



SCHOOL OF MECHANICAL,
INDUSTRIAL & AERONAUTICAL
ENGINEERING

Supersonic flow Separation Over the Shoulder of Cone-cylinder and Hemisphere-cylinder at Incidence

MECN7018 - Research Project

T.K. Chan

Supervisor: Prof. B. Skews

A project report submitted to the Faculty of Engineering and the Built Environment,
University of the Witwatersrand, Johannesburg, in partial fulfilment of the requirements for
the degree of Master of Science in Engineering.

Johannesburg, October 2020



INDIVIDUAL DECLARATION WITH TASK SUBMITTED FOR ASSESSMENT

I, the undersigned, am registered for the course **MECN7018 - Research Project** in the year 2020. I herewith submit the following task "**Supersonic flow Separation Over the Shoulder of Cone-cylinder and Hemisphere-cylinder at Incidence**" in partial fulfilment of the requirements of the above course.

I hereby declare the following:

- I am aware that plagiarism (the use of someone else's work without their permission and / or without acknowledging the original source) is wrong;
- I confirm that the work submitted herewith for assessment in the above course is my own unaided work except where I have explicitly stated otherwise;
- This task has not been submitted before, either individually or jointly, for any course requirement, examination or degree at this or any other tertiary educational institution;
- I have followed the required conventions in referencing the thoughts and ideas of others;
- I understand that the University of the Witwatersrand may take disciplinary action against me if it can be shown that this task is not my own unaided work or that I have failed to acknowledge the sources of the ideas or words in my writing in this task.

Signed this 18th day of October 2020

T.K. Chan

Abstract

The study of supersonic flow and flow separation over slender bodies have been studied extensively in the past where research towards fuselage and projectile body applications were common. However, the research of supersonic flow separation requires emphasis in the 3D domain using computational analytical tools to contribute towards the extensive research of such topic. The purpose of this research is to analyse, in detail, three-dimensional (3D) supersonic flow separation while utilizing computational tools to assist with flow visualization. The supersonic flow separation characteristics and properties over the shoulder of a cone-cylinder and hemisphere-cylinder are investigated at various angles of incidence and Mach numbers.

The hemisphere-cylinder and cone-cylinder have a diameter of 16mm and the cone-cylinder has a cone deflection angle of 25° . The selected flow conditions were M2.8 and M3.2, corresponding to high Reynolds numbers where turbulent flow separation conditions are present. Computational fluid dynamics (CFD) results are validated qualitatively with experimental data obtained from wind tunnel testing. Optics and surface oil flow visualization techniques were used to gather experimental results. Flow separation characteristics and properties are presented as surface flow streamlines, cross sectional contours, flow field streamlines, planar vectors, and 3D surfaces.

The hemisphere-cylinder encountered more complex flow separation characteristics over the shoulder compared to the cone-cylinder at high angles of incidence. The angle of incidence of the slender bodies affected the initiation of flow separation over the shoulder hence generating complex vortical flow structures downstream of the shoulder. The variation of freestream Mach number flow conditions did not indicate significant differences in the flow separation characteristics.

Acknowledgements

I would like to thank my supervisor Prof. Beric Skews for assisting, inspiring and giving me the passion for gas dynamics studies. Your dedication towards gas dynamic and supersonic flow studies is admirable and left me with a strong desire to continue the limitless study of flow research. The excitement you showed from discussing shock waves and supersonic flow was contagious. It was an honour and privilege to have you as a supervisor.

To Dr. Randall Paton, Timothy Missing, Irshaad Mahomed, and my colleagues in the Flow Research Unit, I thank you for your assistance and guidance. Your insights, discussions, and advices were indispensable. Your presence around the laboratory was welcoming and certainly made me feel at home whenever I was working in Flow Research Unit.

I would like to thank Shaun Riekert and the North West Engineering Workshop technicians for manufacturing the test pieces and mounts for my research.

This work was made possible by an allocation of resources (programme number MECH0847) at the Centre for High-Performance Computing (CHPC), a part of the South African National Integrated Cyberinfrastructure System (NICIS). Their resources and constant communication is greatly appreciated.

Finally, I would like to thank my family and friends for the much needed support. Your patience, understanding, and space that you gave helped me complete this research. I am forever grateful for the support that you have given me.

Contents

Declaration	i
Abstract	ii
Acknowledgements	iii
Contents	iv
List of Figures	viii
List of Tables	xv
1 Research Background	1
1.1 Purpose of the Study	1
1.2 Research Background	1
1.3 Research Motivation	2
1.4 Problem Statement	3
2 Research Objectives	4
3 Literature Review	5
3.1 Compressible and Supersonic Flow	5

3.1.1	Oblique Shock Waves	5
3.1.2	Expansion Waves	6
3.2	Flows in the Three-dimensional Domain	7
3.2.1	Conical Flows	7
3.2.2	Flows over blunt bodies	8
3.2.3	Slender Bodies at Incidence	9
3.3	Flow Separation and Topology	11
3.4	Influence of Reynolds Number	13
3.5	Flow Visualisation	15
3.5.1	Experimental Flow Visualization	15
3.5.2	CFD Flow Visualization	16
4	Experimental Method	17
4.1	Experimental Facilities and Apparatus	17
4.1.1	Supersonic Wind Tunnel (SSWT) Facility	17
4.1.2	Mounting block and Test Pieces	21
4.1.3	Shadowgraph and Schlieren Imaging	22
4.1.4	Surface Oil Flow Visualization	22
4.2	Experimental Procedure	23
4.2.1	Storage Pressurization Procedure	23
4.2.2	Test Procedure	24
4.2.3	Shut-down Procedure	25
4.3	Precautions	26
5	Computational Method	27

5.1	Geometry and Meshing	28
5.1.1	Geometry	28
5.1.2	Inflation Layer Mesh Settings	29
5.1.3	Local Mesh Refinement	31
5.2	Model Equations and Solvers	32
5.3	Model Independence	34
5.3.1	Turbulence Model Independence	35
5.3.2	Mesh Independence	36
6	Post-Processing	38
6.1	Experimental Results	38
6.1.1	Optics Flow Visualization	38
6.1.2	Surface Oil Flow Visualization	38
6.2	Computational Results	41
6.2.1	CFD Flow Visualization	41
6.2.2	Data Plots	41
6.3	CFD Model Independence	42
6.3.1	Turbulence Model Independence	42
6.3.2	Mesh Independence	48
7	Results and Discussion	59
7.1	Experimental and CFD Validation	59
7.2	Flow Field Characteristics and Flow Topology	64
7.3	Nose Geometry Influences on the Flow Separation Characteristics and Properties	72
7.4	Body Attitude Influences on the Flow Separation Characteristics and Properties	74

7.5	Mach Number Influences on the Flow Separation Characteristics and Properties	78
8	Conclusions	84
9	Recommendations	85
A	: SSWT Calculation	89
A.1	SSWT Mach number calibration	89
B	: Test Model Design	91
B.1	SSWT Shock Wave Loads	91
B.2	Stress Analysis	91
B.3	Test Model Specifications	93
C	: Experimental Methods and Results	105
C.1	Uncertainty Analysis	105
C.2	Sample Test Log	106
D	: Computational Method and Processing	108
D.1	Model Independence Results	108
D.2	Fluent TUI Commands	115
E	: Additional Results	117
F	: Results for Future Research	121

List of Figures

1.1	Flow over a 2-D surface representing characteristics of flow separation, flow reattachment, and formation of boundary layers. [3]	2
1.2	3-D representation of a cone cylinder with separated flow structure. [5]	3
3.1	$\theta - \beta - M$ plot for 2D supersonic flow over 2D wedges. [7]	6
3.2	$\theta - \beta$ plot comparison for weak shock solution at Mach = 2 for a 2D wedge and cone. [7]	8
3.3	Supersonic flow trajectory over a cone at 0° angle of attack. [7]	8
3.4	Schematic diagram of supersonic flow over a blunt-nose body. [7]	9
3.5	Descriptions of the flow field around the cross-section of a cone at a high angle of attack. [8]	10
3.6	Experimental image of flow separation over a hemisphere-cylinder at 19° angle of incidence and Mach 1.2. [9]	11
3.7	Surface flow topology of hemisphere cylinder as: (a) graph indicating the angle of attack of the body with different Reynolds numbers, (b) visual representation of surface streamline patterns over the body. Note: Labels for each flow condition in (b) correspond to the Reynolds number labels in the graph in A. Several flow conditions were excluded as it was redundant. [10]	13
3.8	Plot of Reynolds number, with respect to the cross-sectional diameter of the body, and angle of attack representing different types of flow separation regimes. [14]	14
3.9	Pressure distribution over a hemisphere-cylinder at $\alpha = 20^\circ$ and $Re = 4.2 \times 10^5$. [15]	14

3.10	Surface patterns over the body represented with: (a) powdered coloured paint mixture (b) titanium-dioxide mixture. [16]	15
4.1	Schematic view of the pressure tank area of the SSWT.	18
4.2	Schematic view of the test section area of the SSWT.	20
4.3	Schematic view with normalized dimensions relative to the diameter for: (a) hemisphere-cylinder, (b) cone-cylinder.	21
4.4	Basic schematic view of the optics setup for the schlieren imaging.	22
5.1	Flowchart of the process for computational methods utilizing Ansys Fluent	27
5.2	Isometric view of domain and hemisphere-cylinder cut-out with labelled surfaces for boundary conditions	29
5.3	Pressure contour and mesh gridlines near the surface of the nose of the hemisphere-cylinder for inflation configured with: (a) a smooth transition, (b) abrupt transition of cell size. Images on the left indicates pressure contours only.	30
5.4	Isometric view of domain and the selected volume (in yellow) within the domain selected for local mesh refinement.	32
6.1	Flow chart indicating steps of processing images obtained from optics flow visualization	39
6.2	Flow chart indicating steps of processing images obtained from surface oil flow visualization	40
6.3	Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder simulated with a: (a) $\kappa-\omega$ SST model, (b) $\kappa-\kappa_L-\omega$ model, (c) Transition SST model, using a coarse mesh.	44
6.4	Mach number contours and vectors plot over the symmetry plane of the hemisphere-cylinder simulated with a: (a) $\kappa-\omega$ SST model, (b) $\kappa-\kappa_L-\omega$ model, (c) Transition SST model, using a coarse mesh.	45
6.5	Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder simulated with a: (a) $\kappa-\omega$ SST model, (b) $\kappa-\kappa_L-\omega$ model, (c) Transition SST model, using a fine mesh.	46

6.6	Mach number contours and vectors plot over the symmetry plane of the hemisphere-cylinder simulated with a: (a) $\kappa - \omega$ SST model, (b) $\kappa - \kappa_L - \omega$ model, (c) Transition SST model, using a fine mesh.	47
6.7	Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder with mesh adaption: (a) 2 levels of refinement, (b) 3 levels of refinement, (c) 4 levels of refinement.	49
6.8	Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with mesh adaption for: (a) 2 levels of refinement, (b) 3 levels of refinement, (c) 4 levels of refinement.	50
6.9	Wall shear surface streamlines and skin friction coefficient contours on leeward side of the hemisphere-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers, (c) 90 layers.	52
6.10	Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the cone-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers , (c) 90 layers.	53
6.11	Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the hemisphere-cylinder with local mesh refinement settings: (a) 0.335 mm (150 circumferential cells), (b) 0.28 mm (180 circumferential cells) , (c) 0.25 mm (200 circumferential cells), (d) 0.228 mm (220 circumferential cells).	54
6.12	Pressure deviations relative to 0.228 mm local mesh size at different circumferential locations at the shoulder of the hemisphere-cylinder.	55
6.13	Pressure deviations relative to 0.228 mm local mesh size at different circumferential locations at 5 mm downstream the shoulder of the hemisphere-cylinder.	56
6.14	Pressure deviations relative to 0.25 mm local mesh size at different circumferential locations at 1 mm downstream the shoulder of the cone-cylinder.	57
6.15	Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the cone-cylinder with local mesh refinement settings: (a) 0.42 mm (120 circumferential cells), (b) 0.335 mm (150 circumferential cells) , (c) 0.28 mm (180 circumferential cells), (d) 0.25 mm (200 circumferential cells).	58
7.1	Comparison of the flow field features at M3.2 for the hemisphere-cylinder at an incidence of 30° presented as: (a) schlieren image with vertical knife edge for experimental results (b) density contours with multiple vertical planes for CFD results.	59

7.2	Flow topology of the hemisphere-cylinder pitched at 30° with M3.2 flow conditions presented as: (a) CFD solution with wall shear surface streamlines, (b) Experimental results as surface oil flow visualization, (c) Combination of both experimental and CFD results.	60
7.3	Surface oil flow visualization of the hemisphere-cylinder pitched at 30° with M2.8 flow conditions as seen from: (a) Top view of the leeward side, (b) Side view.	62
7.4	Comparison of the flow field features at M3.2 for the cone-cylinder at an incidence of 30° presented as: (a) schlieren image with vertical knife edge for experimental results (b) density contours with multiple vertical planes for CFD results.	63
7.5	Flow topology of the cone-cylinder pitched at 30° with M3.2 flow conditions presented as: (a) CFD solution with wall shear surface streamlines, (b) Experimental results as surface oil flow visualization, (c) Surface oil flow visualization viewed from an oblique angle.	64
7.6	Mach number contours and velocity vectors at cross section 20mm from the shoulder for hemisphere-cylinder with 30° incidence and M3.2 flow conditions.	65
7.7	Flow field streamlines over hemisphere-cylinder at 30° incidence and M3.2 indicating:(a) Vortex structure downstream of shoulder, (b) Horn vortex structures by shoulder.	66
7.8	3D density iso-surfaces of supersonic flow features over the hemisphere-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A front oblique view, (b) A rear oblique view.	67
7.9	3D Mach number and turbulence intensity iso-surfaces of flow separation features over the hemisphere-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A rear oblique view, (b) A front oblique view.	68
7.10	Mach number contours and velocity vectors at cross section 35mm from the shoulder for cone-cylinder with 30° incidence and M3.2 flow conditions.	69
7.11	Flow field streamlines over cone-cylinder at 30° incidence and M3.2 flow conditions shown at:(a) A rear oblique view, (b) A frontal oblique view.	69
7.12	3D density iso-surfaces of supersonic flow features over the cone-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A front oblique view, (b) A rear oblique view.	71

7.13	3D Mach number and turbulence intensity iso-surfaces of supersonic flow features over the cone-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A rear oblique view, (b) A front oblique view.	71
7.14	Surface streamlines and pressure contours for: (a) hemisphere-cylinder, (b) cone-cylinder, at 30° angle of incidence and M3.2 flow conditions.	72
7.15	Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of: (a) hemisphere-cylinder, (b) cone-cylinder, at 30° incidence and M3.2 flow condition.	73
7.16	Surface streamlines and pressure contours over of hemisphere-cylinder at M3.2 conditions and: (a) 30° angle of incidence (b) 20° angle of incidence.	75
7.17	Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of the hemisphere-cylinder at M3.2 flow conditions and: (a) 30° incidence, (b) 20° incidence.	76
7.18	Surface streamlines and pressure contours over of cone-cylinder in M3.2 conditions and: (a) 30° angle of incidence (b) 20° angle of incidence.	77
7.19	Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of the cone-cylinder at M3.2 flow conditions and: (a) 30° incidence, (b) 20° incidence.	78
7.20	Surface streamlines and pressure contours for hemisphere-cylinder at 30° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.	79
7.21	Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of hemisphere-cylinder at 30° incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.	80
7.22	Surface streamlines and pressure contours for hemisphere-cylinder at 20° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.	81
7.23	Surface streamlines and pressure contours for cone-cylinder at 30° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.	82
7.24	Cross-sectional plane located at 35mm downstream of shoulder with Mach number contours and velocity vectors of cone-cylinder at 30° incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.	82
A.1	Calibration graph representing the relationship between the veeder counter the test section Mach number. Calibration completed by A. A. Fillipi. [27]	90

B.1	FEM stress results for the cone-cylinder and adjustable strut at 30 incidence	92
B.2	FEM stress results for the sting with applied loads transferred from the test piece and adjustable strut	92
B.3	FEM stress results for the mounting block with applied loads transferred from the stings	93
D.1	Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with inflation layer mesh settings of: (a) 60 layers, (b) 75 layers, (c) 90 layers.	108
D.2	Mach number contours and vectors of the symmetry plane of the cone-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers , (c) 90 layers. .	109
D.3	Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with local mesh refinement settings: (a) 0.335mm (150 circumferential cells), (b) 0.28mm (180 circumferential cells) , (c) 0.25mm (200 circumferential cells), (d) 0.228mm (220 circumferential cells).	110
D.4	Coefficient of pressure plot at different circumferential locations at the shoulder of the hemisphere-cylinder.	111
D.5	Coefficient of pressure plot at different circumferential locations at 5mm downstream the shoulder of the hemisphere-cylinder.	112
D.6	Mach number contours and vectors of the symmetry plane of the cone-cylinder with local mesh refinement settings: (a) 0.42mm (120 circumferential cells), (b) 0.335mm (150 circumferential cells) , (c) 0.28mm (180 circumferential cells), (d) 0.25mm (200 circumferential cells).	113
D.7	Coefficient of pressure plot at different circumferential locations at 1mm downstream the shoulder of the cone-cylinder.	114
E.1	Mach contours of hemisphere-cylinder with 30° incidence and M3.2 flow conditions at cross sections located with reference to the shoulder: (a)2.5mm, (b)5mm, (c)10mm, (d)15mm, (e)20mm, (f)25mm, (g)30mm, (h)35mm.	117
E.2	Mach contours of cone-cylinder with 30° incidence and M3.2 flow conditions at cross sections located with reference to the shoulder: (a)2.5mm, (b)5mm, (c)10mm, (d)15mm, (e)20mm, (f)25mm, (g)30mm, (h)35mm.	117

E.3	Mach contours of hemisphere-cylinder with 30° incidence and M3.2 flow conditions at meridional plane of: (a) 5° , (b) 10° , (c) 15° , (d) 20°	118
E.4	Mach contours of cone-cylinder with 30° incidence and M3.2 flow conditions at meridional plane of: (a) 5° , (b) 10° , (c) 15° , (d) 45°	119
E.5	Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 10° incidence and M3.2 flow conditions	120
E.6	Wall shear surface streamlines and pressure contours of the cone-cylinder at 10° incidence and M3.2 flow conditions	120
F.1	Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 15° incidence and M3.2 flow conditions	121
F.2	Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 18° incidence and M3.2 flow conditions	122
F.3	Wall shear surface streamlines and pressure contours of the cone-cylinder at 25° incidence and M3.2 flow conditions	122

List of Tables

5.1	Matrix of the simulation models required to achieve turbulence model independence	35
5.2	Matrix of the simulation models required to achieve mesh independence . . .	37
A.1	Table of veeder counter and Mach number obtained from calibration test completed by A. A. Fillipi. [27]	89
C.1	Calibration data on the pressure transducers used for the experimental data. Data edited from reference. [4]	106

Nomenclature

γ	Heat capacity constant
γ_{int}	Intermittency factor $[-]$
κ	Turbulent kinetic energy $[m^2/s^2]$
κ_L	Laminar kinetic energy $[m^2/s^2]$
μ_m	Dynamics viscosity of medium $[Ns/m^2]$
ν	Kinematic viscosity $[m^2/s]$
ω	Specific dissipative rate $[m^2/s^3]$
ρ	Density $[kg/m^3]$
τ_w	Shear stress at the wall $[Pa]$
a	Speed of sound $[m/s]$
C	Veeder Counter
d	Characteristic length $[m]$
F	Focus point
F_N	Start-up shock wave force (N)
g_r	Growth rate of the inflation layer $[-]$
h_{IL}	Height of the inflation layer cell $[m]$
M	Mach Number
N_L	Number of inflation layers $[-]$
P_0	Stagnation Pressure (Pa)
q	Dynamic Pressure (Pa)
R	Reattachment point

$RANS$	Reynolds Averaged Navier-Stokes
Re	Reynolds number $[-]$
Re_θ	Momentum thickness Reynolds number $[-]$
S_0	Flow separation point
S_1	Saddle point
S_F	Total surface area (m^2)
S_L	Surface area of lifting surface (m^2)
TUI	Terminal User Interface
u_w	Local velocity at the wall $[m/s]$
V	Velocity $[m/s^2]$
y_+	Dimensionless distance of boundary layer to wall $[-]$
y_{bl}	Distance of the first boundary layer to the wall $[mm]$
CAD	Computer-Aided Design
HD	High Definition
MPa	Mega Pascals
PC	Personal Computer
RAM	Random Access Memory
SOFV	Surface Oil Flow Visualization
SSWT	Supersonic Wind Tunnel

1 Research Background

1.1 Purpose of the Study

The purpose of this research is to contribute to the knowledge of flow separation in the three dimensional (3D) domain that occurs around a slender body travelling at steady supersonic conditions. The study of flow in the 3D domain at various Mach numbers and Reynolds number is well established, however flow separation over the shoulder of a slender body in the 3D domain requires additional research as it is a topic that have not yet been focused on.

1.2 Research Background

Supersonic flow is a flow condition where the flow medium or the moving body exceeds the speed of sound or Mach one. The Mach number is a dimensionless value, representing the speed of the flow, which is defined as:

$$M = \frac{V}{a} \quad (1.1)$$

V is the local velocity in the flow field and M is the Mach number which can be interpreted as a ratio between the local velocity and speed of sound. The speed of sound of the medium (a) is dependant on the temperature and constants related to the medium properties. The Mach number allows for dimensional analysis and is a parameter used for determining the properties of a shock waves. Shock waves are developed in supersonic flow where compression waves coalesce, forming a thin layer that defines flow property discontinuities [1]. Compression waves are formed when supersonic flow encounters a concave surface, whereas isentropic expansion waves are formed when supersonic flow follows a convex surface.

Flow separation is a phenomenon where flow travelling over the body detaches from the surface of the body due to viscous and geometric effects. The body shape and attitude plays a role in defining the flow separation characteristics and the extent of turbulence caused by flow separation. A parameter that describes viscous and turbulent flow regimes is the dimensionless Reynolds number [2], which is defined as:

$$Re = \frac{\rho V d}{\mu} \quad (1.2)$$

Reynolds number (Re) is dependent on the medium density, viscosity, velocity, and characteristic length of the body. It represents a dimensionless value that allows one to perform dimensional analysis. This enables researchers to adapt research findings using geometric and dynamic similarity to translate to application purposes.

Flow separation can occur as a result of immediate geometric properties such as a sharp turn or gradual changes where the aspect ratio plays a role. Flow separation may exist in either

laminar or turbulent flow conditions at localised areas on the surface of a body [3]. Laminar flow separation is seen in figure 1.1, where the velocity profile along the surface of the body can describe localised flow conditions. Flow separation can be located where the zero-velocity line does not coincide with the surface of the body or if there is a sudden decrease of flow velocity in the flow field near the surface. The point where flow separation is initiated is known as the separation point.

The significance of the flow reattachment point, shown in figure 1.1, indicates the formation of a separation bubble. Separation bubbles display properties of adverse pressure gradients and a flow circulation zone. The cause of flow to reattach onto the surface is linked to the Reynolds number where turbulent flow assists in the process of flow reattachment. Turbulent flow conditions are able to delay flow separation over a body as the momentum transfer caused by turbulent eddy-motions enable sufficient kinetic energy for the flow to reattach onto the surface [4].

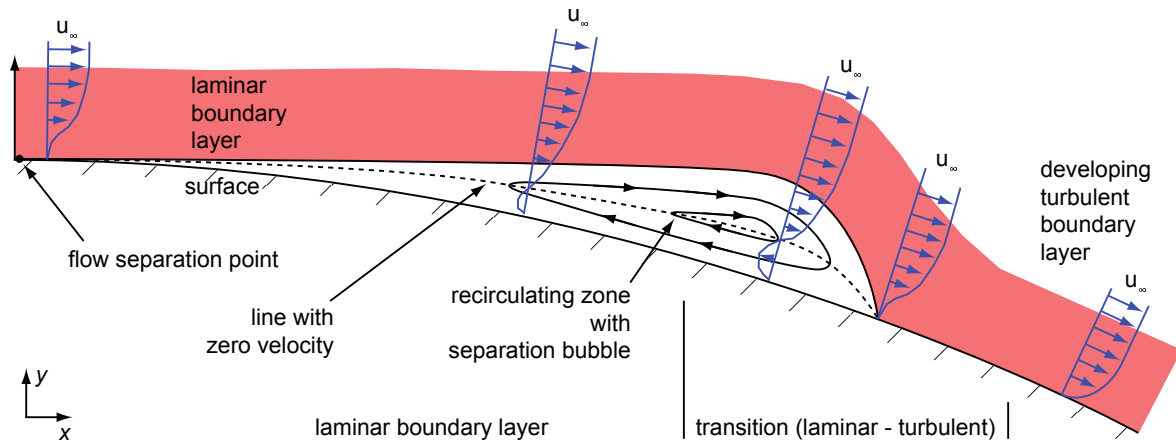


Figure 1.1: Flow over a 2-D surface representing characteristics of flow separation, flow reattachment, and formation of boundary layers. [3]

1.3 Research Motivation

Extensive research has been done in the field of supersonic flow that varies from shock wave studies to boundary layer interactions. Flow separation over specific geometries is well established, but further development of methods to visualize flow separation is proceeding. To further understand the characteristics of flow separation that is formed over a body, it would be relevant to develop computational models that visualize the interactions between the medium and body.

Computational results have been developed recently by Hooseria whom analysed curved shock wave and boundary layer interactions over slender bodies [5], where a representation of a three-dimensional numerical solutions is seen in figure 1.2. This research adapts various

computational methods that can employ similar results that focuses on flow separation at the shoulder of the proposed body.

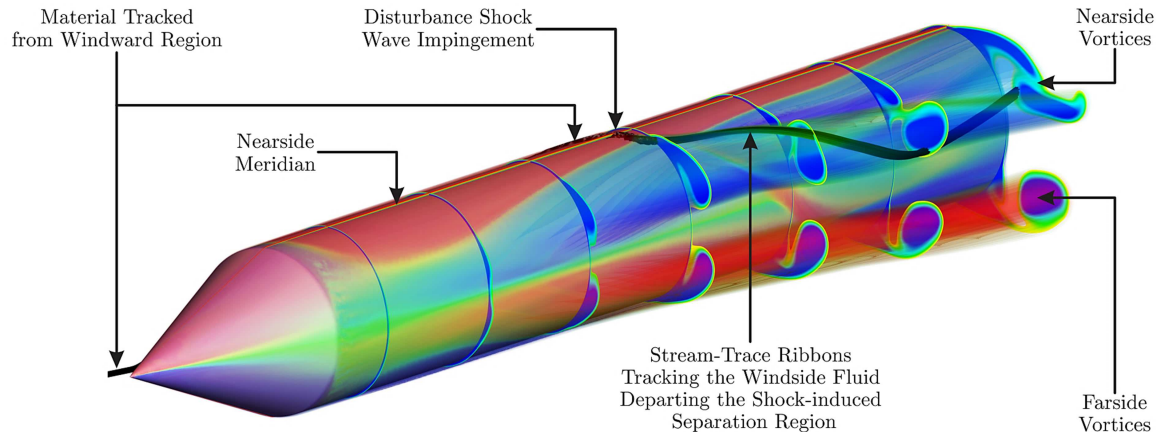


Figure 1.2: 3-D representation of a cone cylinder with separated flow structure. [5]

1.4 Problem Statement

The characteristics and properties of the flow separation over the shoulder of a cone-cylinder and hemisphere-cylinder will be analysed in a supersonic regime. The possibility of the flow separation formations are explored with independent variables such as freestream flow Mach number and angle of incidence of the cone-cylinder and hemisphere-cylinder.

2 Research Objectives

The main objective of this research is to analyse the characteristics and properties of supersonic flow separation over the shoulder of a cone-cylinder and hemisphere-cylinder. In order to satisfy the main objective, the following sub-objectives are considered:

- To validate the numerical solutions of flow separation over the shoulder of a cone-cylinder and hemisphere-cylinder
- To determine the relationship of the characteristics and properties of the flow separation with changes in the free stream Mach number and geometric properties of the body
- To explore different computational flow visualization techniques that model flow separation over the shoulder of a cone-cylinder and hemisphere-cylinder.

Parameters that fall outside of the scope are briefly discussed and not elaborated, however such parameters that lead to further interesting topics are included under Appendix which may contribute to recommendations for further research.

3 Literature Review

3.1 Compressible and Supersonic Flow

Supersonic flows are considered a category of compressible flow that is classified by the Mach number of the local flow field. The basic governing equations for compressible flow are related to conservation laws relating to flow dynamics and thermodynamics. The formation of compression waves is based on the compressive nature of the medium, which is gas for this research. The compressibility of air is related to the particles of air where the density properties are one of the key parameters to consider.

Shock waves are formed from coalescing compression waves generated by means of the flow velocity and the surface geometry of a body. Normal shocks are basic types of shock waves that propagates in a medium by a sudden change in pressure or temperature. The equations for normal shocks are derived by Anderson [6], where fluid dynamic and thermodynamic laws were used. Fluid dynamic and thermodynamic laws provide an understanding of the effects of temperature, pressure, density and Mach across shock waves. Although normal shock waves are not encountered in this research, they form the foundation of supersonic flow fundamentals which allows one to understand the flow characteristics and properties of shock waves. Normal shock waves are considered as one-dimensional (1D) shock wave systems, where the vertical axis is assumed to have infinite length. It is relevant to consider supersonic flow in two-dimensional (2D) which introduces oblique shock waves and expansion waves.

3.1.1 Oblique Shock Waves

Basic oblique shock waves can be considered for concave corner such as wedges of infinite span. When flow travels horizontally to the concave corner, the angled geometry of the wedge provides flow compression. At supersonic speeds, oblique shock waves are formed relative to the wedge angle hence flow downstream the shock wave travels parallel to the wedge surface. The derivation and equation to determine the shock wave angles associated with oblique shock waves is presented by Anderson [6]. The derived equation considers the relationship between the shock wave angle (β), wedge deflection angle (θ), and Mach number (M) which is plotted onto a graph, as seen in figure 3.1.

Figure 3.1 contains information regarding the nature of shock wave in the 2D domain which is also relevant to 3D supersonic flow. The plot indicates maximum θ angles corresponding to each Mach number. When the θ angle is exceeded for a Mach number curve i.e. for $M = 5$ the θ angle exceeds 41° , the shock waves become a curved or bow shock wave offset from the tip of the wedge. An increase in the Mach number allows an increased wedge deflection

angle threshold to maintain an oblique shock wave solution. The presence of two β angle solutions for one θ angle is an indication of a weak and strong shock solution. Weak shock solutions are commonly encountered, whereas strong shock solutions are rarely encountered in 2D shock wave systems as the weak shock waves generally prevail. However, the purpose of strong shock solutions are related to shock wave over blunt bodies. Weak shock waves are associated with high Mach numbers that are transonic or supersonic downstream of the shock wave, whereas strong shock waves are often subsonic downstream of the shock wave.

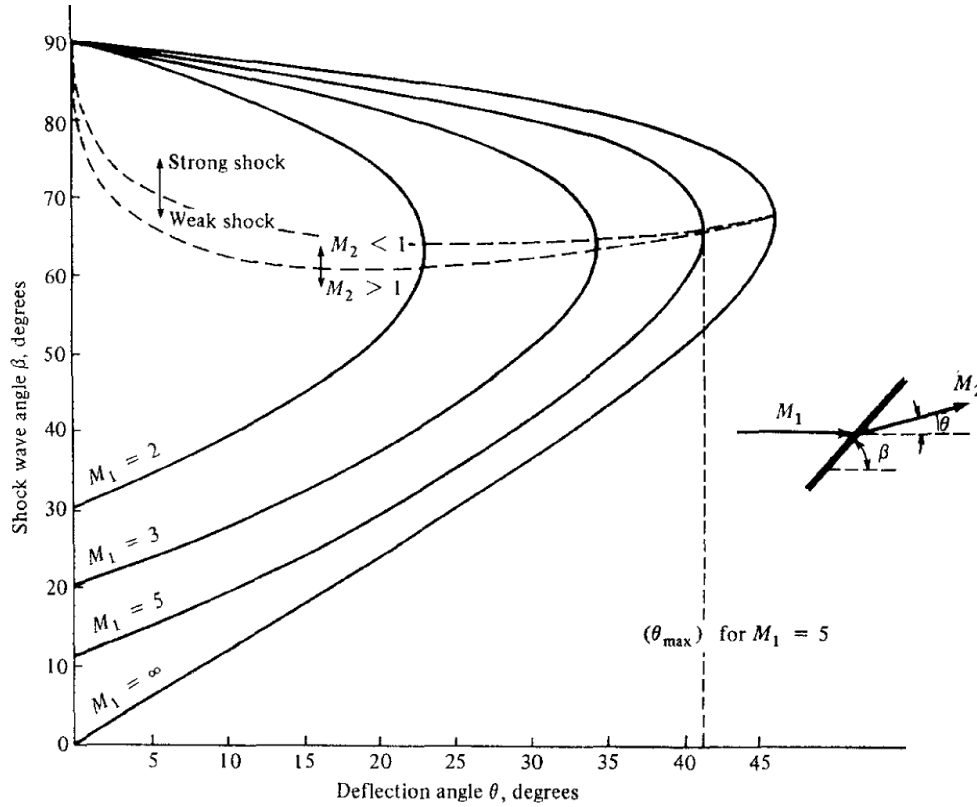


Figure 3.1: $\theta - \beta - M$ plot for 2D supersonic flow over 2D wedges. [7]

3.1.2 Expansion Waves

Expansion waves are another phenomenon that occurs in supersonic flow where the flow encounters a convex corner i.e wall turning downwards. Flow properties in the region experience the opposite effects of compression waves where the flow velocity increases and pressure decreases continuously across each successive wave. An expansion fan is formed in the expanded region where each expansion wave emanates from the vertex of the convex corner. Flows in the region are also isentropic which maintains a gradual change in flow properties across the expansion wave [6]. Note that this is counter-intuitive as the opposite occurs for subsonic flow conditions.

3.2 Flows in the Three-dimensional Domain

Modern 3D flow dynamics remain a contributing research topic that is constantly growing due to the vast amount of complex topics. Some categories of 3D flow dynamics (such as 1D flow, quasi 2D etc.) are based on simplified versions of 3D flow which allows for simplified versions of flow dynamic equations.

3.2.1 Conical Flows

Conical flows are present for cone-shaped bodies at zero angles of incidence or angles of attack. The flow is axisymmetric when the angle of attack is zero and consequently the flow equations can be simplified. The nature of the cone geometry enables equations defining the flow dynamics around the cone to be dependent only on the cone deflection angle. Cones with the same deflection angle as infinite-span wedges have similar flow features, however the supersonic speeds downstream of the shock wave are much greater in magnitude. It can be deduced that the shock wave strength over the cone is weaker in comparison to the shock wave over an infinite-span wedge. The flow trajectory downstream of the shock does not travel parallel to the surface of the cone. The plot for cone geometries are also similar to figure 3.1, where asymptotic curves and two shock solutions are present for a single deflection angle. Figure 3.2 indicates the differences in the $\theta - \beta$ relationship for the 2D wedge and cone of the same deflection angle. It can be deduced that the cone, which encounters 3D effects, allows air to flow laterally, causing less air to accumulate and compress at the nose of the body. The wedge, considered to have infinite span, restricts air from flowing laterally which forces the flow to move in a single plane [7].

Figure 3.3 indicates the flow trajectory downstream of the shock wave for a cone at 0° angle of attack. The flow trajectory downstream of the shock wave is curved and no longer maintains its parallel trajectory with respect to the surface. The flow trajectory eventually becomes parallel with the surface of the cone downstream where flow is subsonic. Another feature for cones is the presence of Mach or sonic line which defines the boundary of supersonic and subsonic flow downstream of the shock wave. The Mach line allows flow properties to change isentropically and continuously.

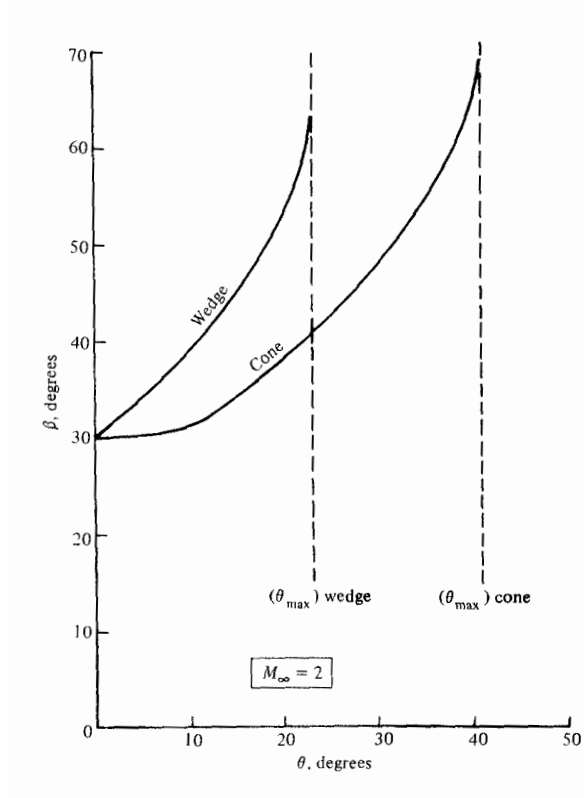


Figure 3.2: $\theta - \beta$ plot comparison for weak shock solution at Mach = 2 for a 2D wedge and cone. [7]

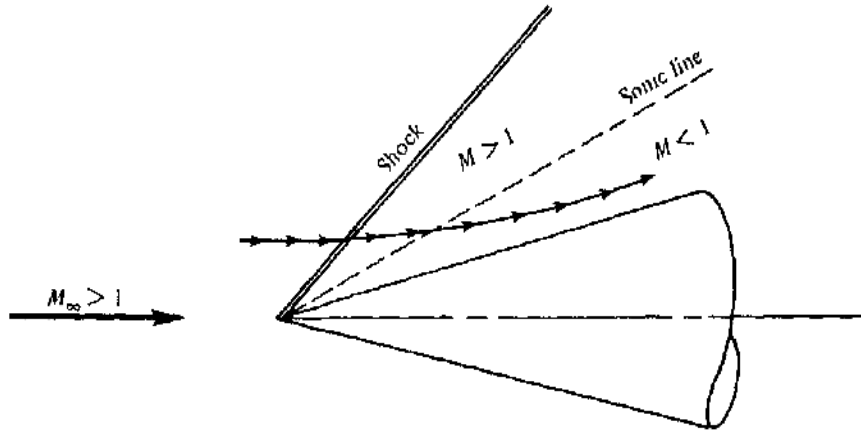


Figure 3.3: Supersonic flow trajectory over a cone at 0° angle of attack. [7]

3.2.2 Flows over blunt bodies

Flows over blunt bodies are common in practical applications. Flow features that are present for blunt bodies are stagnation pressure regions located at the front of the nose that lead to an increase in the aerodynamic loads on the body. In supersonic conditions, blunt bodies develop

strong shock waves at the nose at 0° angle of attack. Curved bow shock waves are developed at the nose which is often seen experimentally for a finite three-dimensional body where it consists of both strong and weak shock wave solutions. The two-solution shock wave for flows over blunt bodies is similar to two-solution shock waves seen in $\theta - \beta$ graphs for cones and two-dimensional wedges. Two distinct flow regions are established downstream of the shock wave near the nose tip. The regions are distinguished as either subsonic or supersonic regions which are separated by a Mach line. Figure 3.4 indicates the shock wave over a blunt body where the weak shock solution corresponds to the curved shock wave region with a lower averaged angle gradient (line $c' - d - e$), where supersonic flow conditions are experienced downstream of the weak shock wave. The strong shock solution (line $a - b - c'$) corresponds to the region near the nose tip where subsonic flow conditions are present downstream of the shock wave. The presence of a blunt nose results in an accumulation of air at the nose which causes a region of stagnation pressure. The stagnation pressure region is characterised by high density regions where the maximum flow density is present.

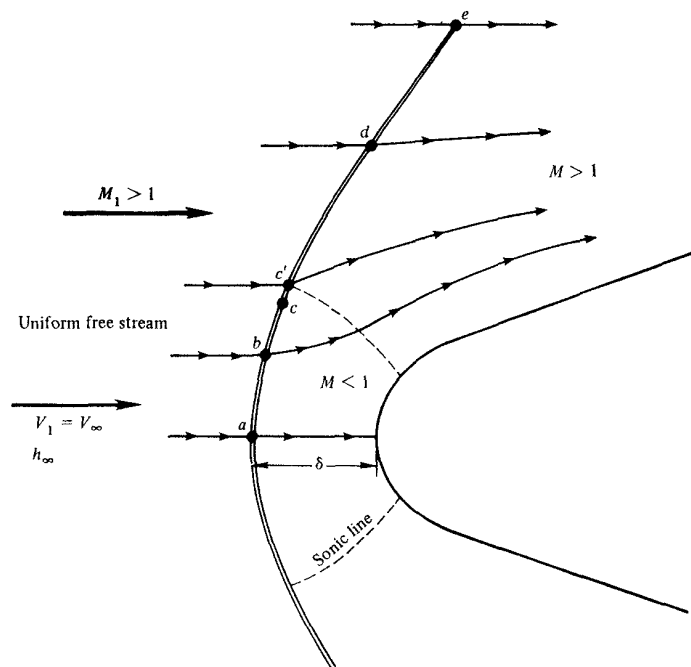


Figure 3.4: Schematic diagram of supersonic flow over a blunt-nose body. [7]

3.2.3 Slender Bodies at Incidence

The complexities of 3D flow domains and parameters requires one to model curved shock waves and 3D flow separations in the form of irregular-shaped surfaces. Such models are required for bodies at an angle of attack where the flow field is limited to only one plane of symmetry, or in some cases no planes of symmetry, for the simplification of flow dynamic equations. A cone at an angle of attack has shock wave angles that are different at each

circumferential or meridional plane of the cone cross section. Supersonic flow over cone bodies encounters similar flow dynamics to blunt-nose bodies when they are at an angle of attack. A common feature for both body types is the entropy gradient at various locations of the shock wave in the flow field due to multiple shock wave angles present [7]. The presence of an entropy gradient is an indication of rotational flow which causes curved streamlines over the surface of bodies.

Flow characteristics adjacent to the surface is affected by the deflection angle (θ_c) and angle of attack (α) of the cone. For cases where $\theta_c > \alpha$, a vortical singularity is present along the vertical symmetry plane on the leeward side of the cone-body. Vortical singularity is a location where flow streamlines converge and form a region with multiple entropy values. For cases where $\theta_c < \alpha$, the vortical singularity detaches from the body and is formed above the leeward side of the cone-body. Flow conditions with vortical singularities may encounter another phenomenon where a weaker shock is formed downstream of the vortical singularity point. A weak imbedded shock wave is formed due to expansion waves present on the leeward side of the cone-body. The features of flow dynamics over cone-bodies are seen in figure 3.5, where additional flow separation features are included in the flow field of the cross-section of the cone-body. Although not shown in the figure, vortex shock waves may form due to severe cross flow conditions that originates from the windward side of the body. Stagnation points and embedded shock waves are features that may result in the formation of cross-flow separation [8].

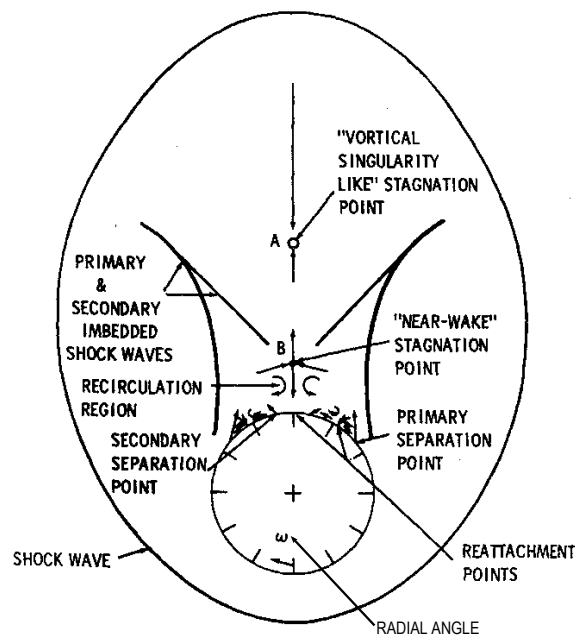


Figure 3.5: Descriptions of the flow field around the cross-section of a cone at a high angle of attack. [8]

3.3 Flow Separation and Topology

The background section on flow separation provided basic understandings of flow separation in the 2D domain but this research requires interpretation of flow separation in the 3D domain. The flow complexity is further exacerbated by the body attitude which renders several simplifying assumptions invalid.

The formation of flow separation can be classified as an open or closed separation region which is dependent on flow trajectory and presence, or absence, of flow reattachment downstream of the separation point. These types of flow separation appear on the leeward side of a symmetrical body at relatively high angles of incidence. Closed flow separation regions are more likely to form from turbulent flow conditions downstream of the flow separation point, where the flow is supplied with sufficient energy to overcome the separated boundary layer. Hsieh [9] investigated flow separation over hemisphere-cylinders at incidence within a Mach number range of 0.6 to 1.5. Results, as seen in figure 3.6, indicated the formation of two flow separation regions over leeward side of the body. The nose separation bubble is classified as the closed separation region and the cross flow separation region is classified as open flow separation region. An additional characteristic of the open flow separation region is the secondary separation lines which are indications of vortex formations in the flow field above surface.

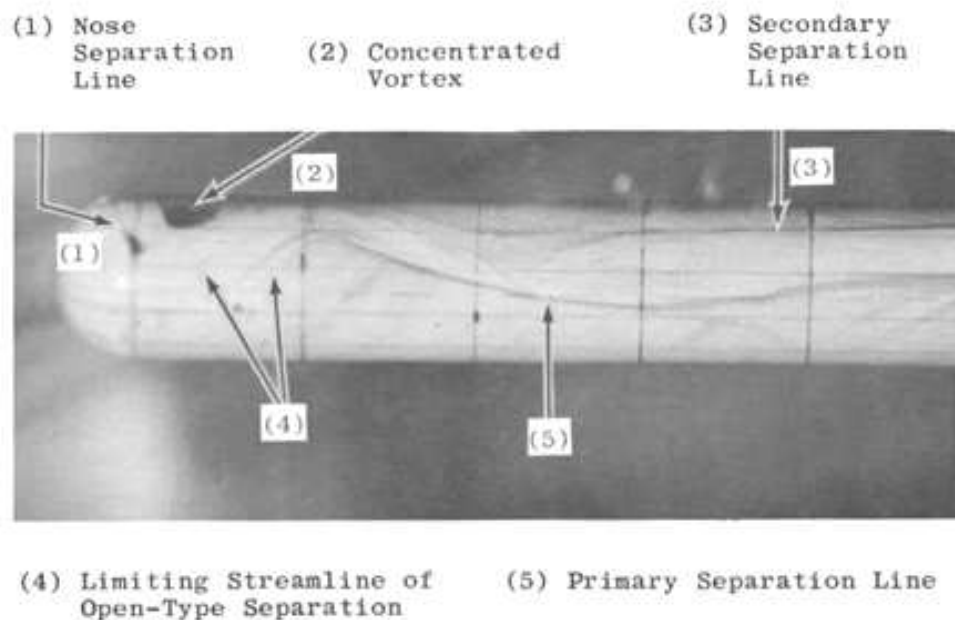


Figure 3.6: Experimental image of flow separation over a hemisphere-cylinder at 19° angle of incidence and Mach 1.2. [9]

Le Clainche et al. studied flow separation characteristics, using flow topology techniques, for flow at low Reynolds numbers [10]. Tobak and Peake [11] deduced that flow topology

at low Reynolds number displayed similar characteristics to flow with high Reynolds number at transonic and low supersonic regimes. Although high Mach numbers are considered for this research, the classifications of flow topology features considered by Le Clainche et al. may prove useful when identifying the features of flow separation. Numerical simulations for hemisphere-cylinders at different angles of incidence and low Reynolds number with incompressible flow were compiled. In figure 3.7, N represents nodes of separation and reattachment, S are saddle points, and F are focus points of the separation bubble.

