)



(C)

Figure 4.12: (A) Wedge Model - 15 degrees half wedge angle (B) Wedge Mode -I 20 degrees half wedge angle (C) Threaded hole in the wedge.

4.2.3. MATERIAL USED:

Material used in order to fabricate the wedge and the cavity is stainless steel. Stainless steel is a metal alloy created by combining steel and chromium. The more chromium in the steel, the more durable it is. Two main reasons to select stainless steel over other part are, stainless steel is highly-resistant to corrosion and is more durable and resistant to abrasion. Stainless steel parts are substantially more resistant to scratches and impact damage than aluminum parts. Stainless steel alloys are increasingly being used in aircraft components that demand high strength while also being able to withstand increased weight.

Table 4.2: Properties of Stainless Steel

PROPERTY	VALUE
Density	23°C g/cm ³
Thermal Conductivity	100°C W/m/°K
Coefficient Thermal Expansion x10 ⁶	51 - 150°C m/m/°C
Tensile - Yield strength (0.2%)	305MPa
Strength	510MPa
Elongation	10%
Young Modulus	137GPa
Compression - Yield Strength (0.2%)	317MPa

CHAPTER 5

RESULTS AND DISCUSSIONS

5.1. COMPUTATIONAL RESULTS:

5.1.1. INITIAL ANALYSIS (WITH AND WITHOUT WEDGE):

5.1.1.1 MACH CONTOUR:

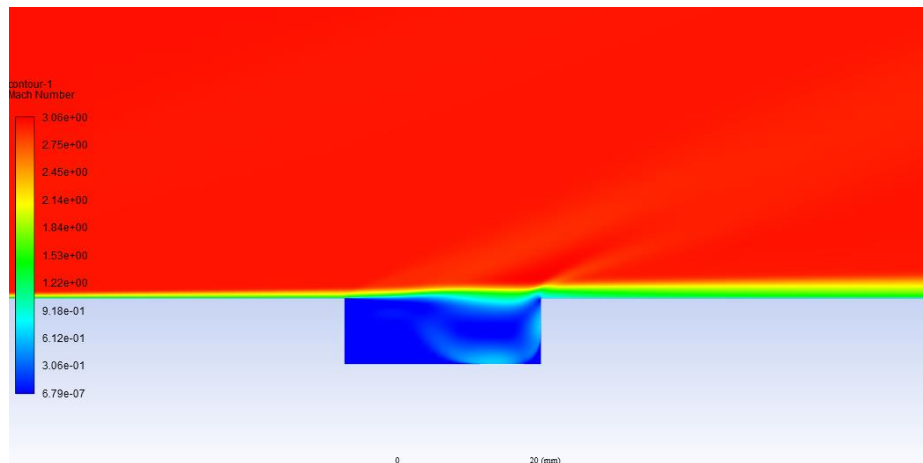


Figure 5.1: Mach contour of open cavity without wedge

The figure above is useful in depiction of the flow within and in the vicinity of the cavity. The development of shear layer occurs for the flow close to the wall. Since this is the study of open cavity, the shear layer does not reach the cavity floor. The expansion of the extent of the shear layer downstream of the flow is observed. There is development of shock wave both in the frontal and hind wall corners of the cavity. The shock wave formed is weak in nature; this is indicated by the faint change in color in the figure above.

The figure below is the depiction of Mach contour in the presence of wedge. The appearance of the shear layer is somewhat similar to the previous case. The shock generated at both the corners of the cavity look much more prominent. The shock wave formed upstream reduces the velocity of the flow and aft of the wave there is a drastic rise in the velocity of the flow, until the flow intercepts with the shock wave

formed downstream. The lift off the shear layer occurs due to the interaction between the shock wave generated from the wedge with the shear layer passing over the cavity. The external flow slightly dips within the cavity close to the hind wall. The shock originated due to the presence of the wedge is clearly seen. Wake region formed behind the wedge is indicated by the drastic drop in the flow velocity. Within the cavity, regardless of the supersonic flow over the domain, the flow remains lower than the speed of sound.

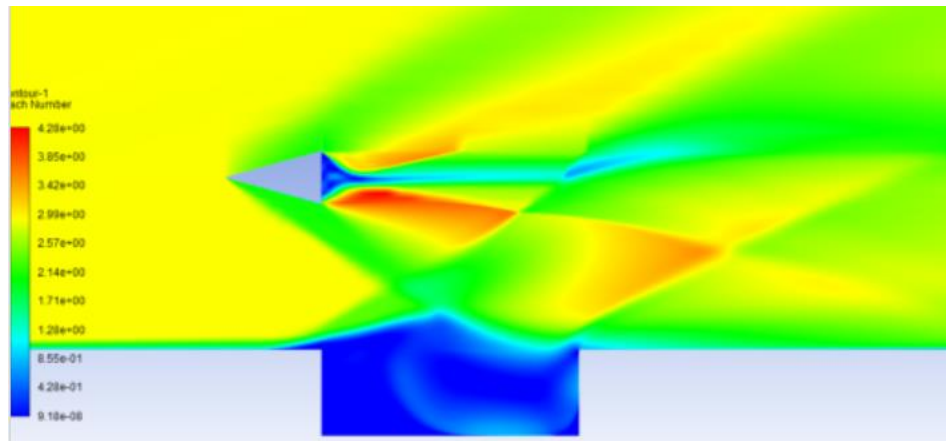


Figure 5.2: Mach contour with wedge

5.1.1.2. RECIRCULATION ZONES

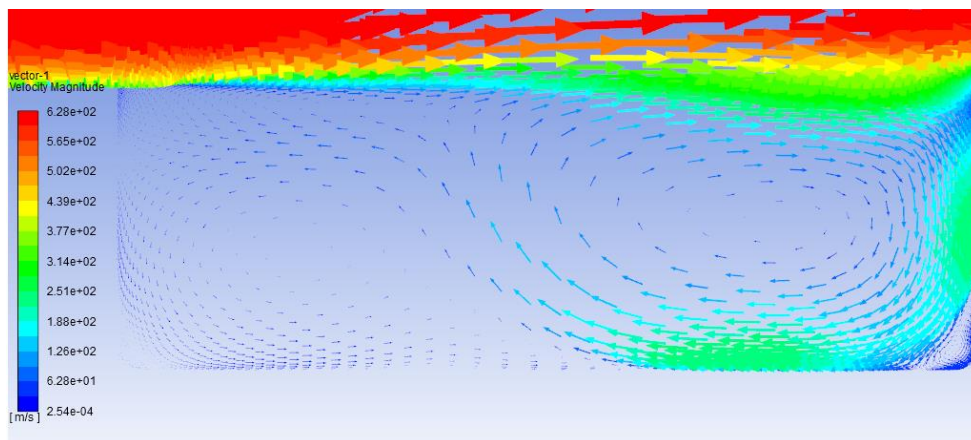


Figure 5.3: Vector plot in the case of absence of wedge.