Saddle points often exist with converging skin friction lines where lines of separation emanate from this point. Flow separation lines do not only have skin friction lines emerging from the saddle point but includes streamlines that converge towards the boundary on either sides. The flow conditions around the saddle point enable one to determine the type of flow separation region present, where a closed separation region consists only of skin friction lines emerging from the saddle point. The existence of a saddle point is a necessary condition for flow separation [12]. The focus point, or spiral node, is only formed in the presence of a saddle point. The dominant skin friction lines that converge towards the focus point are directed by the saddle point where it has either diverged from the saddle point or is converging towards the line connected to the saddle point and focus point. Reattachment points are of similar characteristics to the saddle point, which consists of both converging and diverging skin friction lines. Skin friction lines that are connected by nodal points are referred to as global separation lines which define the boundary of flow separation and may form a surface boundary in 3D flow. Skin friction lines or streamlines converging to a separation line but are not connected to a nodal point are defined as local separation lines. The description of ‘global’ refers to the effect of the separation lines on the flow streamlines. Global separation lines distinctly form a flow boundary between two nodal points of separation hence causing multiple streamlines (on a ‘global’ scale) to be diverted. Local separation lines generally detach from the surface of the body and become a flow field streamline that potentially contributes to the formation of vortices. The definitions of the points are derived from Critical Point Theory and form the backbone of flow separation theory for 3D flow characteristics [13]. It was deduced by Détery [13] that 3D flow separation surfaces were more difficult to predict for blunt nose slender bodies, such as the hemisphere-cylinder, in comparison to sharp-nosed slender bodies where separation lines often emanate from the vertex point of the nose.

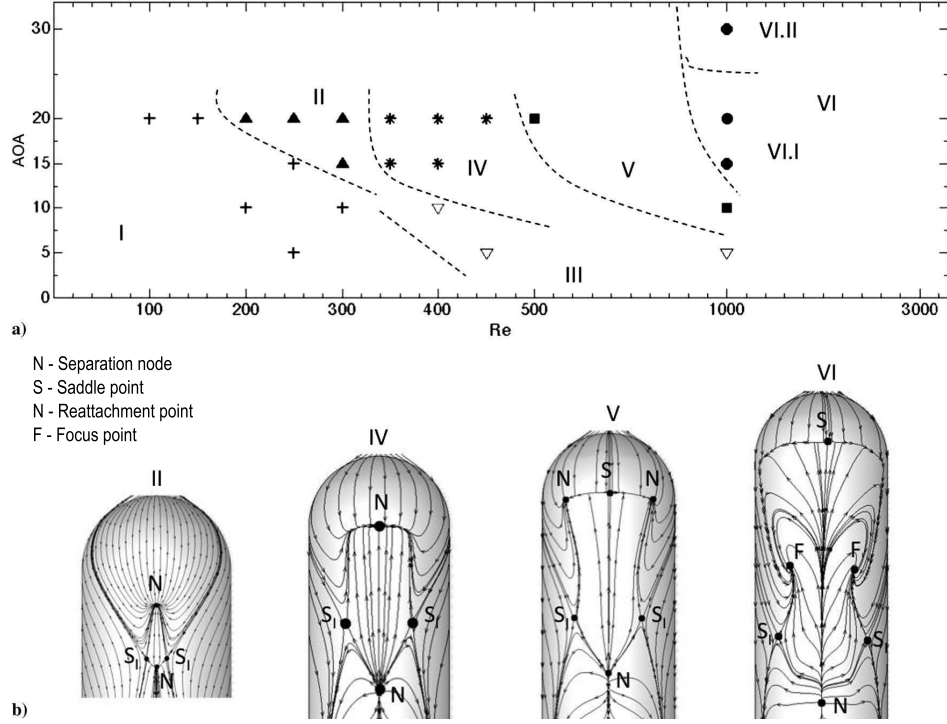


Figure 3.7: Surface flow topology of hemisphere cylinder as: (a) graph indicating the angle of attack of the body with different Reynolds numbers , (b) visual representation of surface streamline patterns over the body. Note: Labels for each flow condition in (b) correspond to the Reynolds number labels in the graph in A. Several flow conditions were excluded as it was redundant. [10]

The significance of the focus points at condition VI, from figure 3.7, represents features of horn vortices. The formation of horn vortices was introduced by Tobak and Peake [12] where vortices were developed as a result of vortex filaments formed from the surface of the nose. Horn vortices can be described as vortices formed on the surface of a body with shape resembling the tip of a horn when considering the outer lines of separation connecting the focus points and saddle point near the nose. The formation of vortex filaments further affect the cross flows dynamics hence leading to a more complex flow pattern downstream of the body. Cross flows over the surface of a body are common for bodies at high angles of attack.

3.4 Influence of Reynolds Number

As defined in previous sections, Reynolds numbers are ratios of inertial to viscous forces and are helpful in defining flow characteristics on the surface. Figure 3.8 indicates the range of Reynolds number for smooth cylinders at various angles of incidence. The regions include flow regimes such as fully turbulent, laminar, and transitional. Although the plot is only valid for low-turbulence streams [14], it can provide an estimation for Reynolds numbers that can maintain turbulent flow separation or laminar flow separation.

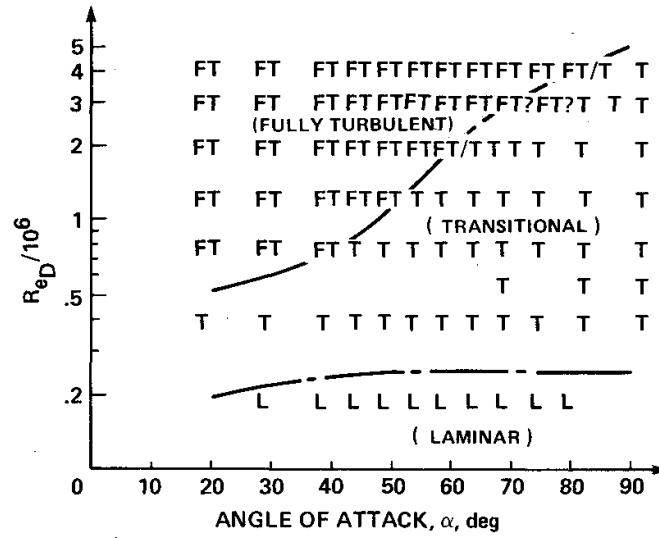


Figure 3.8: Plot of Reynolds number, with respect to the cross-sectional diameter of the body, and angle of attack representing different types of flow separation regimes. [14]

Investigation of subsonic flow separation over hemisphere cylinders at incidence for high Reynolds numbers were completed by Hoang et al. [15]. It was determined that at low angles of incidence, separation bubbles form a ring around the body at the shoulder and become more isolated with an increase in incidence. As Reynolds numbers increased the separation bubble becomes much wider. The pressure distribution over the body was recorded and is illustrated in Figure 3.9. The pressure trends at different angular locations around the circumference of the body is a useful method for determining the properties of separation.

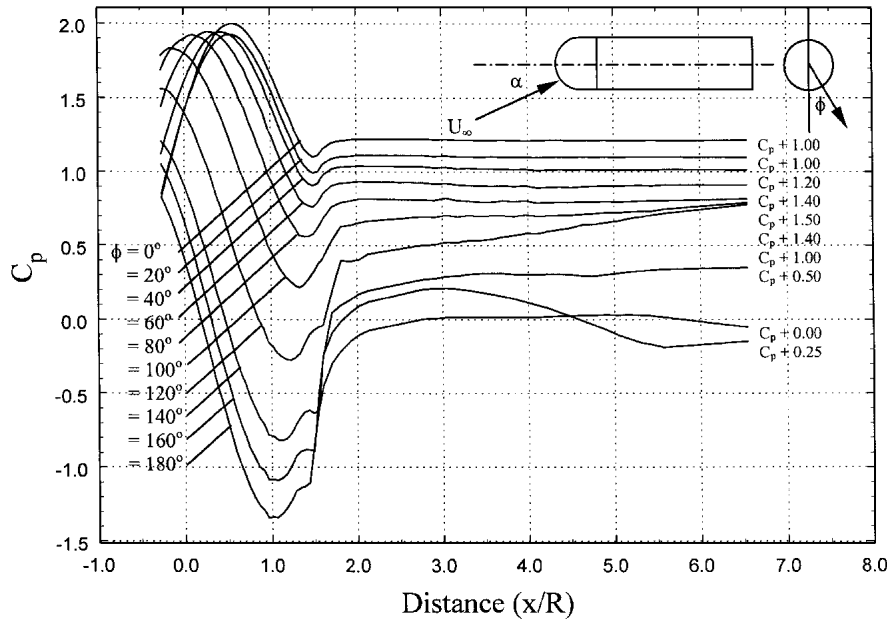


Figure 3.9: Pressure distribution over a hemisphere-cylinder at $\alpha = 20^\circ$ and $Re = 4.2 \times 10^5$. [15]

It is evident that the topic of supersonic flow separation over cone-cylinders and hemisphere-cylinders requires additional research, especially with the current computational tools. Although flow properties over hemisphere-cylinders at low supersonic speeds have been investigated, emphasis on flow separation at much higher supersonic speeds have not yet been focused.

3.5 Flow Visualisation

The presence of shock waves and flow separation requires the use of flow visualization techniques that can capture the characteristics of the flow in a medium. The density gradients in supersonic flow conditions allow one to utilise the medium properties, hence it is possible to analyse the flow using shadowgraph and schlieren photography. The flow can further be analysed using computational methods where the properties can be manipulated to effectively model and visualize the characteristics of flow.

3.5.1 Experimental Flow Visualization

Surface oil flow visualisation is an effective method to determine the surface flow topology experimentally. Hooseria [16] used several mixtures that displayed three-dimensional flow patterns on the surface of slender bodies. The results included surface oil visualisation techniques in colour which showed results that were more distinct and clear than titanium dioxide oil streaks. The comparisons between monochrome and coloured surface oil flow visualisation can be seen in figure 3.10.

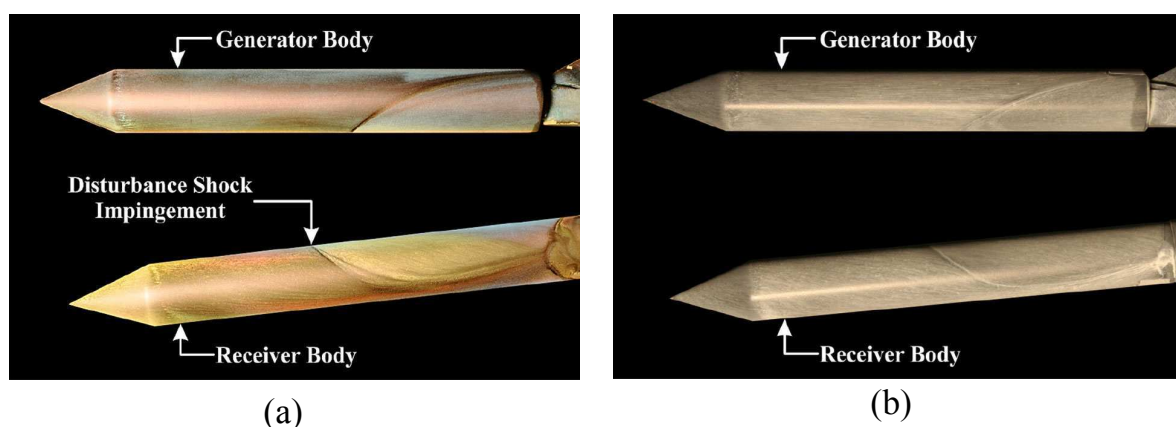


Figure 3.10: Surface patterns over the body represented with: (a) powdered coloured paint mixture (b) titanium-dioxide mixture. [16]

Pressure-sensitive paint was utilised by Boswell [17] to measure surface pressures at Mach 2.46 of a cylindrical afterbody at incidence. Although not as accurate as pressure tappings,

it allows the use of a low-cost method to determine the pressure properties that the body will experience. Boswell also used Mie scattering visual techniques to capture turbulent areas in order to determine flow separation.

3.5.2 CFD Flow Visualization

Computational Fluid Dynamics (CFD) can yield effective and efficient results when studying the characteristics of fluid flow fields. The complexity of three-dimensional flow can be difficult to understand, hence it is necessary to explore various methods to represent 3D flow dynamics. A great amount of effort has been invested into the investigation of surface-based flow visualization by Edmund et al. [18]. Geometric and texture-based techniques of modelling the surface flow properties were several categories that were explored from a review of research papers that focused on 3D flow visualization. Despite the development of various 3D methods of flow visualization modelling, most surface-based flow visualizations were considered a novelty. Hence there is limited number of computational tool options that can effectively model 3D flow solutions as 3D graphic representations of the flow. The 3D graphic representations also require care when the 3D features are constructed as excess information and misuse of 3D modelling tools may reduce the clarity of the results e.g. volume rendering combined with 3D streamlines may be difficult to interpret.

4 Experimental Method

4.1 Experimental Facilities and Apparatus

The Supersonic Wind Tunnel (SSWT) located at the University of Witwatersrand in the North West Engineering Laboratory building on the main campus was used for the experimental tests. A mount for the test pieces was designed to accommodate the dimensions of the test chamber in the SSWT. The SSWT consists of several components that enabled the wind tunnel to operate within a specific range of Mach numbers.

4.1.1 Supersonic Wind Tunnel (SSWT) Facility

The SSWT facility consists of the pressure tank, air dryer unit, SSWT pressure tank, test section, and the diffuser.

Pressure Storage

The pressure tank is pressurized by a reciprocating air compressor and functions as the pressure source for the settling chamber of the SSWT. Pressurized air flows through the air dryer unit, prior to the pressure tank, to ensure that the air contains minimal moisture which may potentially affect the results. A schematic view of the pressure tank and air dryer unit can be seen in figure 4.1. The air dryer unit consists of a heat controller and crystal silica storage. Air is able to pass through the crystal silica storage where the silica crystals absorb the moisture from the air. The silica crystals are dried by a heating element which is automatically adjusted by the heat controller unit. The silica crystals are dried to prevent saturation from constant moisture absorption. The two bleed valve levers are situated below and above the crystal silica storage, and function as a pressure-relief valve for the air dryer unit. The pressure control valve attached to the pressure tank leads to the settling chamber of the SSWT and is controlled by the user from a pressure control box in the SSWT room. The pressure control box adjusts the required stagnation pressure.

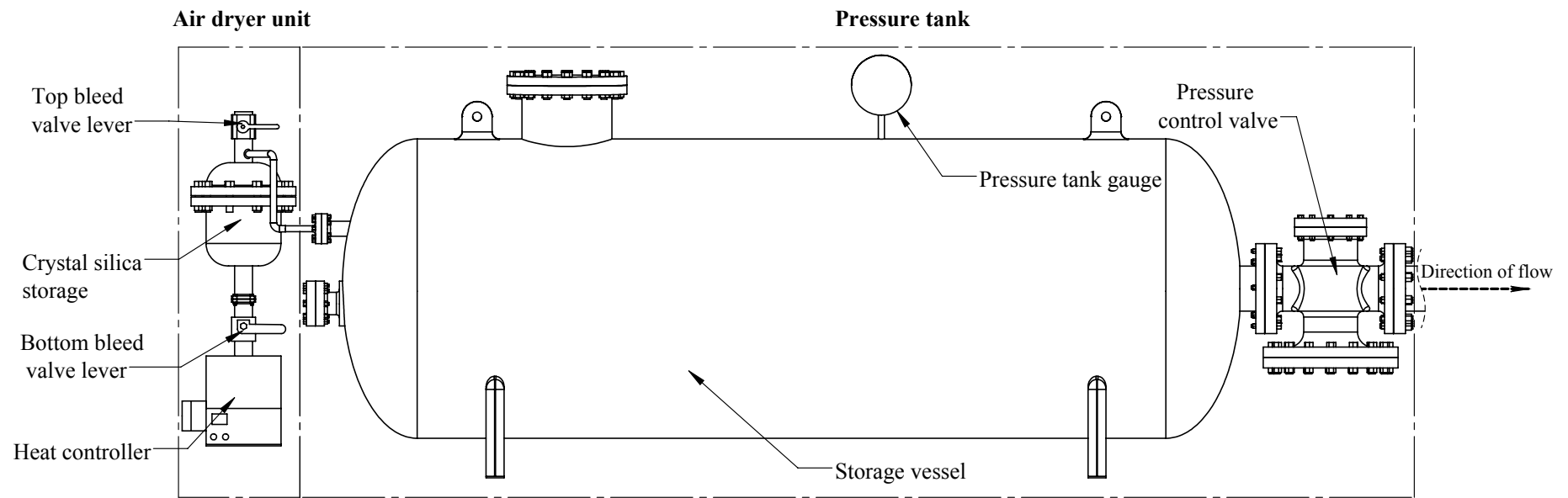


Figure 4.1: Schematic view of the pressure tank area of the SSWT.

SSWT Test Section and Control Room

A schematic view of the SSWT can be seen in figure 4.2. The SSWT pressure tank consists of the expansion nozzle and settling chamber. The settling chamber stores air at the selected stagnation pressure required for the desired Mach number. The pressure gauge above the settling chamber, indicating the stagnation pressure in the settling chamber, consists of a pressure transducer which measures and sends readings to the data acquisition system during the tunnel blow down. The data acquisition system information is accessed in the SSWT software on the controller PC in the SSWT room.

The test section portion of the SSWT consists of the adjustable first throat, test chamber, and sliding block. The first throat, which is a component of the converging-diverging nozzle, is adjusted by the lower block of the throat and can translate horizontally (refer to cut-out section from figure 4.2). This enables the wind tunnel to operate within the Mach number range of 1.5 to 3.5. Translating the bottom block towards the settling test chamber decreases the throat area hence increasing the freestream Mach number, whereas translating the block towards the chamber increases the throat area hence decreasing the Mach number. The position of the bottom block is adjusted using the wheel handle equipped with a veeder counter located below the opening of the test chamber. The veeder counter represents a linear distance that requires calibration to determine the counter value with the corresponding Mach number. The calibration graph for the veeder counter and the Mach number is seen in figure 4.2 under Appendix A.1. The calibration curve was configured in the SSWT software that automatically calculates the required veeder counter number for the selected freestream Mach number. A stagnation pressure was also recommended by the software program, however the calibration curve was completed with a constant stagnation pressure of 250 kPa.

The test chamber has a square cross-section with dimensions of 4×4 inch. A pressure gauge and transducer are located above the test chamber to measure the pressure during a blow-down. A signal is sent to the data acquisition system to record the pressure in test chamber. Two bolt holes are located above the test chamber at the interface between the sliding block and the test chamber which functions as a mounting holes for the mounting block. The test chamber window is made from glass that accommodates for optics flow visualization. The sliding block translates horizontally and is fastened by the wing bolt and nut. The wing bolt is hinged on a bracket fixed to the external frame of the test chamber.

The second throat of the SSWT is located downstream of the sliding block. The second throat, which is a converging-diverging nozzle, returns the flow from supersonic to subsonic conditions before it is routed to the silencer and diffuser unit. The flow is vented through a large duct routed to the roof of the laboratory.

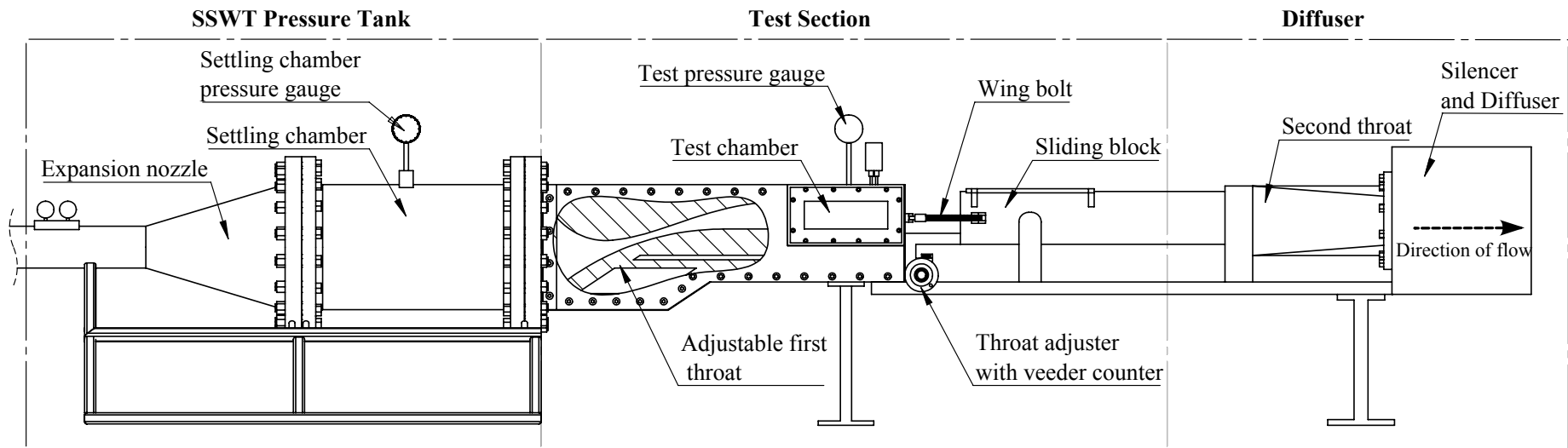


Figure 4.2: Schematic view of the test section area of the SSWT.

4.1.2 Mounting block and Test Pieces

The mounting rig and test pieces are designed to accommodate for the test chamber of the wind tunnel which has a cross-sectional area of $101.6 \text{ mm} \times 101.6 \text{ mm}$. The mounting block consists of two side plates, a bottom plate, and a milled block of aluminium with slots that accommodates bolts to fasten the mounting block to the test chamber of the SSWT. Horizontal plates are mounted to the side plates to accommodate sting rods which extend the test piece towards the centre of the test chamber window. A separator plate is attached at the midpoint of each sting rod to stiffen the sting rod assembly. Bolt holes are located at the ends of each sting rod to attach the test piece and adjustable strut. The design of the mounting rig was based on stress analysis of the test piece and assembly under the maximum expected loads on the structure, which can be seen in Appendices B.1 and B.2. The assembly and drawings associated with the design of the mounting block and test pieces can be seen in Appendix B.3.

The assembly accommodates for four different angles of incidence, which are 0° , 10° , 20° , and 30° , for the hemisphere-cylinder and cone-cylinder. The angles of incidence were adjusted by attaching the strut to different bolt holes located on the top sting rod (refer to the assembly drawing in Appendix B.3). The diameter of the test pieces was 16 mm where the dimensions of the test pieces can be seen in figure 4.3. The selected Mach numbers for the experiment are $M = 2.8$ and $M = 3.2$, both of which enable fully turbulent flow conditions over the test pieces.

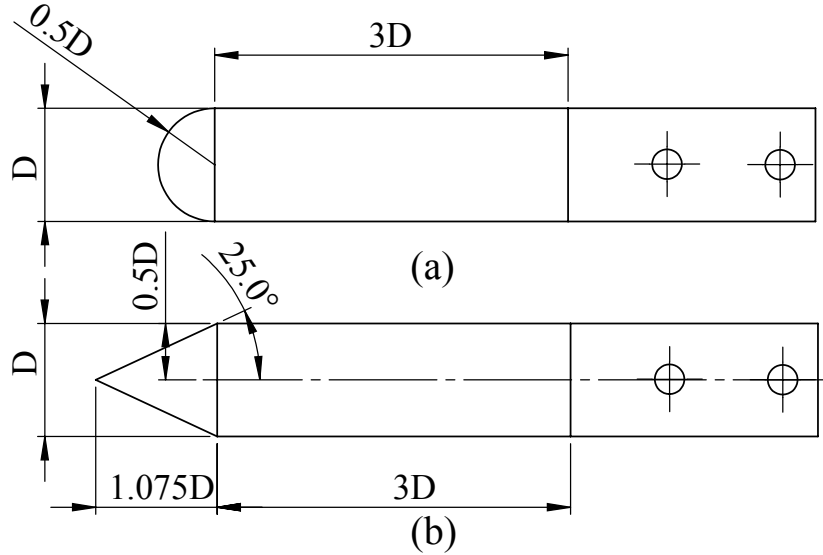


Figure 4.3: Schematic view with normalized dimensions relative to the diameter for: (a) hemisphere-cylinder, (b) cone-cylinder.

4.1.3 Shadowgraph and Schlieren Imaging

The experimental data were recorded visually using schlieren imaging and shadowgraphing techniques. The optics set-up for schlieren imaging can be seen in figure 4.4, where the set-up for shadowgraphing excludes the use of the knife edge near the HD camera. Vertical and horizontal knife edge configurations are tested to determine the most favourable set of results for validity comparisons of the experimental and computational methods.

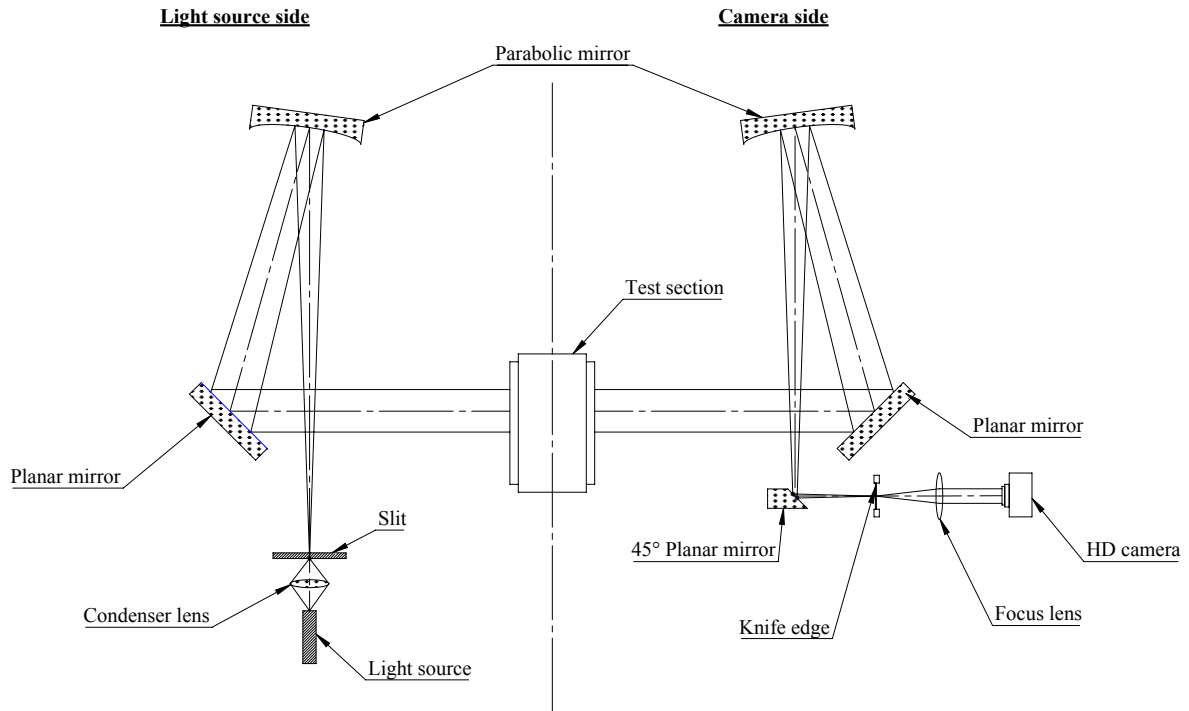


Figure 4.4: Basic schematic view of the optics setup for the schlieren imaging.

The results gathered using optics flow visualization are limited to supersonic flow features, such as the shock waves, and any other features that have sufficient density gradients to alter the light path travelling towards the camera. Optics flow visualization does not indicate areas of flow separation around the body. However, the experimental results are utilized to obtain an estimate of the accuracy of the supersonic flow conditions as these potentially influence the flow separating characteristics.

4.1.4 Surface Oil Flow Visualization

Surface oil flow visualization (SOFV) is a technique where an oil mixture consisting of pigments in a carrier fluid is applied to a model prior to airflow over the body. The viscosity of the mixture interacts with the airflow over the body which create streaks and patterns on the surface of the model. The streaks and patterns that are created on the surface of the body

during or after the wind tunnel blow-down enables one to study the flow separation characteristics over the shoulder and on the body of the model. However, the current methods are limited to results that are qualitative as there is insufficient information given by the pattern and streaks to determine quantitative properties such as wall shear forces.

It is determined by Hooseria [16] that kerosene allowed the oil mixture to evaporate after the wind tunnel blow-down, which creates a surface pattern with the remaining pigments from the oil mixture. Transformer oil is another popular carrier fluid used for SOFV, but it requires images to be recorded during wind tunnel blow-down due its lack of volatility. The use of kerosene is considered more beneficial for this experiment as it allows one to observe and record the streaks with more ease after the wind tunnel blow-down. Several pigments were tested in the oil mixture to yield the most desirable result with distinct separating features to be compared with the CFD results. The pigments that were used are paint powder, chalk powder and dustless chalk powder. Different colours of each pigment were also investigated and compared for contrast. The pigments were ground against water paper with a high grit to allow for even consistency in the particle sizes of the pigments. The test pieces were sprayed black to allow better contrast between the coloured mixtures, and prevented reflections of light which may lower the clarity of the images. Obtaining the mixture with the desired ratio of kerosene and pigments required an iterative process. The mixture ratios outlined by Hooseria [16] give guidance of what ratios may be expected during the iterative process. The paper also mentioned the benefits of coloured pigments and their ability to enable streaks that are more distinct in comparison to white pigments. The comparisons of different pigments used are briefly discussed in later sections.

4.2 Experimental Procedure

The experimental procedure consists of several sets of steps. The sets of procedures are separated into the storage pressurization procedure, test procedure, and the shut-down procedure.

4.2.1 Storage Pressurization Procedure

The pre-test procedure refers to instructions for the pressurization of the pressure tank. The steps of this procedure are specific to a water-cooled reciprocating compressor and pressure vessel. The procedure are as follows:

1. Close top and bottom bleed valve of the air dryer unit.
2. Turn off air dryer unit heaterⁱ.

ⁱThe heater is only on for the first start-up of the compressor

3. Turn on the electric solenoid valve of the compressor.
4. Open the water supply for the compressor by turning the tap three quarters of a revolution.
5. Turn on compressor, rotate switch clockwise to turn on.

Allow the compressor to pressurize the pressure tank until its maximum threshold (approximately 14 bars of pressure) or a specific pressure required for wind tunnel testing. The air compressor will automatically change to standby mode once the pressure vessel has reached 14 bars of pressure. Once the pressure vessel has reached the desired pressure, the shut-down procedure for the compressor is required. The shut-down procedure is as follows:

1. Turn off the compressor by pressing the button down and rotating anti-clockwise to ensure its off.
2. Close water supply.
3. Turn off the electric solenoid valve of the compressor.
4. Open the top bleed valve on the air dryer unitⁱⁱ
5. Close the bottom bleed valve on the air dryer unit.

4.2.2 Test Procedure

Prior to the test procedure for the SSWT, the test chamber should be open. The test procedure for the optics flow visualization is included. It is assumed that the controller PC, pressure controller box, data acquisition system, light source and electrical supply for the siren are turned on before the steps of procedure. Once the controller PC is fully booted up, select the SSWT software program and fill in the required details of the user and research description. The steps of test procedure are as follows:

1. Fasten the mounting block and test piece to the test chamber
2. Close the test chamber and tighten the sliding block with the wing bolt and nuts.
3. Adjust the optics set-upⁱⁱⁱ until the test piece is at the desired position to record the test images.
4. In the SSWT software program, enter the storage tank pressure, desired Mach number, stagnation pressure, test number, and image number (if applicable).

ⁱⁱThe order of opening the bleed valve is important as this may cause silica crystals to be forced into the pressure vessel

ⁱⁱⁱOptics set-up refer to camera settings, camera position, and knife edge settings

5. Set the stagnation pressure of the settling chamber on pressure controller box to 250 kPa^{iv}.
6. Adjust veeder counter to the value displayed on the SSWT software program. The veeder counter is calculated based on the same calibration curve from Appendix A.1.
7. Fill in the test log form which may consists of checks for the previous steps. A sample test log form is seen in Appendix C.2.
8. Sound the siren for 5 seconds twice with a 3-second interval.
9. Turn off the room lights.
10. Select the 'START' option for the blow-down process. The camera trigger is to be pressed 1 to 3 seconds after starting the blow-down.
11. Turn on room lights
12. Save the recorded data on the software program and complete the test log form.
13. Select the 'NEW TEST' option on the software program
14. Loosen the wing bolt nuts and open the test chamber.
15. Detach the mounting block and test piece from the test chamber.
16. If the pressure tank is above 0.8 MPa, repeat steps 1-14.
17. If the pressure tank is below 0.8 MPa, switch off the optics light source and set the stagnation pressure to 0 kPa. Repeat the storage pressurization procedure. Once the pressure tank is pressurized, repeat steps 1-14.

When accommodating for surface oil flow visualization, the oil mixture is applied prior to step 1. An additional step may be required after step 15 to clean the test chamber due to the oil mixture.

4.2.3 Shut-down Procedure

The shut-down procedure for the SSWT are considered once testing has been completed for the day. The shut-down procedure is as follows:

1. Set the stagnation pressure to 0 kPa and switch off the pressure control box.
2. Close the SSWT software program and switch off the controller PC.

^{iv}The value of 250 kPa is specific to the calibration graph. The stagnation pressure may change if a different calibration graph is used.

3. Switch off all electrical equipment and instrumentation as well as the electrical power supply.
4. Check test chamber is open
5. Check compressor is switched off
6. Check bleed valves of the air dryer unit is open
7. Switch on the heater for the air dryer unit.

The steps of procedure defined in this section corresponds to the 'End of Day Checklist' on the test log form as seen in Appendix C.2.

4.3 Precautions

- Ensure that the test pieces are secured tightly and all fasteners are secure.
- Ensure that the test chamber is clear of debris and is closed prior to blow-down.
- Wear hearing protection during the SSWT blow-down and when releasing the pressure from the air dryer unit.
- Always sound the siren before the blow-down to warn persons within the laboratory.
- Switch off the light source of the optics equipment when not in use. Long periods of operating time may cause the light source to overheat and possibly resulting in the bulb exploding.
- Use a microfibre cloth or soft dust-free cloth to clean the test chamber window to avoid scratching the glass.

5 Computational Method

The computational method or computational fluid dynamics (CFD) refers to theoretical equations solved by numerical methods defined by algorithms in software programs such as ANSYS® Fluent® . The algorithms are able to solve the flow field dynamics within a domain, hence producing accurate solutions that indicate the characteristics and properties of the flow field around the defined bodies. Access to high performance computers are required for CFD simulations as the models generated for this research require significant processing power to reduce simulation time and a large amount of random-access memory (RAM) to accommodate for model file size. The process for generating models for CFD simulations are grouped as: (1) geometry with mesh settings and (2) solver settings expanded in detail in later sections. Grid independence and convergence are requirements when generating solutions from CFD simulations. These are also outlined in later sections. Figure 5.1 shows a flowchart that summarizes the process of generating CFD solutions. The feedback loop represents the iterative process of mesh and solver independence.

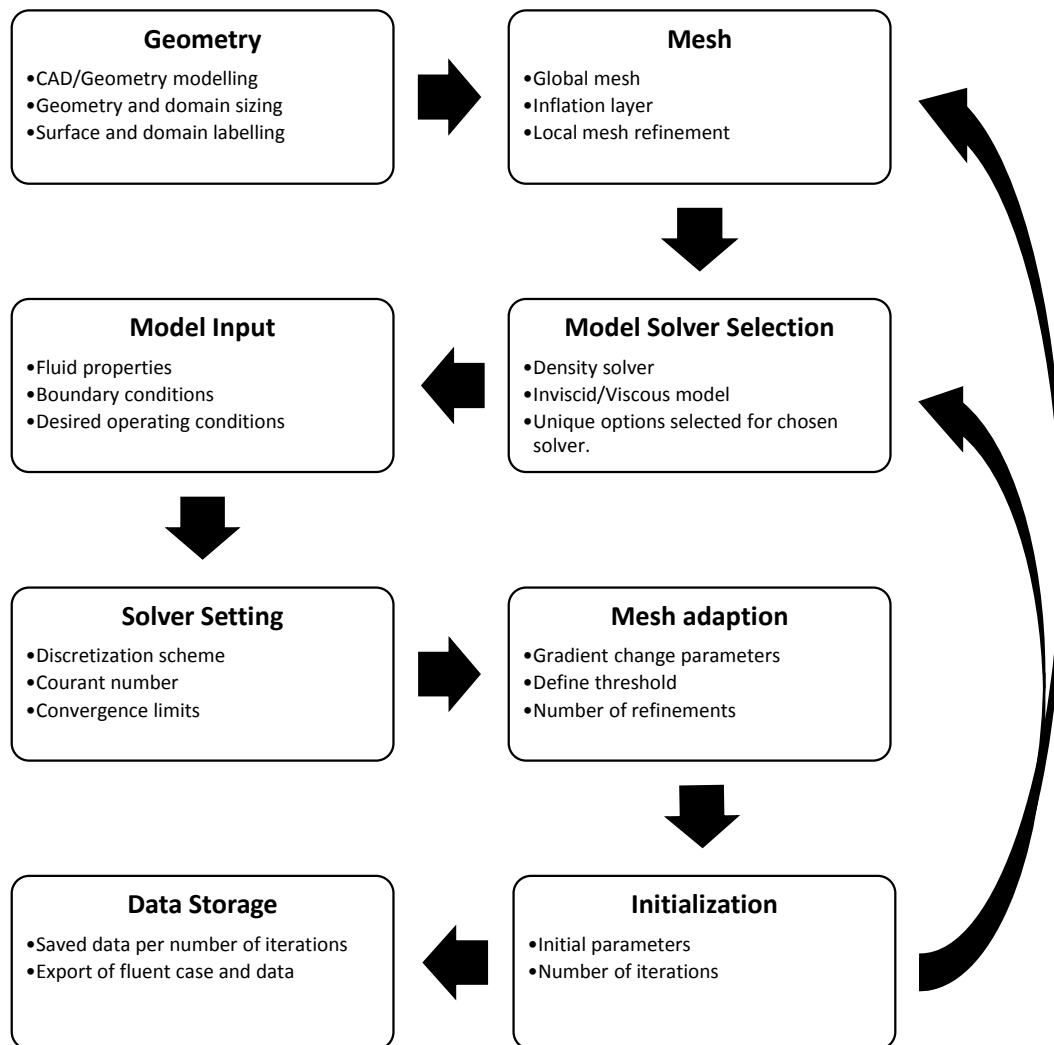


Figure 5.1: Flowchart of the process for computational methods utilizing Ansys Fluent

5.1 Geometry and Meshing

5.1.1 Geometry

The geometry for the CFD models consist of both the domain size and geometry of the body of interest for this research i.e. hemisphere-cylinder or cone-cylinder. A computer-aided design (CAD) program was used to construct the domain and the hemisphere- or cone-cylinder with dimensions similar to experimental method set-up of the test chamber. The domain has one characteristic length upstream of the nose of body, two characteristic lengths downstream of the trailing edge of the cylinder, and three characteristic lengths normal to the shoulder of the nose and cylinder. When the body is pitched at an angle of incidence, the characteristic lengths normal to the body may decrease. This only occurs at the windward side of the body where flow separation does not occur. The characteristic lengths are relatively small as the shoulder of the hemisphere- and cone-cylinder are the area of interest. The flow Mach number in the domain are expected to be high and mostly supersonic downstream of the shock wave, hence it may be acceptable to have relatively small characteristic lengths downstream of the body. Limiting the domain size is beneficial as it decreases the required simulation time to solve the CFD model. Note that the domain size is specifically used for pressure far-field boundary conditions which is describe later. This ensures that shock waves and flow does not reflect at the boundaries of the domain and affect the flow near the body. The surface of the body is modelled as a cut-out from the 3D domain where the cut-out is pitched over a range of angles of incidence between 0° to 30° . The maximum limit of 30° angle of incidence ensured that flow remained symmetric about the vertical plane, hence the domain was modelled as a half-model domain instead of a full-model domain. After the geometry of the domain was defined, selected surfaces and domain volumes were labelled as the symmetry plane, pressure far-field, wall and fluid domain. The labelled surfaces and volumes were for boundary conditions that were required to be defined when creating the model in Ansys Fluent. The geometry of the domain with the labels is seen in figure 5.2.

The meshing process for the CFD model is one of several important processes that may affect the accuracy of the solution. There are various mesh settings one may select to generate a mesh that is acceptable for Fluent solvers. The meshing process outlined in the upcoming sections was found to be more robust in comparison to the default settings provided by the Ansys mesh modelling software. Poor mesh qualities are undesirable for many solvers in the Ansys Fluent program i.e. inflation layers (high aspect ratio cell near the wall of the surface) may cause an inviscid solver in Fluent to diverge hence causing errors in the flow field. It is also important to choose the correct mesh at certain locations as it may be necessary to obtain the flow characteristics in certain areas of the flow field i.e. inflation layers are necessary for viscous flow solvers as the velocity profile near the wall may require dense layers to capture the velocity profile despite having cells with a high aspect ratio.

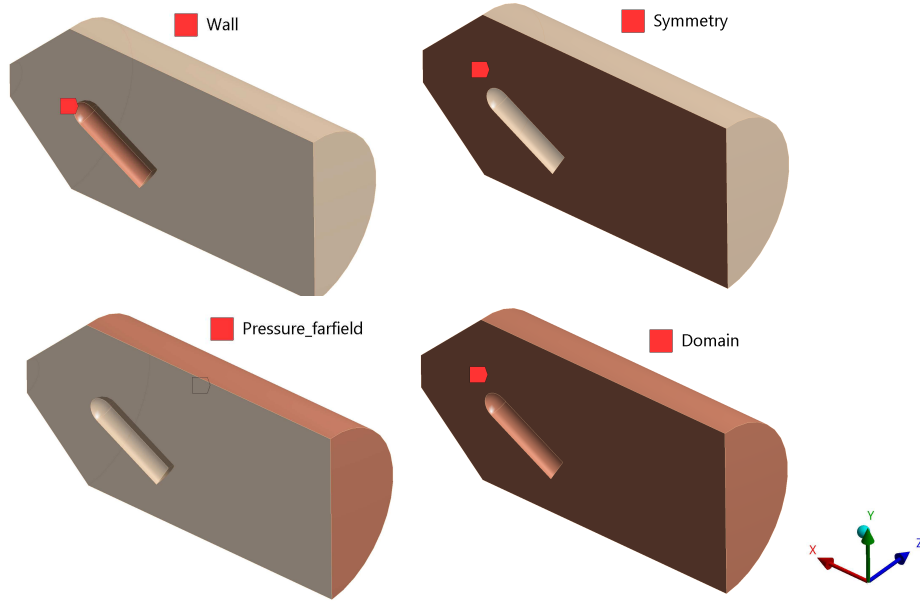


Figure 5.2: Isometric view of domain and hemisphere-cylinder cut-out with labelled surfaces for boundary conditions

5.1.2 Inflation Layer Mesh Settings

The inflation layer mesh setting is a vital parameter for turbulence or viscous models which is able to determine the boundary layer flow near the surface. Inflation layers are layers of dense cells that extend outwards from the surface or boundary. Each layer of cells follow an exponential growth in cell height where cells are closely packed together near the boundary. The cells with the greatest aspect ratio is adjacent to the boundary and increases in volume while maintaining its orthogonal dimension i.e. height of each successive inflation layer cell increases. Inflation layer cells play a role in determining the characteristics of flow separation hence influencing the flow field around the body. The default setting in the meshing program often selects the smooth transition setting for the growth rate of the inflation layers. Depending on the global and local (areas near the surface of the wall) mesh settings, the smooth transition setting may cause the mesh quantity to increase rapidly. This may cause a loss of solving efficiency and robustness as the meshed model becomes unnecessarily large. To decrease the mesh quantity while maintaining the mesh quality to accurately model the boundary layer near the surface, the first inflation layer cell was specified with a growth rate and number of inflation layers. The first inflation layer, which is measured as a length, was calculated using equation 5.1 [19]. Equation 5.1 calculates the y^+ parameter and is similar to Reynolds number equation (Equation 1.2). This dimensionless parameter is related to the distance of the boundary layer at the surface of the body hence considered for resolution of the inflation layers [20].

$$y^+ = \frac{u_w y_{bl}}{\nu} \quad (5.1)$$

u_w is the velocity near the wall, y_{bl} is the first inflation layer height, and ν is the kinematic viscosity of air. The velocity near the wall can be calculated using equation 5.2 and was derived by White [2].

$$u_w = \sqrt{\frac{\tau_w}{\rho}} \quad (5.2)$$

τ_w is the shear stress at the wall. Equation 5.1 can be manipulated to obtain y_{bl} which is the height of the first cell near the wall. The y_+ values are required to be below 1 for the turbulence models considered for this research but may vary for selected turbulence models such as the $\kappa - \epsilon$ turbulence model. The y^+ values were checked after the model was solved to ensure that value was less than 1. It is desired that y^+ values are less than one for near-wall flow models which allows flow separation characteristics to be modelled correctly [20].

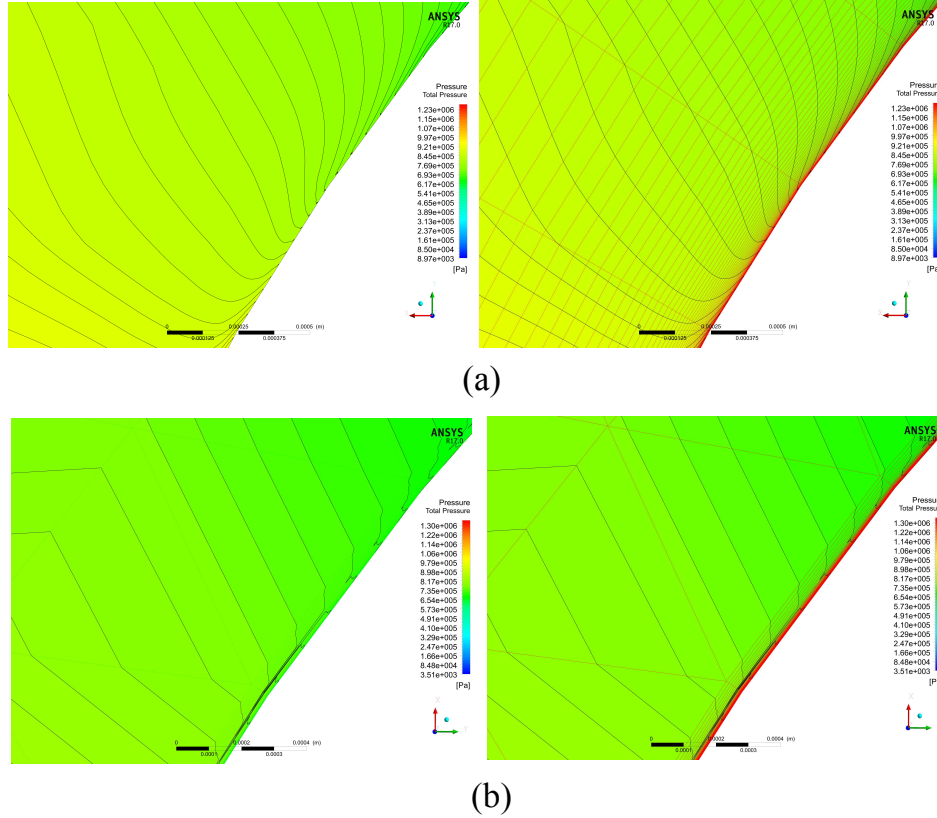


Figure 5.3: Pressure contour and mesh gridlines near the surface of the nose of the hemisphere-cylinder for inflation configured with: (a) a smooth transition, (b) abrupt transition of cell size. Images on the left indicates pressure contours only.