The formation of recirculation zones within the cavity is one of the interesting concepts that have been sought after by researchers, as these greatly influence the mixing properties of the fluid, in the case of flame holder applications. Formation of two zones is noticed, one close to the hind wall, which rotates in the clockwise direction and another close to the frontal wall of the cavity, which rotates in the anti-clockwise direction. These are termed as primary and secondary recirculation zones. Also close to the corner that is formed as a result of the intersection of the cavity floor and cavity hind wall, the velocity drops drastically. This point emerges due to the shape of the recirculation zones, which is curved at the edges.

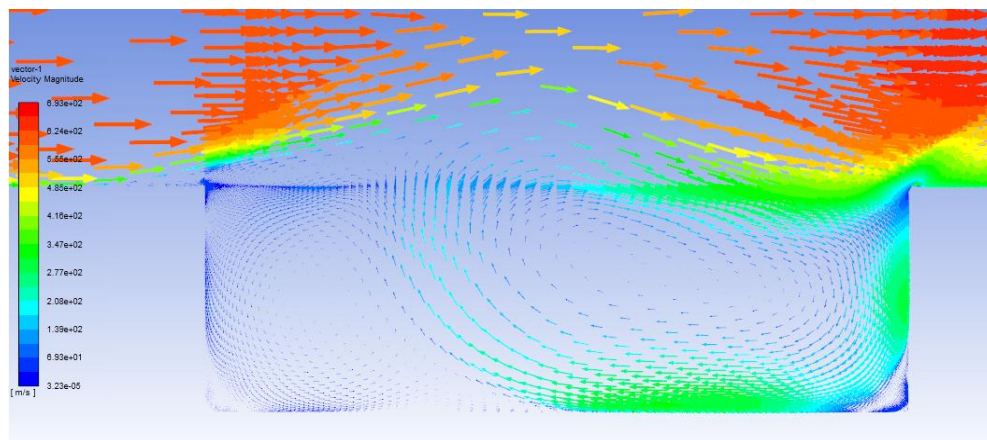


Figure 5.4: Vector plot in the case of presence of the wedge

Now after the interaction of the shock from the wedge, the shear layer has been lifted up. The expansion of the primary recirculation zone is observed. This leads to reduction in the extent of the other zone. All other aspects including the direction of the rotation remains the same with that of the previous case. This reduction of the extent of the secondary zone can ultimately lead to better fuel-air mixing, in the case of flame holders.

5.1.1.3. PRESSURE PLOT:

The graph displayed below only calculates the distribution of pressure with respect to the horizontal span of the cavity wall. The overall coefficient of pressure value (C_p)

increased twice in the presence of the wedge. The change in pressure is not observed in the frontal wall (from 0 inches to 0.35 inches). There is a slight dip in the pressure value close to lower right corner of the cavity. This dip becomes quite prominent on the presence of shock wave emerging from the wedge. Exactly at the lower corner of the cavity hind wall (hind wall extends from 0.35 to 1.4 inches), there appears a peak in the pressure value. This is due to the stagnation point that was discussed in Section 5.1.1.2. Along the cavity hind wall the pressure drops and a sudden peak is noticed at the corner of the downstream wall. This is explained by the origination of the shock wave in that location. This peak value is nearly 8 times greater for with wedge case.