It is necessary to define the growth rate in the inflation layer settings to ensure that the transition of the inflation layer and surrounding cells adjacent to the inflation layer are smooth. The importance of a smooth transition is seen in figure 5.3 where it is evident that solutions are affected by the transition of the inflation layer. In figure 5.3a, a smooth transitioned inflation layer enables smooth contour lines whereas the rapid cell transition

(seen in figure 5.3b) show contour lines with kinks causing faulty solutions near the wall. The growth rate can be calculated by considering the following equation:

$$h_{IL} = y_{bl}(g_r)^{N_L} \quad (5.3)$$

h_{IL} is the height of the last inflation layer cell adjacent to the cell in the flow field, g_r is the growth rate of the inflation layer, and N_L is the number of defined inflation layers. The growth rate was calculated by equating the h_{IL} to the adjacent mesh size which was either defined by the global mesh size setting or the local mesh refinement size setting. The number of inflation layers affects the flow separation characteristics hence it was considered as a parameter for mesh independence which is detailed in later section.

5.1.3 Local Mesh Refinement

The local mesh refinement refers to meshing methods that target specific regions in the domain to increase the mesh density and quality. This allows for a more efficient mesh and reduces the time required to solve the flow dynamics. Local mesh refinement may also allow for a coarser global mesh setting. The body of influence setting was selected to refine the selected volume in the domain, which is highlighted in yellow in figure 5.4. The selected volume targets a larger area of refinement on the leeward side of the hemisphere-cylinder where flow separation and vortices are expected to develop. A portion of the volume covers the windward side of the body to include the flow dynamics modelling the cross flow. Cross flow and the trajectory of flow on the windward side of the body may affect the onset of flow separation on the leeward side of the body. The volume downstream of the hemisphere-cylinder was only refined in regions near the base and wake of the body. However mesh refinement was not prioritized at such locations as the region of interest was primarily near the shoulder of the body. This was considered a trade-off as the simulated solutions may not indicate accurate results in separated turbulent regions downstream of the base of the cylinder. The selected region for mesh refinement was scaled to twice the diameter size of the hemisphere-cylinder. The local cell size setting for the mesh refinement were based on the number of cells around the circumference of the cylinder i.e. 0.28 mm corresponds to 90 cell around the semi-circumference of the cylinder. This parameter was considered for mesh independence as it is an important mesh setting that may affect the accuracy of the solution. The area where the shock wave is present in the domain did not require local mesh refinement as mesh adaption methods were used, which is detailed in later sections.

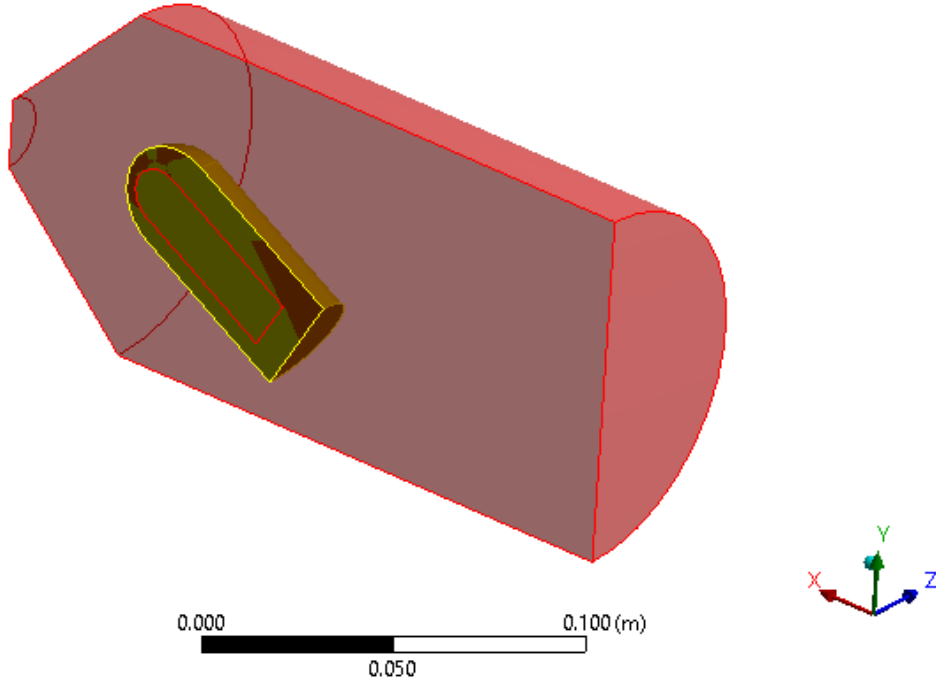


Figure 5.4: Isometric view of domain and the selected volume (in yellow) within the domain selected for local mesh refinement.

5.2 Model Equations and Solvers

Turbulence models are used to model the flow dynamics affected by viscous properties of flow on a surface. However, a universal turbulence model applicable for all types of flow applications does not exist. Each type of turbulence model relies on certain parameters that are defined in its algorithm hence influencing various variables such as simulation time, computational resources, solution accuracy, application of flow and flow physics [19]. Reynolds Averaged Navier-Stokes (RANS) turbulence models are effective due to its economical methods of modelling relatively complex flows and are the only types of solvers considered for this research. Navier-Stokes equations can solve flow dynamics in a 3D domain and are considered exact solutions as no approximations are made to reduce the number of equations and terms [6]. Navier-Stokes equations need to include energy equations for it to accurately solve compressible viscous flows. RANS turbulence models are based on Navier-Stokes equations but includes mean and fluctuating parts of flow variables. The equations are further averaged to yield the Reynolds-stress tensor term which includes the effect of turbulent motion [21]. This term is either solved using the Eddy Viscosity Models (EVM) or Reynolds Stress Models (RSM). EVM assumes that the gradients of velocity are proportional to Reynolds stress which reduces the number of unknown terms. RSM considers transport equations that can solve for all six unknowns that exist in the Reynolds-stress tensor term. RANS turbulence models considered for this research use EVM to solve for the Reynolds-stress term. Different

categories of RANS turbulence models are distinguished by the number of equations considered to solve the flow dynamics. The transport equation and treatment of the turbulence characteristics influences the number of equations utilized in the model. A steady-state flow solution method was selected for the different types of turbulence models considered for model independence. The turbulence models include $\kappa - \omega$ shear stress transport (SST), $\kappa - \kappa_L - \omega$ transition, and transition SST models.

The $\kappa - \omega$ SST turbulence model solves for the flow dynamics by the boundary layers near the wall of the body with relatively good predictions [22]. The flow dynamic equations of the $\kappa - \omega$ SST model were derived by Menter [23]. The $\kappa - \omega$ SST turbulence model uses two equations to model the flow dynamics. The equations are based on an empirical method and determines the turbulent kinetic energy (κ) and specific dissipation rate (ω). The algorithm of the $\kappa - \omega$ SST turbulence model is able to utilize exact equations to model the flow dynamics and only activates the empirical equations when flow is within close proximity to the wall, enabling a more robust solver [22]. The $\kappa - \kappa_L - \omega$ solver is derived from the $\kappa - \omega$ turbulence model where an additional transport equation is used to model the laminar kinetic energy (κ_L) for near wall flow conditions [24]. The three-equation turbulence model can solve the transition dynamics of flow present in the boundary layer where flow may change from laminar to turbulent flow conditions. The transition SST turbulence model is also derived from the $\kappa - \omega$ SST turbulence model by Langtry and Menter [25]. Two additional transport equations are included to model the intermittency factor (γ_{int}) and the onset transition criteria, in the form of momentum thickness Reynolds number (Re_θ). These extra equations increase the modelling capabilities of the turbulence model by considering the characteristics of flow separation [22]. The γ_{int} transport equation considers the characteristic of turbulent flow separation where flow occurs intermittently between laminar and turbulent flow conditions. This equation was introduced by Suzan et al. [26] where the transport equation was found to be in good agreement with separated flow and transition boundary layer. The Re_θ transport equation solves for the thickness of the boundary layer which captures information about the boundary layer. The coupling of the γ and Re_θ transport equations is presented by the research paper by Menter et al. [27] where the implementation of both transport equations allows for a robust method to blend the freestream flow conditions with the boundary layer.

When the desired turbulence model was selected, the flow properties and conditions were defined. A density-based implicit solver was considered as the model involved compressible flow dynamics. Flow properties that were defined in the model included the ideal gas properties, ambient gas density, and dynamic viscosity. Boundary conditions are assigned to the labelled surfaces and volume which are vital as these parameters specify the flow conditions simulated in the model. The defined boundary conditions include pressure far-field (defining the freestream flow conditions), symmetry (treatment of a surface to mirror flow dynamics), domain, and wall. Discretization schemes that approximates the flow dynamics are defined where the trade-off between the accuracy and simulation time was considered. A second-order scheme was considered sufficiently accurate as the first-order scheme displayed

much larger residuals for solution convergence, which is undesirable. The Courant number is a measure representing the transfer of flow information across a grid cell at a given time step, hence a parameter affecting the simulation time and convergence of the model. Values were chosen within the range of 0.3 and 0.9 for the Courant number. Although the Courant number requires time steps for the solver, the steady-state flow solution method accounts for the parameter through its time-marching method to achieve solution convergence.

As mentioned in the previous section, mesh adaption settings and commands were used in Ansys Fluent as the mesh adaption process is solution dependant. Mesh adaption was necessary and enabled an efficient method to model the characteristics of shock waves which requires areas with dense cells to capture the steep gradients of temperature, density, velocity and pressure. Mesh adaption was controlled through commands initiated by the selected number of iterations where the selected number of iterations exceeded the number of iterations required for solution convergence. Thresholds were defined for each of the flow parameter considered for mesh refinement. The cells that are marked undergo refinement by division of the original cell i.e one cell divides into four cells per level of mesh refinement. Mesh adaption not only improves the modelling of shock waves as a thin boundary but affects the resolution of the flow separation characteristics. Shock waves near the surface of the wall may also affect the flow separation characteristics hence obtaining mesh independence using mesh adaption was necessary. Hybrid initialization was used to calculate the initial input required for the model solver. Hybrid initialization utilizes rapidly solved Euler equations to approximate the initial flow field properties. This option benefited by decreasing the simulation time as the solutions converged much quicker. After the completion of the model simulation, data files were used for analysis in Ansys CFD-Post. A sample terminal user interface (TUI) command, which are commands used to define the model in Ansys Fluent, can be found in Appendix D.1.

5.3 Model Independence

Model independence refers to a set of solutions that does not have significant differences after varying certain model parameters, such as mesh parameters or turbulence model types. This process is required for CFD methods to ensure that the results are sufficiently accurate and has converged hence the model solutions are independent of variances in the mesh quality or the turbulence model. To prove model independence, assumptions are made to constrain the number of simulation models as it was not necessary to simulate each slender body type for each angle of incidence and Mach number. The assumptions are as follows:

- Slender bodies at 30° angle of incidence was assumed to have a higher complexity of flow separation characteristics in comparison to lower angles of incidence. Hence if model independence is obtained at 30° angles of incidence, it is assumed that models

with lower angles of incidence at the same freestream Mach number and nose shape have also obtained model independence.

- The flow separation characteristics are assumed to have a higher degree of complexity at higher Mach numbers, given that the flow conditions remain fully turbulent. Hence if model independence is obtained at Mach 3.2, it is assumed that models at lower supersonic Mach numbers have also obtained model independence given that the geometrical configuration is the same.

Mesh and turbulence model independence conditions are elaborated in the next section where acceptable tolerances are discussed.

5.3.1 Turbulence Model Independence

Turbulence model independence required the different types of models introduced in the previous section to be simulated and compared. The models are constrained to hemisphere-cylinders as preliminary simulation models indicated that hemisphere-cylinders had a more complex flow characteristics, hence sufficient to prove turbulence model independence for both nose types. The models were simulated with two different types of mesh, such as the coarse and fine mesh. The purpose of using a coarse mesh was to determine the desired solver type to be utilized for the mesh independence whereas the fine mesh type determined whether turbulence model independence was obtained. It was necessary to cross-check the models with a finer mesh, which was based on mesh independence, as the variation of the mesh caused distinct discrepancies for the comparisons of each turbulence model. Table 5.1 represents a matrix indicating the number of simulations required to complete turbulence model independence. The tolerance of error for turbulence model independence was not gauged quantitatively, with plots and properties, as semi-quantitative solutions indicating the flow characteristics were sufficient.

Table 5.1: Matrix of the simulation models required to achieve turbulence model independence

Geometry		Model Settings	
Body type	Incidence ($^{\circ}$)	Mesh Type	Turbulence Model Type*
Hemisphere-cylinder	30	Coarse	KWSS
			KKLW
			TSST
		Fine	KWSS
			KKLW
			TSST

* KWSS: $\kappa - \omega$ SST solver, KKLW: $\kappa - \kappa_L - \omega$ solver, TSST: Transition SST solver

5.3.2 Mesh Independence

Mesh independence required a thorough approach as CFD solutions are sensitive to the quality of the mesh due to its reliance on numerical solvers. Mesh parameters such as the number of inflation layers, local mesh size, and the number of mesh adaption levels were considered. Mesh adaption were used by means of activation through specified iterations i.e. mesh adaption commands that activated after every 4000 iterations would have two levels of mesh adaption if 10000 iterations was completed for the simulation. By changing the number of iteration of the model, the levels of adaption can be varied. The local mesh refinement considered the number of circumferential cells around the body. This required further adjustments for growth rate settings for the inflation layers to ensure a smooth transition.

Mesh independence were considered for both the hemisphere-cylinder and cone-cylinder, where solutions are compared both quantitatively and qualitatively. Quantitative solutions are analysed over the leeward side of the body along the symmetry plane as flow separation occurs over such regions. Quantitative analysis is only done for local mesh refinement comparisons as the variation of this mesh parameter affects the solution by the greatest amount. Qualitative solution comparisons, such as surface contours and flow topology, are done for all mesh parameters. The various selections of mesh parameter settings can be seen in table 5.2 and shows that the local mesh refinement is dependent on the outcome of the inflation layer and mesh adaption settings. This hierarchal arrangement of the mesh independence process is preferred as it reduces the required total simulation time i.e. the mesh quantity differences were less for inflation layers than local mesh refinement. The chosen mesh parameters are discussed in the post-processing section where it is supported by CFD solutions.

Table 5.2: Matrix of the simulation models required to achieve mesh independence

ID No.	Geometry		Mesh Parameters				
	Body type	Incidence(°)	Inflation layers	Levels of adaption	No. of cells per circumference (mm)		
M1	Hemisphere-cylinder	30	60	-	-		
M2			75	-	-		
M3			90	-	-		
M4			Chosen based on M1-M3	2	-		
M5				3	-		
M6				4	-		
M7				Chosen based on M4-M6	150 (0.335)		
M8					180 (0.28)		
M9					200 (0.25)		
M10					220 (0.228)		
M11	Cone-Cylinder		60	-	-		
M12			75	-	-		
M13			90	-	-		
M14			Chosen based on M11-M13	Chosen based on M4-M6	120 (0.42)		
M15					150 (0.335)		
M16					180 (0.28)		
M17					200 (0.25)		

6 Post-Processing

The results obtained from both the experimental and computational methods requires processing to enable clear results for analysis of the flow dynamics.

6.1 Experimental Results

The raw images gathered from wind tunnel testing required processing to enhance the clarity of the recorded images. Image processing was completed using the Matlab image-processing package due to its ability to precisely select and modify pixels using matrices. Note that post-processing for the experimental results do not alter the image results but only enhance the quality of the image, hence the integrity of the experimental results are maintained.

6.1.1 Optics Flow Visualization

The image processing for the optics flow visualization results required adjustments to the contrast and sharpness of the image. It was also necessary to convert the image into gray-scale images, as no colour was used for the optics set-up, and cropping the image to exclude redundant information from the images. The steps of processing the optics flow visualization image is seen in figure 6.1.

6.1.2 Surface Oil Flow Visualization

The image processing for the surface oil flow visualization results required adjustments to the background and enhancement of the streaks on the surface of the body. It is necessary to refine the edges of the pigment streaks to assist in comparisons of the flow topology and surface oil flow visualization. The presence of a coloured image involved multiple sets of matrices that represent different shades of coloured pixels. The background of the images was masked by excluding pixels that did not fall within a certain numerical range defining the colour of the pigments, as the background had a different shade of colour. To enhance the edges of the pigment streaks, a high-pass image filter was used which also adjusted the contrast of the colour of the pigment streaks. The 3×3 filter matrix was adjusted by means of an iterative process as the variation of the pigment colours were sensitive to the filter. The high filter was typically defined by a single high positive value at the centre of the matrix with the rest of as negative values. Figure 6.2 indicates the steps of processing the surface oil flow visualization results.

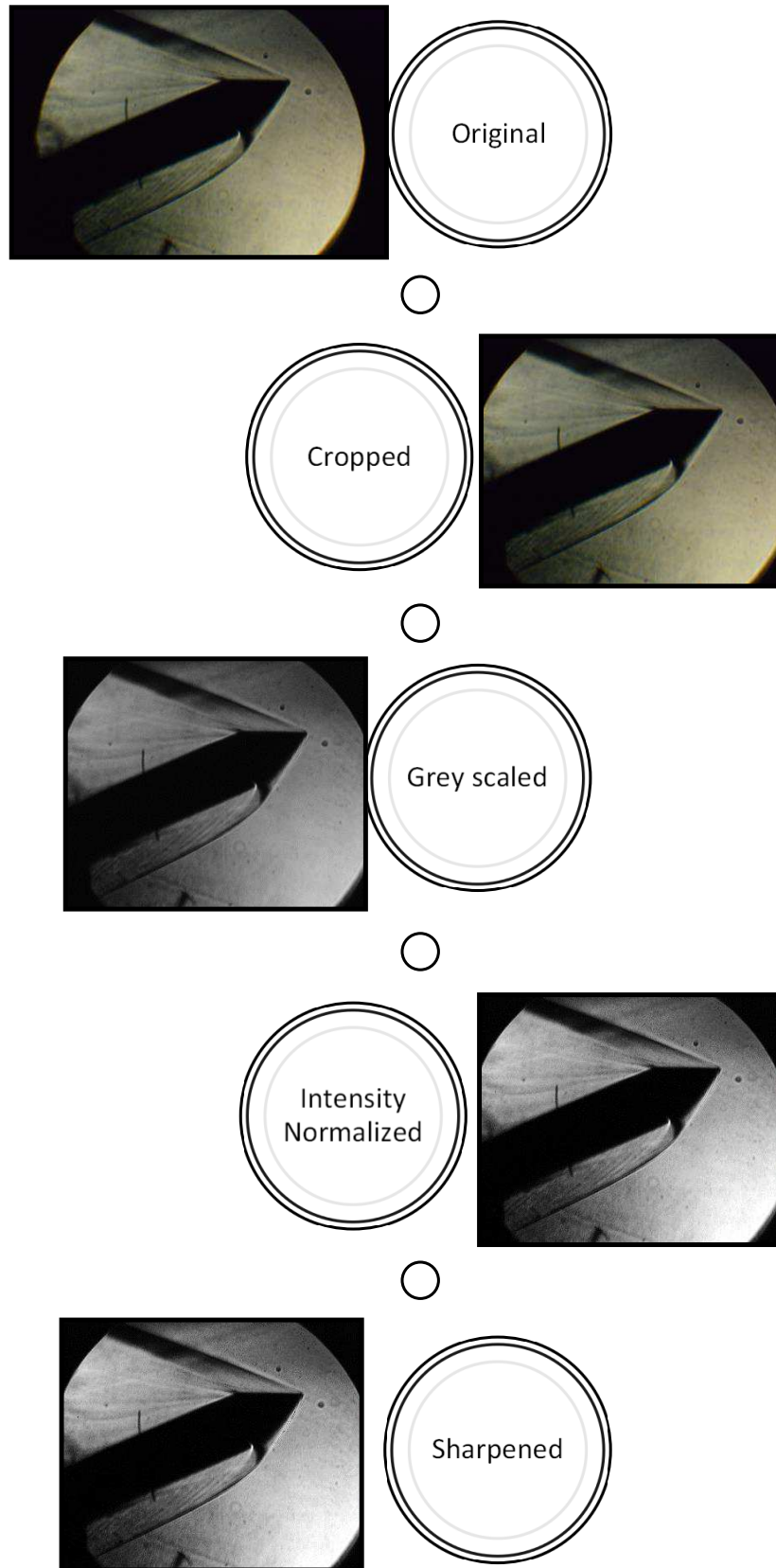


Figure 6.1: Flow chart indicating steps of processing images obtained from optics flow visualization

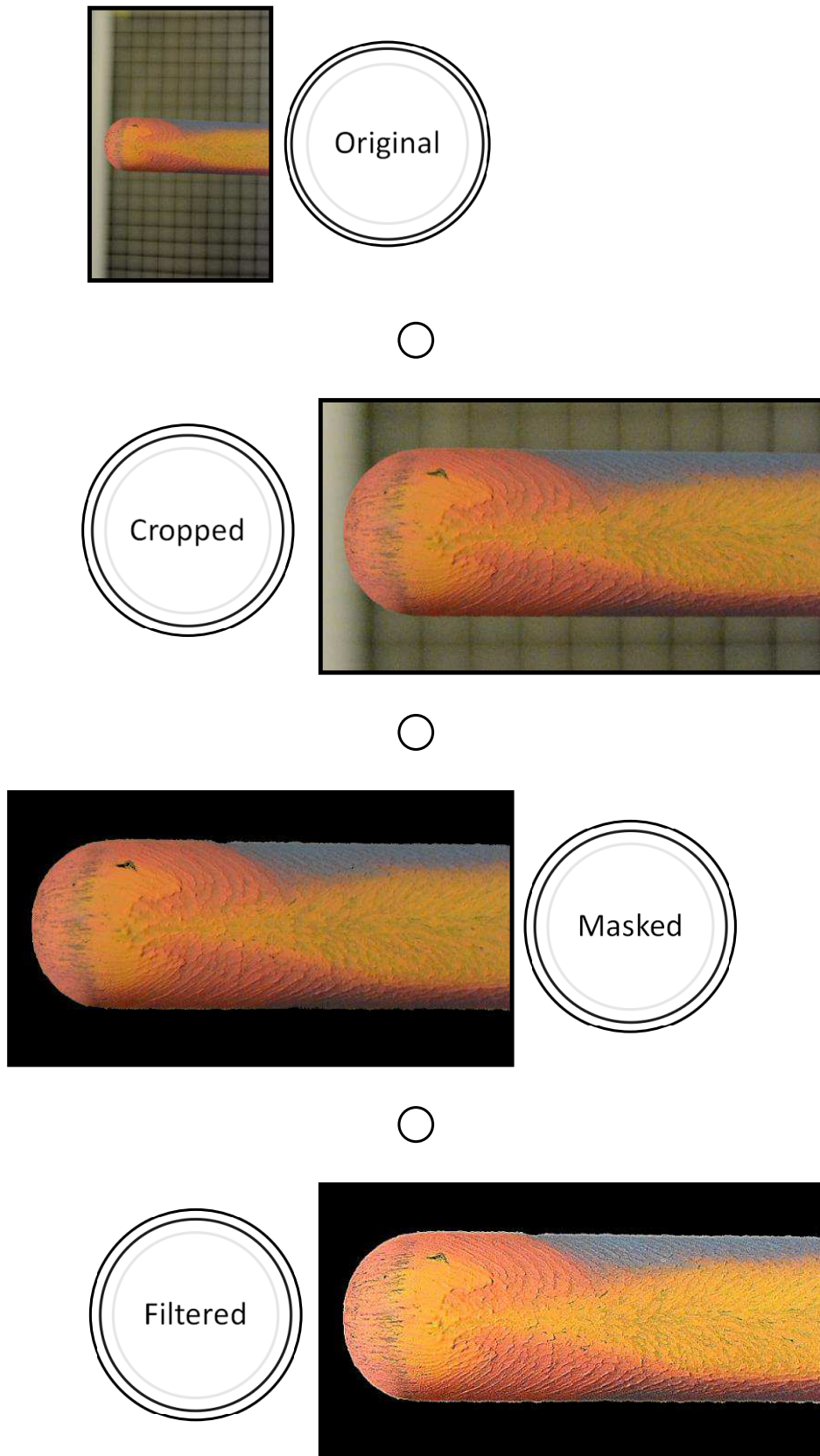


Figure 6.2: Flow chart indicating steps of processing images obtained from surface oil flow visualization

6.2 Computational Results

The results from computational simulations required processing to visually present the flow characteristics. The Ansys CFD-Post software was used as it provided several functions that allowed flow properties to be shown on generated surfaces.

6.2.1 CFD Flow Visualization

CFD flow visualization can present results in the form contours and surfaces with a selection of flow properties. Contour plots at various longitudinal cross-sections of the body provide sufficient information regarding the development of the flow separation. A colour scheme was used for the contours to present flow properties quantitatively. Common flow properties include temperature, Mach number, pressure, or density. Vertical planes are included to provide information from another perspective to supplement the 3D surfaces that are represented as iso-surfaces. Iso-surfaces indicate supersonic and flow separation features which enables 3D representation of these phenomena. Surface streamlines on the slender bodies were important for this research as they provided information on locations where flow separation occurred and development of vortices on the surface of the body. Flow field streamlines and surface vectors are considered to determine the trajectory of the flow in certain areas.

Model independence relied on the processing of flow visualization results for qualitative comparisons. It was sufficient to compare the flow topology and Mach number contours with vectors about the symmetry plane as distinct discrepancies in the flow separation characteristics were evident.

6.2.2 Data Plots

Quantitative plots are utilized as additional information to support the semi-quantitative results generated for flow visualization. Plots may contain flow properties such as pressure, density, and Mach number over local positions of the body. Pressure plots around the circumference of the slender bodies are considered for mesh independence to compare the differences between each mesh. Pressure properties are sufficient for mesh independence as this parameter is affected by the onset of flow separation hence a good representation of the discrepancies of the flow dynamics. Aerodynamic properties were not plot as they were considered auxiliary information. The study of aerodynamic properties for different nose geometry and incidences of slender bodies are beyond the scope of this research.

6.3 CFD Model Independence

For all models created for both the mesh and turbulence model independence, flow velocity of Mach 3.2 and incidence of 30° were considered for both the hemisphere-cylinder and cone-cylinder.

6.3.1 Turbulence Model Independence

The turbulence model independence follows the matrix, as seen in table 5.1, which considered the hemisphere-cylinder for three different turbulence models. Qualitative results are presented as surface streamlines with skin friction coefficient contours on the leeward side of the body. Specific attention was considered for the nose flow separation line occurring near the shoulder of the hemisphere-cylinder as the initiation of the flow separation affected the flow dynamics downstream. Mach number contours and vectors about the symmetry plane are shown to detail the effects of the flow separation near the nose of the body.

Figures 6.3 and 6.4 are results from the coarse mesh models indicating the comparisons between the solver types in the form of flow topology and flow field characteristics respectively. Clear distinctions of the nose separation line are evident where flow separation seems to occur near the shoulder for the transition SST solver. An absence of a developing vortex system near the shoulder is apparent for the $\kappa - \omega$ SST and $\kappa - \kappa_L - \omega$ models, and the flow separation by the nose are delayed further downstream of the shoulder. The nose separation line only exists for the $\kappa - \omega$ SST and transition SST models whereas a separation point is present for the $\kappa - \kappa_L - \omega$ model. The variation of the nose separation lines influences the initiation of cross separation due to an open flow separation type, hence the transition SST model has an overall larger flow separation region. The flow separation regions by the symmetry plane are indicated in figure 6.4 where the transition SST solver shows an elongated flow separation region. Flow vectors do not show large discrepancies between the solvers as their resolution of the vector arrows are insufficient. The supersonic features such as the bow shock wave and embedded shock are similar, which was expected as transport equations for κ and ω are able to model the flow dynamics away from the wall accurately. The $\kappa - \omega$ SST solver also indicates a slightly greater Mach number value in comparison to the other two solvers. The discrepancies between the turbulence models indicate that turbulence model independence was not achieved for coarse mesh settings hence the transition SST turbulence model was used for mesh independence. A fine mesh model for each turbulence model was considered, after mesh independence was completed, as there was a probability that turbulence model independence can be achieved and may provide clearer similarities between each turbulence model.

Figures 6.5 and 6.6 are solutions from fine mesh models that indicate the comparisons of the turbulence model types in the form of flow topology and flow field characteristics respectively.

The fine mesh models were based on the selected mesh model for mesh independence hence it was a mesh type optimized for the transition SST model. The differences of the nose separation lines are more distinct between the different type of turbulence models with a finer mesh. The $\kappa - \kappa_L - \omega$ model generated the widest nose separation line that spanned half of the diameter of the cylinder, hence resulting in a larger flow separation region. The $\kappa - \omega$ SST model developed a pair of vortices in the flow separation region with a cross flow separation downstream similar to the transition SST solver. The transition SST solver indicated less flow topology differences when the coarse and fine mesh models were compared. The fine mesh model displayed points, such as saddle points and focus points, that were more distinct and the core of the vortex seemed more developed. The distinct nose separation line contributed to greater flow separation regions, as seen in figure 6.6, where the $\kappa - \omega$ SST and $\kappa - \kappa_L - \omega$ models are greater compared to the coarse-mesh models. The transition SST had minimal differences compared to its coarse mesh counterpart, however the flow separation regions are clearer with a greater mesh resolution. The flow separation region for the $\kappa - \kappa_L - \omega$ model showed jagged surface flow patterns and was highly turbulent downstream of the shoulder as shown in the flow topology results. It is unlikely that the flow dynamics would develop such flow characteristics as historical experimental results did not encountered such flow characteristics.

Turbulence model independence between the three selected models was not achieved and the solver with the greatest accuracy cannot be deduced without comparisons to the experimental results. The four-equation transition SST model was prioritized to generate CFD solutions for validation as the solver was considered to have greater accuracy based on the quantity of transport equations used to model flow separation and boundary layers. RANS solvers with additional transport equations can be considered, however this was not necessary as it will be indicated later that the transition SST solver is sufficient for flow separation analysis.

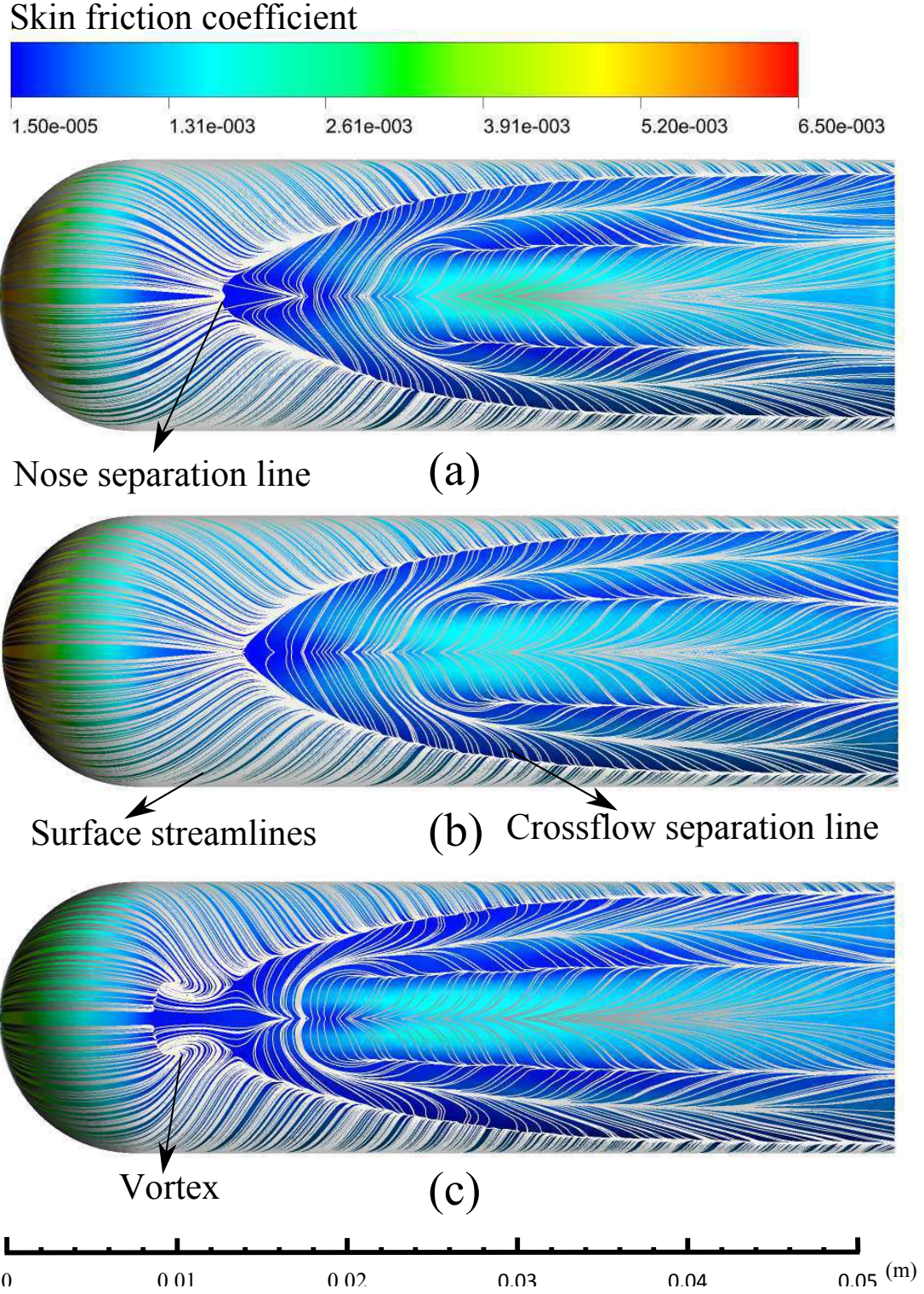


Figure 6.3: Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder simulated with a: (a) $\kappa - \omega$ SST model, (b) $\kappa - \kappa_L - \omega$ model, (c) Transition SST model, using a coarse mesh.

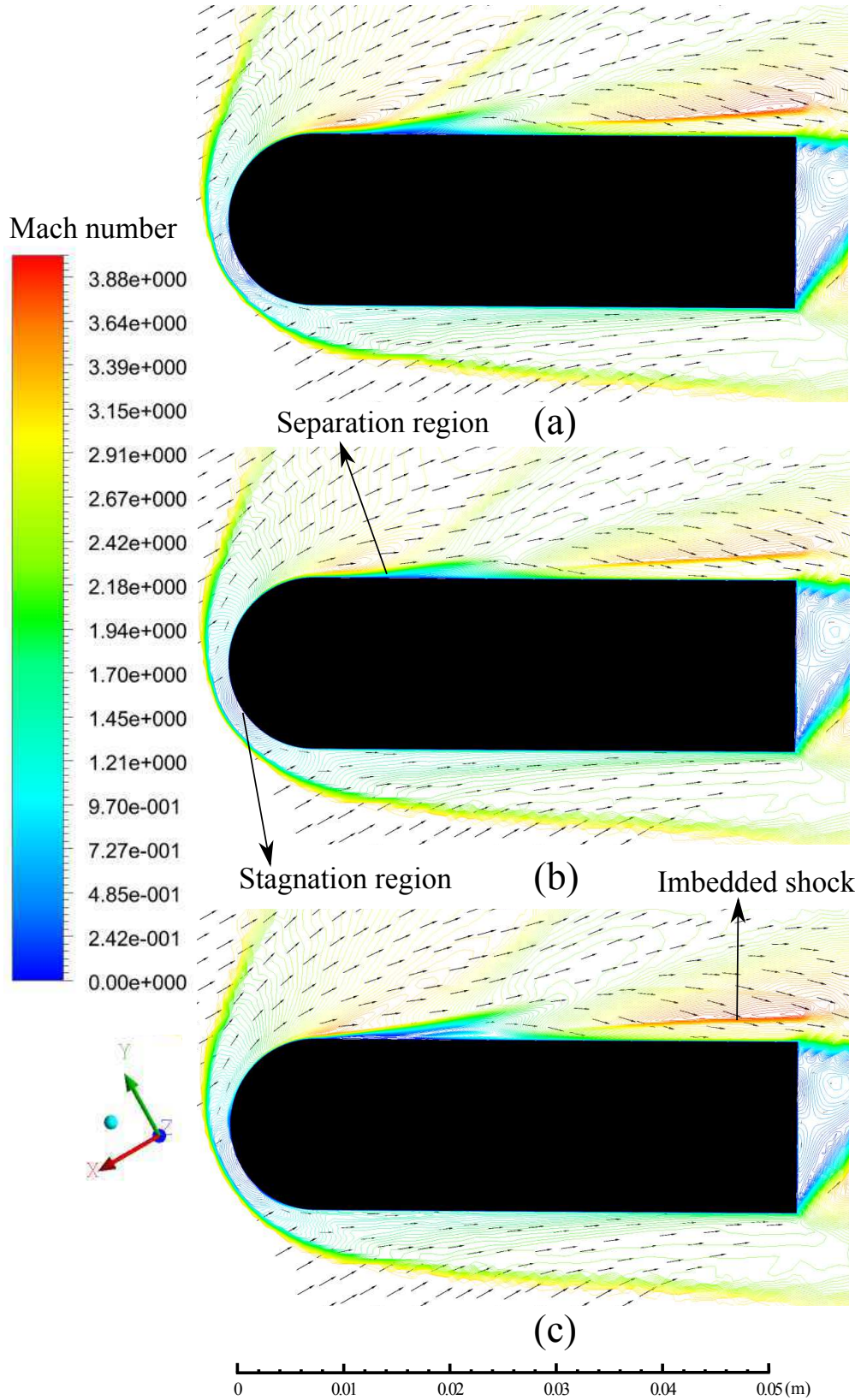


Figure 6.4: Mach number contours and vectors plot over the symmetry plane of the hemisphere-cylinder simulated with a: (a) $\kappa - \omega$ SST model, (b) $\kappa - \kappa_L - \omega$ model, (c) Transition SST model, using a coarse mesh.

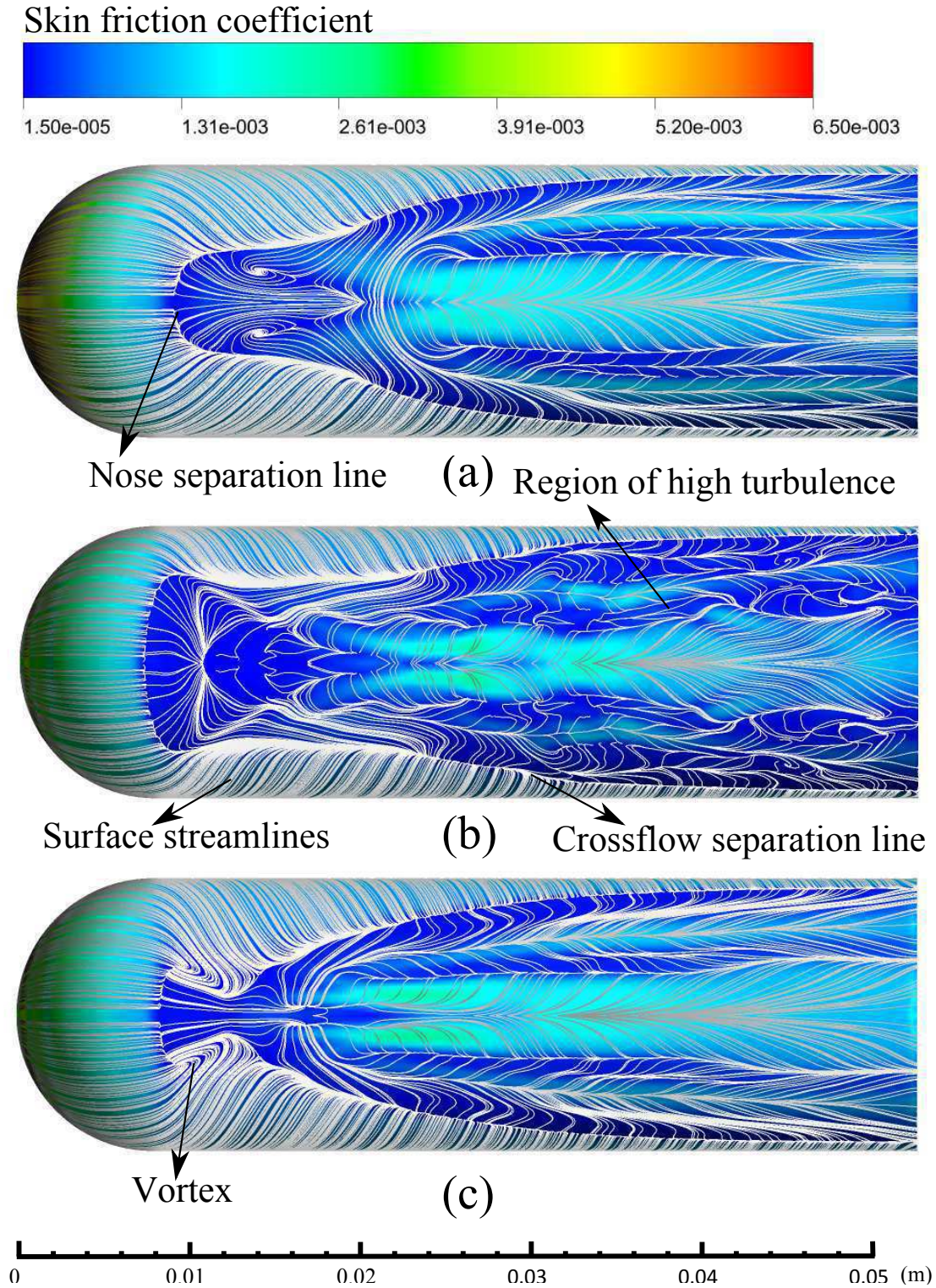


Figure 6.5: Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder simulated with a: (a) $\kappa - \omega$ SST model, (b) $\kappa - \kappa_L - \omega$ model, (c) Transition SST model, using a fine mesh.

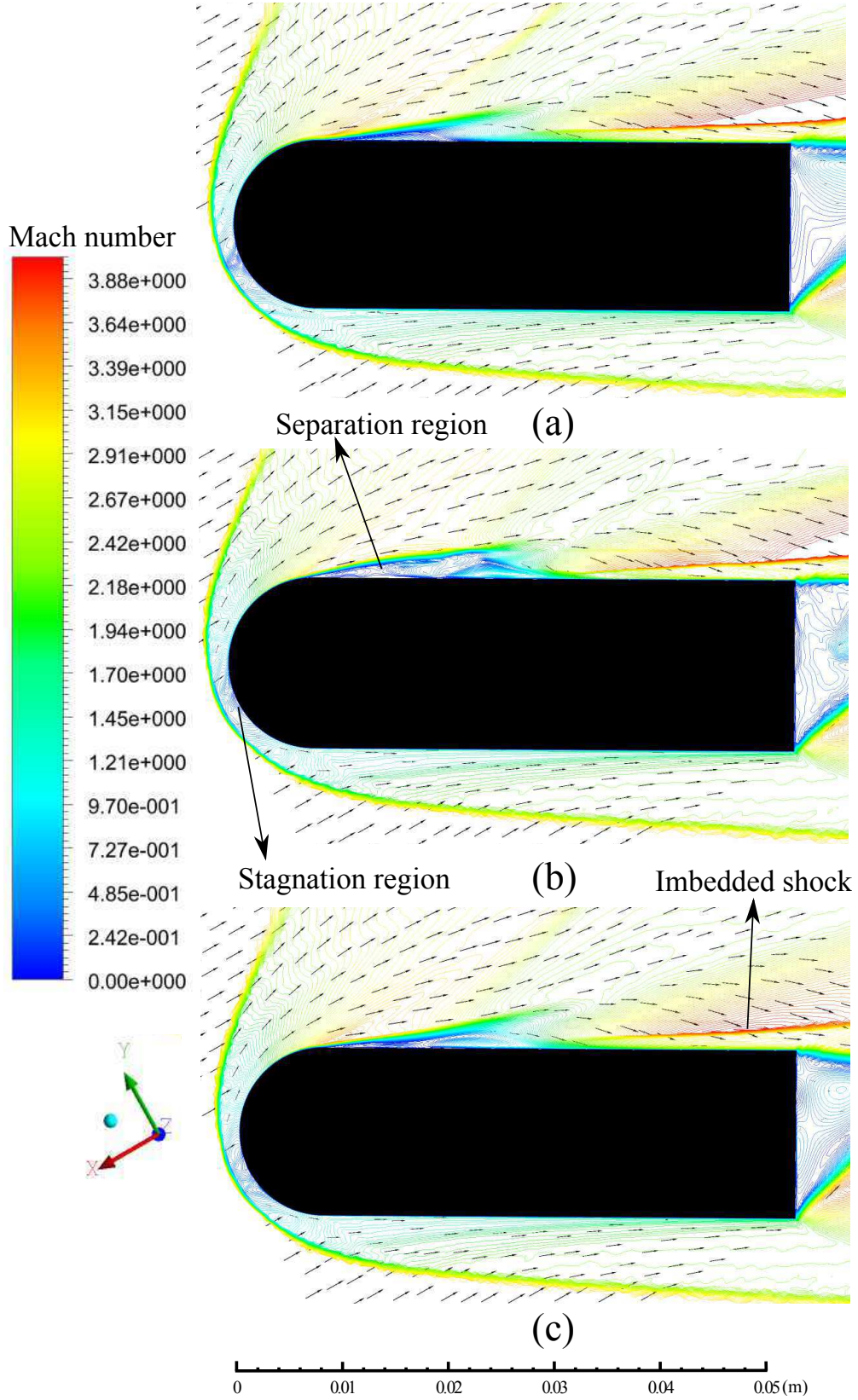


Figure 6.6: Mach number contours and vectors plot over the symmetry plane of the hemisphere-cylinder simulated with a: (a) $\kappa - \omega$ SST model, (b) $\kappa - \kappa_L - \omega$ model, (c) Transition SST model, using a fine mesh.

6.3.2 Mesh Independence

The mesh independence follows the matrix, as seen in table 5.2, which considered both the hemisphere-cylinder and cone-cylinder. Qualitative results are presented and assessed in the same manner as turbulence model independence. Quantitative results, in the form of pressure plots and percentages of deviation, are presented for the local refinement cases.

The first set of mesh independence solutions considers the variation of inflation layers as seen in figure 6.9, which is for the hemisphere-cylinder. The flow topology indicates under-developed nose separation lines that are subtly different between each inflation setting. The flow topologies between the 75 inflation layers and 90 inflation layers settings are more similar to the 60 inflation layer setting, as the nose separation lines are straighter. The cross flow streamlines also seem more dense near the nose separation line for the 75 and 90 inflation layers setting. The flow field about the symmetry plane did not indicate any significant differences despite the subtle difference between each inflation layer setting and can be seen in figure D.1 under Appendix D.1. Although the discrepancies are minor, it can be deduced that the similarities between the 75 and 90 inflation layer settings were sufficient for mesh independence. Note that the solutions may experience greater changes from local mesh refinement and mesh adaption settings as the solutions are more sensitive to these parameters. Hence it was acceptable to consider an inflation layer mesh setting of 75 layers for the hemisphere-cylinder to progress with the next mesh parameter which was mesh adaption settings.

The inflation layer mesh settings comparison for the cone-cylinder models are indicated in figure 6.10 where the results were similar with subtle changes in the flow separation regions. Nose separation lines and cross flow separation lines are similar with minimal differences in the distribution of skin friction coefficient. The lack of discrepancies between the different inflation layer mesh settings translated to almost identical flow field dynamics about the symmetry plane, as seen in figure D.2 under Appendix D.1. The only discrepancy is located by the shoulder near the symmetry plane where a more defined diamond-shaped streamline pattern was formed for the 75 and 90 inflation layer mesh setting compared to the 60 inflation layer setting. Although there were no significant differences between the 60 and 75 inflation layer setting, the 75 layer was still chosen for further progress of mesh independence.