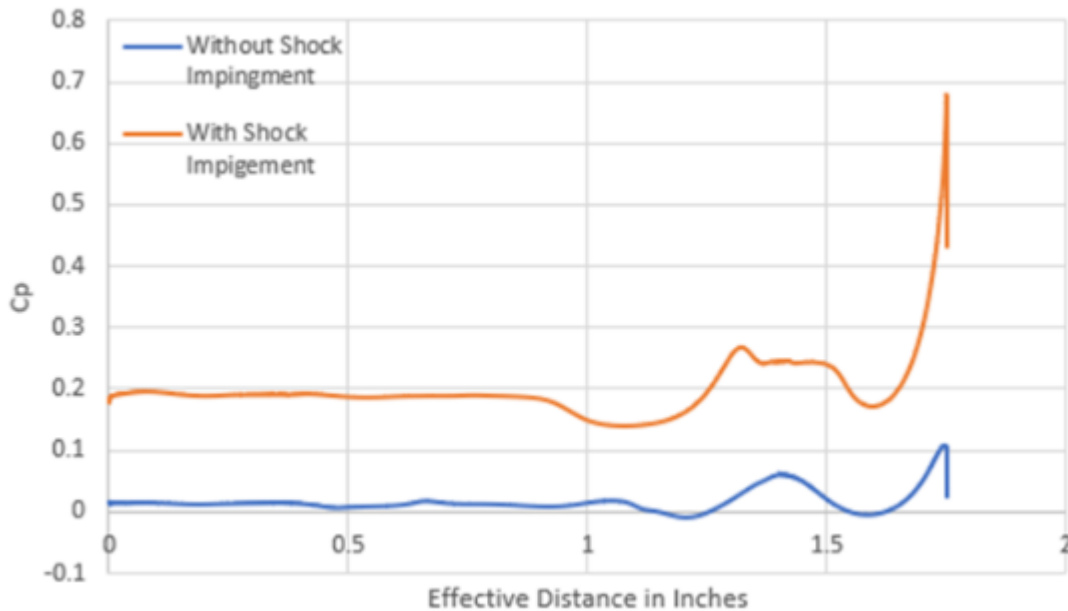


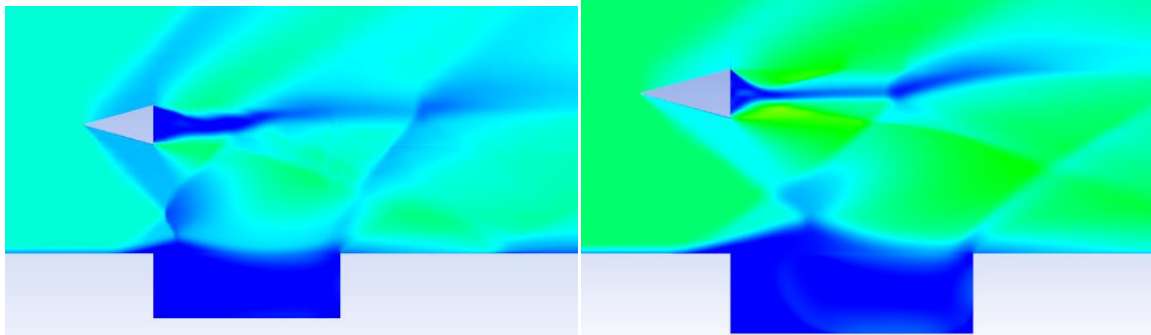
Figure 5.5: Length of the cavity wall vs. Value of C_p

5.1.2. VARYING MACH NUMBER ANALYSIS:

5.1.2.1. MACH CONTOUR:

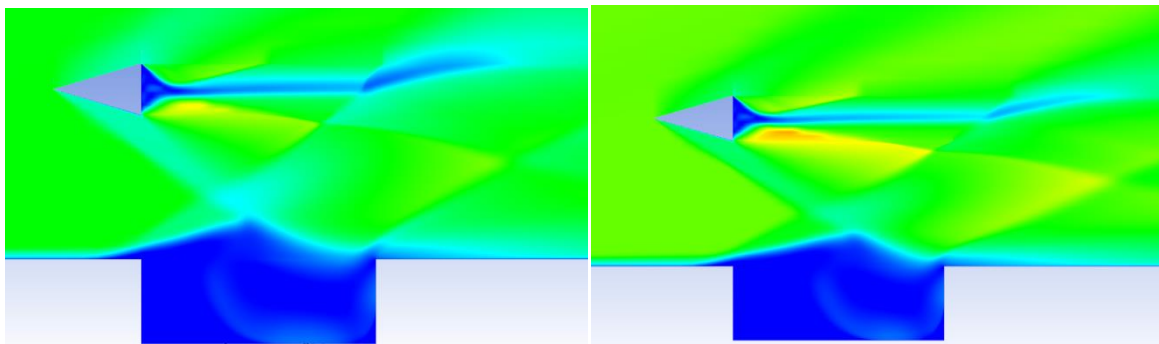
From the figure depicted below the change in the value of the free stream Mach number clearly results in the decrement of the wave angle. But from the shock wave relations mentioned earlier, the strength of the shock (the pressure ratio front and behind the wave) is influenced by both Mach number and shock angle. However the

change in shock angle is very negligible in this case. Therefore, as the Mach number increases the strength of the shock increases with it. The increase in the height to



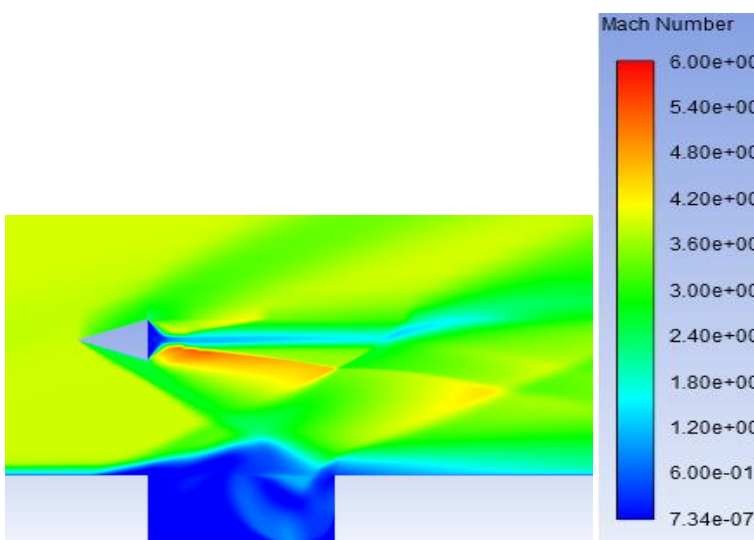
(A)

(B)



(C)

(D)



(E)

(F)

Figure 5.6: (A) Mach 2 (B) Mach 2.5 (C) Mach 3 (D) Mach 3.5 (E) Mach 4 (F) Color Map for Mach Contour

which the shear layer rises above the cavity is not significant as the Mach number increases. And as the value of Mach number is increased the intensity of plunging of the external flow within the cavity rises. At Mach 4, we can observe the external flow finally mixes with the internal flow. The point at which the interaction of shock wave from wedge and the shear layer moves further behind with rising Mach number. Also the shock formed at the separation corners of the cavity moves the closer the downstream wall i.e. reduction in the shock angle. The wake region aft of the wedge appears like tail. The tail tends to become thinner as there is a growth in the value of Mach number. The wake region tail interacts with the shock wave generated from the cavity upper corner and the tail is deflected upwards.