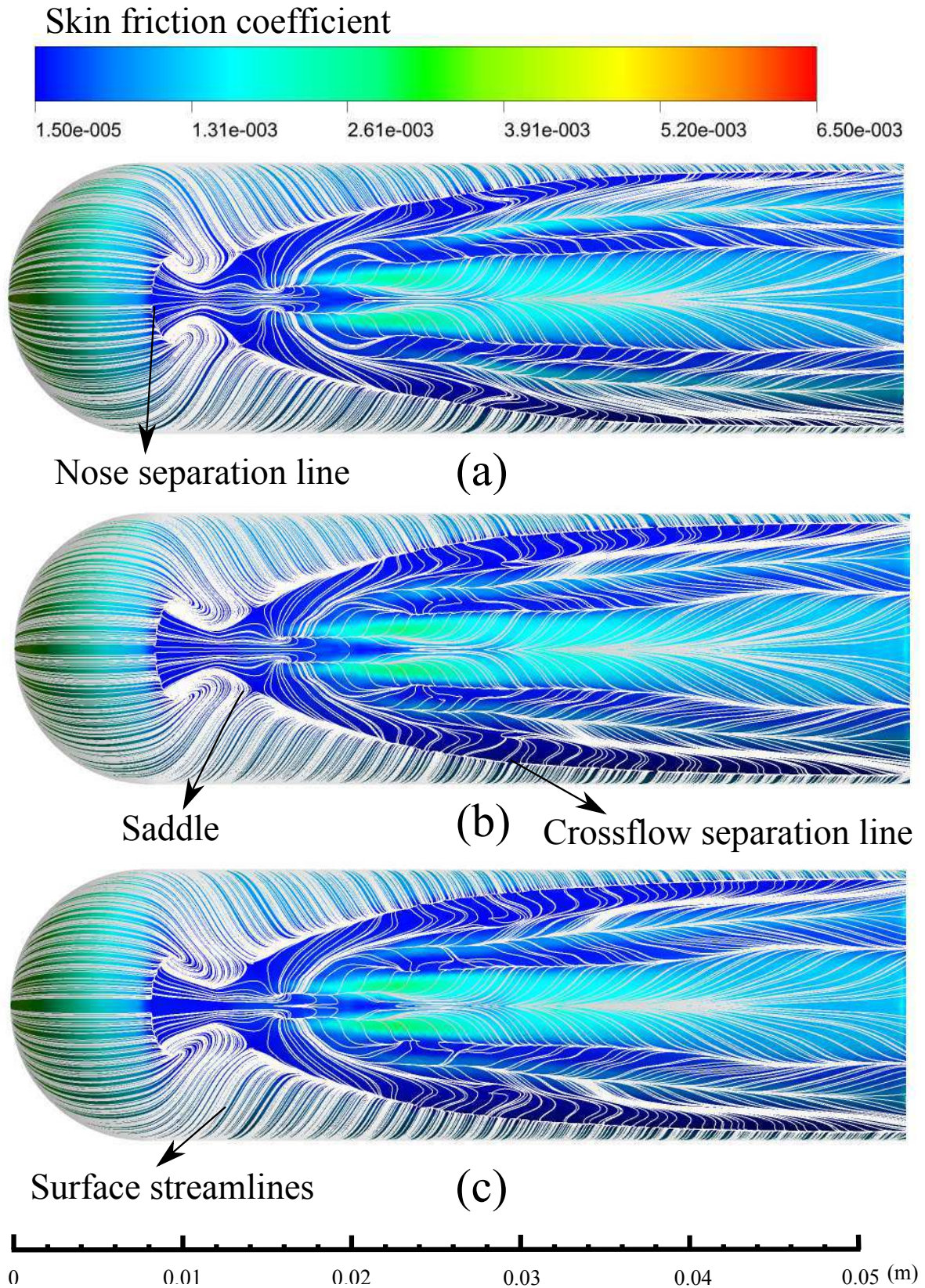


Figure 6.7: Wall shear surface streamlines and skin friction coefficient contours on the leeward side of the hemisphere-cylinder with mesh adaptation: (a) 2 levels of refinement, (b) 3 levels of refinement, (c) 4 levels of refinement.

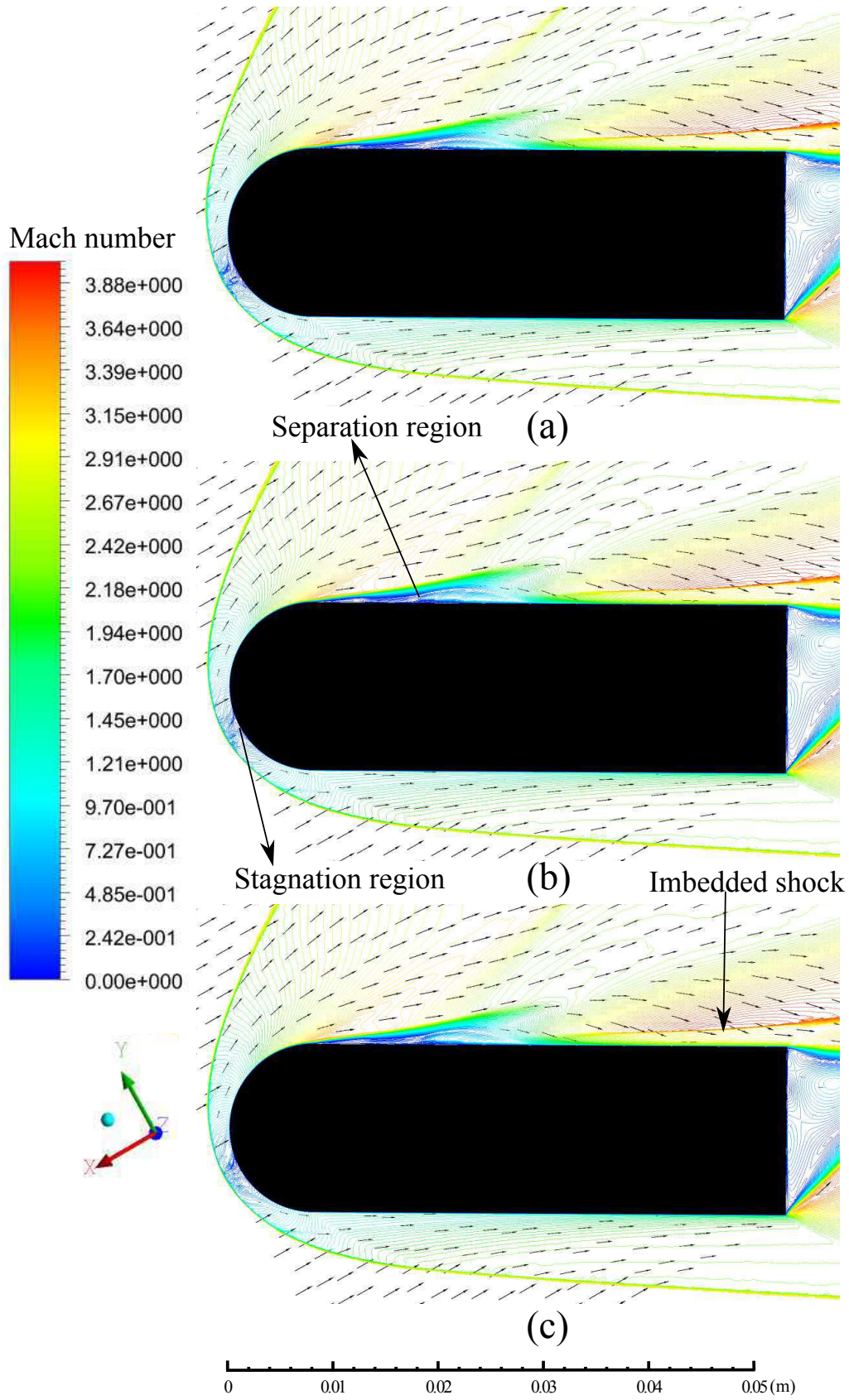
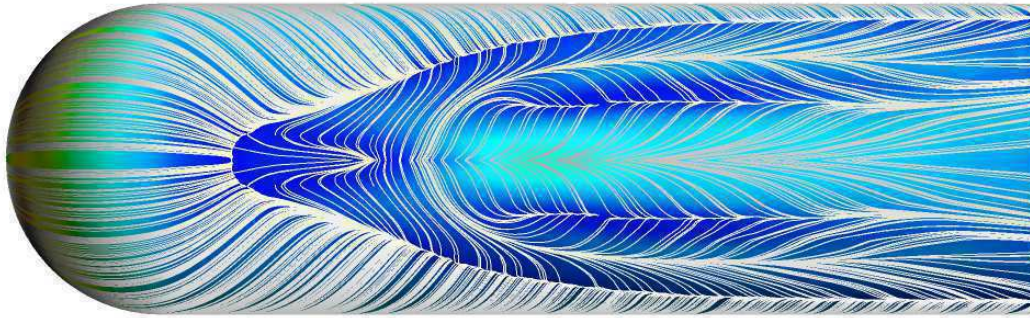
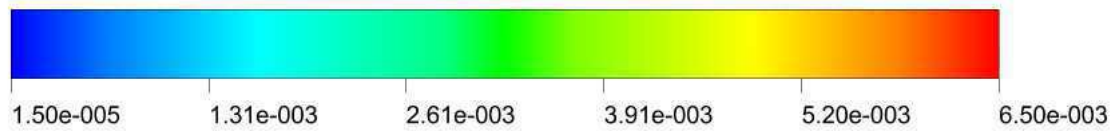


Figure 6.8: Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with mesh adaption for: (a) 2 levels of refinement, (b) 3 levels of refinement, (c) 4 levels of refinement.

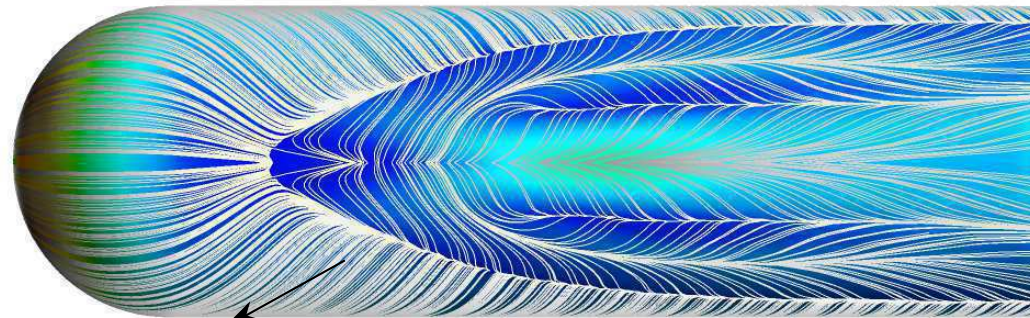
The set of solutions for the comparison of the mesh adaption settings are indicated in figures 6.7 and 6.8. The implementation of mesh adaption resulted in a greater development of the flow separation characteristics and its potential to generate a vortex near the shoulder. The increased level of mesh adaption affected the nose separation line and the saddle point located further downstream. Higher levels of mesh adaption resulted in straightened nose separation lines and saddle points were located closer to the symmetry plane. Discrepancies were present in the flow field, as seen in figure 6.8, where a greater mesh resolution enabled stagnation pressure regions with smoother contours around the nose. The shock waves, expansion waves, and flow separation boundaries all have thinner contour density lines with greater levels of mesh adaption. The skin friction coefficient over the leeward side remains similar, hence an indication that mesh adaption affects regions with large gradients instead of the entire flow field. The discrepancies between three and four levels of adaption were not significant, hence it was acceptable to consider three levels of mesh adaption for the hemisphere-cylinder to progress with the next mesh parameter which was local mesh refinement. The same number of mesh adaption was applied to the cone-cylinder models as this enabled the same thin shock wave and boundary characteristics.

The solutions for the local mesh refinement of the hemisphere-cylinder are shown in figure 6.11. Discrepancies between each mesh refinement setting are apparent in the structure of the surface streamlines. The changes in the surface streamlines also affects the saddle and focus points by the shoulder. There are nuances in the nose separation lines which were likely affected by the horn vortex structure on the surface. The nose separation line for the 0.335 mm mesh size indicated a much longer separation line causing the vortex cores to be located further apart. A coarser mesh resulted in the surface streamlines to be somewhat lacking, despite the same number of streamlines used for each mesh model. An indication of this was the saddle point not being distinct near the shoulder. The surface streamlines converging towards the saddle point and the vortex core differed where the finer mesh had a greater density of surface streamlines converging to such points. The cross flow separation lines and the distribution of skin friction coefficient were similar, hence indicating that local mesh refinement mainly caused differences near the shoulder of the hemisphere-cylinder. The flow field characteristics at the symmetry plane, as seen in figure D.3 under Appendix D.1, did not indicate significant differences between each mesh type as the discrepancies of the nose separation lines were subtle.

Skin friction coefficient



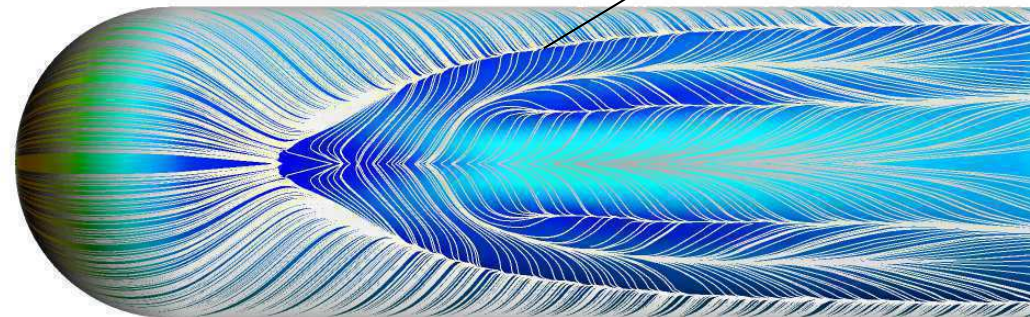
(a)



Surface streamlines

(b)

Crossflow separation line



(c)

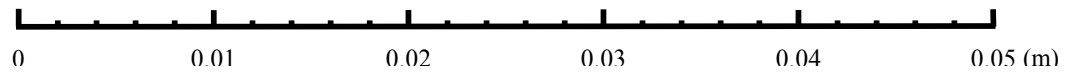


Figure 6.9: Wall shear surface streamlines and skin friction coefficient contours on leeward side of the hemisphere-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers, (c) 90 layers.

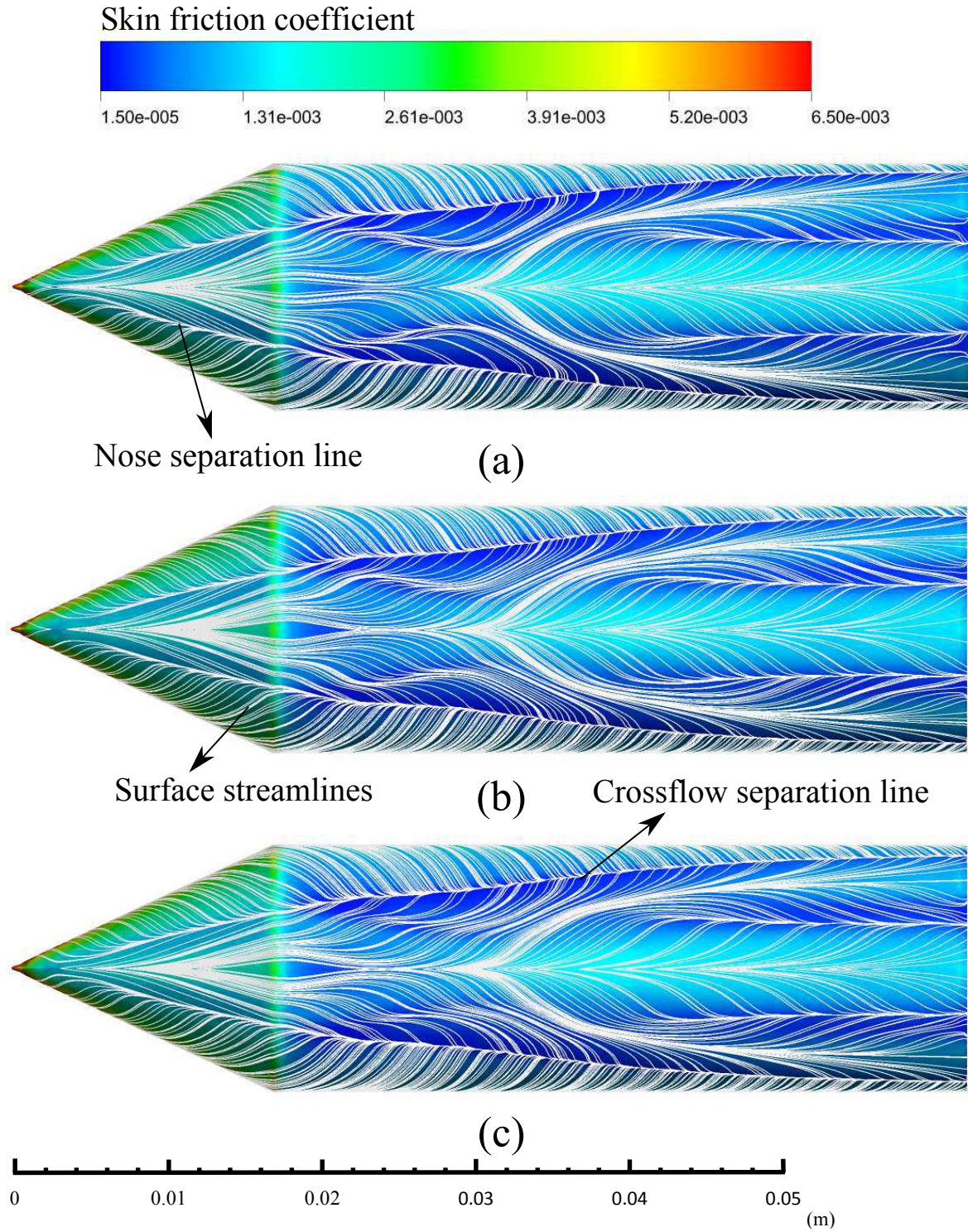


Figure 6.10: Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the cone-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers , (c) 90 layers.

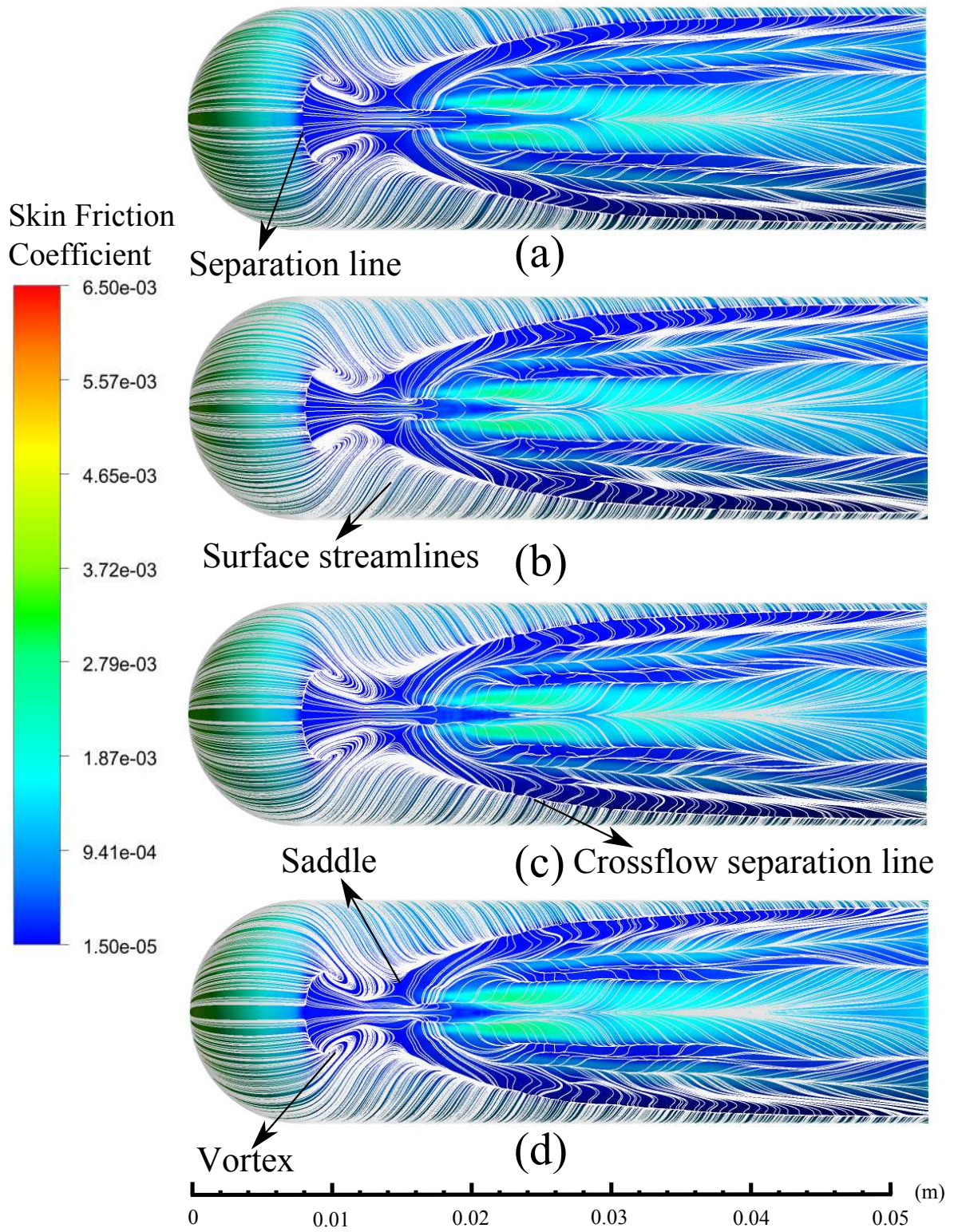


Figure 6.11: Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the hemisphere-cylinder with local mesh refinement settings: (a) 0.335 mm (150 circumferential cells), (b) 0.28 mm (180 circumferential cells), (c) 0.25 mm (200 circumferential cells), (d) 0.228 mm (220 circumferential cells).

Quantitative results were extracted from the CFD solutions and were specifically obtained at circumferential cross-sections located at the shoulder and 5 mm downstream of the shoulder for the hemisphere-cylinder. The circumference of the shoulder was selected as this was the area of interest for this research hence accuracy is vital for this region. The 5 mm offset from the shoulder was chosen as this region coincides or within close proximity to the location of the saddle point that is influenced by the cross flow streamlines and the nose flow separation. The data for 5 mm shoulder offset circumference would further indicate whether the flow field downstream the shoulder would have a tendency to converge or diverge. The pressure deviations were calculated in relation to the 0.228 mm mesh size which was assumed to have greater accuracy due to a greater mesh resolution. The pressure deviations at the shoulder and 5 mm downstream of the shoulder for the hemisphere-cylinder are seen in figures 6.12 and 6.13 respectively, which corresponds with figures D.4 and D.5 in Appendix D.1 indicating the coefficient of pressure (C_p) plot around its respective circumference. The maximum acceptable pressure deviation was considered as 5%, hence any deviation values exceeding the limit were deemed unacceptable for mesh independence. It is seen that the 0.335 mm mesh size for the hemisphere-cylinder had exceeded the limit of 5% at 20° from the leeward side at the shoulder, whereas other data points around the circumference were below the 5% deviation limit. Although the deviation for the 0.25 mm mesh size has a larger pressure deviation in comparison to the 0.28 mm mesh size, the discrepancies seen in the qualitative results were also taken into consideration. The pressure deviation at the shoulder circumference indicated a gradual decrease in pressure deviation towards the windward side of the body, whereas the pressure deviation trends at the 5 mm offset circumference were more irregular.

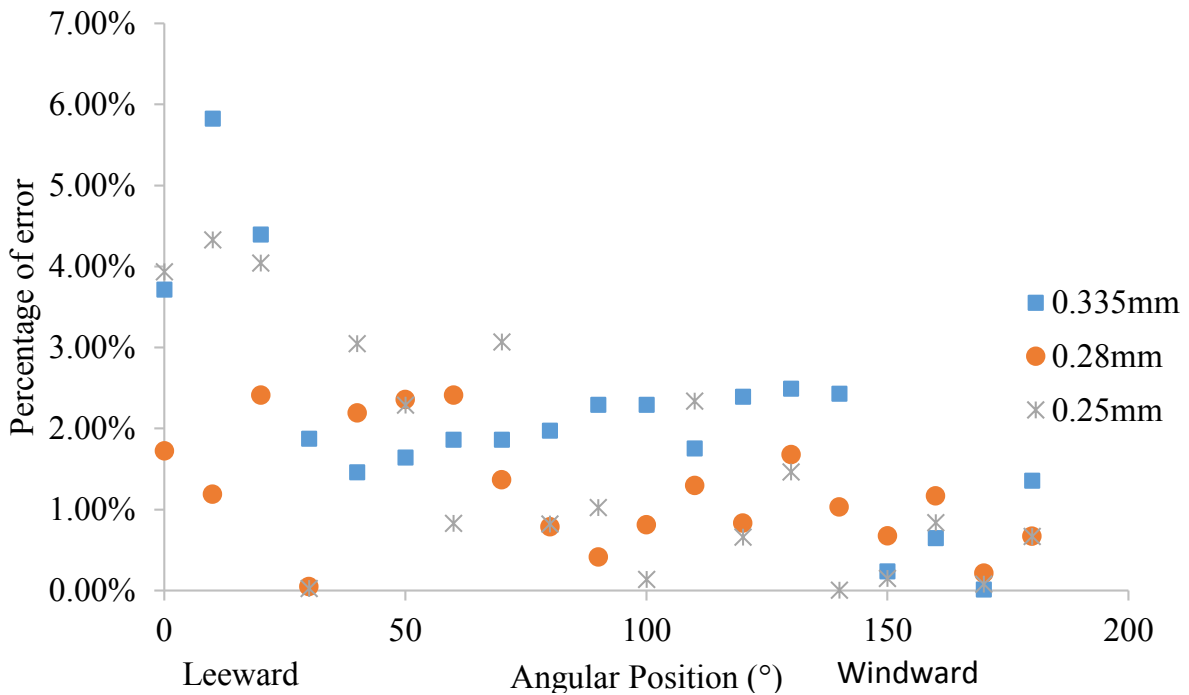


Figure 6.12: Pressure deviations relative to 0.228 mm local mesh size at different circumferential locations at the shoulder of the hemisphere-cylinder.

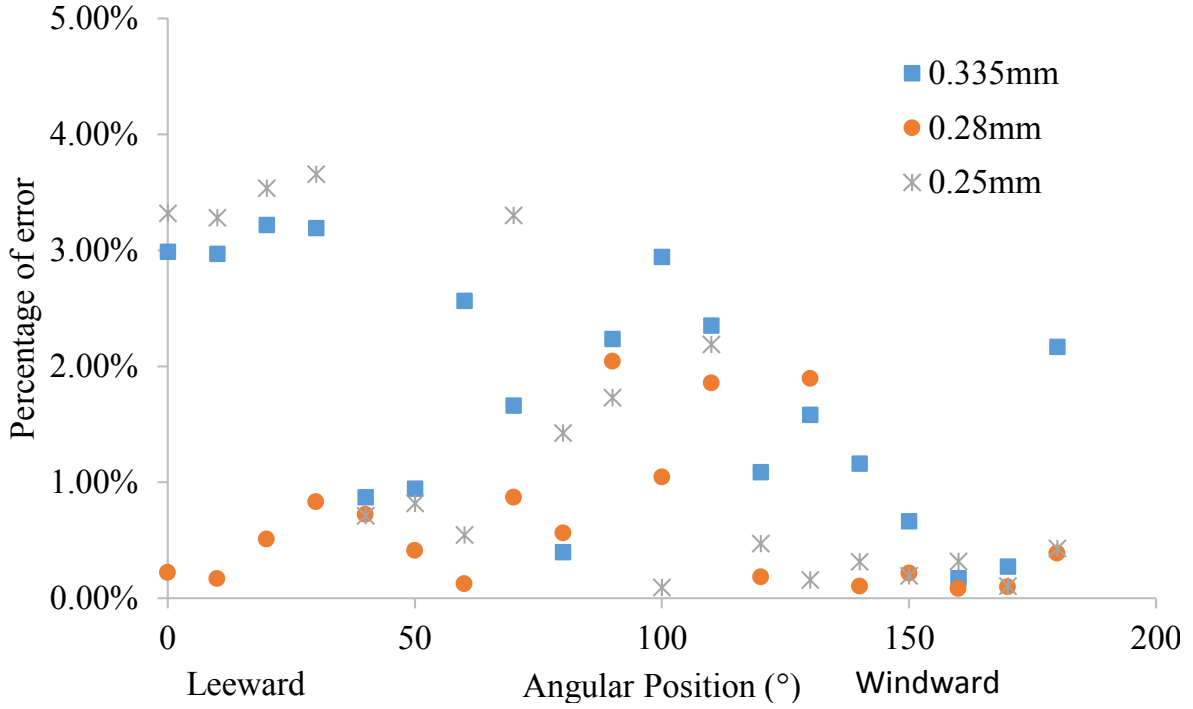


Figure 6.13: Pressure deviations relative to 0.228 mm local mesh size at different circumferential locations at 5 mm downstream the shoulder of the hemisphere-cylinder.

Both the qualitative and quantitative results for the hemisphere-cylinder indicates that the 0.25 mm mesh size was acceptable to proceed with analysis of the flow separation characteristics and properties.

The solutions for the local mesh refinement of the cone-cylinder are shown in figure 6.15. The nose separation and cross flow separation lines are similar where the surface streamlines accumulate at the flow separation regions. The skin friction coefficient distribution are similar where regions of greater skin friction coefficient, such as the nose, are captured with minimal discrepancies. The only discernible difference is the density of the streamlines between the 0.42 mm and 0.335 mm mesh size in the flow separation region downstream of the shoulder, at approximately 10 mm downstream of the shoulder. The variation of density for the surface streamlines likely existed due to the greater resolution of the surface mesh. The lack of discrepancies translate to minimal differences in the flow field, as seen in figure D.6 under Appendix D.1.

Quantitative results for the cone-cylinder were extracted at the cross section located 1 mm downstream of the shoulder. The shoulder was not chosen as the presence of the kink affected the accuracy of extracting the data due to irregular mesh patterns by the shoulder. The pressure deviation for the 1 mm-offset cross section around the circumference for the cone-cylinder is indicated in figure 6.14. The same deviation tolerance of 5% was applied for the cone-cylinder where there are indications that the 0.42 mm and 0.335 mm mesh did not comply with the criterion. The 0.42 mm mesh exceeds the limit between the range of 40°

and 60° which was the region where flow separation occurred, whereas the 0.335 mm mesh exceeds the limit at the windward side region.

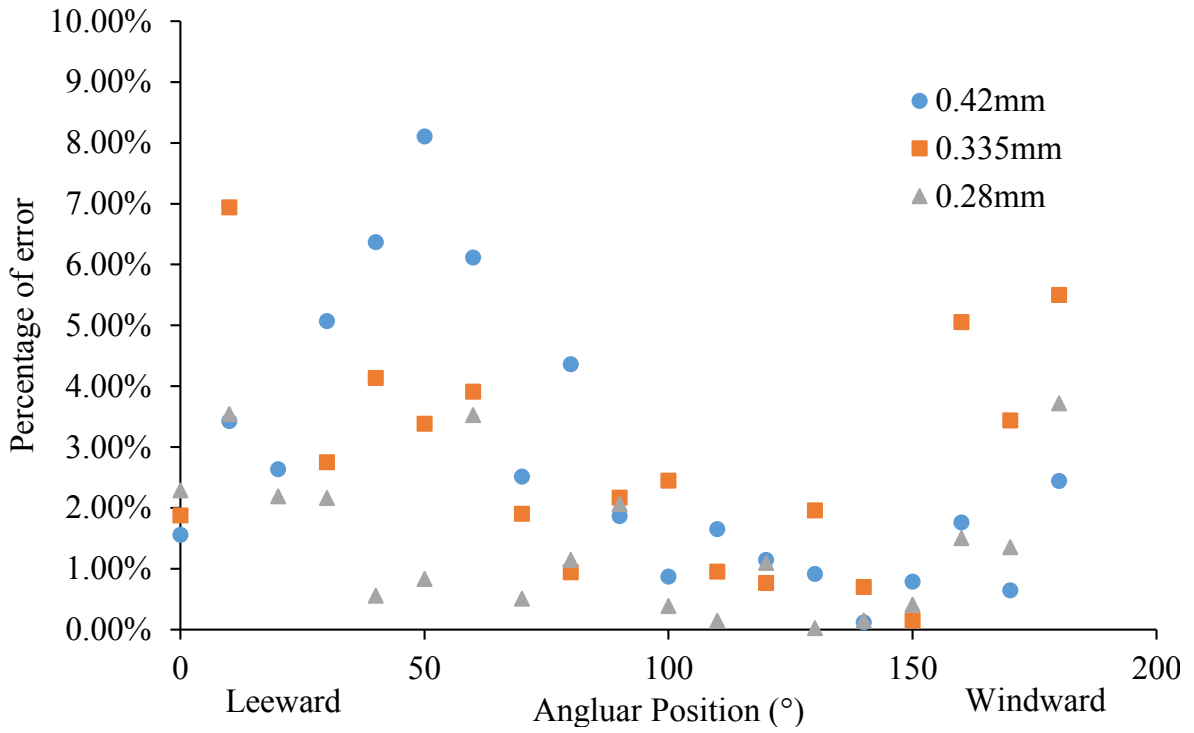


Figure 6.14: Pressure deviations relative to 0.25 mm local mesh size at different circumferential locations at 1 mm downstream the shoulder of the cone-cylinder.

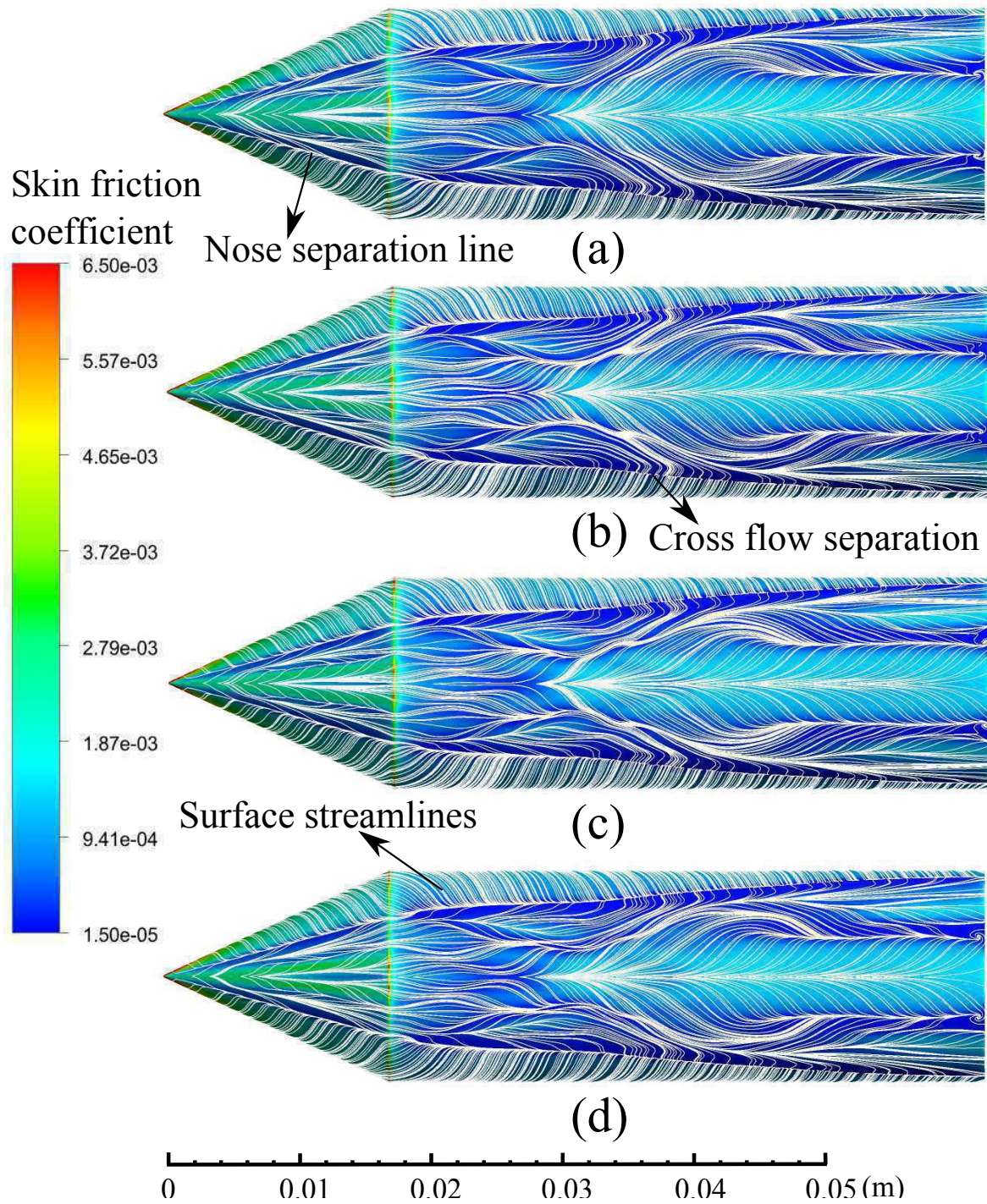


Figure 6.15: Wall shear surface streamlines and skin friction coefficient contours on the wall surface of the cone-cylinder with local mesh refinement settings: (a) 0.42 mm (120 circumferential cells), (b) 0.335 mm (150 circumferential cells) , (c) 0.28 mm (180 circumferential cells), (d) 0.25 mm (200 circumferential cells).

It can be deduced that the 0.28 mm mesh results are within an acceptable deviation tolerance for the quantitative comparison, hence has achieved mesh independence.

7 Results and Discussion

The findings from the experimental and computational methods of this research are presented in this section where comparisons are drawn in relation to the objectives of this research.

7.1 Experimental and CFD Validation

The validation of the CFD results with the experimental results are compared with two sets of results, one describing the flow field characteristics and the other describing the flow topology over the slender bodies. The comparison of the bodies at an angle of incidence of 30° at M3.2 flow conditions are considered. An assumption was made that validating for such configuration and flow condition was sufficient to validate configurations at incidences lower than 30° and Mach numbers lower than M3.2, while maintaining Reynolds numbers greater than 0.5×10^6 , for the same body shape. The assumption was based on the rationale that the conditions of 30° incidence and M3.2 freestream had more complex flow dynamics hence flow dynamics that were assumed to be less complex, by variation of incidence and flow condition, may be considered as valid.

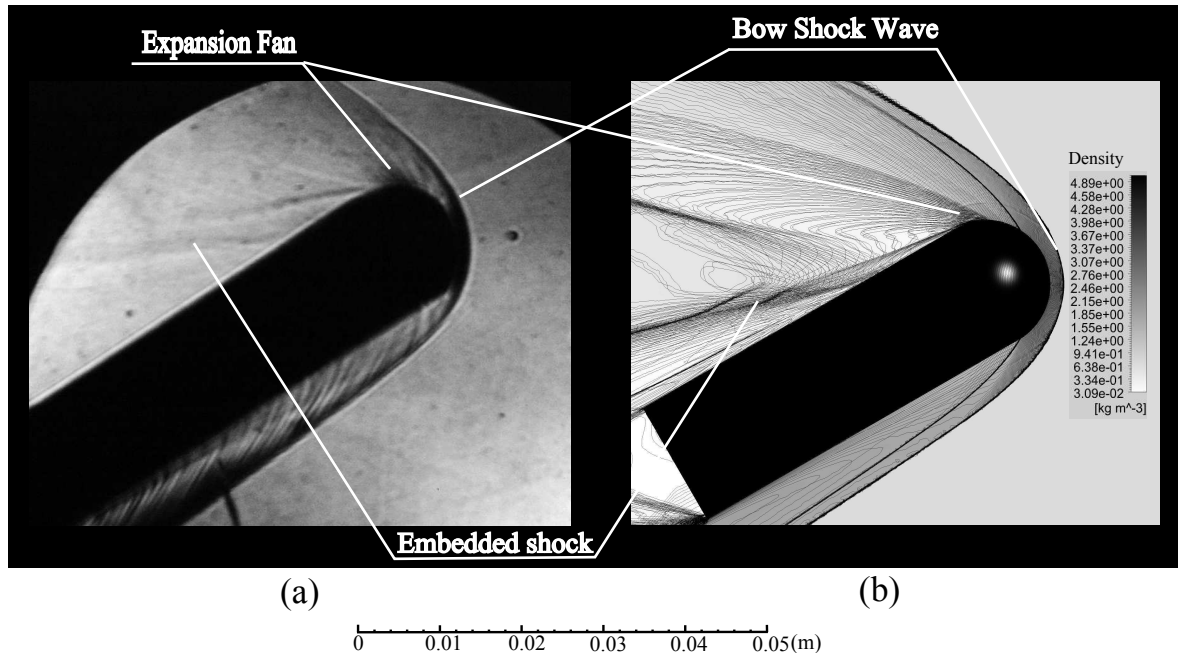


Figure 7.1: Comparison of the flow field features at M3.2 for the hemisphere-cylinder at an incidence of 30° presented as: (a) schlieren image with vertical knife edge for experimental results (b) density contours with multiple vertical planes for CFD results.

The comparisons of the flow field characteristics for the hemisphere-cylinder are seen in figure 7.1, which are represented as a schlieren image and density contours with selected multiple

vertical planes. The bow shock wave formed at the nose and expansion waves over the leeward shoulder for the CFD results are similar to the experimental results. Discrepancies are seen between the CFD and experimental results for the embedded shocks and multiple shock wave lines by the windward side of the cylinder. The embedded shock above the leeward side of the cylinder in the CFD results is steeper than the experimental results, which was affected by a slight increase in the incidence during the wind tunnel blow-down for the experimental results. The increased angle of the incidence was caused by the lack of stiffness in the structure of the assembly which allowed the hemisphere-cylinder to deflect during wind tunnel blow-down, however the increased incidence did not exceed 2° . The multiple shock wave lines on the windward side seen in the experimental results are caused by the 3D effects of the nose where each line corresponds to a local angle on the curvature of the nose. The CFD results only consists of two shock wave lines as the image consists of two planes that represent the shock wave features in order to reduce clutter from the contour lines in the image.

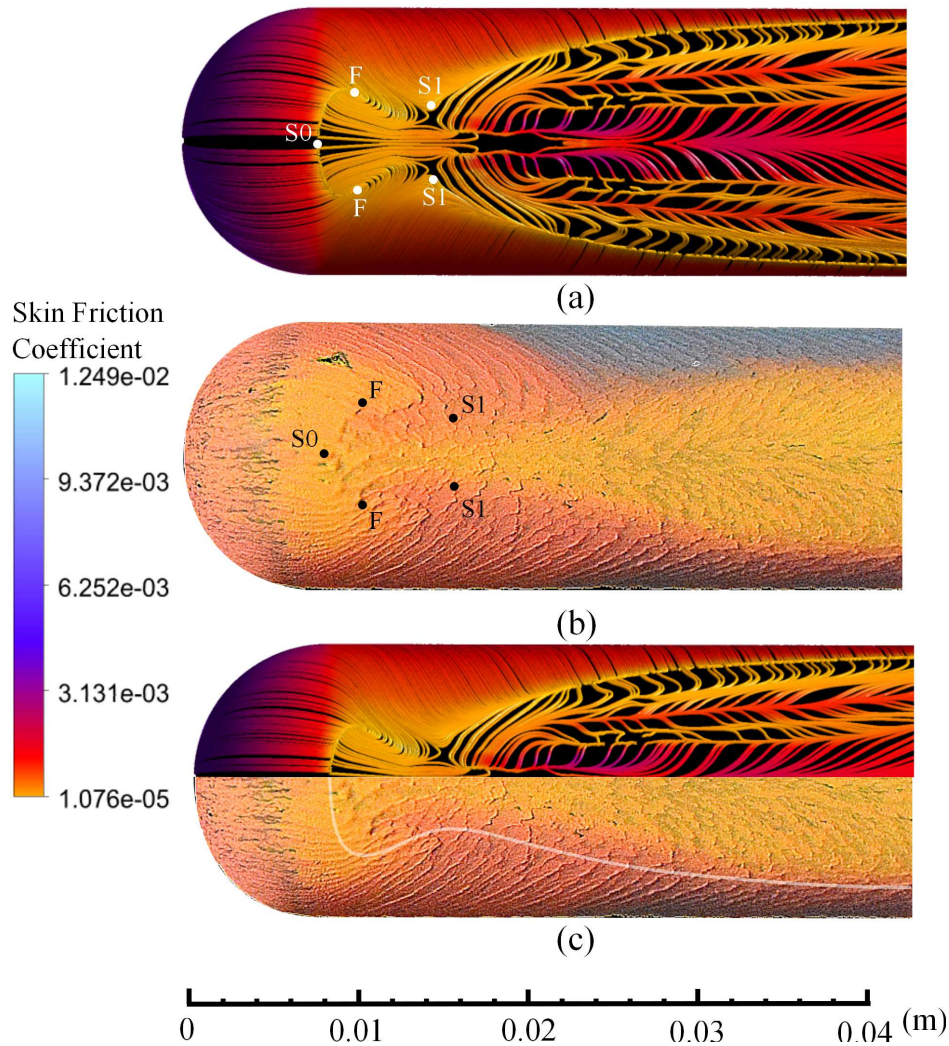


Figure 7.2: Flow topology of the hemisphere-cylinder pitched at 30° with M3.2 flow conditions presented as: (a) CFD solution with wall shear surface streamlines, (b) Experimental results as surface oil flow visualization, (c) Combination of both experimental and CFD results.

The comparisons of the flow topology for the hemisphere-cylinder at 30° incidence and M3.2 flow conditions are seen in figure 7.2. The CFD results indicate wall shear surface streamlines as skin friction coefficients with additional labelled points defining the features of flow separation occurring over the surface of the body. The surface oil flow visualization results are refined images with similar labelled points where both images represent the leeward side of the hemisphere-cylinder. The labelled points S_0 , F , and S_1 represents the separation point, focus point, and saddle point respectively. Each point in the experimental results was deduced from close observation of the streak patterns and accumulation of pigments of the oil mixture. The separation point was located by finding the region with a thick accumulation of pigments and the absence of streak patterns in such regions. The saddle point was located by following the converging streaks of pigment, specifically the cross flow pattern, until the lines become less distinct where the pigments are smudged or lines seemed parallel to the lateral plane. Foci points are located by considering regions with pigment accumulations and lightly patterned streaks which are present due to the nature of vortices. The clarity of the streaks by the foci points are poor as the wall shear forces in such regions are lower compared to windward side and sides of the hemisphere-cylinder. The ‘separation’ line, indicated as the white line in figure 7.2, was determined by outlining the areas where the pigment streaks have ceased.

The location of the separation and foci points for the CFD results are similar to the experimental results, where the formation of horn vortices are shown by the accumulation of the pigments. The saddle point for the experimental results are located approximately 2mm further downstream in comparison to the CFD results. The discrepancy of the saddle point locations are likely caused by the slight increase in the incidence. Discrepancies for the primary separation lines, white separation line and the perimeter of the separation region in the figure 7.2, are present where the experimental results are wider compared to the CFD results. However, the separation line in the experimental results correspond more closely to the red-orange boundary of the surface streamlines (In figure 7.2a) hence wall shear or skin friction coefficient decreases as flow approaches the symmetry plane from the windward side. The outlined ‘separation’ line in figure 7.2c indicates the wall shear threshold where the mixture was not able to create clear streaks due to the lack of wall shear forces in the region. This is seen more clearly in figure 7.3 where the 30° incidence and M2.8 flow condition was considered for the experimental results. The greater wall shear, contributed by an increased velocity through expansion waves, experienced at the nose resulted in the thinning of pigments near the shoulder. As flow separates, associated with lower velocities hence lower wall shear, pigments are accumulated at various regions on the body which varies in thickness depending on the severity of the flow separation. The roughly distributed pigments are regions indicating sufficiently low wall shear that prohibits the mixture from streaking with greater detail, and is a feature of initiation or potential flow separation. The skin friction coefficient distribution, as seen in figure 7.2a, is similar to the pigment thinning regions in the experimental results where the pigments in the separation region are evidently thinner, in figure 7.2b, than regions on the sides of the cylinder.

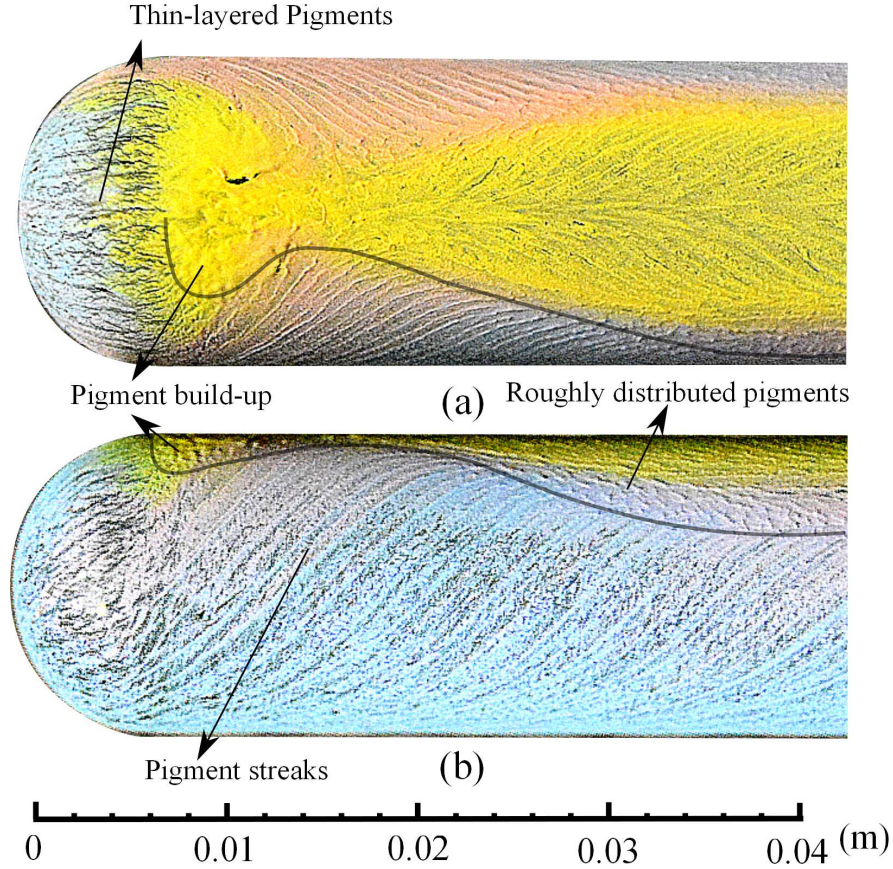


Figure 7.3: Surface oil flow visualization of the hemisphere-cylinder pitched at 30° with M2.8 flow conditions as seen from: (a) Top view of the leeward side, (b) Side view.

The similarities between the computational and experimental results for the hemisphere-cylinder are sufficient to prove validation for the simulated CFD results for this research. Discrepancies that exist are minor and have been elaborated to provide explanations of such discrepancies. It was necessary to consider CFD results, rather than experimental results, for further analysis as the information provide by experimental results are limited.

The comparisons of the flow field characteristics of the cone-cylinder at an incidence of 30° and M3.2 flow conditions are seen in figure 7.4. The shock waves generated from the tip of the nose in the CFD results are similar to experimental results where the shock wave is shown to be slightly stronger, due to a darker shade, at the windward side. Multiple shock wave lines emanating from the nose of the body, as seen from figure 7.4a, are an effect of 3D flow dynamics on the cone-cylinder, similar to the hemisphere-cylinder. The CFD results consist of three planes representing the shock wave lines that proves the 3D effect in the simulated model. Similar to the hemisphere-cylinder, the cone-cylinder experienced an incidence increase that potentially affected the results. A flow field discrepancy caused by the incidence increase was the embedded shock which appeared more steep in the CFD results than the experimental results. Another discrepancy was the absence of expansion waves in the experimental result which was likely caused by the sensitivity of the optics

set-up. Figure 7.4a indicates darker regions at the bottom of the image near the windward side, hence indicating an increased sensitivity. Whereas the region at the top of the image near the leeward side was much brighter which affects the visibility and sensitivity of density gradients. With a lower sensitivity at the leeward side, due to the uneven distribution of light intensity from the vertical knife edge set-up, the expansion waves were not clearly shown over the leeward shoulder from schlieren imaging.

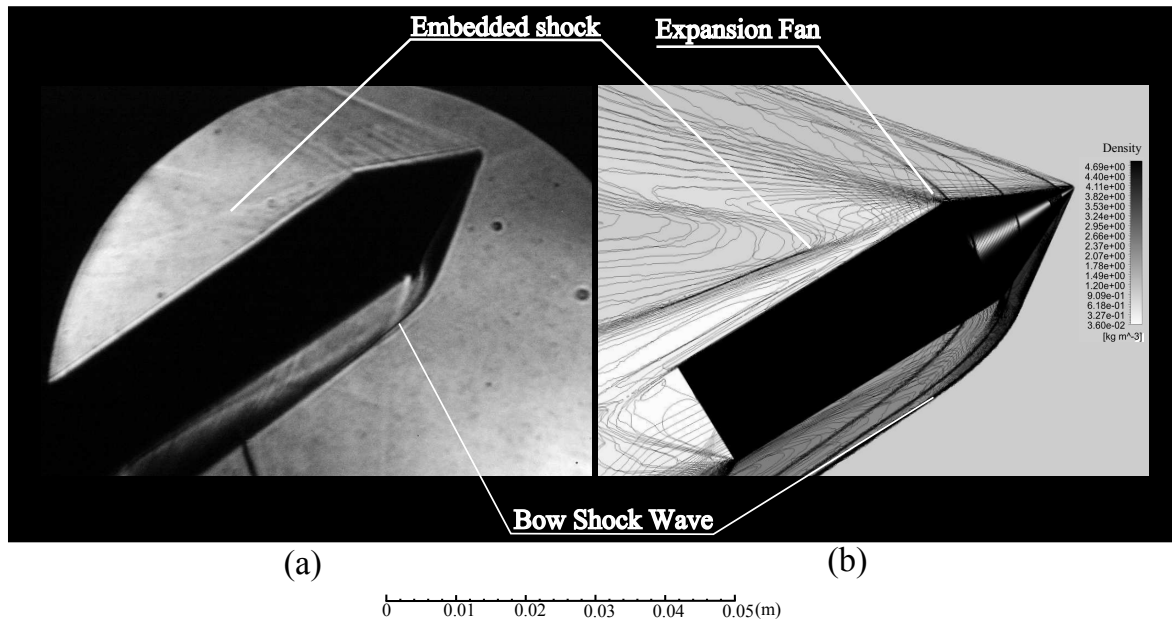


Figure 7.4: Comparison of the flow field features at M3.2 for the cone-cylinder at an incidence of 30° presented as: (a) schlieren image with vertical knife edge for experimental results (b) density contours with multiple vertical planes for CFD results.

The comparisons of the flow topology for the cone-cylinder at 30° incidence and M3.2 flow conditions is seen in figure 7.5. The flow separation characteristics over the cone-cylinder are relatively simple where separation line was mostly straight and was the main characteristics that can be compared for the experimental and CFD results. The separation line, indicated as the black line in figure 7.5b, was outlined by locating the termination of the pigment streaks on the leeward side. Figure 7.5c indicates a clear image of the flow separation line location which was deduced by locating the rough distribution of the pigments. The CFD results indicated similar flow separation lines to the experimental results where the location of the lines did not vary significantly despite the slight differences in the incidence. Pigment thinning at the shoulder of the cone-cylinder corresponds to the darker shaded areas of red at the shoulder in the CFD results, which was caused by an increase in flow velocity from expansion waves.

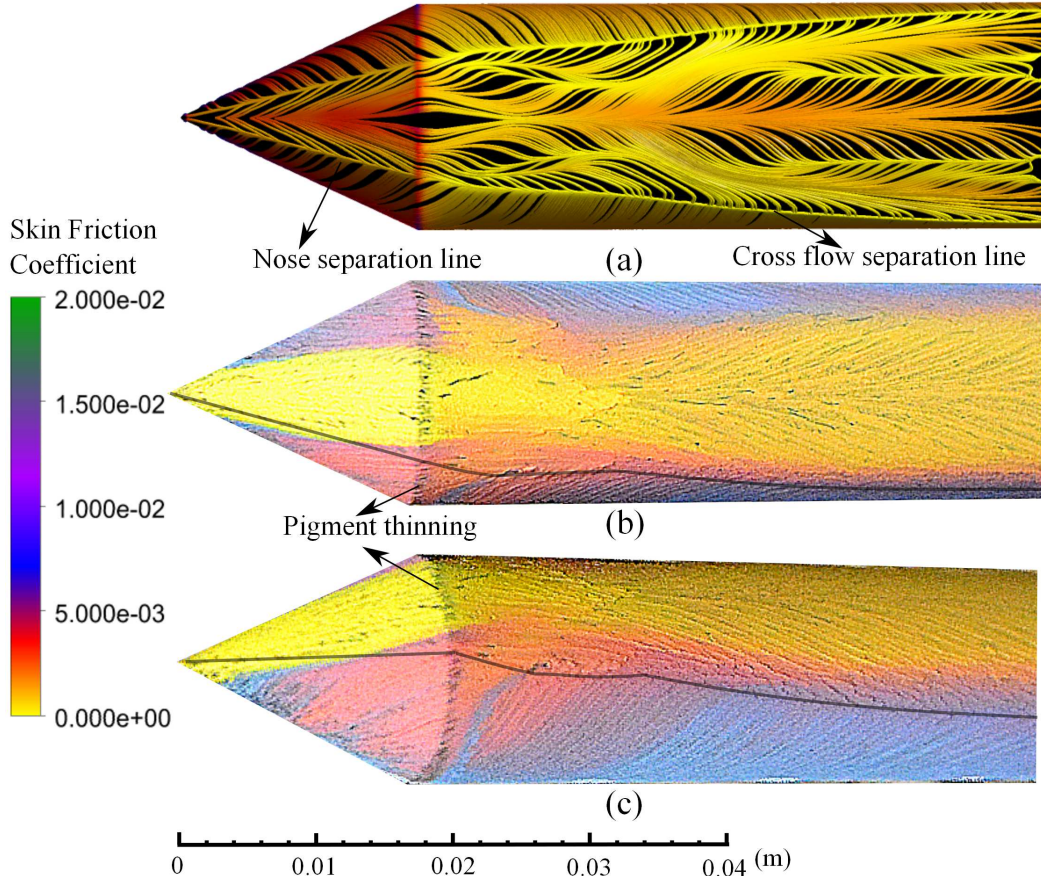


Figure 7.5: Flow topology of the cone-cylinder pitched at 30° with M3.2 flow conditions presented as: (a) CFD solution with wall shear surface streamlines, (b) Experimental results as surface oil flow visualization, (c) Surface oil flow visualization viewed from an oblique angle.

The similarities between the computational and experimental results for the cone-cylinder are sufficient to prove validation of the simulated CFD results for this research. Discrepancies that exists in the flow field features were minor and have been elaborated to provide explanations for the cause.

7.2 Flow Field Characteristics and Flow Topology

The characteristics of flow separation and supersonic flow are analysed, prior to geometric and flow condition comparisons, to establish an understanding of the mechanisms causing such phenomena. The configuration with the maximum angle of incidence and M3.2 flow conditions are considered for both the hemisphere-cylinder and cone-cylinder as the flow dynamics are assumed to be more complex. In the previous section, flow topology and density contours were introduced to determine the characteristics of the flow field and the flow separation features. Flow visualization results are presented in the form of 3D features

to elaborate on the flow separation features and supersonic flow dynamics that occur in flow separation regions.

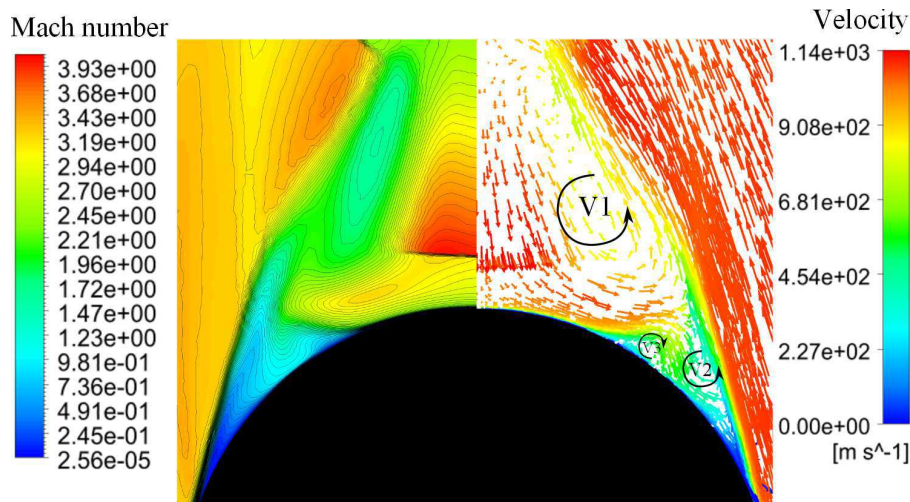


Figure 7.6: Mach number contours and velocity vectors at cross section 20mm from the shoulder for hemisphere-cylinder with 30° incidence and M3.2 flow conditions.

Prior to 3D results, cross-sectional planes along the longitudinal axes are studied to establish the flow dynamics and properties. Consider figure 7.6 which indicates the cross-sectional plane located at 20mm downstream of the shoulder of the hemisphere-cylinder. The low Mach number, blue regions, on the side and a portion of the top of the cylinder indicates flow separation. An additional weak shock, known as an embedded shock, is formed above the top of the cylinder. The vectors are able to indicate the trajectory of the flow in the cross-sectional plane, hence determining the direction of the developed vortices. The labelled vortex structures V_1 , V_2 , and V_3 are primary vortex, tertiary vortex, and the secondary vortex respectively. The primary vortex was formed by the primary separation contributed by the cross flow originating from the windward side of the hemisphere-cylinder. The secondary vortex was further developed by the flow trajectory of the anti-clockwise primary vortex impinging on the surface of the cylinder. The impinged flow is forced laterally hence forming the secondary vortex with a clockwise circulation. The presence of the embedded shock plays a significant role in the circulation of the secondary vortex which is discussed further in the comparisons section. An additional tertiary vortex was formed for the hemisphere-cylinder which is affected by flow features upstream near the shoulder, such as the flow separation and horn vortices on the surface. The basic structure of the flow field at a 2D cross-sectional plane enables one to determine the expected flow field downstream of the shoulder, hence it supplements the 3D results that will be considered next.

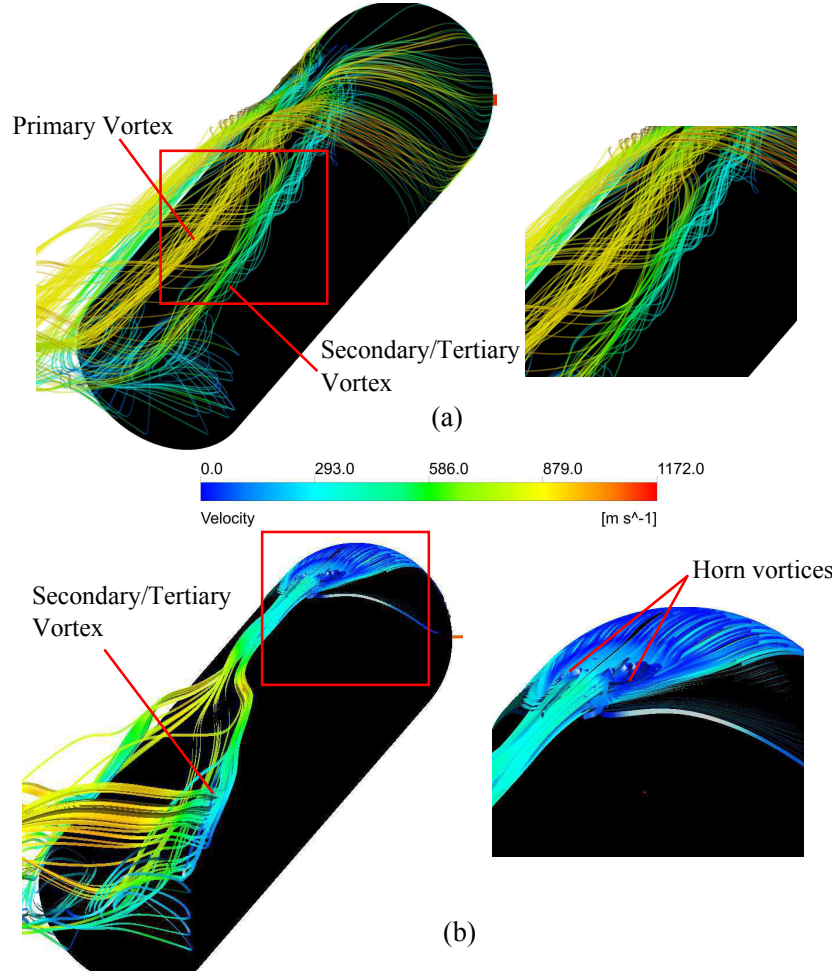


Figure 7.7: Flow field streamlines over hemisphere-cylinder at 30° incidence and M3.2 indicating:(a) Vortex structure downstream of shoulder, (b) Horn vortex structures by shoulder.

The flow field streamlines for the hemisphere-cylinder at 30° incidence and M3.2 flow condition are indicated in figure 7.7. The streamlines provides the flow structure that allows one to establish the location of various flow features such as vortices. The primary vortex, which are the yellow streamlines in figure 7.7a, are induced by cross flow separation where the streamlines originate near the windward side shoulder. The primary vortex experiences higher velocities which is contributed by the flow expansion occurring over the leeward side of the cylinder. The secondary and tertiary vortices are combined for this geometric configuration and are seen in figures 7.7a and 7.7b as green-blue streamlines. The mixing of the streamlines are caused by nose flow separation and horn vortices located upstream. The horn vortices developed by the shoulder remains relatively close to the surface of the body and do not shed laterally. The streamlines that form the horn vortices tend to follow a trajectory parallel to the symmetry plane first then converges towards the secondary/tertiary vortices further downstream. The formation of the secondary and tertiary vortices are elaborated when considering the flow topology structures in the parameter comparisons section.

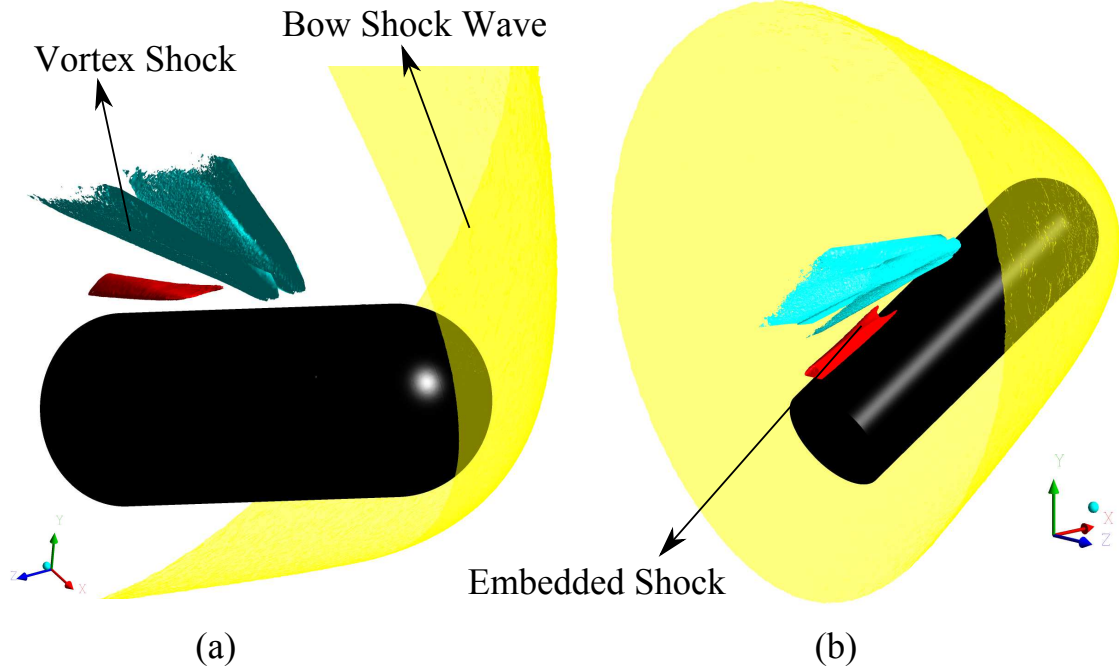


Figure 7.8: 3D density iso-surfaces of supersonic flow features over the hemisphere-cylinder at 30° incidence and M3.2 flow conditions viewed from: (a) A front oblique view, (b) A rear oblique view.

The flow separation and supersonic flow features indicated as 3D iso-surfaces are indicated in figures 7.8 and 7.9 respectively. The iso-surfaces for both set of results were configured to correspond to multiple cross-sectional planes along the longitudinal axis of the hemisphere-cylinder where the cross-sectional planes are indicated in figures E.1 and E.3 under Appendix E. As indicated in figure 7.8, a spherical bow shock wave is formed by combination of high incidence angle and spherical nose. The flow velocity downstream of the bow shock maintains supersonic speeds, with exception to the stagnation region at the front of the nose where a sonic line exists. Despite the flow separation occurring over the shoulder of the hemisphere-cylinder, the flow velocity on the leeward side is able to increase its velocity by expansion and form an embedded shock downstream of the shoulder on the leeward side. The vortex shock formed further above the embedded shock originates from the combination of the primary vortex and flow expansion. The primary vortex acts as a flow boundary where compression is experienced as flow travels downstream of the shoulder. The vortex shock weakens further downstream as the primary vortex is lifted from the leeward side surface and flow expansion gradually dissipates. The surface shape of the vortex shocks are angled inwards towards the symmetry plane, which indicates the shocks are formed on the outer surfaces of the primary vortices where compression of the cross flow occurs.

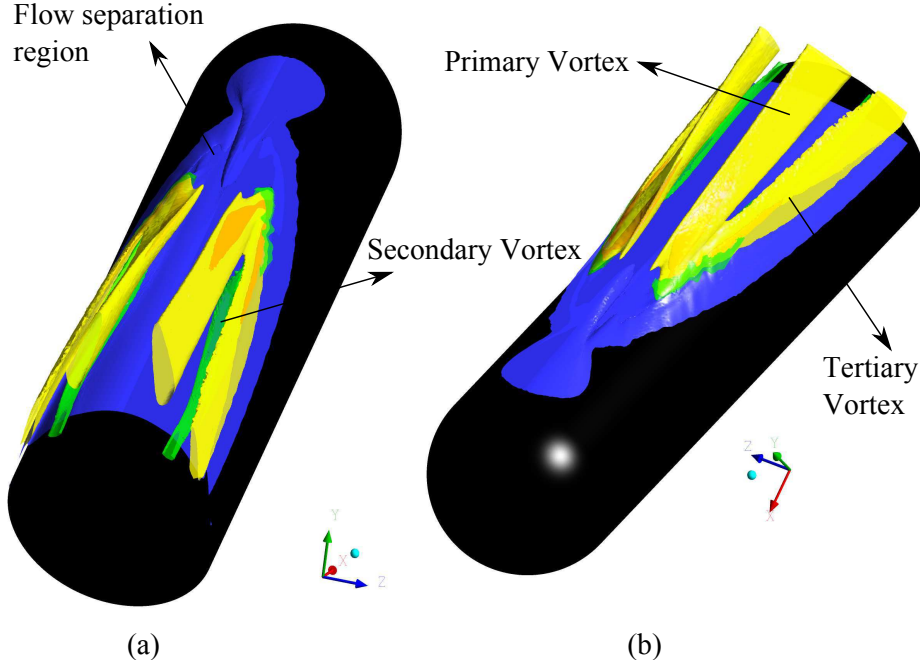


Figure 7.9: 3D Mach number and turbulence intensity iso-surfaces of flow separation features over the hemisphere-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A rear oblique view, (b) A front oblique view.

3D iso-surfaces for the hemisphere-cylinder at 30° incidence are indicated in figure 7.9. The flow separation regions are representations of regions with low Mach number and relatively high turbulent conditions. The saddle-shaped flow separation regions at the shoulder are indications of strong attached cross flow that contributes to the formation of vortex structure at the shoulder. The vortical structures were based on turbulence intensity iso-surfaces that were selected with reference to the velocity and pressure contours. The primary vortices are seen to shed from the leeward side downstream of the saddle-shaped flow separation region where the orange region seen in figure 7.9a indicate regions with greater turbulence intensity. The region of greater turbulence intensity does not extend downstream due to the presence of the embedded shock that reduces the flow velocity in the primary vortex. The secondary vortex was initially formed as the primary vortex was formed near the strong turbulence intensity region. However, as flow travels downstream, the secondary vortex was influenced by the formation of the tertiary vortex. The presence of the tertiary vortex further affects the diameter of the secondary vortex filament to accommodate a large elongated tertiary vortex filament next to the cross flow separation line. The presence of multiple vortex structures above the leeward side results in complex flow patterns in the flow separation region. This is characterised by multiple separation lines on the surface.

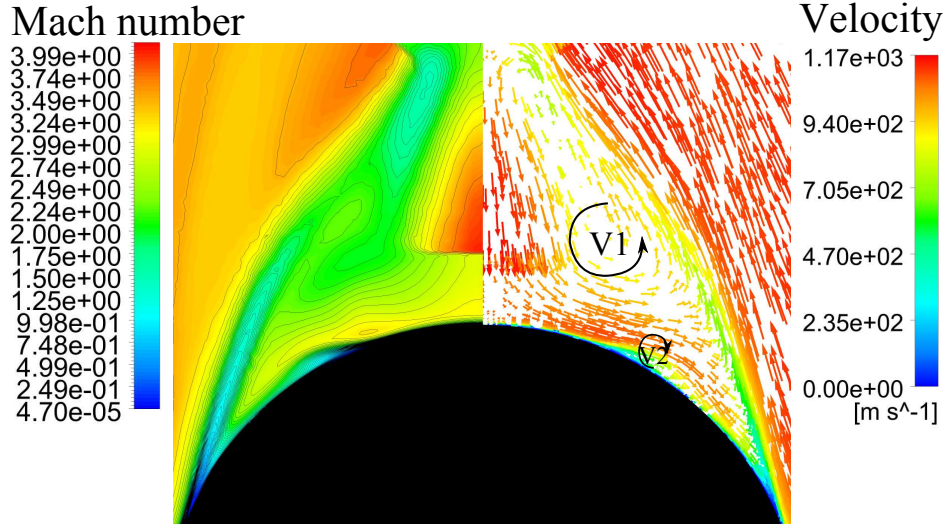


Figure 7.10: Mach number contours and velocity vectors at cross section 35mm from the shoulder for cone-cylinder with 30° incidence and M3.2 flow conditions.

The cross-sectional plane, located at 35mm from the shoulder, for the cone-cylinder at 30° incidence and M3.2 flow condition is indicated in figure 7.10. The specified cross-sectional plane was selected as cross-sectional planes upstream by the shoulder did not indicate distinct characteristics of the secondary vortex, hence indications of flow characteristics with less complexity compared to the hemisphere-cylinder. The embedded shock wave and primary vortex are distinct where the maximum Mach number above the top of the cylinder is above the freestream velocity of M3.2. A secondary vortex is present but does not seem to be fully developed as the velocity in the separated region is relatively high due to expansion.

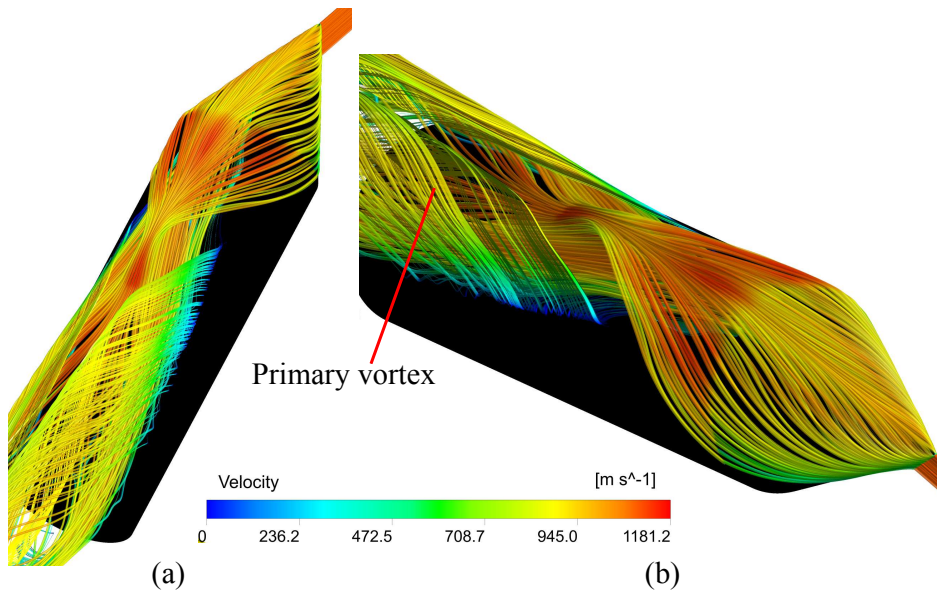


Figure 7.11: Flow field streamlines over cone-cylinder at 30° incidence and M3.2 flow conditions shown at: (a) A rear oblique view, (b) A frontal oblique view.

The 3D streamlines for the cone-cylinder at 30° incidence and M3.2 flow conditions is indicated in figure 7.11. The dense streamlines travelling from the nose of the cone experience severe cross flow where flow tends to converge to the symmetry plane, which the vortical singularity point exists in the flow field. The streamlines eventually diverges laterally downstream where it forms part of the primary vortex induced by the cross flow separation from windward flow travelling towards the leeward side.

The flow separation and supersonic flow features for the cone-cylinder at 30° are indicated in figures 7.12 and 7.13 as 3D iso-surface. Similar to the hemisphere-cylinder, the iso-surfaces were constructed from a combination of pressure, density, and turbulence intensity iso-surfaces with reference to multiple cross-sectional planes as seen in figures E.2 and E.4 in appendix E. Figure 7.12 indicates the iso-surfaces for the cone-shaped shock wave, embedded shock and vortex shock. The cone-shaped shock does not form a full revolution as the 30° angle of incidence exceeds the cone deflection angle of 25° , which causes a portion of the revolution to remain as compression waves. Hence the flow over the leeward side of the nose are able to travel at a greater velocity without the presence of a shock wave. The embedded shock and vortex shock are present due to flow expansion over the leeward side and the formation of the primary vortex respectively.

The 3D flow separation features are indicated in figure 7.13. The flow separation region indicates flow at low Mach number, which is considerably less for the hemisphere-cylinder. This was contributed by the greater velocity conditions experienced over the leeward side and the absence of a tertiary vortex structure which decreases the flow velocity in local regions. Flow separation occurs at the nose where the 25° meridional plane is collinear with the nose flow separation line. The immediate flow separation occurring at the nose is contributed by the high flow velocity over the leeward side of the nose which has sufficient energy to initiate flow separation at the nose. The mechanism of flow separation over the cone-cylinder at different angles of incidence is elaborated in the next section where the flow topology provides evidence of the initiation of flow separation in relation to the incidence. The primary vortex is developed further downstream of the shoulder where the main contributors are the cross flow separation and the converging flow on the leeward side of the cone-cylinder. The primary vortex is elongated with a turbulence intensity that is much lower compared to the hemisphere-cylinder. As flow travels downstream, the secondary vortex is developed where the flow impinging on the leeward side has gathered sufficient velocity to induce secondary flow separation.

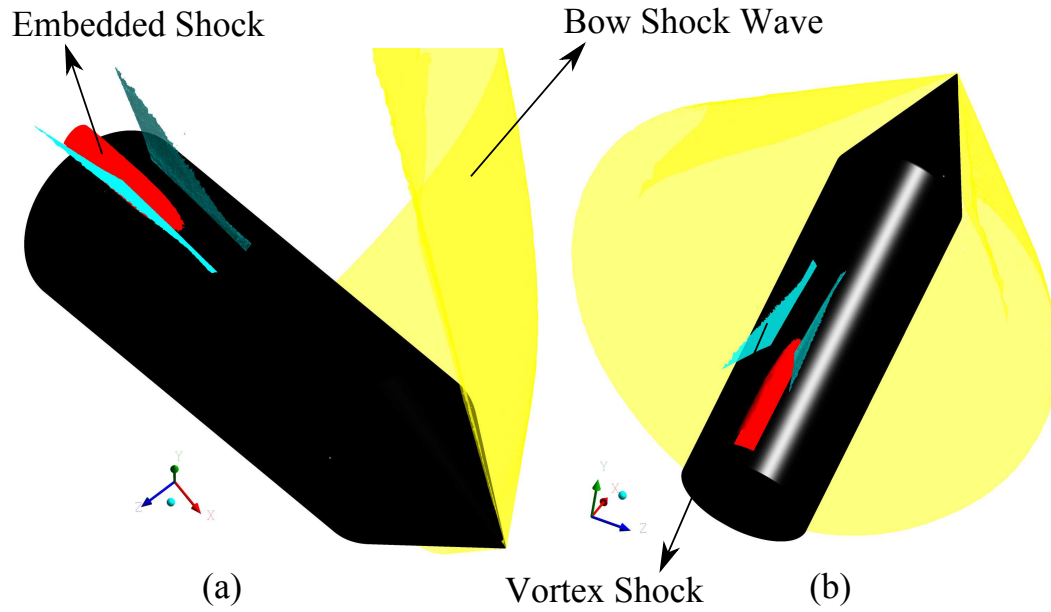


Figure 7.12: 3D density iso-surfaces of supersonic flow features over the cone-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A front oblique view, (b) A rear oblique view.

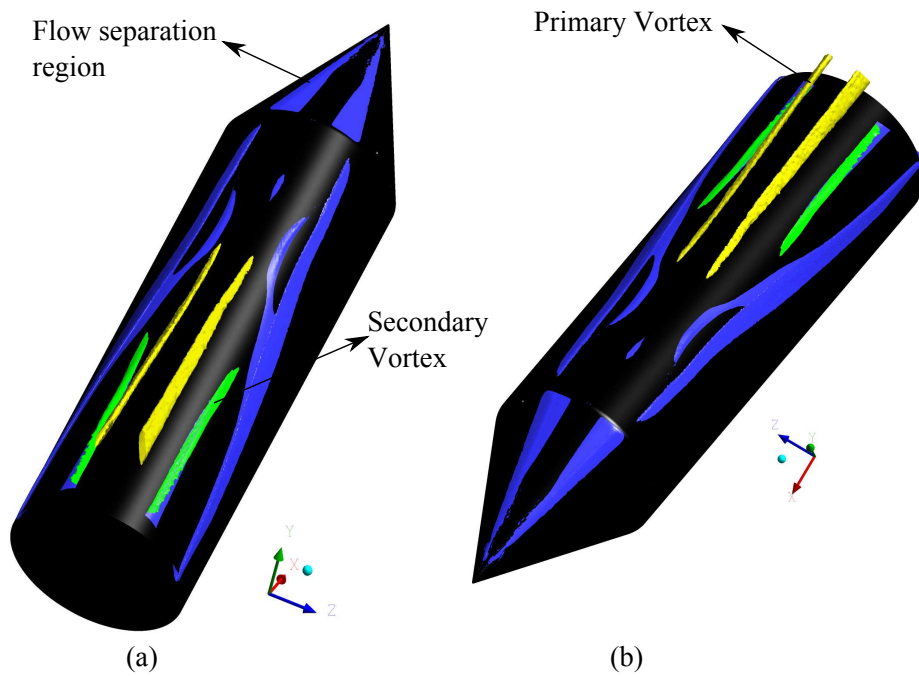


Figure 7.13: 3D Mach number and turbulence intensity iso-surfaces of supersonic flow features over the cone-cylinder at 30° incidence and M3.2 flow conditions viewed from:(a) A rear oblique view, (b) A front oblique view.

7.3 Nose Geometry Influences on the Flow Separation Characteristics and Properties

The results presented in this section are specifically aimed at flow separation characteristics, hence comparisons of the geometric parameters, and later the Mach number parameter, are constrained to flow topology and the cross sections indicating the Mach number and velocity vectors. The selected types of result indicates the most distinct characteristics of flow separation. For ease of discussion, the terms high, medium, and low angles of incidence refers to 30° , 20° , and 10° of incidence.

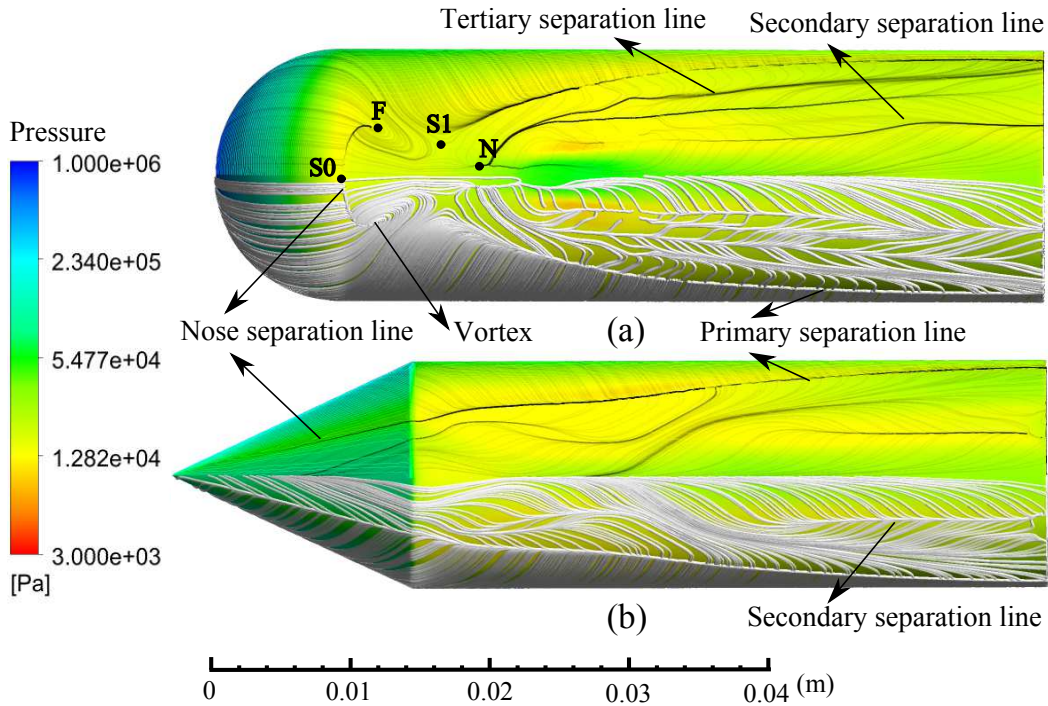


Figure 7.14: Surface streamlines and pressure contours for: (a) hemisphere-cylinder, (b) cone-cylinder, at 30° angle of incidence and M3.2 flow conditions.

The comparisons of the hemisphere-cylinder and cone-cylinder at the same high angles of incidence and Mach numbers are indicated in figures 7.14 and 7.15. Brief comparisons were mentioned in the previous section, hence additional details for the nose shape comparisons are considered. The locations of flow separation near the shoulder are a distinct difference which caused several features of the flow topology to be affected as seen in figure 7.14. The flow separation at the shoulder of the hemisphere-cylinder resulted in the formation of flow separation features such as focus (F), saddle (S_1), and reattachment (N) points on the surface of the body. The presence of the saddle and reattachment points are significant as it contributes to the formation of the tertiary vortex, which corresponds to the tertiary separation line, seen in figure 7.14a. The typical characteristics of the saddle and reattachment points include converging and diverging streamlines. The saddle point is located at the starting point of

the cross flow separation hence can be interpreted as the interface of the nose flow separation and cross flow separation. The reattachment point indicates streamlines converging from the saddle point, hence forming an alternate flow system that affects the flow dynamics in the separated region. Both tertiary and secondary separation lines emanate from the reattachment point hence indicating features of a tertiary and secondary vortex systems. The cone-cylinder does not consist of flow separation points by the shoulder and has a pair of separation lines emanating from the tip of the cone, which can be considered as the separation point. The streamlines in the separated region are wavy where the formation of a secondary separation line occurs further downstream of the shoulder. The pressure distribution over leeward side of the hemisphere-cylinder consists of darker shades of green along the symmetry plane region, indicating slightly higher pressures compared to the cone-cylinder. The region with multiple separation lines on the leeward side of the hemisphere-cylinder also increases the pressure distribution, despite experiencing lower velocities from the tertiary vortex system.

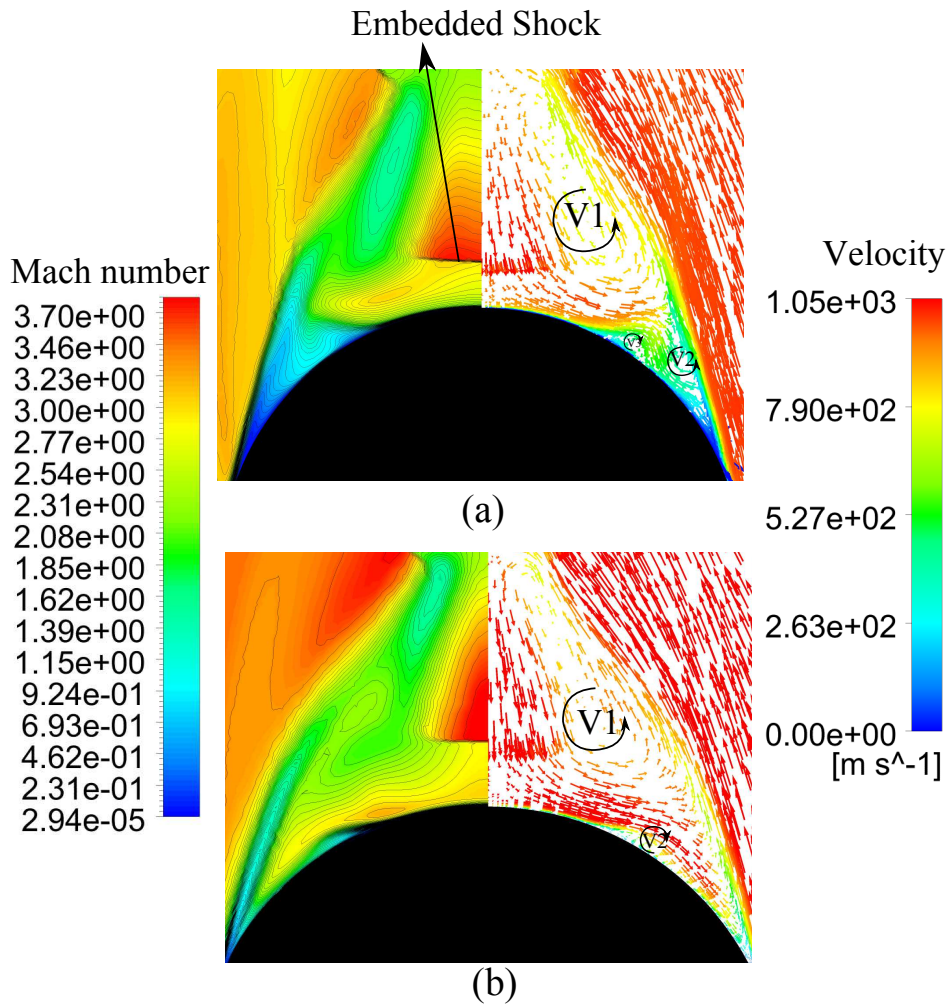


Figure 7.15: Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of: (a) hemisphere-cylinder, (b) cone-cylinder, at 30° incidence and M3.2 flow condition.

The flow field comparison of the hemisphere-cylinder and cone-cylinder at high incidence taken at the cross-section 35mm from the shoulder is indicated in figure 7.15. The presence of the tertiary vortex for the hemisphere-cylinder has a distinct effect on the Mach number regions where the flow velocity is greatly decreased in magnitude. The absence of the tertiary vortex in the cone-cylinder allows flow to maintain relatively high supersonic flow conditions. The embedded shock is present for both slender body types where the embedded shock for the hemisphere-cylinder is slightly laterally longer. The embedded shock affects the secondary vortex by reducing the flow velocity that approaches the surface. Hence the reduced flow velocity resulted in a reduced vortex size which may favour conditions for the tertiary vortex. This is apparent for the hemisphere-cylinder as the tertiary vortex (V_2) seems much larger in size in comparison to the secondary vortex (V_3). The region above the embedded shock for the cone-cylinder consists of a darker shade of red in comparison to the hemisphere-cylinder, indicating a region with greater Mach number flow. This is affected by the higher flow velocity over the leeward side due to the presence of compression waves on the leeward nose.

7.4 Body Attitude Influences on the Flow Separation Characteristics and Properties

The variation in the angle of incidence is considered in this section where it is one of the vital parameters that affected the flow separation characteristics. Selected angles of incidence are considered as some configurations, such as low angles of incidence, did not contribute sufficiently to the discussion due to the absence of nose flow separation. The results for 10° angle of incidence are seen in figures E.5 and E.6 in Appendix E.

The comparisons of the hemisphere-cylinder at high and intermediate angles of incidence with the same flow conditions are indicated in figures 7.16 and 7.17. Prior to the flow topology and flow field conditions, it was recognized that the shock wave over the hemisphere-cylinder at intermediate incidence encounters a weaker bow shock at the windward side of the cylinder compared to the high incidence configuration. The flow topology for the hemisphere-cylinder at different angles of incidence is indicated in figure 7.16, where the differences in flow patterns over the surface are distinct. The saddle point for the intermediate incidence is located further downstream of the shoulder and almost in-line with the reattachment point. The location of the saddle point for the intermediate incidence configuration is partially contributed by cross flow trajectory, which is affected by the lower angle of incidence. The reattachment points are situated at similar locations for both incidences, despite the difference in the saddle point location. The nose flow separation line for the intermediate incidence configuration is narrower compared to the high incidence configuration, hence forming a smaller separated region. The nose separation line is also located further downstream of the shoulder, which is indicative that the nose flow separation is delayed with a decrease of incidence. The presence of the saddle and reattachment points correspond to the presence of a secondary and

tertiary vortex system, which are evident from the existence of both secondary and tertiary separation lines seen on the leeward side. Foci points are not seen for the intermediate incidence configuration as the vortices near the shoulder are not developed, hence there are fewer tertiary separation lines in the separated region. The pressure distribution on the leeward side for the intermediate incidence configuration are greater and relatively uniform along the symmetry plane. Hence the pressure distribution in the separated region tends to be more predictable at lower angles of incidence.