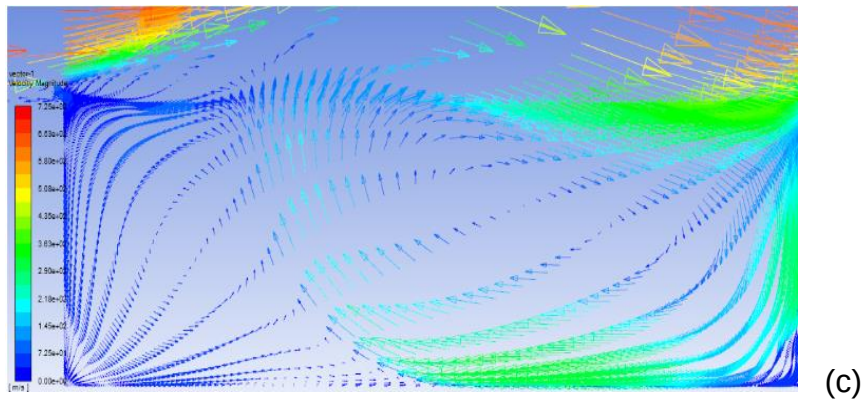
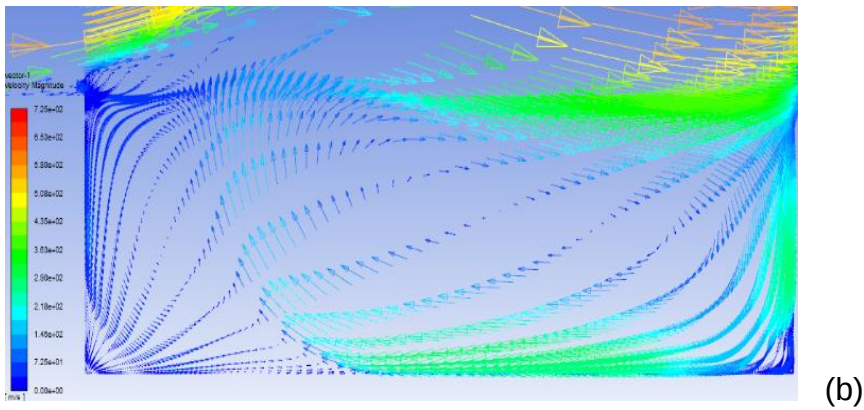
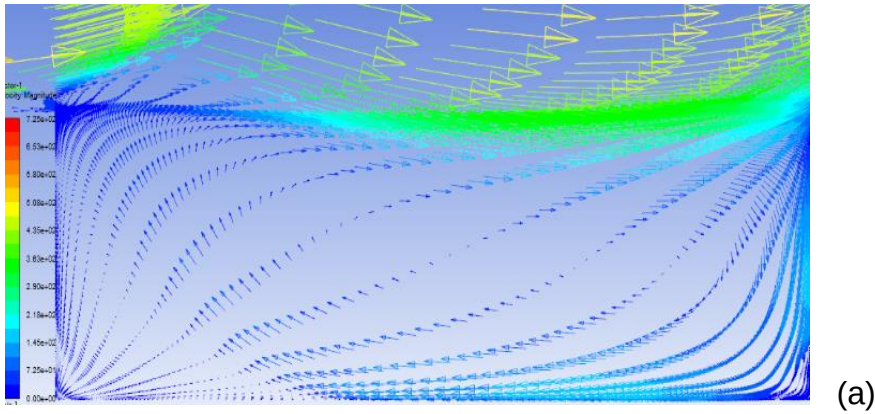
Table 5.1: Values of shock angle and strength of the shock wave for different Mach number.

Mach Number	Shock Angle	Strength of the shock wave
2	45.34	2.1946
2.5	36.94	2.4675
3	32.24	2.8215
3.5	29.19	3.233
4	27.06	3.6972

5.1.2.2. RECIRCULATION ZONES

There are two recirculation zones within the cavity as mentioned in section 5.1.1.2. 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 lift 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.



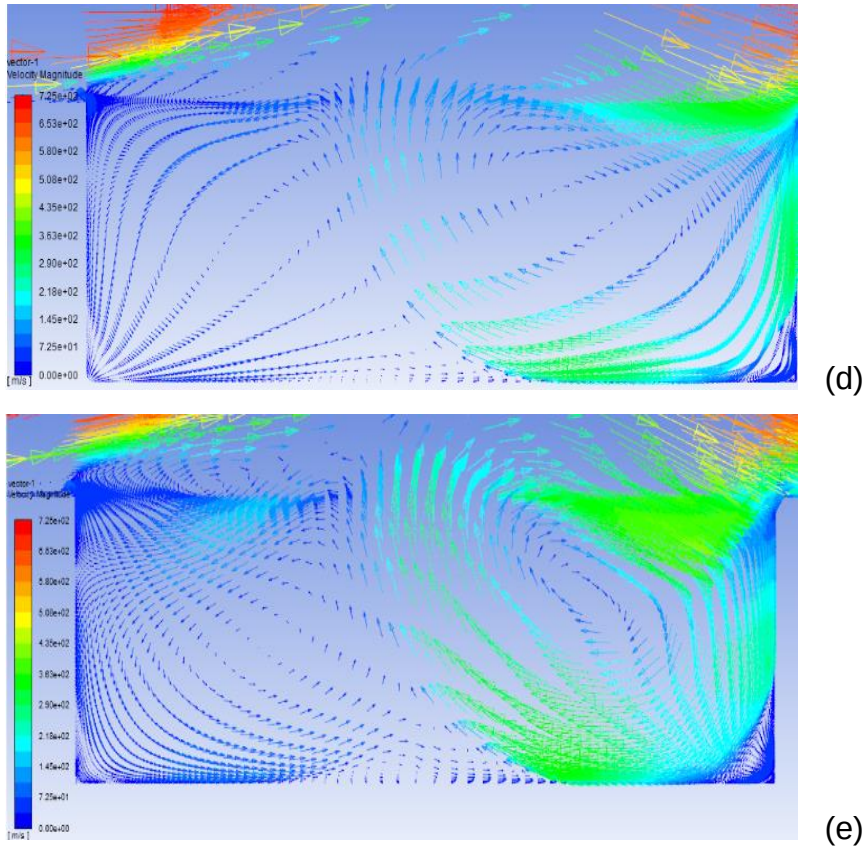


Figure 5.7: Vector plots for (A) Mach 2 (B) Mach 2.5 (C) Mach 3 (D) Mach 3.5 (E) Mach 4

5.1.2.3. PRESSURE PLOT:

The general Mach number and pressure relationship is clearly seen in the graph i.e. as Mach number increase pressure decreases. The general trend as explained in section 5.1.1.3 is observed. As the Mach number drops, the dip in pressure noticed on the cavity floor becomes restricted in extent. The peak noticed at the downstream upper corner is the highest for Mach 3 (before which there was an increase from Mach 2 and after Mach 3 the value drops). So the stretch of the shock formed at the separation corner of the shear layer is the strongest for Mach 3.