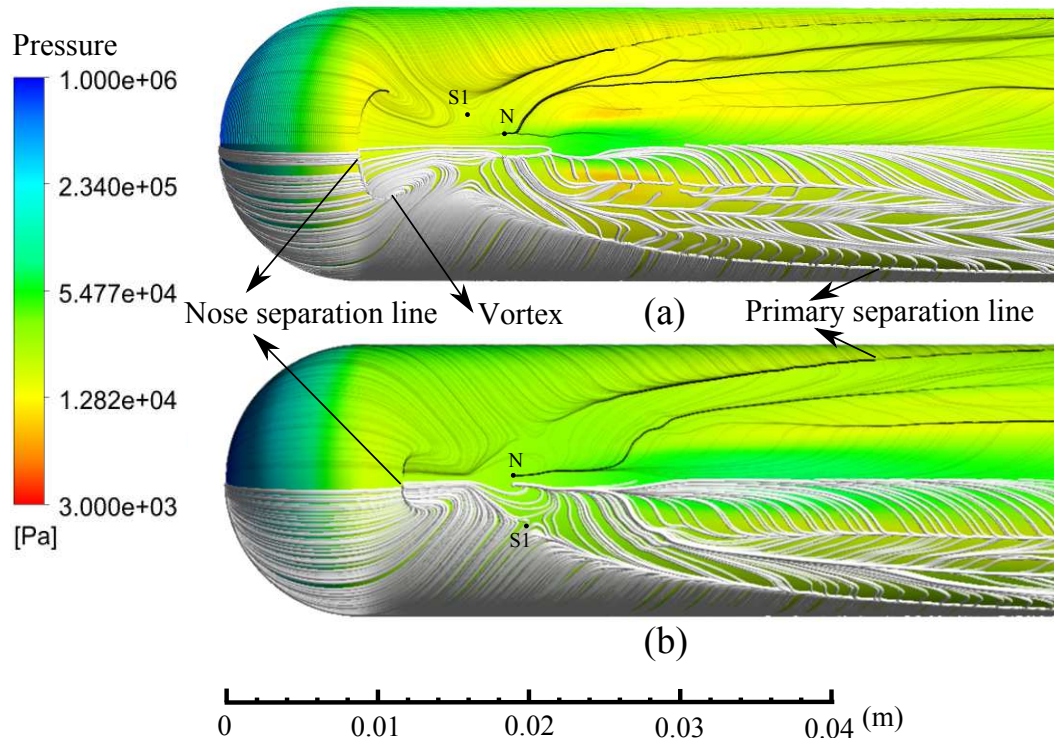


Figure 7.16: Surface streamlines and pressure contours over of hemisphere-cylinder at M3.2 conditions and: (a) 30° angle of incidence (b) 20° angle of incidence.

The flow field comparison of the hemisphere-cylinder high and intermediate incidences taken at the cross-section 35mm from the shoulder is indicated in figure 7.17. The primary vortex and embedded shock are the main differences in the flow field for variations in the incidence. The primary vortex for the intermediate incidence configuration is more circular compared to the more elongated primary vortex in the high incidence configuration. The circular-shaped vortex results in the formation of smaller secondary and tertiary vortex systems as less flow is directed downwards towards the leeward side of the surface. The embedded shock is absent for the intermediate incidence configuration as flow expansion is not as severe compared to the high incidence configuration. Instead of a flow with a lower velocity affected by the embedded shock, flow on the leeward side impinges on the surface along the symmetry plane, hence resulting in regions of lower Mach number for the intermediate incidence configuration. The lowered Mach number corresponds with the increased pressure distribution in the separated region.

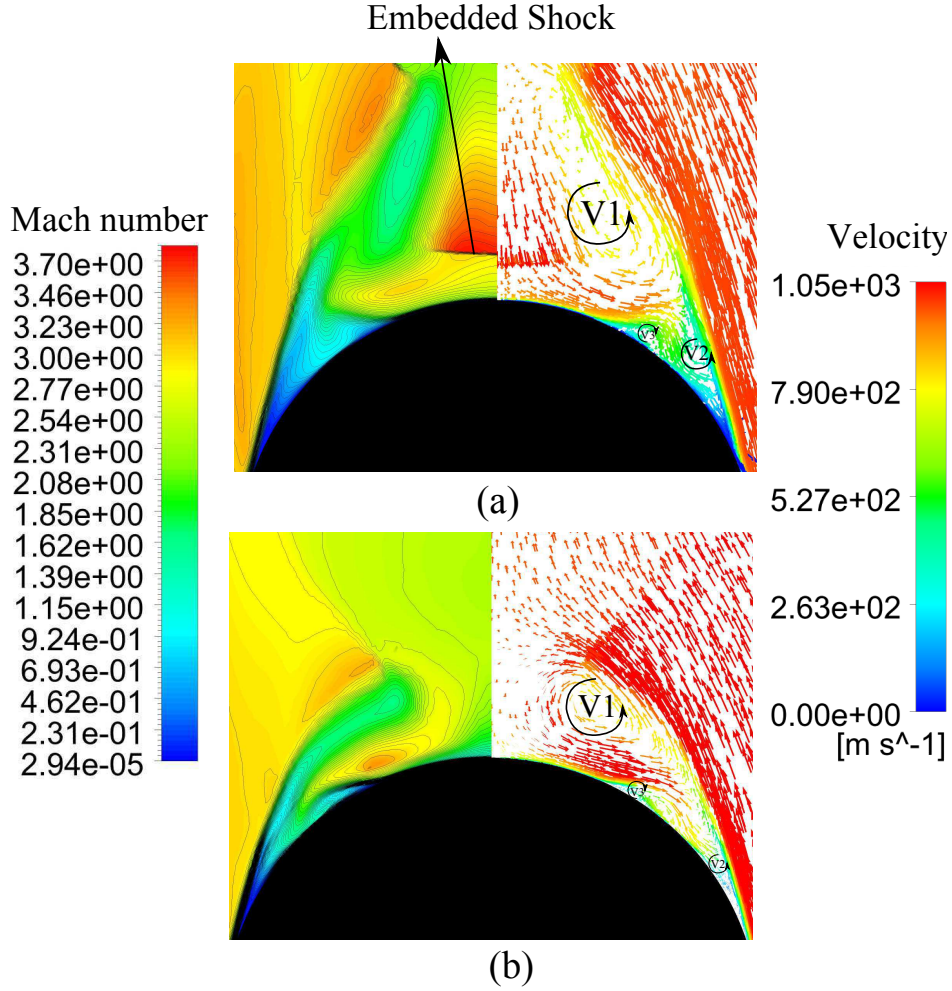


Figure 7.17: Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of the hemisphere-cylinder at M3.2 flow conditions and: (a) 30° incidence, (b) 20° incidence.

The comparisons of the flow topology for the cone-cylinder at high and intermediate angles of incidence with the same flow conditions are indicated in figure 7.18. The distinct difference for the intermediate and high incidence configurations is the presence of the nose separation line. The dense streamlines over the intermediate incidence configuration of the cone-cylinder are indicative that flow separation over the nose is non-existent. However, cross flow separation remains where S_0 indicates the origin of the cross flow separation point. Cross flow separation remains present for the intermediate incidence configuration as the incidence was sufficient to allow flow to separate around the cylinder. It is relevant to recall the high incidence configuration of the cone-cylinder where a full-revolution shock wave was not encountered as the leeward side of the nose experienced a compression wave rather than a shock wave. The significance of the compression waves on the leeward side of the cone enabled the formation of flow separation over the nose of the cone. The intermediate incidence configuration encountered a full-revolution cone shock as the maximum effective cone deflection angle maintains a compression deflection i.e. positive angle of deflection relative

to freestream vector. Although a weaker shock solution is encountered at the leeward side of the cone for the intermediate incidence configuration, the flow downstream of the shock decreased sufficiently to inhibit flow separation over the nose. Hence flow over the nose was unable to separate from the surface for intermediate angles of incidence for cone-cylinders with a cone deflection angle of 25° . The secondary separation lines are similar for different incidence configurations, which is indicative that the secondary vortex systems are mainly dependent on the cross flow separations.

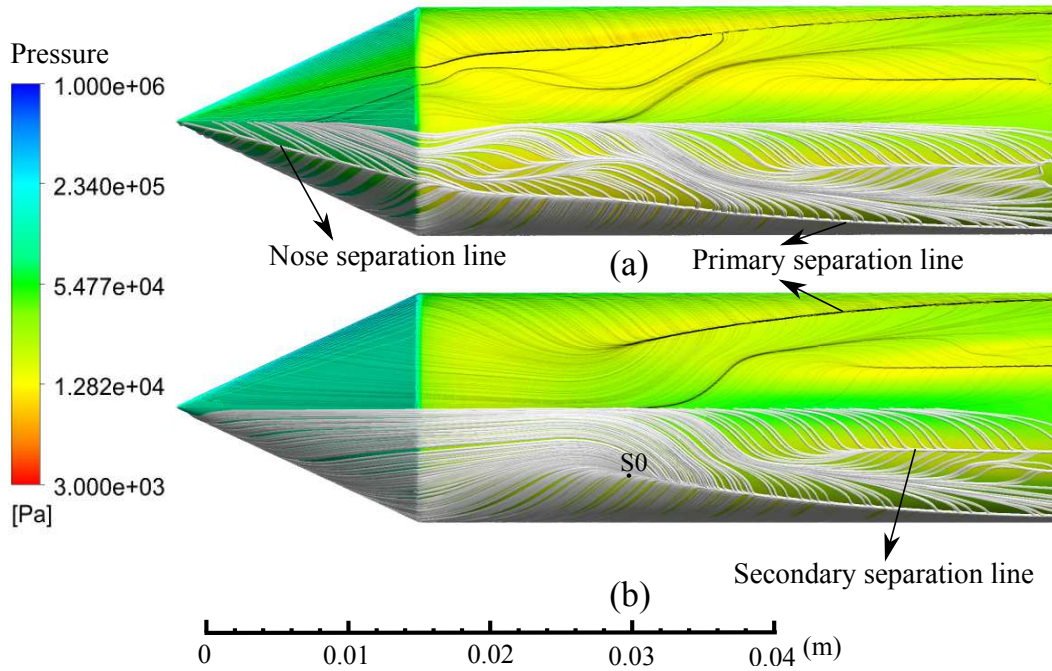


Figure 7.18: Surface streamlines and pressure contours over of cone-cylinder in M3.2 conditions and: (a) 30° angle of incidence (b) 20° angle of incidence.

The comparisons of the flow field characteristics for the cone-cylinder at high and intermediate angles of incidence with the same flow conditions are indicated in figure 7.19. Although the flow topology results indicated that the secondary and tertiary separation lines were similar in the flow separated region for the intermediate and high incidence configurations, the vortex systems were noticeably different above the leeward side of the cone-cylinder at different incidences. The primary vortex system is more circular for the intermediate incidence configuration where cross flow separation region is smoothly merged with the primary vortical structure. The secondary vortex structure in the intermediate incidence configuration is subtly greater in diameter, despite having a slightly lower Mach number and flow velocity directly above the leeward side of the cylinder. However, the circular-shaped primary vortex of the intermediate incidence configuration prevails over the greater velocity conditions as the flow vectors were more horizontal, which promotes the flow to shear over the surface more frequently instead of impinging at the leeward side surface.

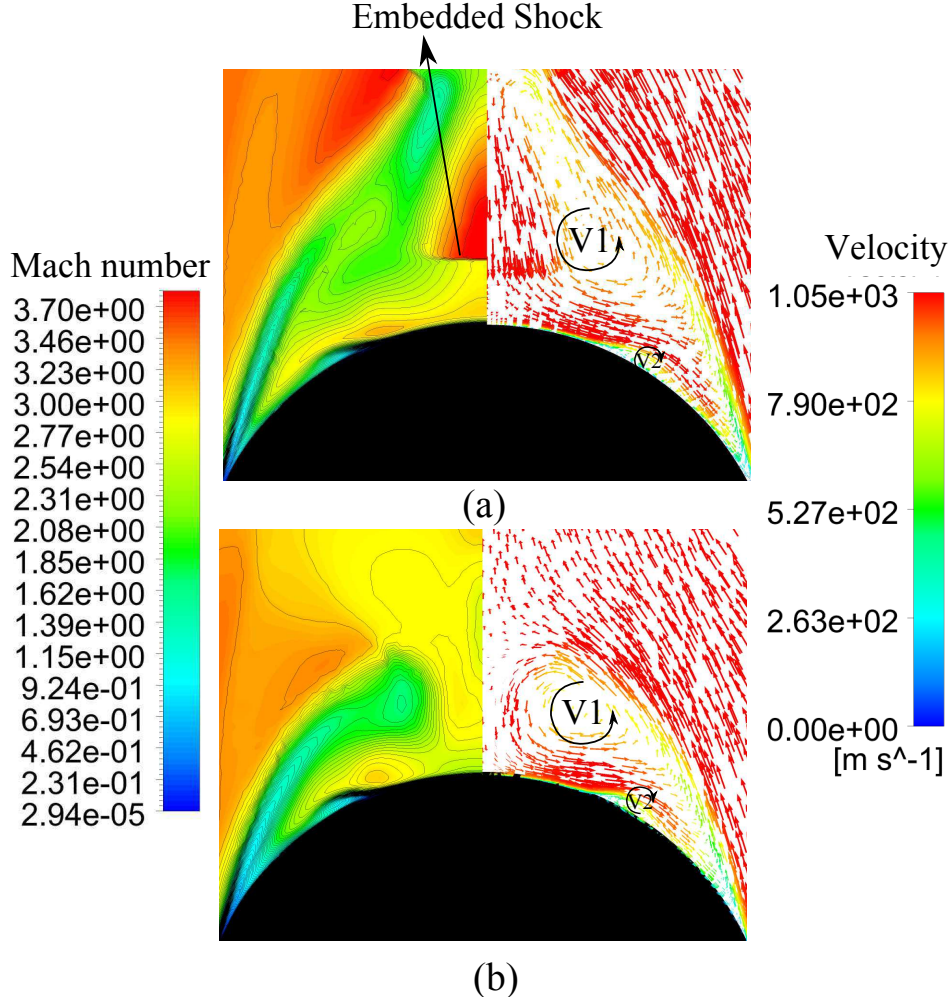


Figure 7.19: Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of the cone-cylinder at M3.2 flow conditions and: (a) 30° incidence, (b) 20° incidence.

7.5 Mach Number Influences on the Flow Separation Characteristics and Properties

The variation of Mach number is considered for this section to determine the differences in flow separation characteristics. It was established in the previous section where flow separation at the nose or shoulder were constrained to certain angles of incidence. Mach number comparison are only considered for incidence configurations where nose or shoulder flow separation occurs.

The comparisons of different Mach numbers for the hemisphere-cylinder at high angles of incidence are considered in figures 7.20 and 7.21. The variation in the freestream Mach number affects the flow downstream of the bow shock wave which also varies in geometry.

The flow topology for the comparison of M2.8 and M3.2 flow conditions for the hemisphere-cylinder are indicated in figure 7.20. There are subtle differences in the flow topology for the hemisphere-cylinders at high angles of incidence where the shape and locations of flow separation features differ. The nose flow separation line and cross flow streamlines converging to the focus points for lower Mach numbers are more curved compared to the high Mach number flow condition. The curved nature of the streamlines means that the saddle points are located wider apart which affects the cross flow separation region. The interface of the nose separation region and cross flow separation forms a kink at the saddle point. The lower Mach number flow condition consists of two reattachment point where secondary reattachment point (N_1) is located in the same location as the reattachment point for the high Mach number condition. The presence of N_1 indicates a transition towards the multiple tertiary flow separation lines in the separated region. The flow velocity in the separated region, for the higher Mach number condition, allows for the condition of N_1 to become the primary reattachment while the other reattachment points dissipate which results in the formation of more irregular tertiary separation line. The primary reattachment point (N_2) for the lower Mach number condition is formed as a result of dominant vortices generated near the shoulder. Rather than converging at N_1 , surface streamlines tend to converging further downstream at N_2 which allows for diverging streamlines to form a more distinct tertiary separation line. The pressure distribution over the leeward side of the hemisphere-cylinder at different Mach number conditions does not differ significantly.

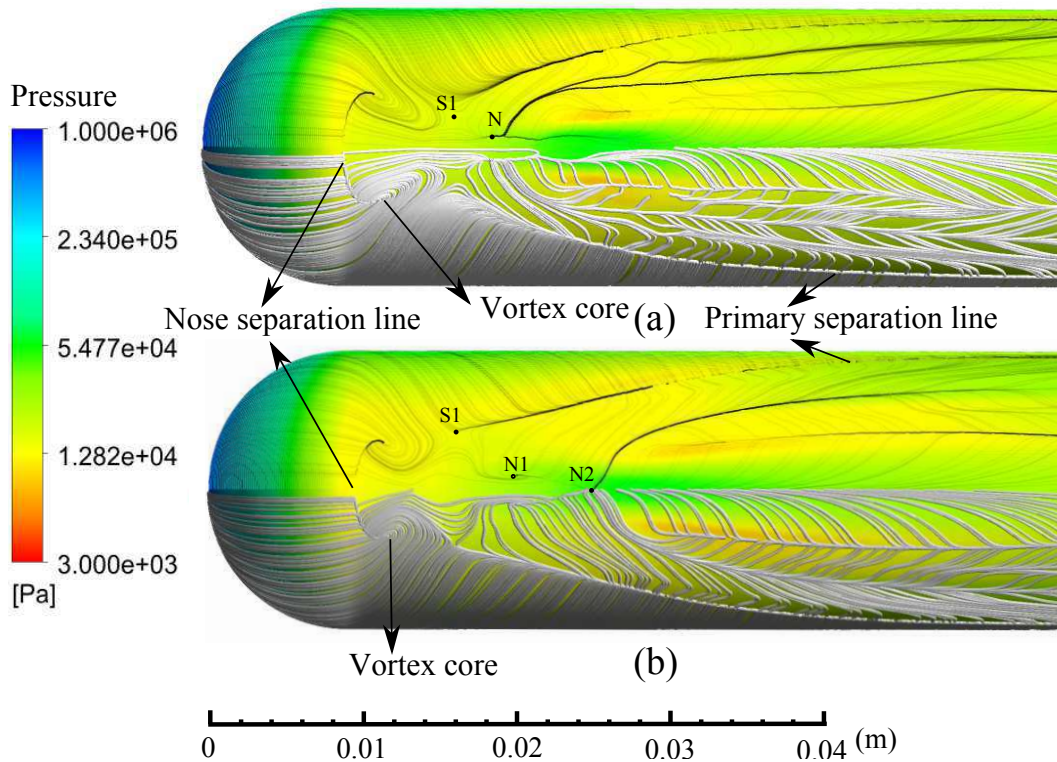


Figure 7.20: Surface streamlines and pressure contours for hemisphere-cylinder at 30° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.

The flow field comparison of the hemisphere-cylinder at M2.8 and M3.2 flow conditions taken at the cross-section 35mm from the shoulder is indicated in figure 7.21. The Mach number contours indicate an expected difference in the flow conditions where the higher Mach condition has regions of greater Mach flow values with exceptions to the vortex systems. The embedded shock wave has similar dimensions where the Mach numbers above the embedded shock is similar to the difference of the freestream Mach number. The Mach number contours indicates a darker shade of blue (lower Mach number) located at the primary separation line hence an indicating that the cross flow separation was stronger for the high Mach number flow condition. However, this did significantly change the vortex structure compared to the low Mach number condition. Despite the M2.8 flow condition containing a single distinct tertiary separation line in the separated region, the tertiary vortex did not indicate any significant difference in vortex shape.

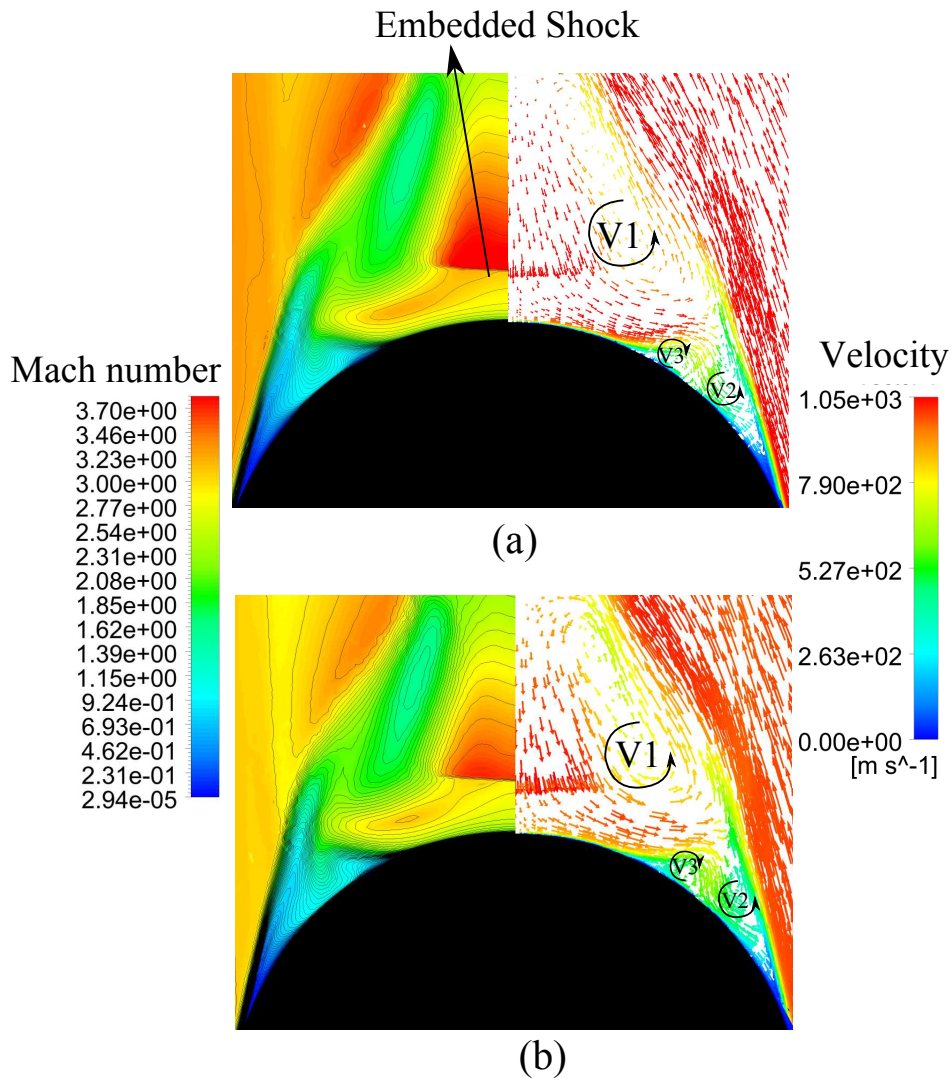


Figure 7.21: Cross-sectional plane located at 35mm downstream of the shoulder with Mach number contours and velocity vectors of hemisphere-cylinder at 30° incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.

The comparisons of different Mach numbers for the hemisphere-cylinder at intermediate angles of incidence are considered in figure 7.22. The discrepancies of the flow separation features with variations in Mach number flow conditions are more subtle and in some cases similar for intermediate incidence configurations. The nose flow separation line are slightly curved for the lower Mach number condition. The location of the saddle point and the cross flow separation line are similar, hence the secondary and tertiary separation lines in the separated region are also similar. The lack of differences for the hemisphere-cylinder at intermediate of incidence for difference Mach number flow conditions are indicative of the exacerbated flow separation characteristics experienced at higher angles of incidence as the differences were more noticeable at higher incidences.

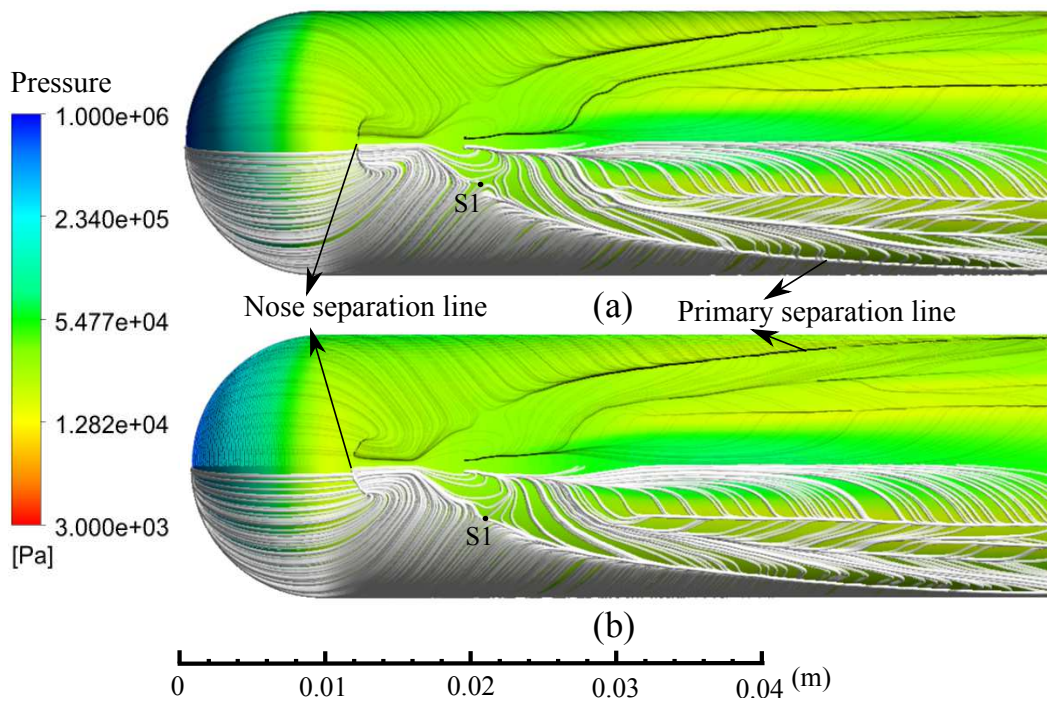


Figure 7.22: Surface streamlines and pressure contours for hemisphere-cylinder at 20° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.

The comparisons of different Mach numbers for the cone-cylinder at high angles of incidence are considered in figures 7.23 and 7.24. The differences in the flow separation characteristics are expected to be even more subtle in comparison to the hemisphere-cylinder as there are less flow separation features. The flow topology comparisons for the cone-cylinder at M2.8 and M3.2 flow conditions are indicated in figure 7.23. The primary separation and secondary lines are identical for both Mach number conditions. The vortex systems seen in figure 7.24 are also similar with minimal differences in the shape of the primary and secondary vortex structures. The difference of the flow conditions are evident from the Mach number contours where the M2.8 flow conditions has a wider embedded shock above the leeward side of the cylinder. However, the difference in the flow velocity is not sufficient to change the flow separation characteristics.

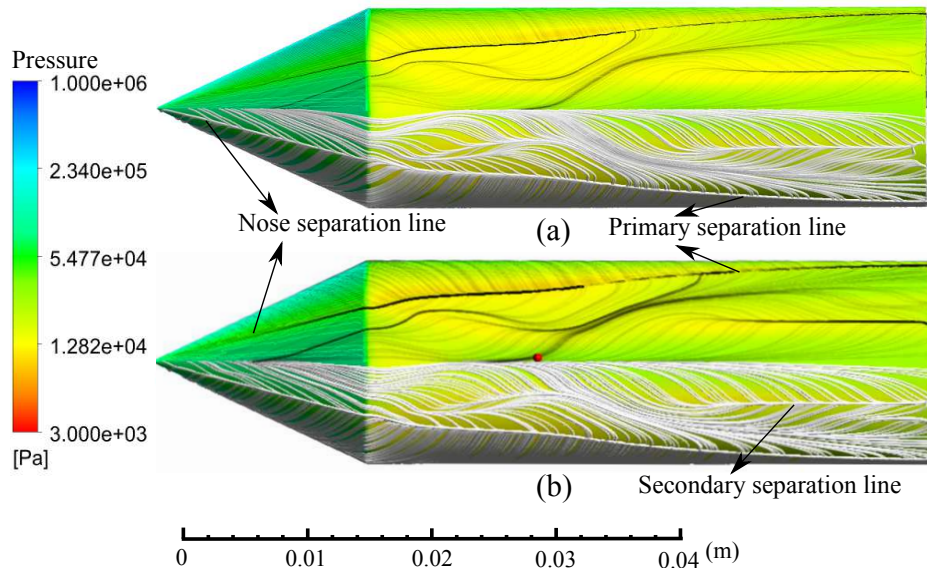


Figure 7.23: Surface streamlines and pressure contours for cone-cylinder at 30° angle of incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.

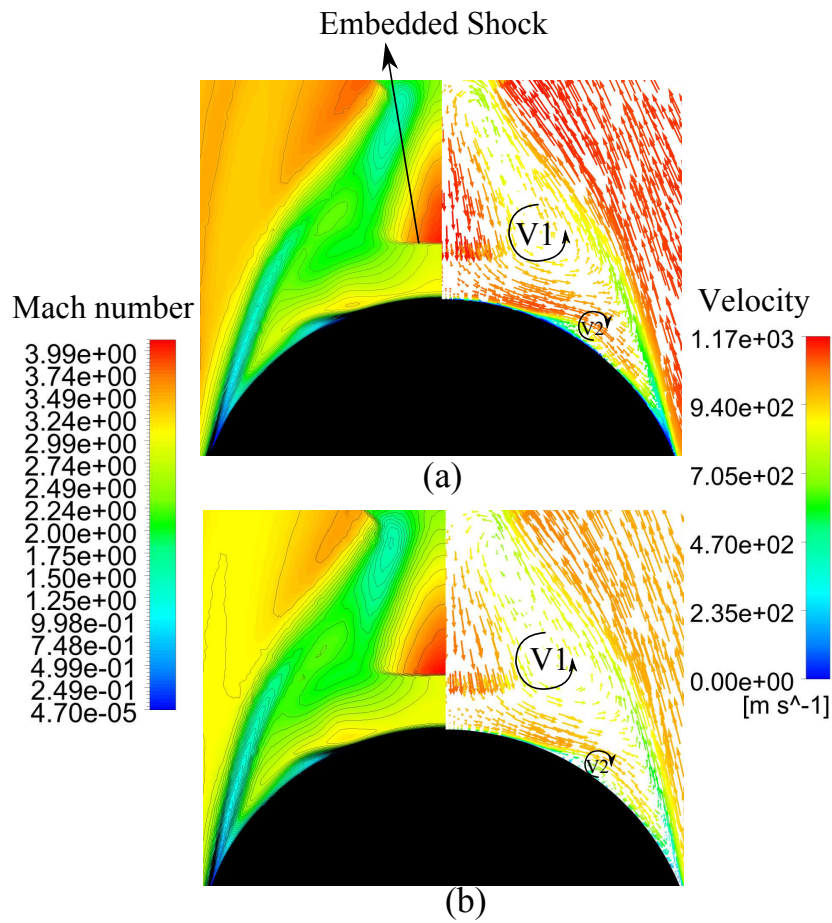


Figure 7.24: Cross-sectional plane located at 35mm downstream of shoulder with Mach number contours and velocity vectors of cone-cylinder at 30° incidence and: (a) M3.2 flow conditions, (b) M2.8 flow conditions.

The differences for Mach number are subtle in comparison to geometric parameters over the slender bodies. The apparent similarities for the Mach number variations are likely due to the relatively small changes in Reynolds number which is a parameter that dictates flow separation characteristics. The chosen Mach number flow conditions correspond to a fully turbulent type flow separation, hence significant differences in flow separation are unlikely.

8 Conclusions

The conclusions for the analysis of the supersonic flow separation characteristics and properties over the shoulder for the hemisphere-cylinder and cone-cylinder are as follows:

- Flow separation occurred over the shoulder of the hemisphere-cylinder at intermediate and high angles of incidence at M2.8 and M3.2 supersonic flow conditions, while no flow separation occurred over the shoulder at low angles of incidence.
- Flow separation occurred at the nose for the cone-cylinder at high angles of incidence with M2.8 and M3.2 flow conditions.
- The complexity of flow separation structures was increased with increased angles of incidence for the hemisphere-cylinder at high constant Mach number conditions.
- The nose flow separation occurring over cone-cylinders were limited by the relation of the cone deflection angle and angle of incidence.
- The difference in high Mach number, while maintaining high Reynolds numbers, did not yield significant flow separation differences for the hemisphere-cylinder. However, subtle differences in the flow separation features were evident at intermediate and high angles of incidence.
- The difference in high Mach number, while maintaining high Reynolds numbers, did not yield significant flow separation differences for the cone-cylinder.

9 Recommendations

The recommendations arising from this research are as follows:

- The nose shapes of the slender bodies considered for this research were simple shapes that provided CFD flow visualization for the supersonic flow separation characteristics. Modern application of slender bodies may require more complex nose shapes that are considered optimized for its function, hence it is recommended that more complex nose shape, such as elliptical or ogive nose shapes, be explored to study the flow dynamics of flow separation.
- Reynolds number is a vital parameter that dictates the flow separation characteristic. Flow separation at high Reynolds numbers have already been explored in this research for hemisphere-cylinders and cone-cylinders, however they have not been explored at lower Reynolds numbers that are in the transitional and laminar regions. Hence exploring such conditions provides knowledge of the relationship of Reynolds number regimes with the flow separation characteristics.
- At high angles of incidence, slender bodies experience vortices. developing downstream of the leeward side. However, if the angle of incidence were to approach higher angles of incidence, turbulent conditions may cause the flow downstream to be asymmetrical which would affect the aerodynamic properties of the slender bodies. It is not known whether the flow separation near the shoulder would also be affected at such conditions hence it may be recommended to explore to unsymmetrical flow separation dynamics that may occur at high angles of incidence.

References

- [1] F. M. White. *Fluid Mechanics*, chapter 9: Compressible Flow, pages 623–695. McGraw Hill, 7th edition, 2011. ISBN 978-007-131121-2.
- [2] F. M. White. *Fluid Mechanics*, chapter 7: Flow Past Immersed Bodies, pages 469–521. McGraw Hill, 7 edition, 2011. ISBN 978-007-131121-2.
- [3] H. Sturm, G. Dumstorff, P. Bushe, D. Westermann, and W. Lang. Boundary layer separation and reattachment detection on airfoil by thermal flow sensors. *Sensors*, 12:14292–14306, 2012.
- [4] M. Jahanmiri. Laminar separation bubble: its structure, dynamics and control. Technical report, Chalmers University of Technology, Göteborg, Sweden, 2011.
- [5] S. Hooseria. *Three-dimensional Curved Shock Wave Interactions with Slender Bodies at Incidence*. Doctoral thesis, School of Mechanical, Industrial, and Aeronautical Engineering, 2015. University of Witwatersrand.
- [6] J. D. Anderson. *Fundamentals of Aerodynamics*. McGraw-Hill, New York, 5th edition, 2011.
- [7] J. D. Anderson. *Modern Compressible Flow with Historical Perspective*. McGraw-Hill, New York, 2nd edition, 2001.
- [8] R. H. Feldhuhn, A. E. Winkelmann, and L. Pasiuk. An experimental investigation of the flow field around a yawed cone. In *AIAA 3rd Fluid and Plasma Dynamics Conference*, 1971. Los Angeles, California.
- [9] T. Hsieh. An investigation of separated flow about a hemisphere-cylinder at incidence in the Mach number range from 0.6 to 1.5. In *15th Aerospace Sciences Meeting*, page 179.
- [10] S. Le Clainche, D. Rodríguez, V. Theofilis, and J. Soria. Formation of three-dimensional structures in the hemisphere-cylinder. *AIAA Journal*, 54(12):3884–3894, 2016.
- [11] D. J. Peake and M. Tobak. Three-dimensional interactions and vortical flows with emphasis on high speeds. Technical report, Advisory Group For Aerospace Research And Department Neuilly-sur-Seine (France), 1980.

- [12] M. Tobak and D. Peake. Topology of two-dimensional and three-dimensional separated flows. In *12th Fluid and Plasma Dynamics Conference*, page 1480, 1979.
- [13] J. M. Détery. Robert Legendre and Henri Werlé: toward the elucidation of three-dimensional separation. *Annual review of fluid mechanics*, 33(1):129–154, 2001.
- [14] P. J. Lamont. Pressures around an inclined ogive cylinder with laminar, transitional, or turbulent separation. *AIAA Journal*, 20(11):1492–1499, 1982.
- [15] N. T. Hoang, O. K. Rediniotis, and D. P. Telionis. Hemisphere cylinder at incidence at intermediate to high Reynolds numbers. *AIAA journal*, 37(10):1240–1250, 1999.
- [16] S. J. Hooseria and B. W. Skews. Colour surface flow visualisation of interfering slender bodies at Mach 3. *Journal of Visualization*, 18(3):413–423, 2015.
- [17] B. A. Boswell and J. C. Dutton. Flow visualizations and measurements of a three-dimensional supersonic separated flow. *AIAA journal*, 39(1):113–121, 2001.
- [18] M. Edmunds, R. S. Laramée, G. Chen, N. Max, E. Zhang, and C. Ware. Surface-based flow visualization. *Computers & Graphics*, 36(8):974–990, 2012.
- [19] ANSYS Fluent. Fluent 17 users guide. *Lebanon, USA*, 2016.
- [20] S. M. Salim and S. Cheah. Wall y^+ strategy for dealing with wall-bounded turbulent flows. In *Proceedings of the international multiconference of engineers and computer scientists*, volume 2, pages 2165–2170, 2009.
- [21] G. Alfonsi. Reynolds-averaged Navier-Stokes equations for turbulence modeling. *Applied Mechanics Reviews*, 62(4), 2009.
- [22] ANSYS Fluent. Ansys Fluent theory guide 15.0. *ANSYS, Canonsburg, PA*, 33, 2013.
- [23] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 32(8):1598–1605, 1994.
- [24] D. K. Walters and D. Cokljat. A three-equation eddy-viscosity model for Reynolds-averaged Navier–Stokes simulations of transitional flow. *Journal of fluids engineering*, 130(12), 2008.
- [25] R. Langtry and R. Menter. Correlation-based transition modelling for unstructured parallelized computational fluid dynamic codes. *AIAA Journal*, 47:2894–2906, 2009.
- [26] Y. B. Suzen, P. G. Huang, L. S. Hultgren, and D. E. Ashpis. Predictions of separated and transitional boundary layers under low-pressure turbine airfoil conditions using an intermittency transport equation. *Journal of Turbomachinery*, 125(3):455–464, 2003.
- [27] F. R. Menter, R. Langtry, and S. Völker. Transition modelling for general purpose CFD codes. *Flow, turbulence and combustion*, 77(1-4):277–303, 2006.

- [28] A. A. Fillipi. *Effect of internal surface curvature on steady axisymmetric shock waves*. Masters dissertation, School of Mechanical, Industrial, and Aeronautical Engineering, 2017. University of Witwatersrand.
- [29] L. Kirkup. *Experimental Methods: An Introduction to Analysis and Presentation of Data*. Wiley, New York, 1st edition, 1996.

Appendix A : SSWT Calculation

A.1 SSWT Mach number calibration

The calibration of the SSWT is based on varying the veeder counter and determining the correspond Mach number that is obtained from the throat configuration. A constant stagnation pressure is used for the following calibration curve done by A. A. Fillipi [28]. The above

Table A.1: Table of veeder counter and Mach number obtained from calibration test completed by A. A. Fillipi. [27]

Veeder counter	Mach Number
141	3.6
206	3.4
239	3.3
273	3.2
309	3.1
348	3.0
388	2.9
432	2.9

table represents data obtained from calibration tests. The values are plot as seen in Figure A.1, which follows a 3rd-order trend. The 3rd-order equation is represented as:

$$M = (-6.992 \times 10^{-10})C^3 + (2.514 \times 10^{-6})C^2 - 0.004C + 4.118 \quad (\text{A.1})$$

Where M is the Mach number and C is the veeder counter.

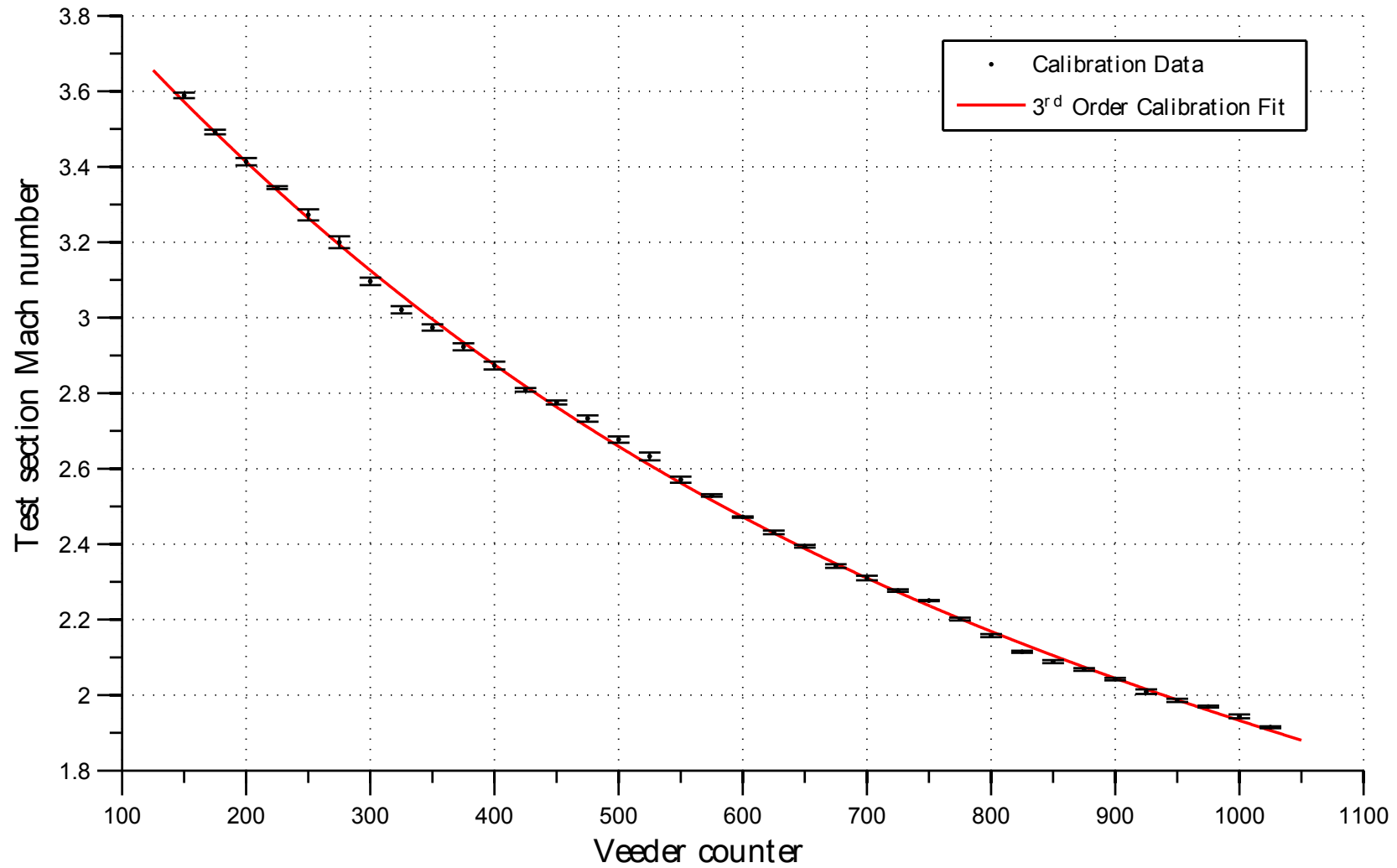


Figure A.1: Calibration graph representing the relationship between the veeder counter the test section Mach number. Calibration completed by A. A. Fillipi. [27]

Appendix B : Test Model Design

B.1 SSWT Shock Wave Loads

The SSWT loads calculated for the test piece and mounting block design are based on a conservative method where a start-up shock and the dynamic pressure of the flow over the test piece are combined. The total load is applied at the center of mass on the test piece hence exerting a bending, shear, and axial load on the test piece. To ensure the maximum load is considered, the loads over a cone-cylinder is determined as it has a greater area in comparison to the hemisphere-cylinder.

The start-up shock wave load is derived from normal shock theory where the pressure ratio in relation to the Mach number is considered. For the equations relating to normal shocks, consider Anderson [6] for the relation of temperature, pressure, and density with Mach number. Start-up shock loads can be derived by considering pressure, area, and force. Hence manipulating the stagnation and Mach number parameters to obtain equation B.2.

$$F_N = \frac{7P_0 S_F (M^2 - 1)}{6(1 + 0.2M^2)^{\frac{3.5}{\beta}}} \quad (\text{B.1})$$

Where F_N is the start-up shock wave force, P_0 is the stagnation pressure, S_F is the total surface area of the cone-cylinder, β is a ratio as defined below and is specific for Mach numbers above 3.

$$\beta = (0.9 + 0.1 \frac{S_L}{S_T}) \quad (\text{B.2})$$

S_L is the surface area of lifting surfaces present on the test piece which is zero as lifting surfaces are not considered for this research.

The dynamic pressure can be consider by applying the same stagnation pressure ratio to the dynamic pressure equation seen in Bernoulli's equation [6]. Hence obtaining equation B.3.

$$q = \frac{0.7P_0}{(1 + \frac{\gamma-1}{2}M^2)^{\frac{\gamma}{\gamma-1}}} \times M^2 \quad (\text{B.3})$$

Where γ is the constant for heat capacity ratio considered for ideal gases. With the equations derived above, the calculated loads on the test piece were determined as 422.8N for the drag or horizontal force and 244.1N for the lift or vertical force.

B.2 Stress Analysis

The stress analysis for the mounting block assembly, sting rod, and test piece assembly were initially completed using basic stress calculations. Ansys[®] Workbench[™] was further used

in order to complete Finite Element Method(FEM) analysis to confirm the stress on the assembly and model. The stress for each component did not exceed a safety factor of 2, which was deemed acceptable for further experimental testing.

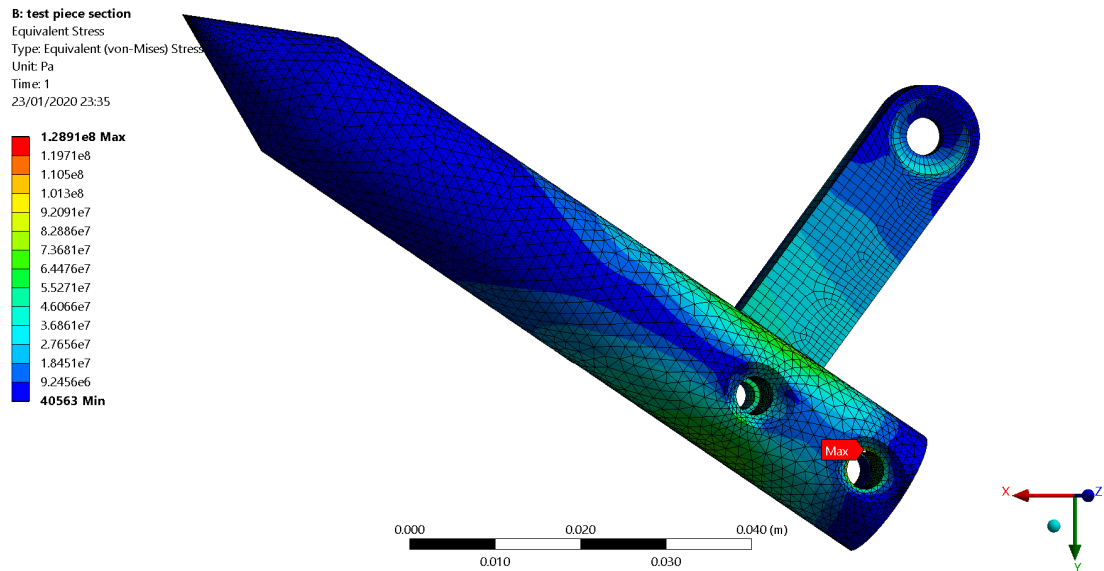


Figure B.1: FEM stress results for the cone-cylinder and adjustable strut at 30 incidence

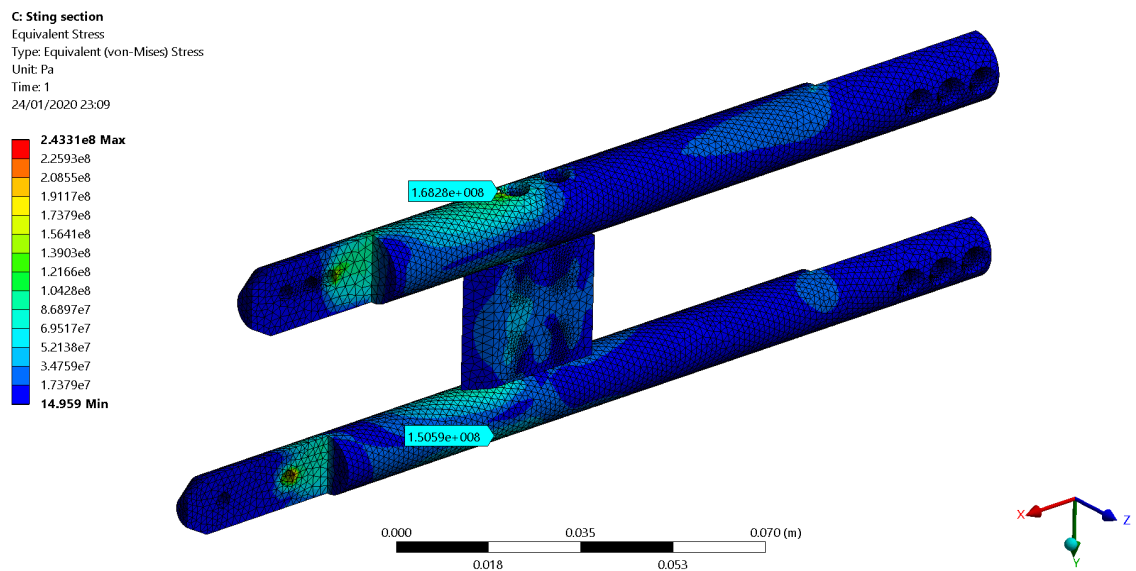


Figure B.2: FEM stress results for the sting with applied loads transferred from the test piece and adjustable strut

G: Version 2- mounting block
 Equivalent Stress
 Type: Equivalent (von-Mises) Stress
 Unit: Pa
 Time: 1
 24/01/2020 23:22

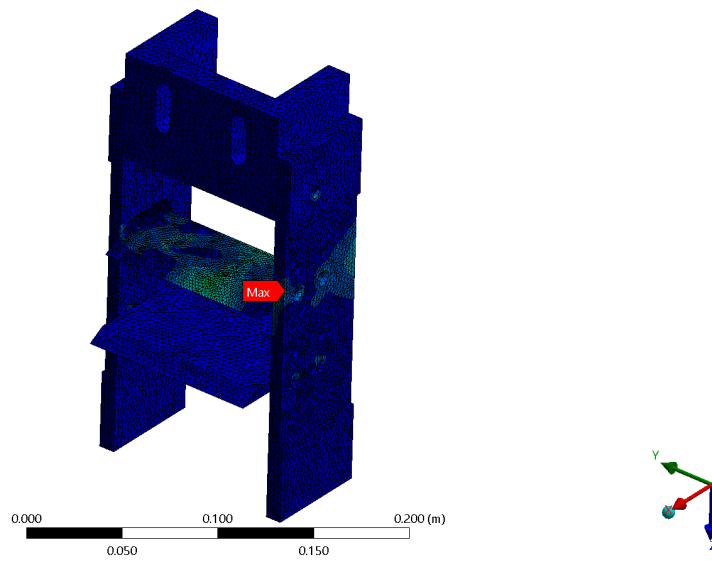
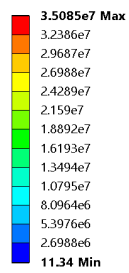
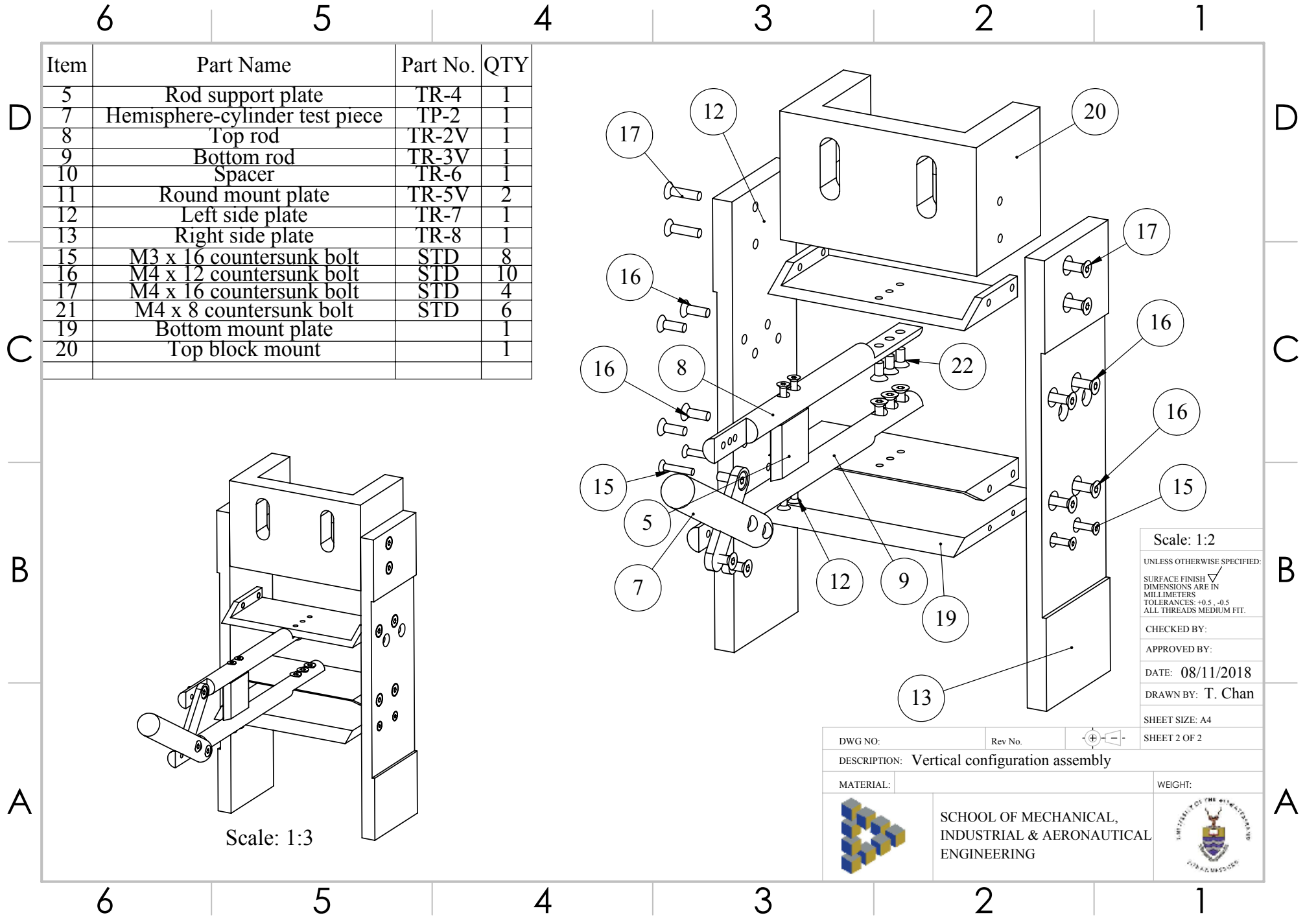


Figure B.3: FEM stress results for the mounting block with applied loads transferred from the stings

B.3 Test Model Specifications

The assembly and drawings with dimensions for the test model follows on the next page.



D

C

B

A

6

5

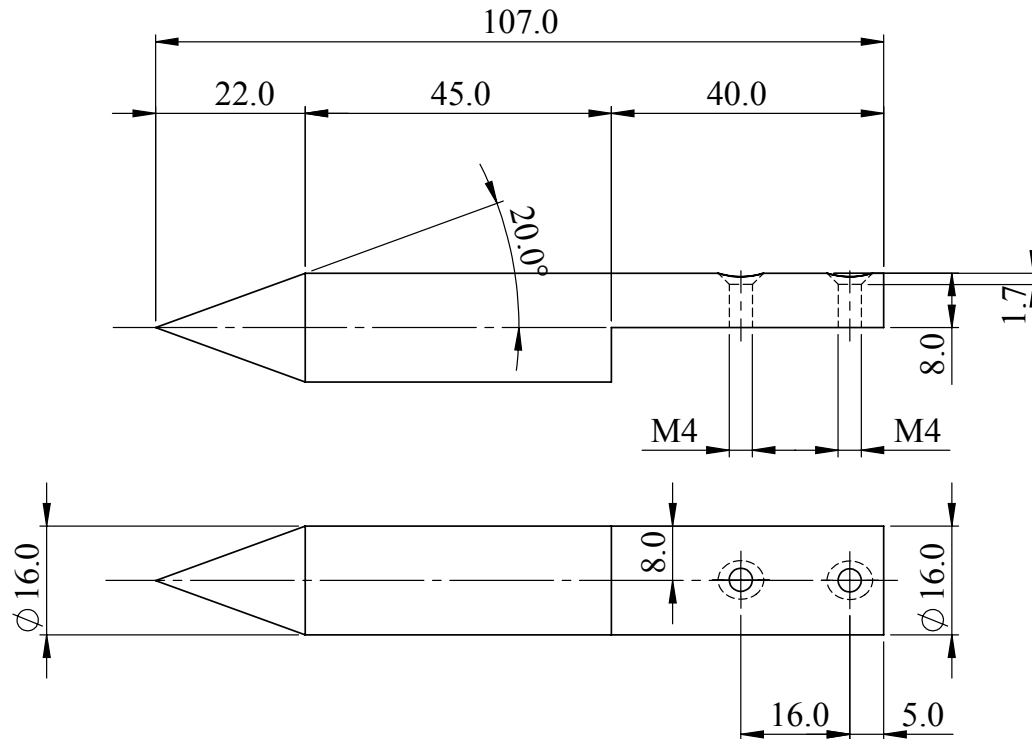
4

3

2

1

Item	Quantity	Part No.	Description
1	1	TP1	Cone-cylinder test piece



Scale: 1:1
UNLESS OTHERWISE SPECIFIED: SURFACE FINISH $\checkmark$ DIMENSIONS ARE IN MILLIMETERS TOLERANCES: +0.5, -0.5 ALL THREADS MEDIUM FIT.
CHECKED BY:
APPROVED BY:
DATE: 17/05/2018
DRAWN BY: T. Chan
SHEET SIZE: A4
SHEET 1 OF 1

DWG NO: 1 of 10	Rev No.		
DESCRIPTION: Cone-cylinder			
MATERIAL: Brass			WEIGHT:
	SCHOOL OF MECHANICAL, INDUSTRIAL & AERONAUTICAL ENGINEERING		

D

C

B

A

6

5

4

3

2

1

D

C

B

A

6

5

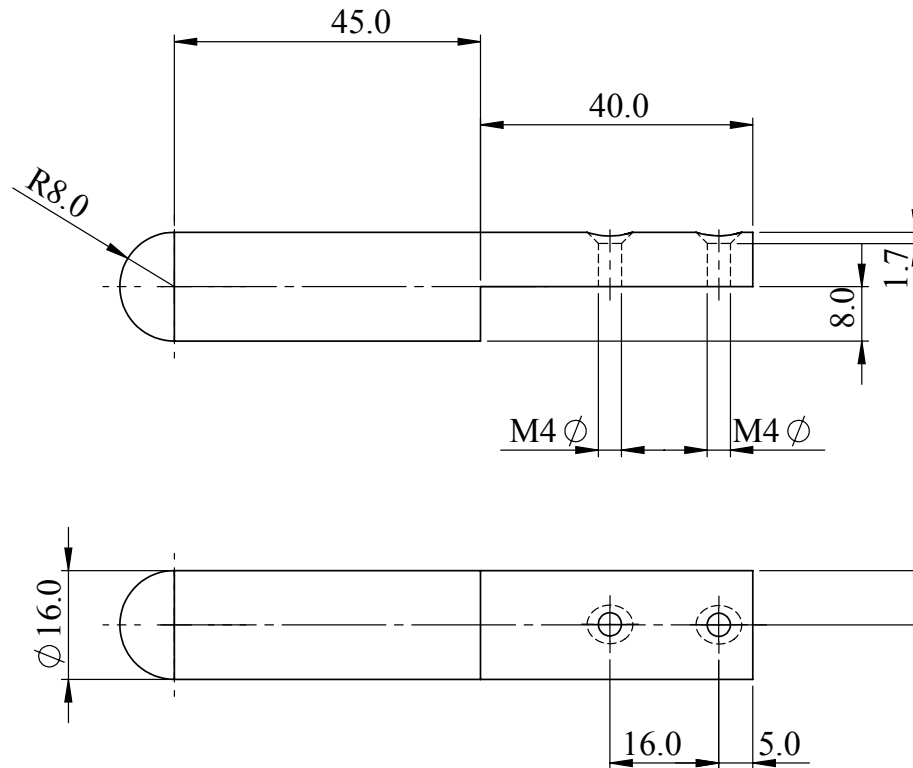
4

3

2

1

Item	Quantity	Part No.	Description
1	1	TP2	Hemisphere-cylinder test piece



Scale: 1:1
UNLESS OTHERWISE SPECIFIED: SURFACE FINISH $\checkmark$ DIMENSIONS ARE IN MILLIMETERS TOLERANCES: $+0.5, -0.5$ ALL THREADS MEDIUM FIT.
CHECKED BY:
APPROVED BY:
DATE: 17/05/2018
DRAWN BY: T. Chan
SHEET SIZE: A4
SHEET 1 OF 1

DWG NO: 2 of 10	Rev No.	
DESCRIPTION: Hemisphere-cylinder		
MATERIAL: Brass		WEIGHT:
	SCHOOL OF MECHANICAL, INDUSTRIAL & AERONAUTICAL ENGINEERING	

D

C

B

A

6

5

4

3

2

1

D

C

B

A

6

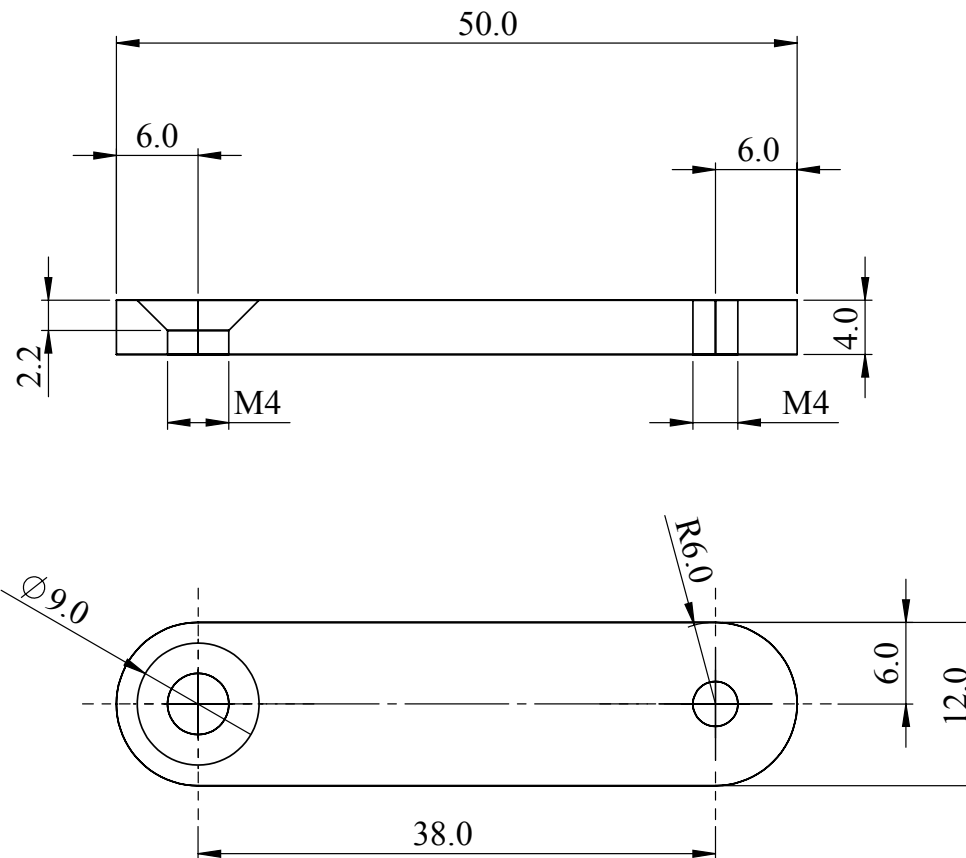
5

4

3

2

1



Item	Quantity	Part No.	Description
1	1	TR1	Adjuster link mounted to rod

D

C

B

A

Scale: 2:1
UNLESS OTHERWISE SPECIFIED: SURFACE FINISH $\checkmark$ DIMENSIONS ARE IN MILLIMETERS TOLERANCES: +0.5, -0.5 ALL THREADS MEDIUM FIT.
CHECKED BY:
APPROVED BY:
DATE: 17/05/2018
DRAWN BY: T. Chan
SHEET SIZE: A4
SHEET 1 OF 1

DWG NO: 3 of 10	Rev No.		SHEET 1 OF 1
DESCRIPTION: Adjuster link			
MATERIAL: Brass			WEIGHT:
	SCHOOL OF MECHANICAL, INDUSTRIAL & AERONAUTICAL ENGINEERING		

D

C

B

A

6

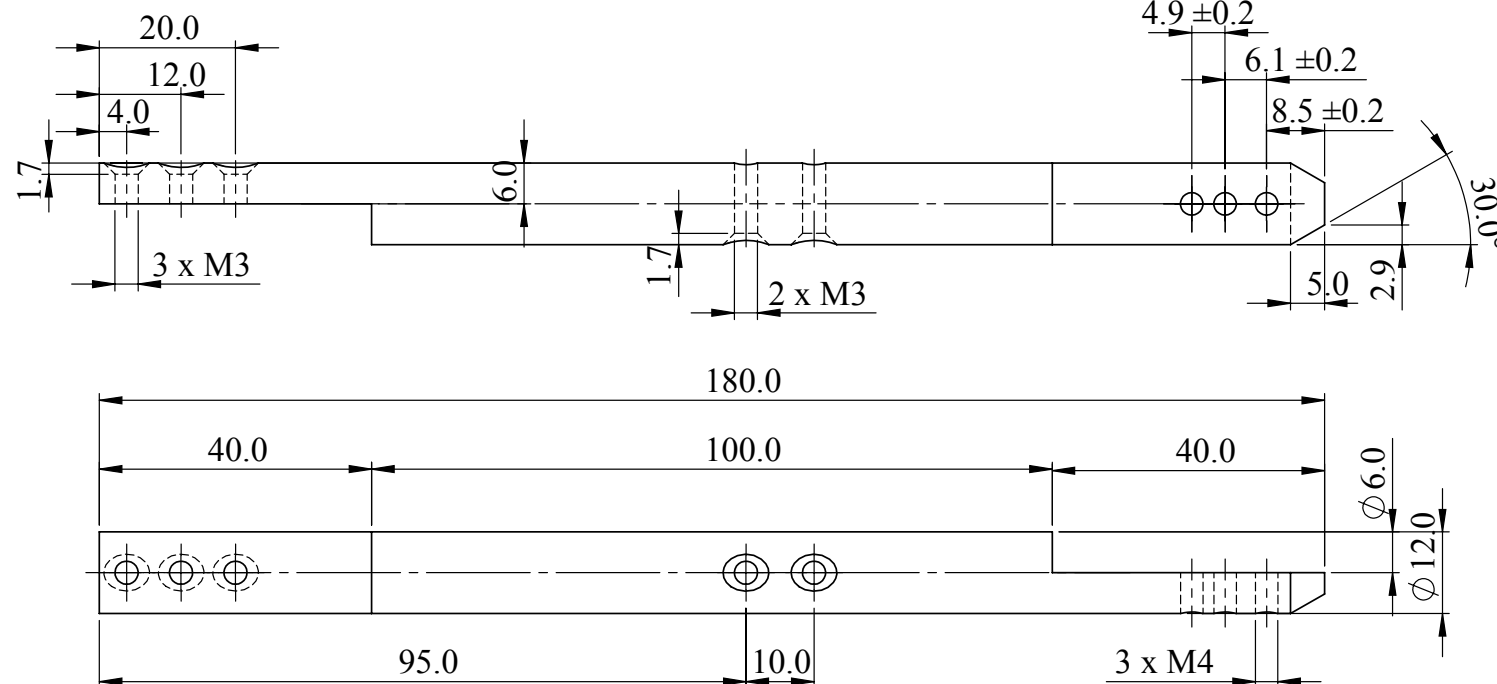
5

4

3

2

1



Item	Quantity	Part No.	Description
1	1	TR3	Top rod of sting config.

Scale: 1:1

UNLESS OTHERWISE SPECIFIED:
 SURFACE FINISH $\checkmark$
 DIMENSIONS ARE IN
 MILLIMETERS
 TOLERANCES: +0.5, -0.5
 ALL THREADS MEDIUM FIT.

CHECKED BY:

APPROVED BY:

DATE: 17/05/2018

DRAWN BY: T. Chan

SHEET SIZE: A4

SHEET 1 OF 1

DWG NO: 5 of 10

Rev No.



DESCRIPTION: Top rod

MATERIAL: Mild steel

WEIGHT:



SCHOOL OF MECHANICAL,
 INDUSTRIAL & AERONAUTICAL
 ENGINEERING



D

C

B

A

6

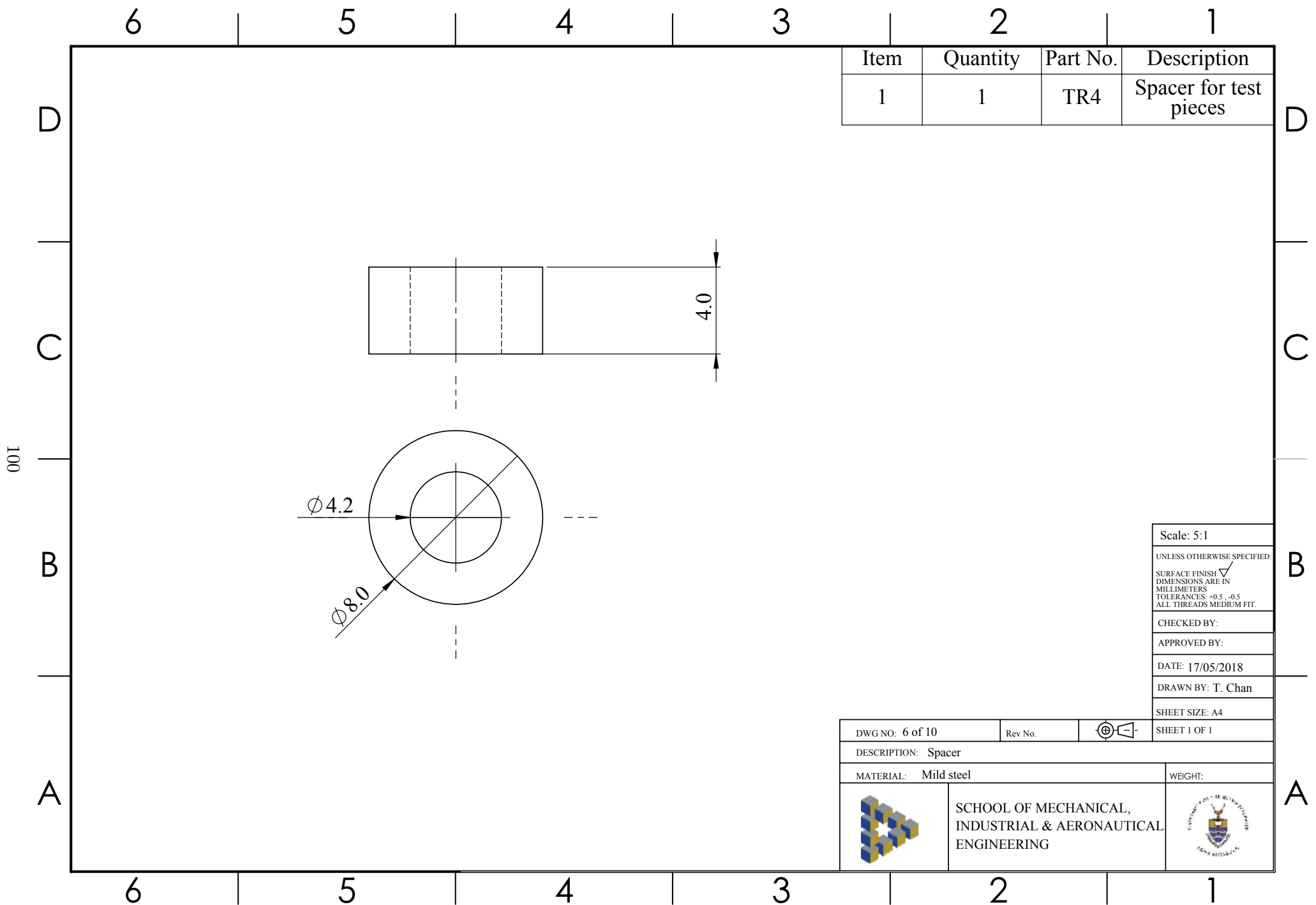
5

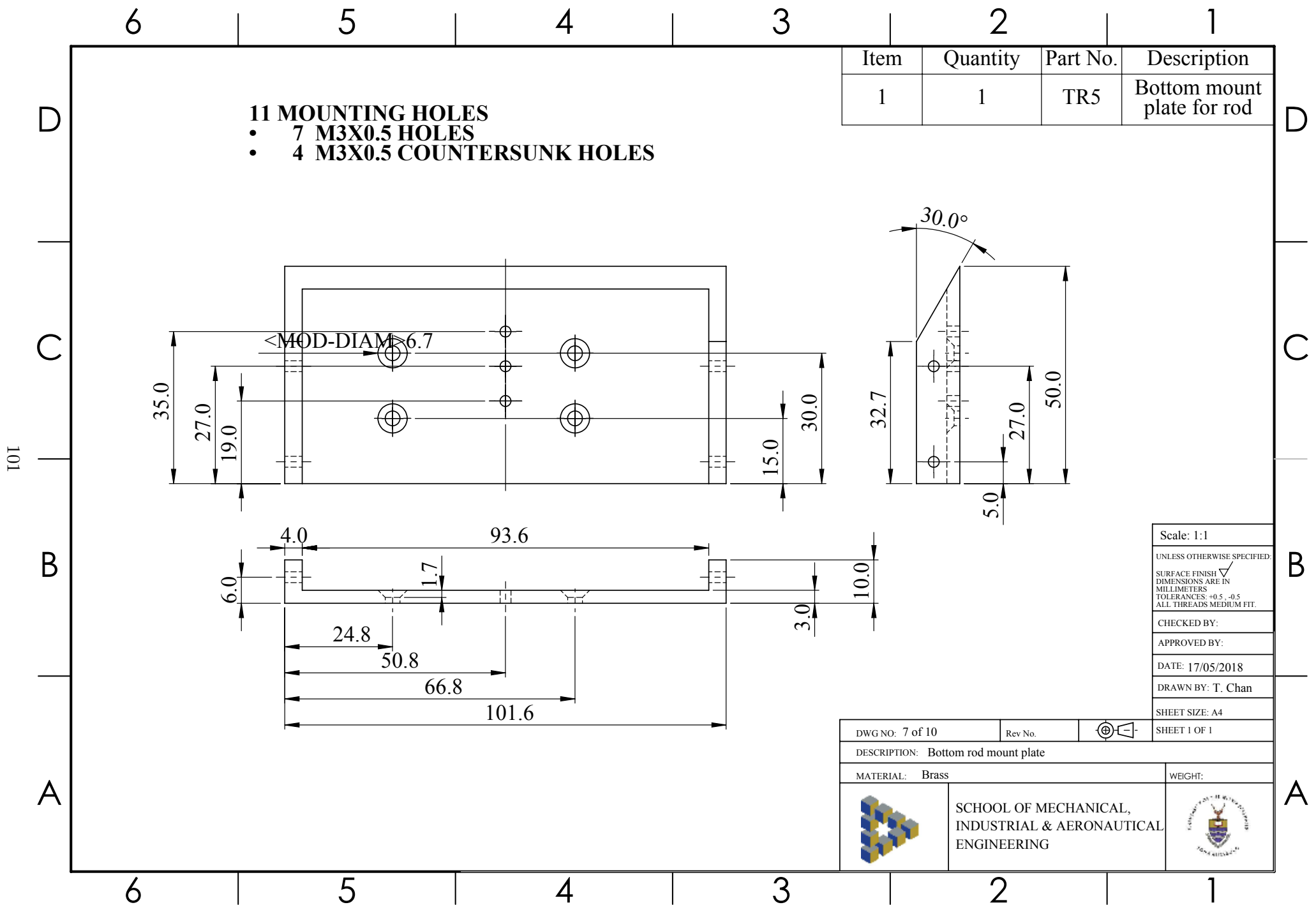
4

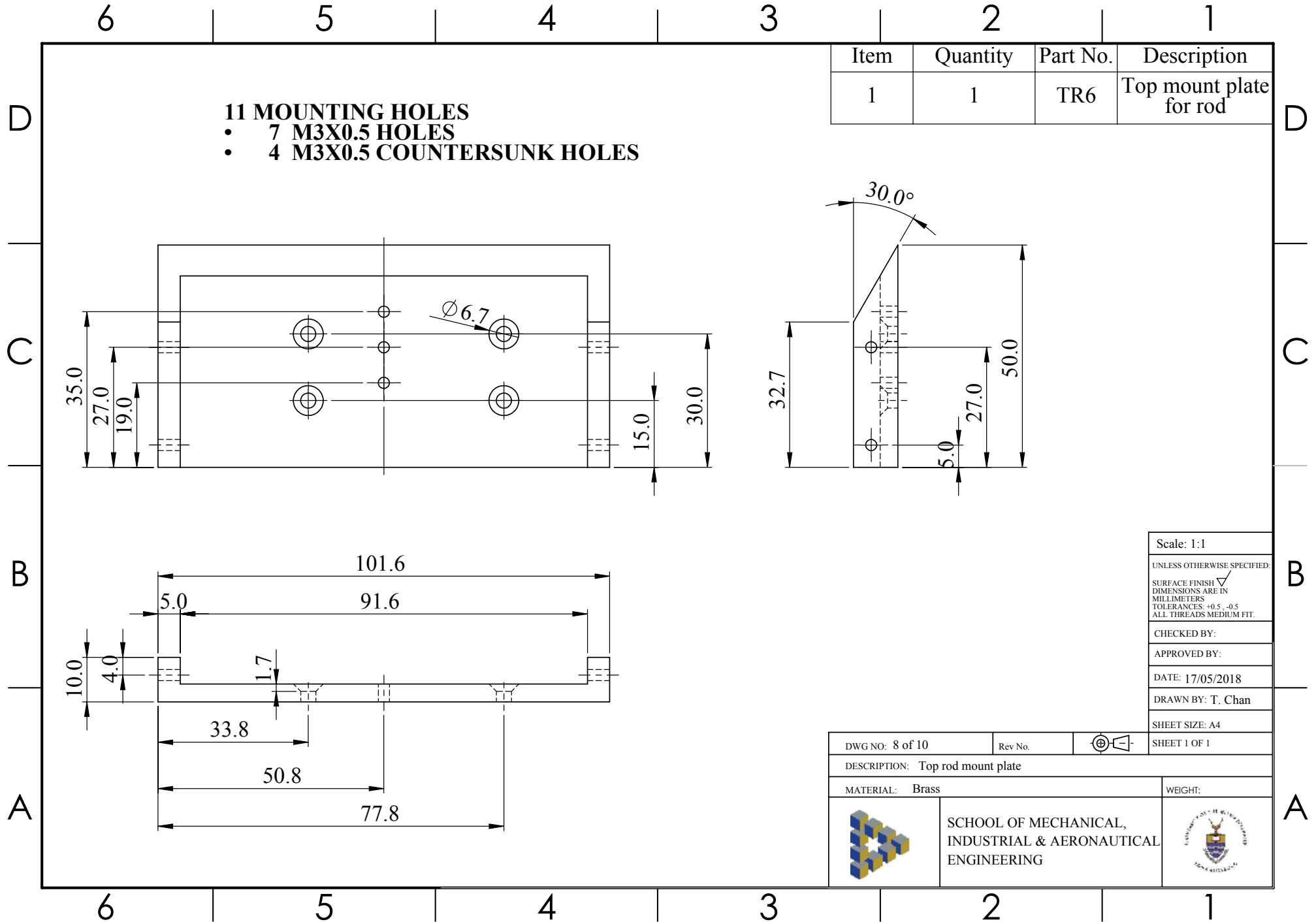
3

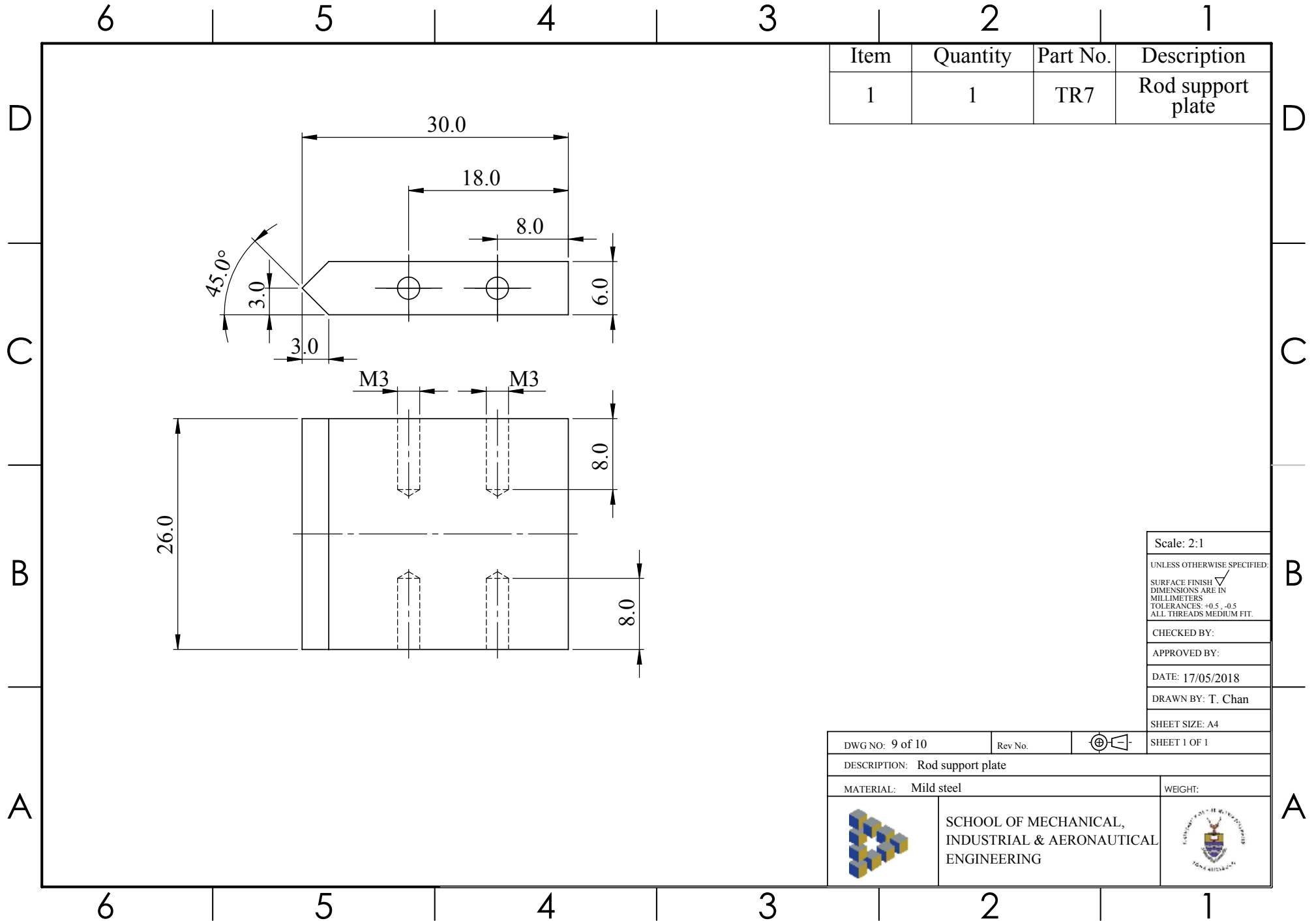
2

1









Item	Quantity	Part No.	Description
1	1	TR7	Rod support plate

Scale: 2:1

UNLESS OTHERWISE SPECIFIED:
 SURFACE FINISH $\checkmark$
 DIMENSIONS ARE IN
 MILLIMETERS
 TOLERANCES: +0.5, -0.5
 ALL THREADS MEDIUM FIT.

CHECKED BY:

APPROVED BY:

DATE: 17/05/2018

DRAWN BY: T. Chan

SHEET SIZE: A4

SHEET 1 OF 1

DWG NO: 9 of 10

Rev No.



DESCRIPTION: Rod support plate

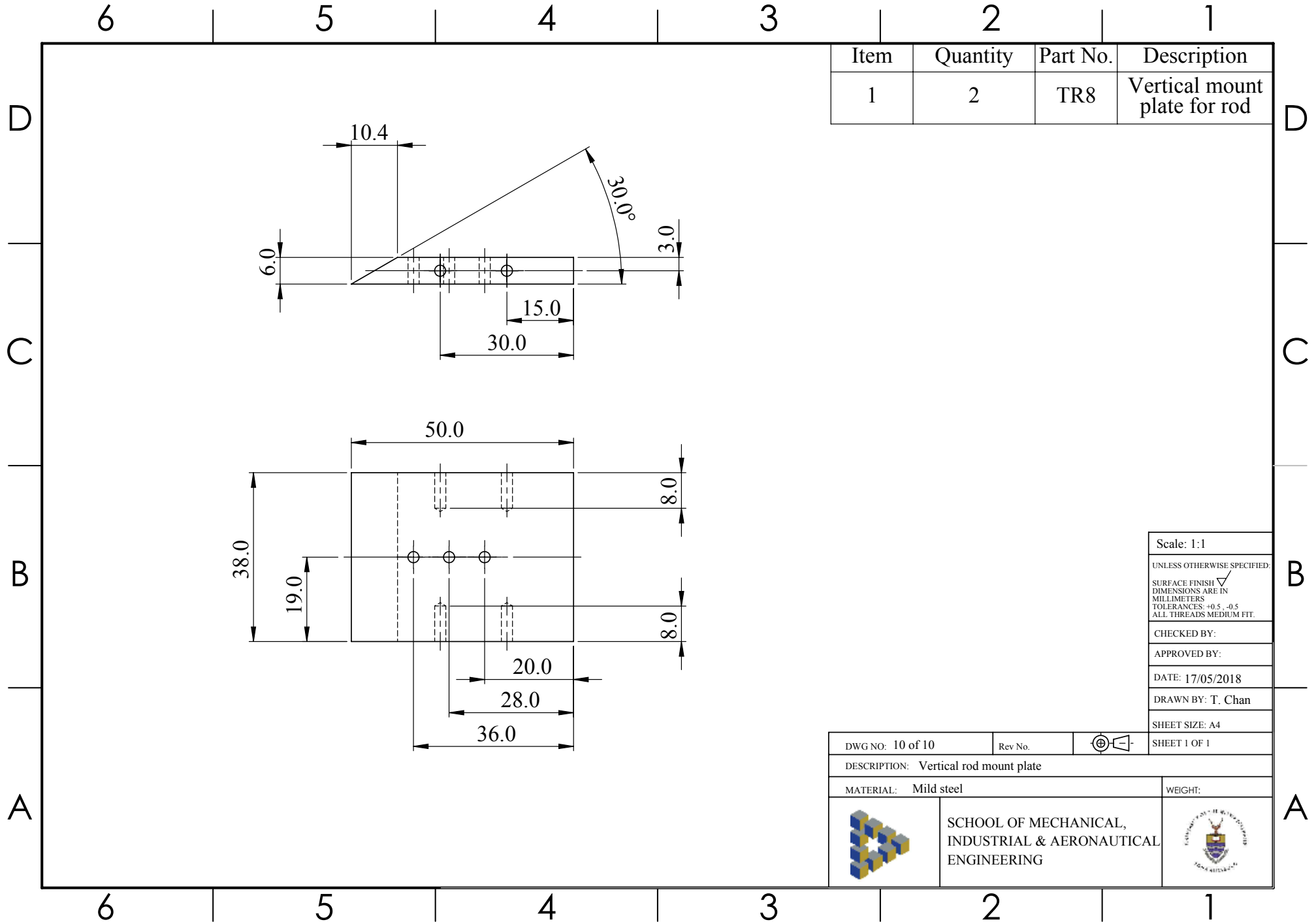
MATERIAL: Mild steel

WEIGHT:



SCHOOL OF MECHANICAL,
 INDUSTRIAL & AERONAUTICAL
 ENGINEERING





Appendix C : Experimental Methods and Results

C.1 Uncertainty Analysis

The uncertainty analysis for the experimental results are specific for the determination of the Mach number utilized for each test recorded. The variables that play a role in the uncertainty of the results include the veeder counter, pressure transducer readings from the both the stagnation chamber and the testing chamber and the voltage readings for the data acquisition system.

The uncertainty of the pressure transducers have been completed by Hooseria [5], where the equations that were considered are as follows:

$$P_0 = m_{P_0} V_{P_0} + c_{P_0} \quad (C.1)$$

$$P = m_P V_P + c_P \quad (C.2)$$

Where m_{P_0} and m_P are the gradients representing the transducer calibration data for the static port and the test chamber pressure port respectively. The variables c_{P_0} and c_P are the constants or y-intercepts of the linear fit of the transducers for the wind tunnel pressure and static pressure respectively. Applying the method by Kirkup [29] to determine the uncertainty analysis, the maximum and minimum error for the transducers are as follows:

$$(P_0)_{max/min} = \left(m_{P_0} \pm \Delta m_{P_0} \right) V_{P_0} + \left(c_{P_0} \pm \Delta c_{P_0} \right) \quad (C.3)$$

$$(P)_{max/min} = \left(m_P \pm \Delta m_P \right) V_P + \left(c_P \pm \Delta c_P \right) \quad (C.4)$$

The data for the pressure transducers are seen in table C.1, which was obtained from the thesis by Hooseria [5] whom used regression statistic methods to obtained an estimation of the pressure transducer readings. The calibration data represents values that compliment equations C.1 and C.2, hence allowing one to determine the voltage output with the corresponding pressures that are measured from the pressure transducers. Note that for the case of uncertainty analysis, the voltage is assumed to be a constant where the variables m_P and c_P effects the deviations of the pressure readings that translates to the uncertainty of the Mach number. The uncertainty error for the Mach number was determined by Hooseria [5] to be approximately 2.3%, which is affected by the uncertainty of the pressure transducer readings.

Table C.1: Calibration data on the pressure transducers used for the experimental data. Data edited from reference. [4]

Transducer type	m (kPa/V)	Δm (kPa/V)	c (kPa)	Δc (kPa)
Wind tunnel pressure	97.119	0.595	80.501	3.995
Static pressure	22.242	0.544	70.505	3.652

An additional uncertainty error is considered for the veeder counter which is relevant to the calibration curve seen in figure A.1. Equation A.1 determines the Mach number with the corresponding selected veeder counter, hence a resolution of ± 1 is applied to the veeder counter value (C). The equation for the maximum and minimum Mach number values with the applied uncertainty deviation is as follows:

$$M_{max/min} = (-6.992 \times 10^{-10})(C \pm \Delta C)^3 + (2.514 \times 10^{-6})(C \pm \Delta C)^2 - 0.004(C \pm \Delta C) + 4.118 \quad (C.5)$$

The uncertainty error of the veeder counter on the Mach number was determined to be 0.13%, which is applied to the total uncertainty error as seen below:

$$\eta_T = \sqrt{\eta_P^2 + \eta_V^2} \quad (C.6)$$

Where η_P is the uncertainty error for the pressure transducer and η_V is the uncertainty error for the veeder counter. The total uncertainty error, which was predominantly based on the pressure transducer error, was 2.304% hence translating to $M = 3.2 \pm 0.07373$.

C.2 Sample Test Log

The sample test log follows onto the next page.

Flow Research Unit

Test Log-

Supersonic Wind Tunnel

Date

27/03/2019

If In Order	✓
If Faulty	X
No Tests	-

Document No.		SSWT-04
Revision No.		2
Revision Date		2016.05.20

Test	Test #	Compressor Manually Switched Off	Both Bleed Valves Attached to the Dryer are Open	Test Model Secured	Sliding Section Secured with Wing Nuts	Camera Image Number (if Required)	Throat Veeder Counter Value	Venue Clear of Unauthorised Persons	Pressure in Tank	Stagnation Pressure	Successful Run	Comments	Signature	Responsible Person
1	1548	✓	✓	✓	✓	3788	273	✓	1400	250	✓	CC-30I-M32-P-Schlieren (OKE) & OV (YRBChalk 2:1 Halo)		
2	1549	✓	✓	✓	✓	3870	273	✓	1210	250	✓	HC-30I-M32-P-Schlieren (OKE) & OV (WChalk 2:1)		
3	1550	✓	✓	✓	✓	3953	432	✓	1100	250	✓	HC-30I-M28-P-Schlieren (OKE) & OV (YRBChalk 2:1)		
4	1551	✓	✓	✓	✓	4028	273	✓	900	250	✓	CC-30I-M32-P-Schlieren (OKE) & OV (YRBChalk 2:1)		
5	1552	✓	✓	✓	✓	4104	273	✓	1320	250	✓	CC-20I-M32-P-Schlieren (OKE) & OV (ORBChalk 2:1)		
6	1553	✓	✓	✓	✓	4181	273	✓	1200	250	✓	CC-10I-M32-P-Schlieren (OKE) & OV (YRBChalk 2:1)		
7	1554	✓	✓	✓	✓	4260	273	✓	1070	250	✓	CC-0I-M32-P-Schlieren (OKE) & OV (YRBChalk 2:1)		
8														
9														
10														
11														
12														
13														
14														
15														
16														
17														
18														
19														
20														

End of Day Checklist

Item	✓/X
Pressure supply off	
All instrumentation switched off	
Optical equipment covered	
Electrical supply switched off	
Compressor switched off if nobody else is using it	
Bleed valves of dryer open	
The air supply heater turn on	
The test section is open	
Test venue locked	

End of Day

Time of Conclusion

Responsible Person

Signature (RP)

.....

.....

.....

Appendix D : Computational Method and Processing

D.1 Model Independence Results

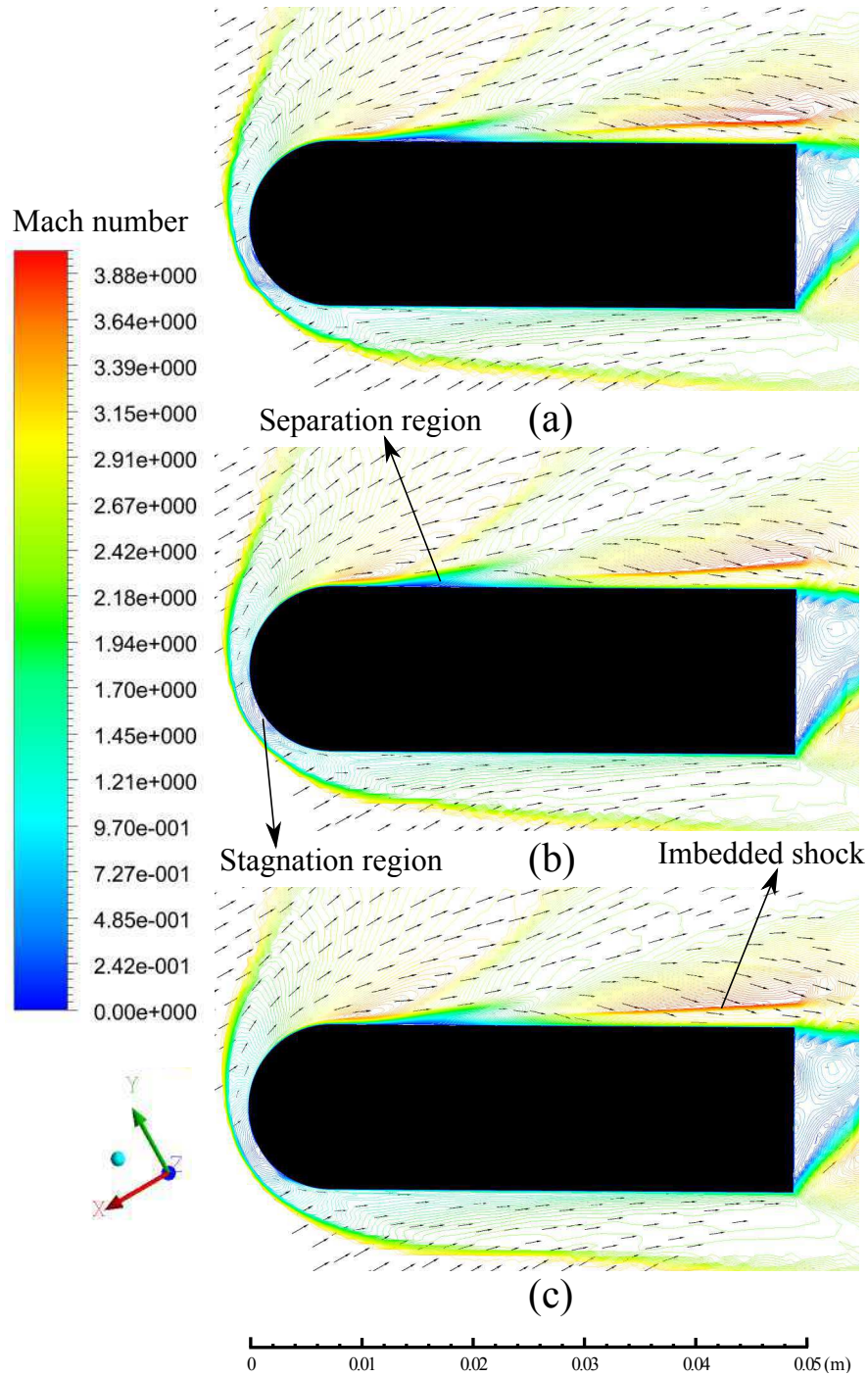


Figure D.1: Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with inflation layer mesh settings of: (a) 60 layers, (b) 75 layers, (c) 90 layers.