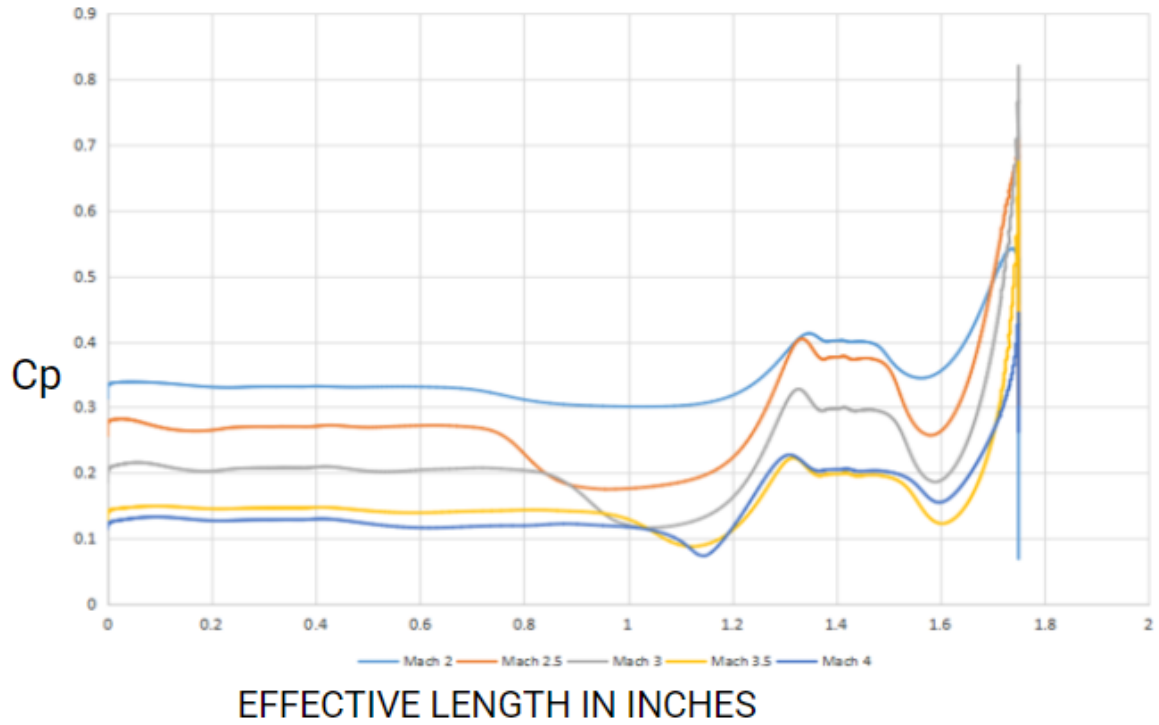
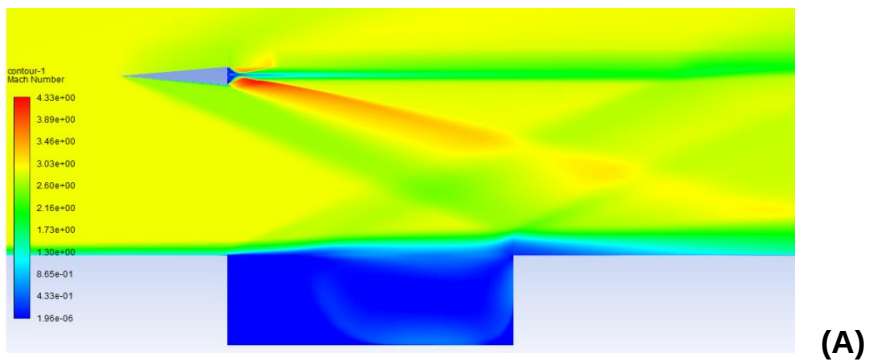
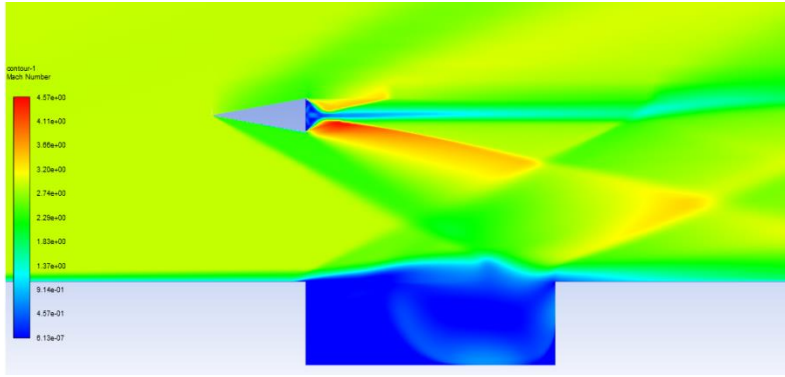


Figure 5.8: Horizontal span vs. C_p graph

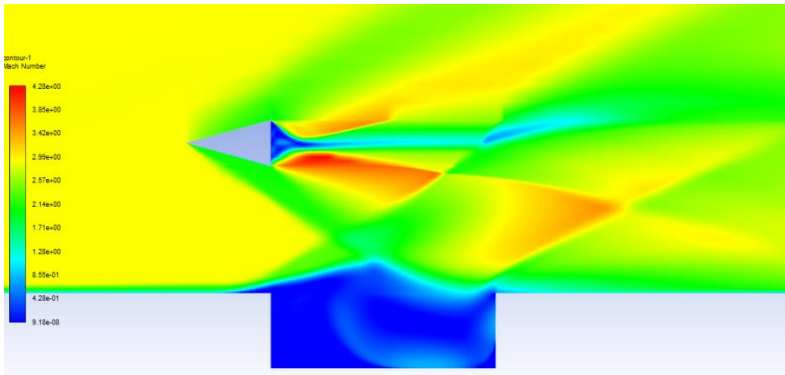
5.1.3. VARYING WEDGE ANGLE:

5.1.3.1. MACH CONTOUR:

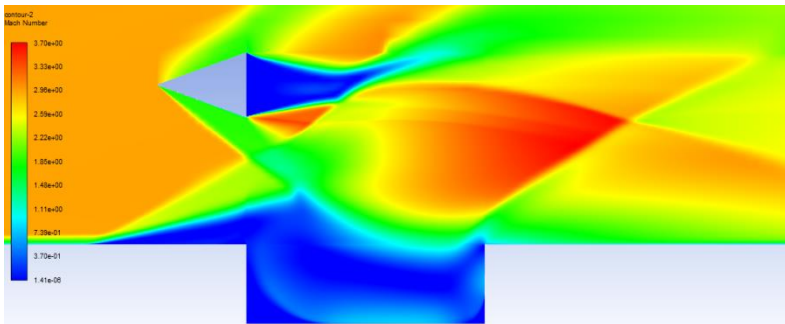




(B)



(C)



(D)

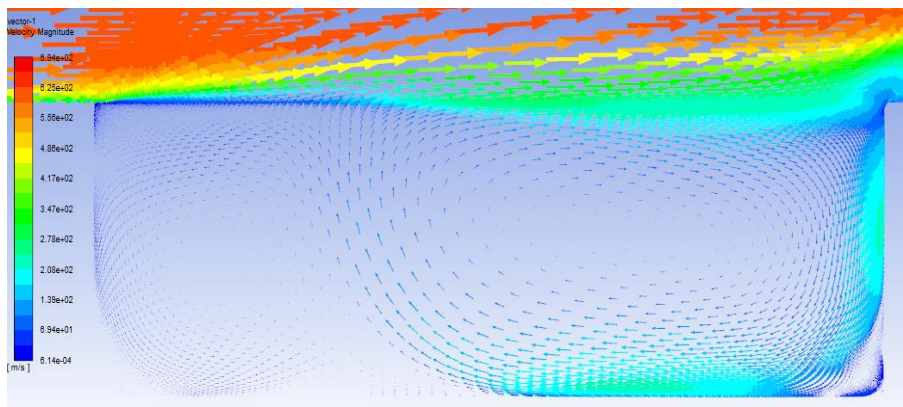
Figure 5.9: Mach contours for (A) 5 deg (B) 10 deg (C) 15 deg (D) 20 deg

For 5 degree half wedge, the interaction of shock wave from the wedge behind the cavity hind wall. There is an absence of significant rise in the height of shear layer. For every 5 degree increase in half wedge the rise in the height of the shear layer increases and also the shear layer seeps into the cavity close to the hind wall. The increment in the wedge angle resulted in the increment of the height of lift up of shear layer. As depicted from the contours, the primary recirculation zone widened. At the last case (D) the entire cavity is filled with only one zone. Shock angle varied directly with respect to the shock strength.

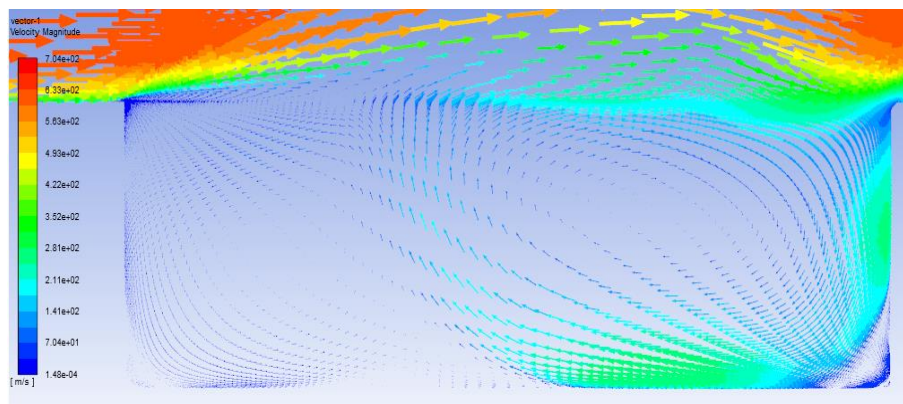
Table 5.2: Shock angle vs. shock strength of different half wedge angle

HALF WEDGE ANGLE	SHOCK ANGLE	SHOCK STRENGTH
5	23.13	1.4539
10	27.38	2.0544
15	32.24	2.8215
20	37.76	3.7712

5.1.3.2. RECIRCULATION ZONE:



(A)



(B)

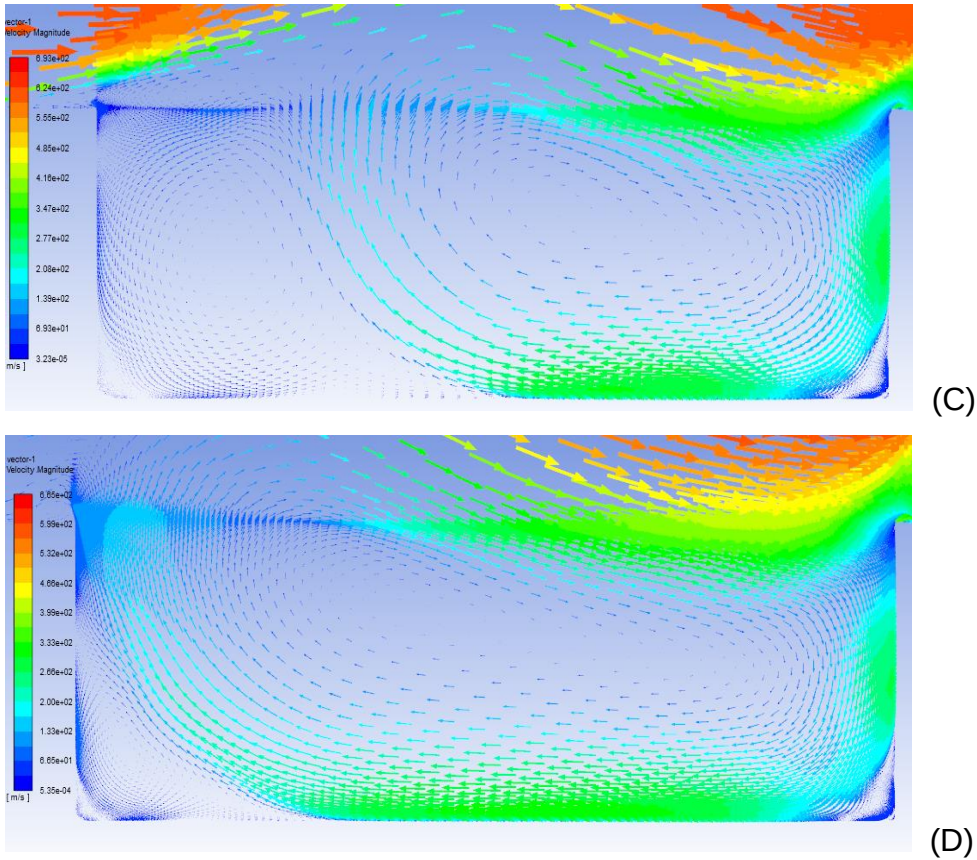


Figure 5.10: Vectors plots of half wedge angle of (A) 5 deg (B) 10 deg (C) 15 deg (D) 20 deg

Figure 5.10 displays the recirculation zones within the cavity. With every increment in the value of half wedge angle, there is a growth in the extent of the primary recirculation zone. At 20 degree half wedge angle, the entire cavity is nearly filled with only one recirculation zone. The primary zone is noticed to rotate at a velocity greater than the velocity of the other zone. The former rotates at speed greater than Mach 1 and the latter rotates at velocity less than the velocity of sound.

5.1.3.3. PRESSURE PLOT:

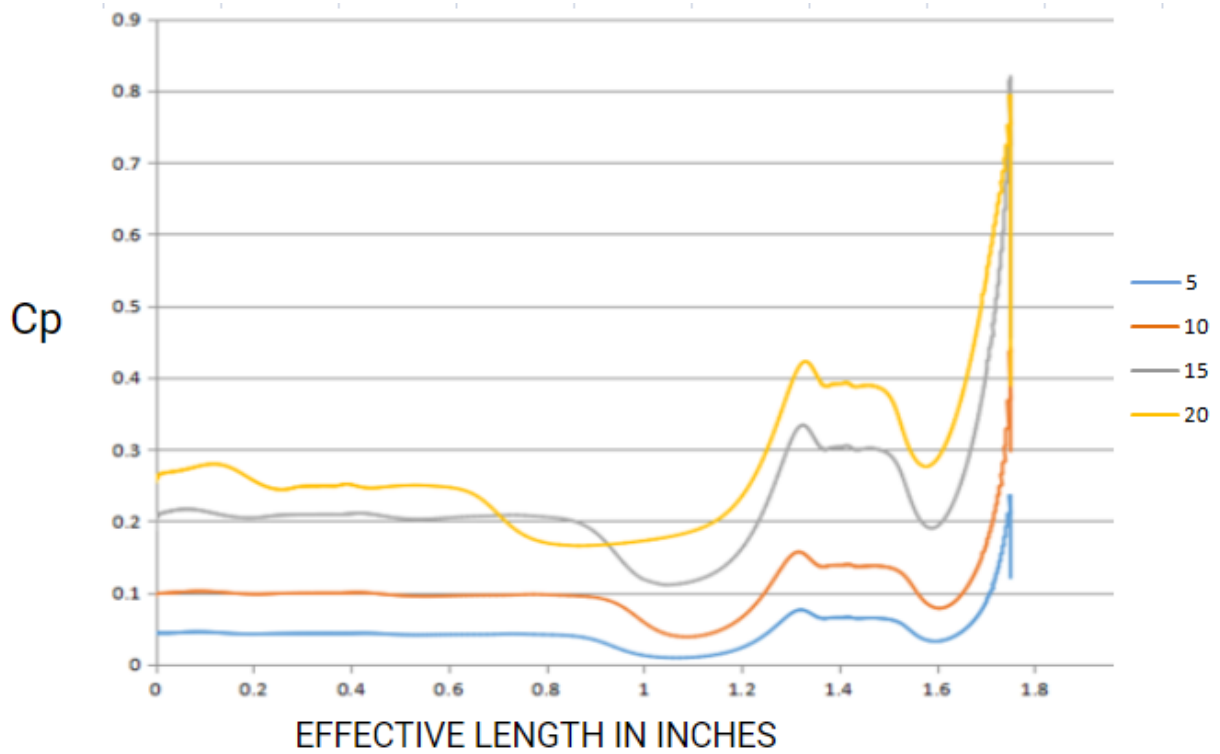


Figure 5.11: Effective length in inches vs. C_p graph

The trend followed by the previous cases was also observed in these cases. The wedge angle varies directly with the pressure coefficient. As the primary recirculation zone expands, its effects are felt by the frontal wall due to proximity. The varying values of C_p on the frontal wall for the 20 degree case are due to this expansion. The width of the pressure drop at the cavity wall, as demonstrated in figure 5.11, is also in accordance with the above stated phenomenon. The velocity of the zone close to bottom wall is always higher than the rest of the regions of the zone. Thus, the pressure drops in that locality.

5.2. EXPERIMENTAL RESULTS:

The experimental analysis was done the shock that was generated from the wedge was visible via the schlieren shadowgraph apparatus. The geo tag photos of the model are given below.

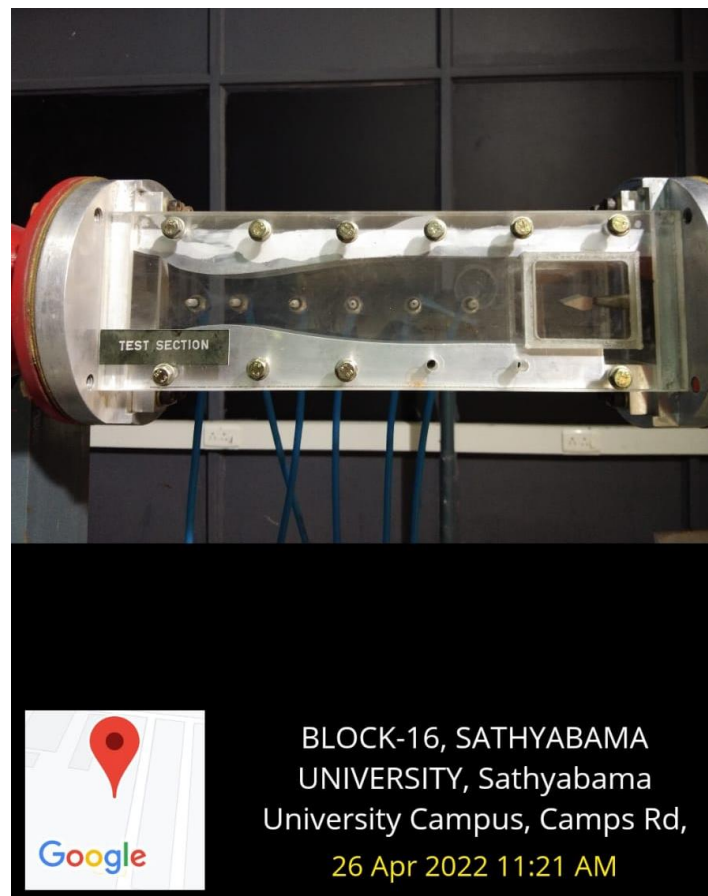


Figure 5.12: Geo tag photo of the cavity and wedge model fixed to the test section

The figure 5.13 depicts the visualization of the shock obtained. The entire image is inverted and hence the cavity is notices at the top at the figure. The wedge was focused along with the cavity. The shock formed over the cavity cannot be visualized due to the presence of double acrylic layer on the side. The tunnel Mach pressure was found out to be Mach 1.05 and hence the shock that can be formed at this speed remains detached to the wedge. This has to be far away from the region of focus of

the schlieren apparatus. Hence, no shock is visible in the figure 5.13. the compression wave that is formed towards the end of the body is due to the turning effect from the hind tip of the wedge to the extension (provided for fixture).

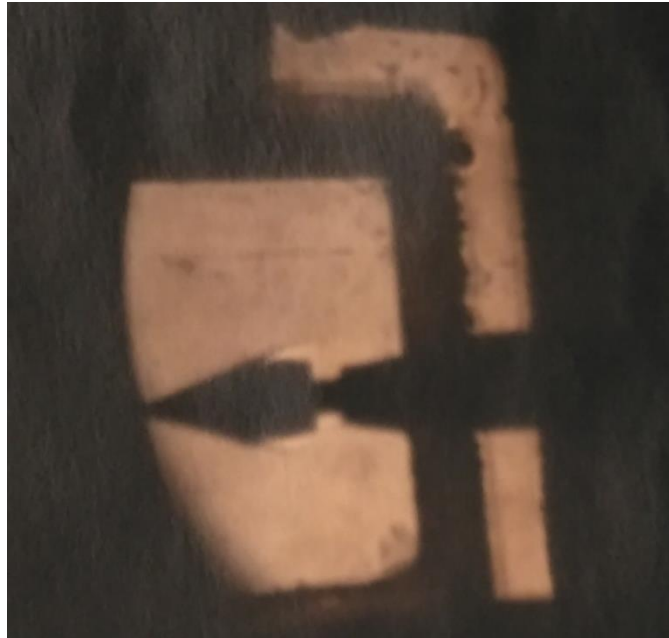


Figure 5.13: Visualization using shadowgraph

CHAPTER 6

CONCLUSION

The project's goal is to study the impact brought about by an incident shock wave on the cavity flow attributes. Since cavity is an unavoidable appearance on any airborne vehicle structure, this study is essential for many applications. A basic rectangular cavity is modeled with L/D ratio 3, OR 1 and hind ramp wall angle 90 degrees. A wedge is also modeled with height 10mm. Computational analysis of the flow field was accomplished through Ansys-Fluent. First the effects of the incidents shock were studied. Next, the analysis was further carried out by varying the value of Mach number. Thirdly, the final analysis of varying wedge angle was carried out. For each of the cases, pressure plots, Mach contours and vector plots are obtained. These are for understanding the pressure distribution over the cavity wall, the visualization of the shock waves that originates from both the separation corner and the wedge and the recirculation zones within the cavity respectively. The pressure plots revealed a similar trend for all cases but with varying values for each case. The Mach contours depicted the position of incident of shock on the shear layer. The vector plot highlighted the difference in the recirculation zone in each case. The experimental analysis was carried out in the wind tunnel. However the desired Mach number (Mach number used for computational analysis) couldn't be achieved. The general conclusions are as follows:

1. The Mach number is directly proportional to the strength of the shock. So in order to obtain a higher strength shock, the operation Mach number should be increased.
2. The shock angle, on the other hand, doesn't follow the above case. Therefore, in order to obtain lower shock angle, the operational Mach number must be increased.
3. The wedge angle has a direct relationship with both the strength and the angle of the shock. As the wedge angle increase, shock angle and strength of the shock increases.

4. The largest primary recirculation zone is obtained for lowest Mach number and highest wedge angle.

5. The pressure of the cavity floor is an inverse variant of Mach number and direct variant of wedge angle.

6. The incident shock induces rising up of shear layer, due to increase in pressure below the layer.

REFERENCES

- [1] Adrian Pudsey, Hideaki Ogawa, Mathew Bricallia , and Tim Roosa, Numerical investigation of upstream cavity enhanced fuel mixing in scramjet combustors, *Acta Astronautica*, 169, 2020, 50-65
- [2] Ajay Dhankarghare, Jayachandran T. and Muruganandam T.M., Study of pressure oscillations in open strut cavity, *AIAA*, January 2020, 1-9
- [3] Ananthanarayanan Veeraragavan, and Will O. Landsberg, Inclining and aerodynamically contouring a scramjet cavity's upstream wall, *Aerospace Science and Technology*, 2019, 1-11
- [4] Aravind Vaidyanathan, Chintan Panigrahi, and Manoj T. Nair, Effects of subcavity in supersonic flow, *Physics of fluids*, Vol.31, Issue 3, March 2019
- [5] Baurle R. A., Gruber M. R., Hsu K.-Y., and Mathur T., Fundamental Studies of Cavity-Based Flameholder Concepts for Supersonic Combustors, *Journal of Propulsion and Power*, Vol.17, No.1, January-February 2001
- [6] Bellary S. A. I., Devabrata Sahoo, Raghavendra N. V., Sampada Lamb, and Sanchayata Nayal, Computational analysis of novel cavity-based flameholder designs for supersonic combustion engines, *IOP Conf. Series: Materials Science and Engineering*, 814, 2020, 1-9
- [7] Chung-Jen Tarn, Paul D. Orkwis, and Peter J. Disimile, Algebraic Turbulence Model Simulations of Supersonic Open-Cavity Flow Physics, *AIAA Journal*, Vol.36, No.1, November 1996, 2255-2260

- [8] Corin Segal, Frank W. Barnes, and Qiuya Tu, Fuel–Air Mixing Experiments in a Directly Fueled Supersonic Cavity Flameholder, *Journal of Propulsion and Power*, Vol.32, No.2, March 2016
- [9] Craig Whitec, Rodrigo C. Palharini, and Thomas J. Scanlonb, Chemically reacting hypersonic flows over 3D cavities: Flowfield structure characterization, *Computers and Fluids*, 165, 2018, 173-187
- [10] Edwards J.A., and Zhang X., Computational Analysis of Unsteady Supersonic Cavity Flows driven by thick shear layers, *Aeronautical journal*, November 1988, 365-374
- [11] Lawrence Ukeiley, and Nathan Murray, Velocity and surface pressure measurements in an open cavity, *Experiments in Fluids*, 38, April 2005, 656-671