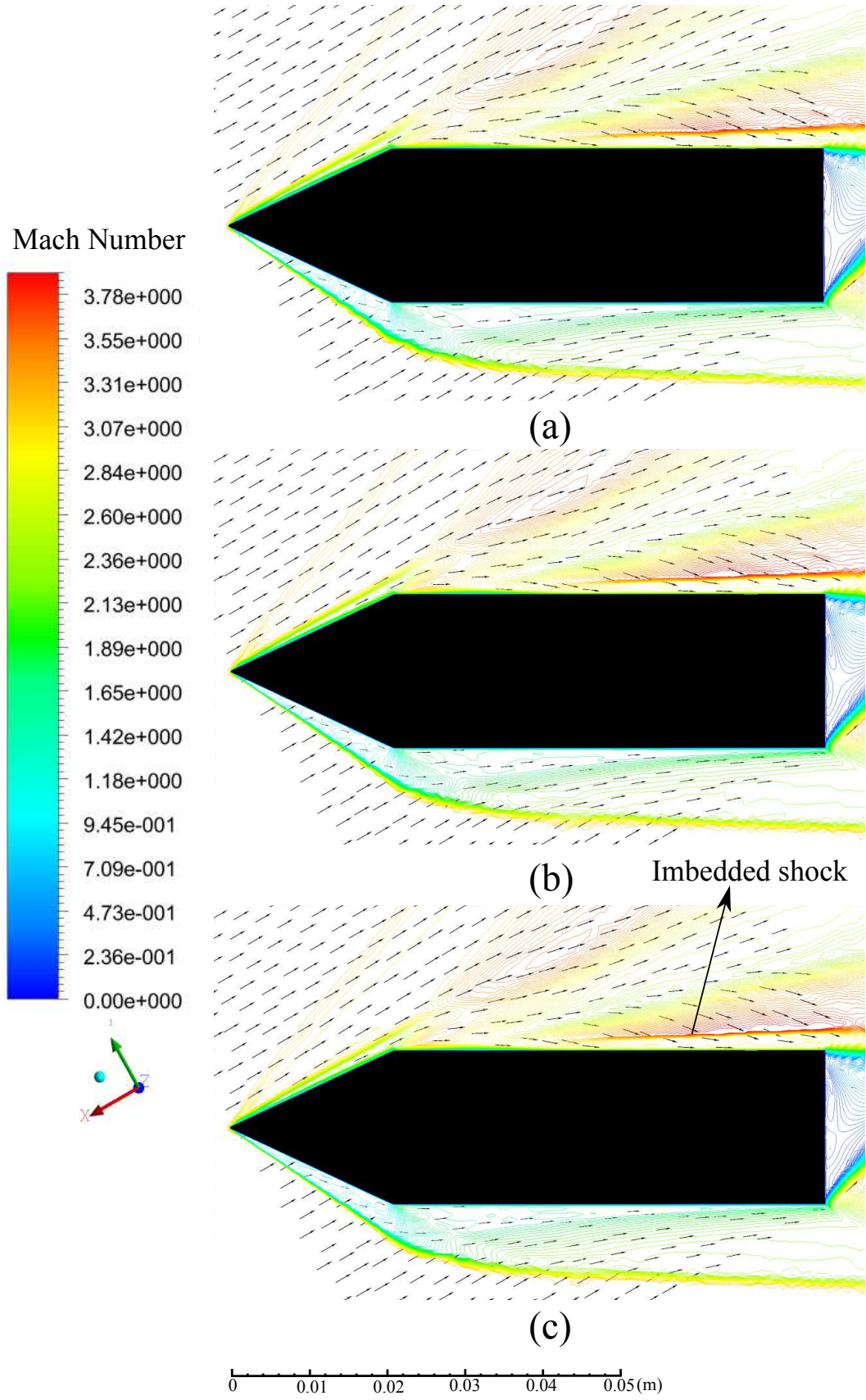


Figure D.2: Mach number contours and vectors of the symmetry plane of the cone-cylinder with inflation layer mesh settings: (a) 60 layers, (b) 75 layers , (c) 90 layers.

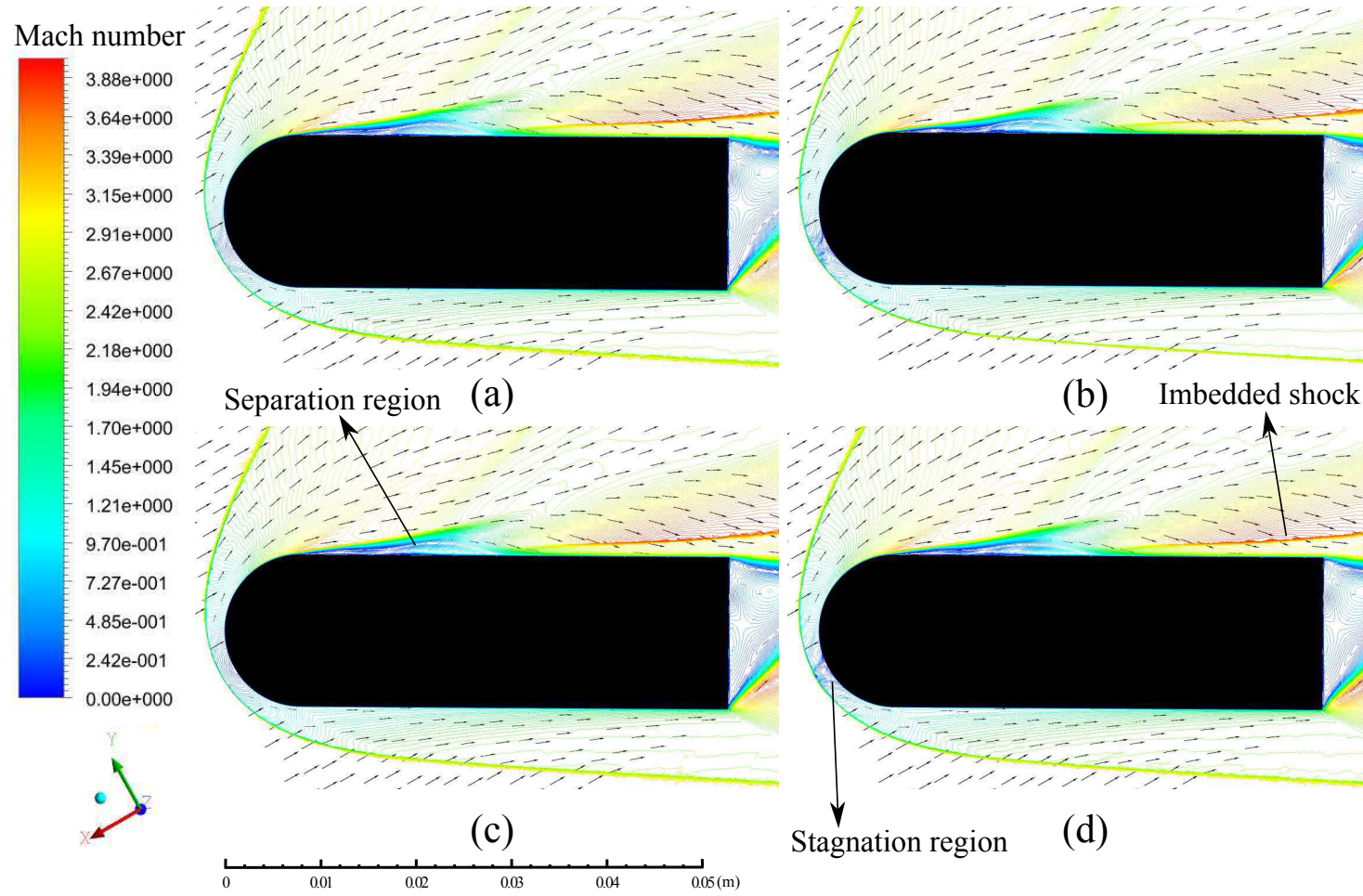


Figure D.3: Mach number contours and vectors of the symmetry plane of the hemisphere-cylinder with local mesh refinement settings: (a) 0.335mm (150 circumferential cells), (b) 0.28mm (180 circumferential cells), (c) 0.25mm (200 circumferential cells), (d) 0.228mm (220 circumferential cells).

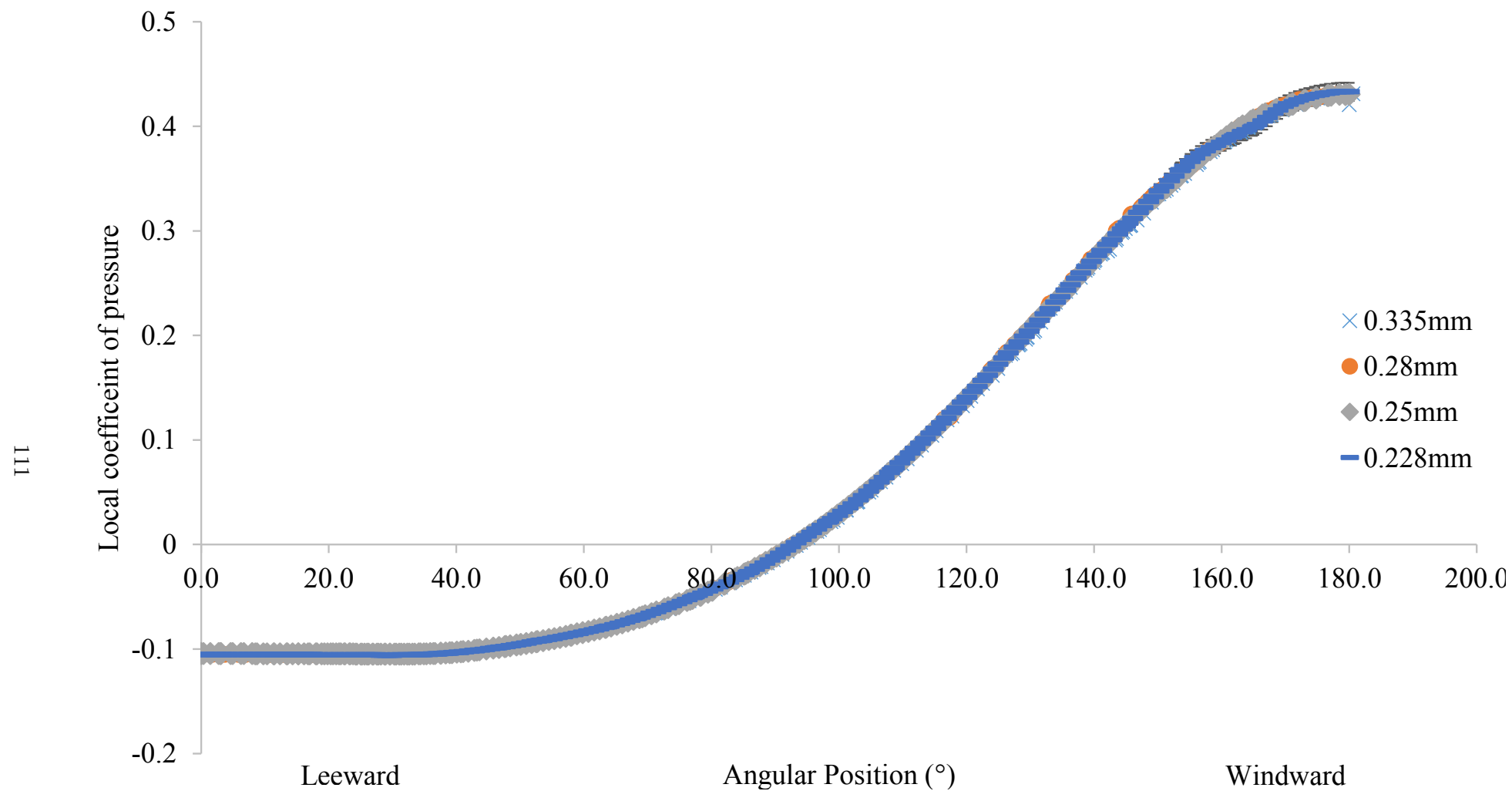


Figure D.4: Coefficient of pressure plot at different circumferential locations at the shoulder of the hemisphere-cylinder.

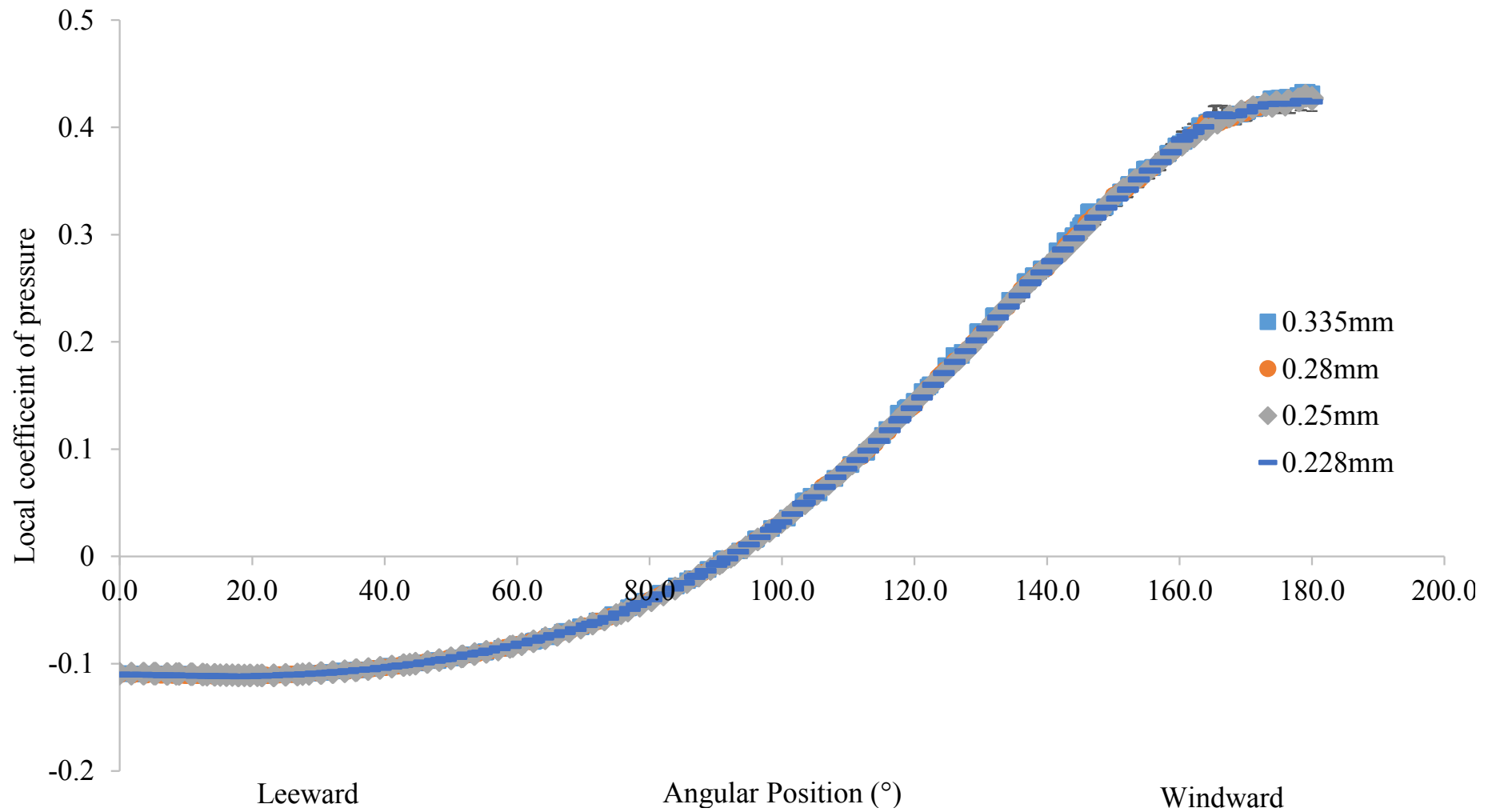


Figure D.5: Coefficient of pressure plot at different circumferential locations at 5mm downstream the shoulder of the hemisphere-cylinder.

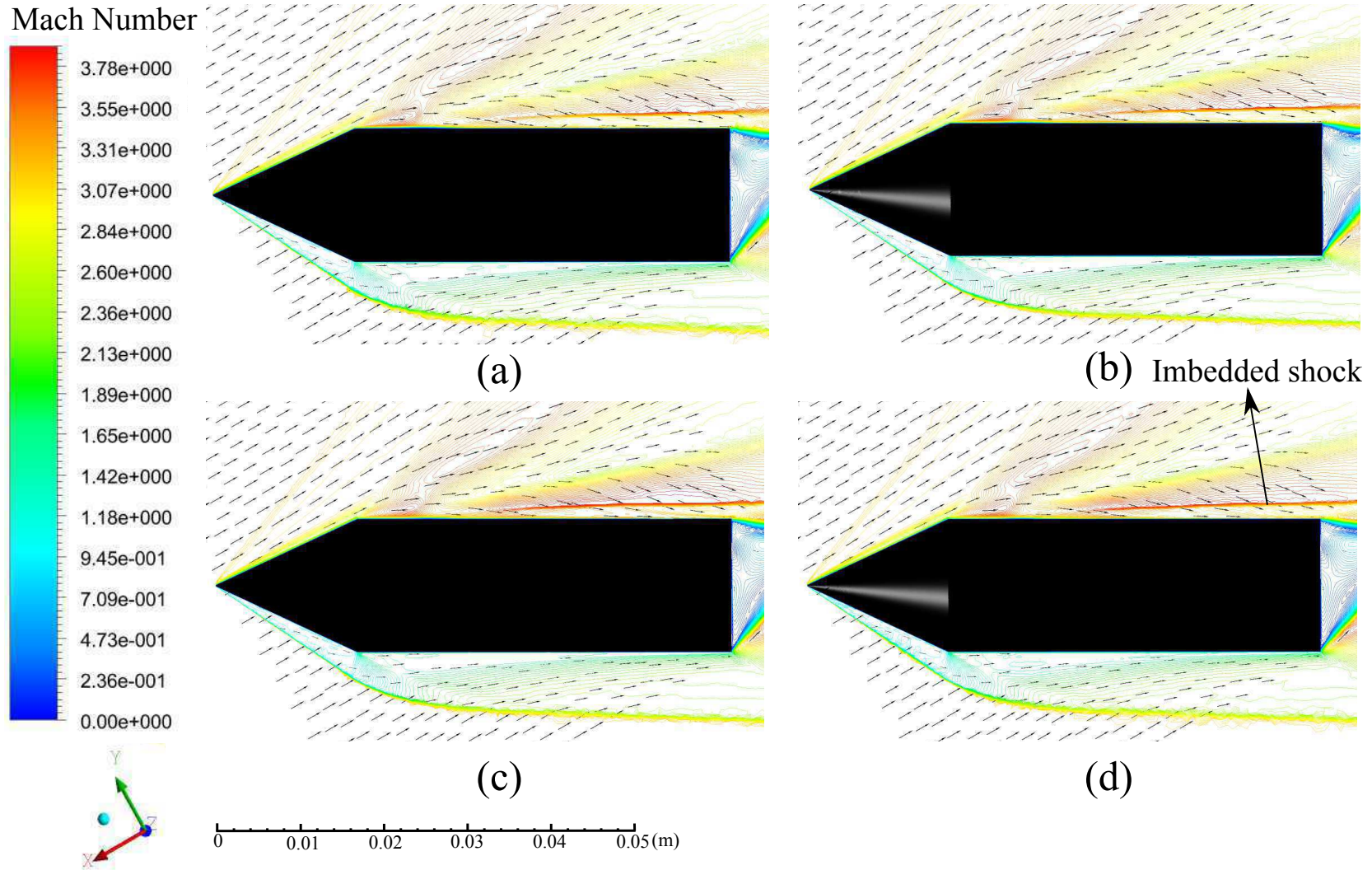


Figure D.6: Mach number contours and vectors of the symmetry plane of the cone-cylinder with local mesh refinement settings: (a) 0.42mm (120 circumferential cells), (b) 0.335mm (150 circumferential cells), (c) 0.28mm (180 circumferential cells), (d) 0.25mm (200 circumferential cells).

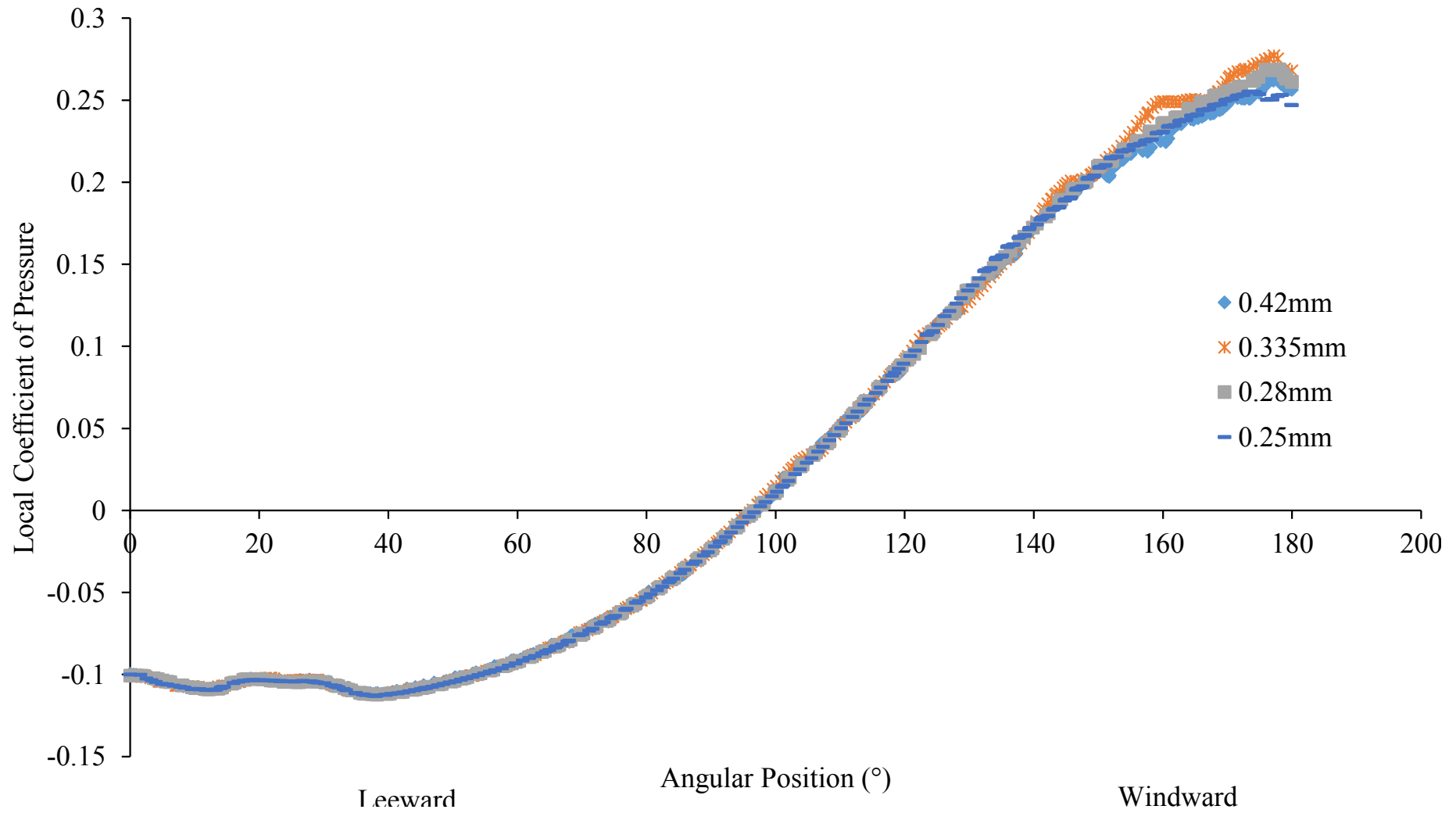


Figure D.7: Coefficient of pressure plot at different circumferential locations at 1mm downstream the shoulder of the cone-cylinder.

D.2 Fluent TUI Commands

```
1| file read-case "/mnt/lustre/users/tchan/Hemi-cylinder/Final/30inc/Hemi30_meshref_028mm.cas"
2| file replace-mesh "/mnt/lustre/users/tchan/Hemi-cylinder/Final/20inc/Hemi20_mesh028mm.msh"
3| file write-case /mnt/lustre/users/tchan/Hemi-cylinder/Final/20inc/Hemi20_M28_028mm
4| define models solver density-based-implicit yes
5| define models steady
6| define models energy yes
7| define models viscous transition-sst yes
8| define models viscous curvature-correction yes
9| define models viscous turbulence-expert kato-launders-model yes
10| define models viscous turbulence-expert production-limiter yes 5
11| define materials change-create air air yes ideal-gas no no yes sutherland three-coefficient-method
    1.716e-05 273.11 110.56 no no no
12| define operating-conditions operating-pressure 0
13| define boundary-conditions pressure-far-field pressure_farfield no 84000 no 2.8 no 293 yes no -1 no 0
    no 0 no no yes no 1 2 5
14| define boundary-conditions fluid domain no no no no no 0 no 0 no 0 no 0 no 1 no no no no no
15| define boundary-conditions symmetry symmetry
16| solve set flux-type 0
17| solve set discretization-scheme amg-c 1
18| solve set discretization-scheme k 1
19| solve set discretization-scheme omega 1
20| solve set discretization-scheme intermit 1
21| solve set discretization-scheme retheta 1
22| solve set high-order-term-relaxation enable yes
23| solve set high-order-term-relaxation options relaxation-factor 0.5
24| solve set courant-number 1
25| solve set under-relaxation k 0.8
26| solve set under-relaxation omega 0.8
27| solve set under-relaxation solid 1
28| solve set under-relaxation turb-viscosity 1
29| solve set under-relaxation intermit 1
30| solve set under-relaxation retheta 1
31| solve monitors residual check-convergence yes yes yes yes yes yes yes yes yes
32| solve monitors residual convergence-criteria 1e-5 1e-5 1e-5 1e-5 1e-7 1e-7 1e-7 1e-7 1e-7
33| solve initialize set-hyb-initialization general-settings 20 0.7 0.7 relative no no no
34| solve initialize hyb-initialization
35| solve set reporting-interval 10
36| solve iterate 7000
37| file write-case-data /mnt/lustre/users/tchan/Hemi-cylinder/Final/20inc/Hemi20_M28_028mm_7000.gz
38| solve execute-commands add-edit curv_tempadapt 4000 "iteration" "adapt mark-with-gradients
    temperature curvature 2 no 0.012 0 no 0 0 0"
39| solve execute-commands add-edit curv_machadapt 4000 "iteration" "adapt mark-with-gradients mach-
    number curvature 2 no 0.012 0 no 0 0 0"
40| solve execute-commands add-edit curv_intermitadapt 4000 "iteration" "adapt mark-with-gradients
    intermittency curvature 2 no 0.02 0 no 0 0 0"
41| solve execute-commands add-edit curv_velocityadapt 4000 "iteration" "adapt mark-with-gradients
    velocity-magnitude curvature 2 no 0.008 0 no 0 0 0"
42| solve execute-commands add-edit grad_pressureadapt 4000 "iteration" "adapt mark-with-gradients
    pressure space-gradient 2 no 0.04 0 no 0 0 0"
43| solve execute-commands add-edit grad_incylinderref 4000 "iteration" "adapt mark-inout-cylinder yes
    no 0.11 0.15 -0.023 0 0 0 0.015"
44| solve execute-commands add-edit set_incylindermask 4000 "iteration" "adapt change-register-type 5"
45| solve execute-commands add-edit grad_sphereref 4000 "iteration" "adapt mark-inout-sphere yes no
    0.15 0 0 0.015"
46| solve execute-commands add-edit set_spheremask 4000 "iteration" "adapt change-register-type 6"
47| solve execute-commands add-edit grad_outcylinderref 4000 "iteration" "adapt mark-inout-cylinder no
    no 0.105 0 -0.023 -0.023 0 0 0.1"
48| solve execute-commands add-edit set_outcylindermask 4000 "iteration" "adapt change-register-type 7"
49| solve execute-commands add-edit combinereg 4000 "iteration" "adapt combine-registers 0 1 2 3 4 5 6
    7)"
```

```

50| solve execute-commands add-edit adaptreg 4000 "iteration" "adapt adapt-to-register 10 0 0 yes"
51| solve iterate 4000
52| file write-case-data /mnt/lustre/users/tchan/Hemi-
   cylinder/Final/20inc/Hemi20_M28_028mm_11000.gz
53| solve iterate 4000
54| file write-case-data /mnt/lustre/users/tchan/Hemi-
   cylinder/Final/20inc/Hemi20_M28_028mm_15000.gz
55| solve iterate 4000
56| file write-case-data /mnt/lustre/users/tchan/Hemi-
   cylinder/Final/20inc/Hemi20_M28_028mm_19000.gz
57| solve execute-commands disable curv_tempadapt
58| solve execute-commands disable curv_machadapt
59| solve execute-commands disable curv_intermitadapt
60| solve execute-commands disable curv_velocityadapt
61| solve execute-commands disable grad_pressureadapt
62| solve execute-commands disable grad_incyylinderref
63| solve execute-commands disable set_incyylindermask
64| solve execute-commands disable grad_sphereref
65| solve execute-commands disable set_spheremask
66| solve execute-commands disable grad_outcylinderref
67| solve execute-commands disable set_outcylindermask
68| solve execute-commands disable combinereg
69| solve execute-commands disable adaptreg
70| solve set convergence-acceleration-for-stretched-meshes yes 50
71| solve iterate 2500
72| solve set convergence-acceleration-for-stretched-meshes yes 75
73| solve iterate 2500
74| file write-case-data /mnt/lustre/users/tchan/Hemi-
   cylinder/Final/20inc/Hemi20_M28_028mm_24000.gz

```

Appendix E : Additional Results

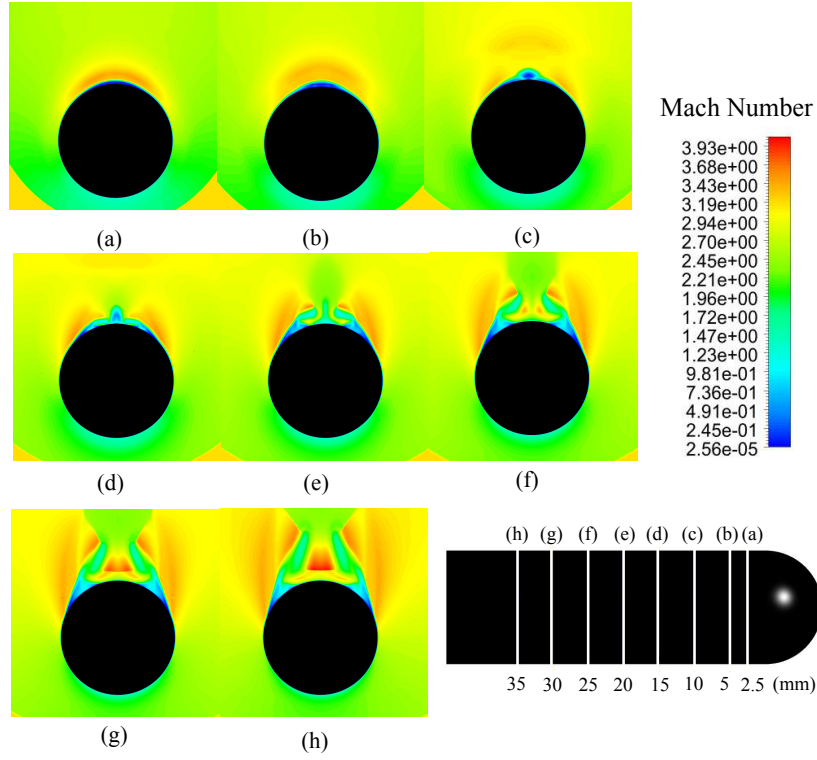


Figure E.1: Mach contours of hemisphere-cylinder with 30° incidence and M3.2 flow conditions at cross sections located with reference to the shoulder: (a)2.5mm, (b)5mm, (c)10mm, (d)15mm, (e)20mm, (f)25mm, (g)30mm, (h)35mm.

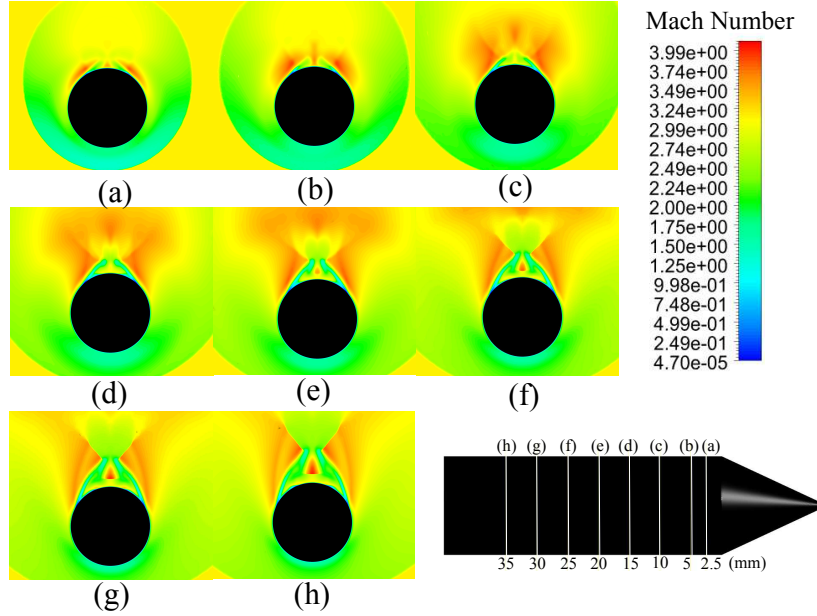


Figure E.2: Mach contours of cone-cylinder with 30° incidence and M3.2 flow conditions at cross sections located with reference to the shoulder: (a)2.5mm, (b)5mm, (c)10mm, (d)15mm, (e)20mm, (f)25mm, (g)30mm, (h)35mm.

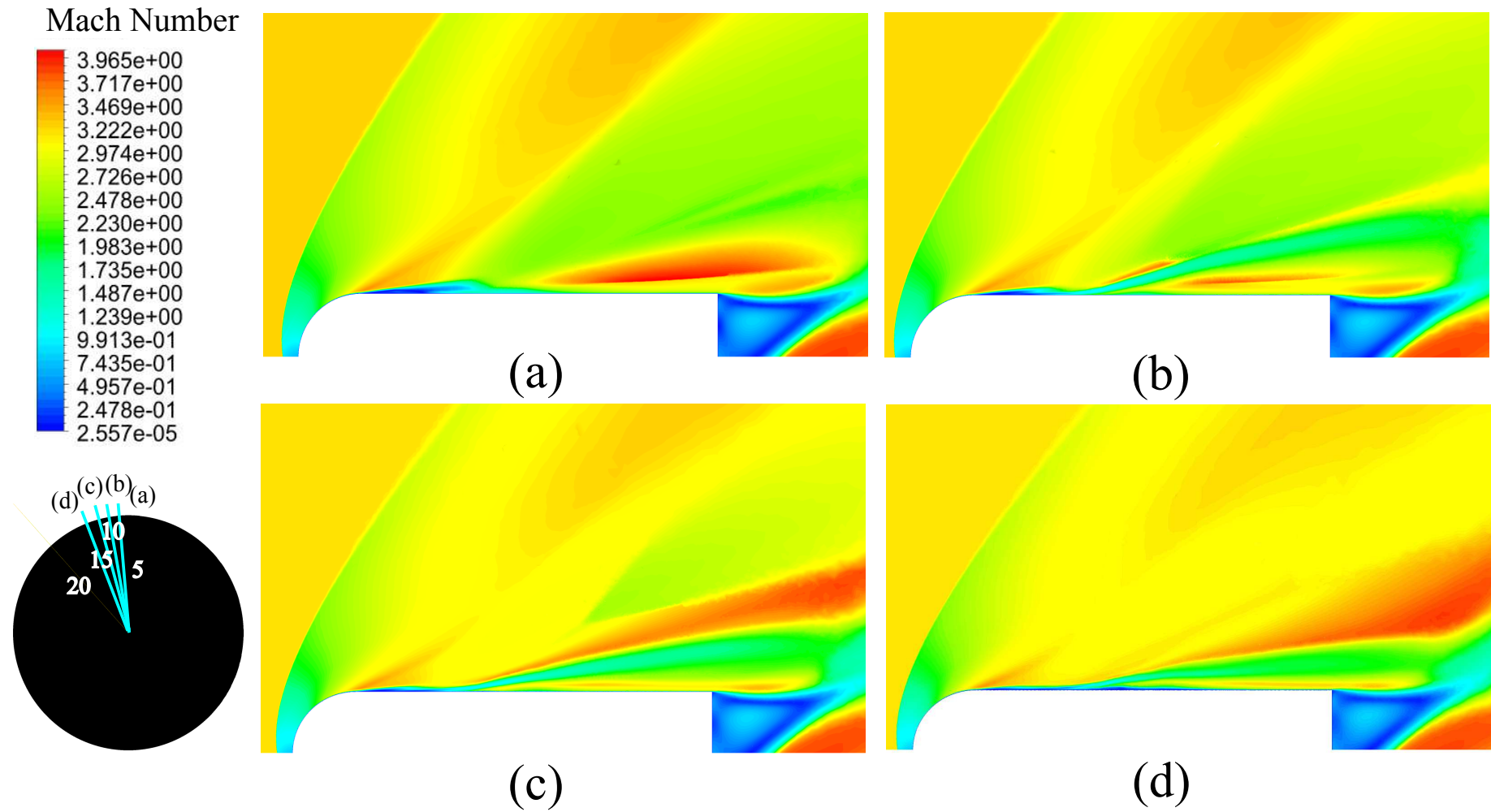


Figure E.3: Mach contours of hemisphere-cylinder with 30° incidence and M3.2 flow conditions at meridional plane of: (a) 5° , (b) 10° , (c) 15° , (d) 20°

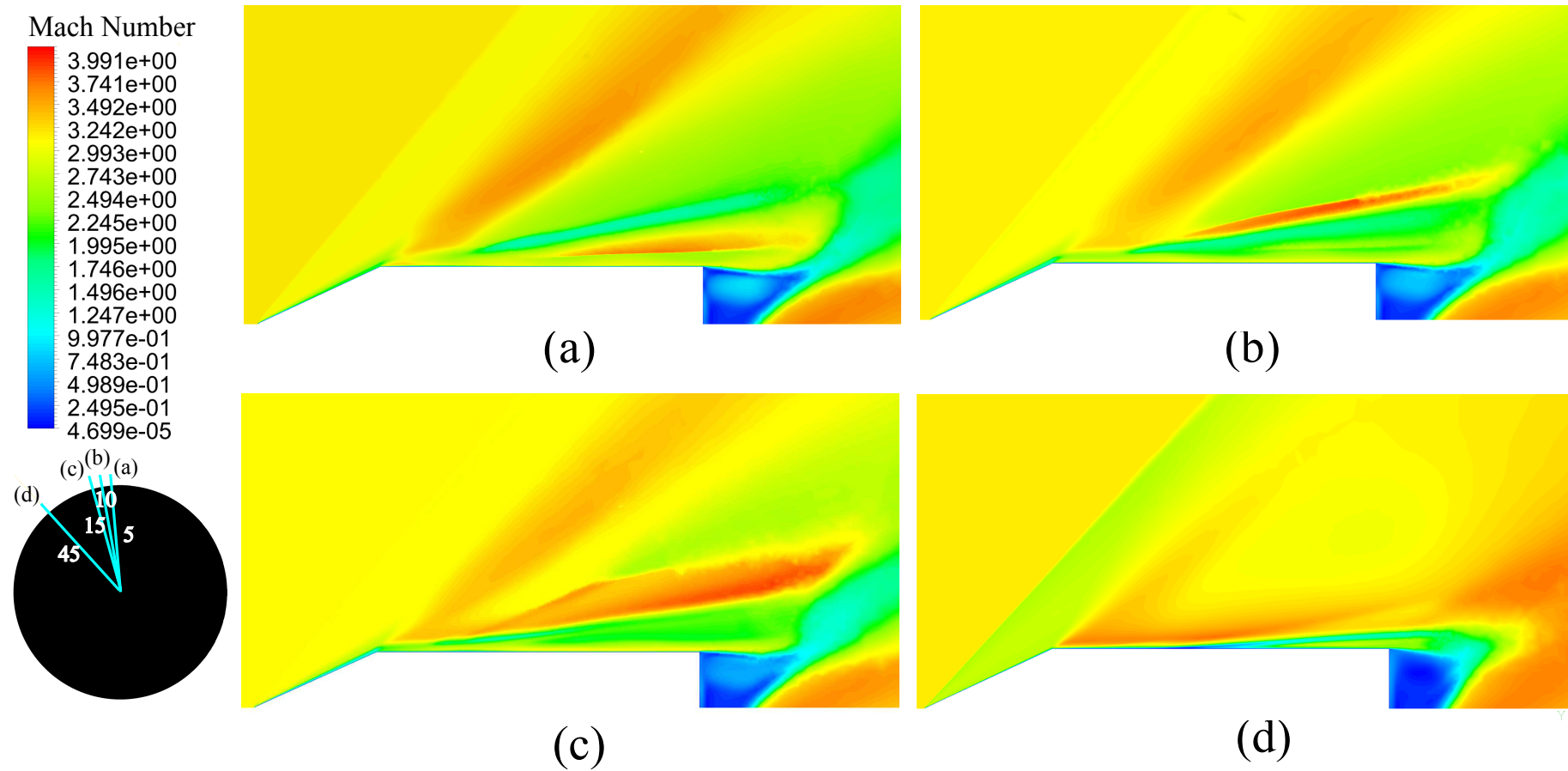


Figure E.4: Mach contours of cone-cylinder with 30° incidence and M3.2 flow conditions at meridional plane of: (a) 5° , (b) 10° , (c) 15° , (d) 45°

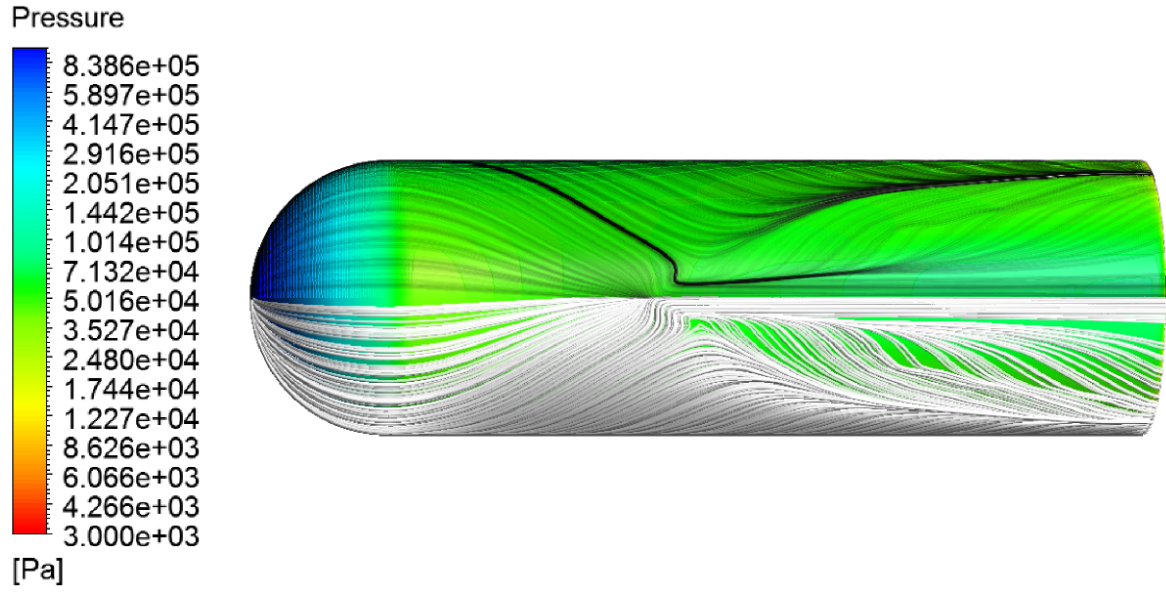


Figure E.5: Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 10° incidence and M3.2 flow conditions

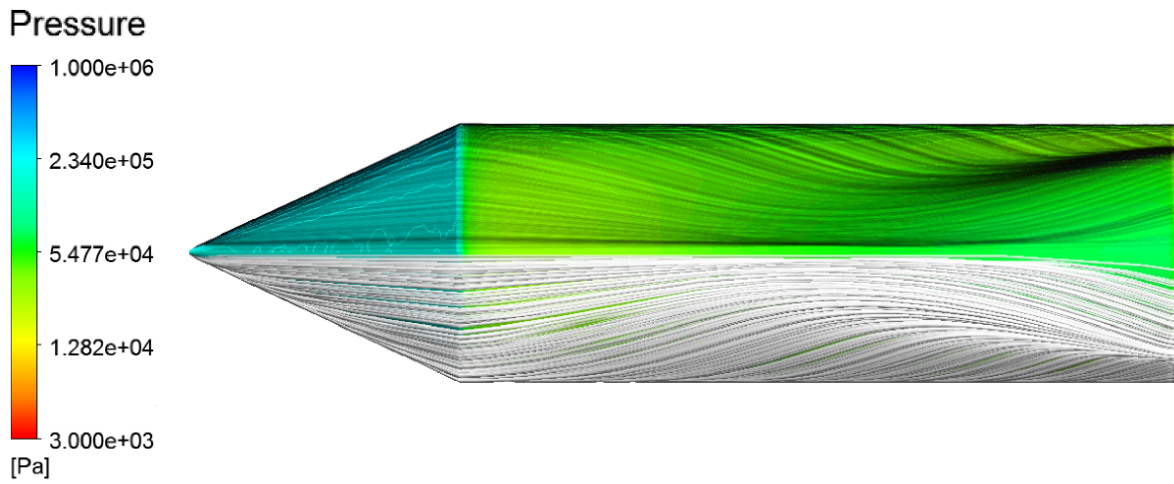


Figure E.6: Wall shear surface streamlines and pressure contours of the cone-cylinder at 10° incidence and M3.2 flow conditions

Appendix F : Results for Future Research

The purpose of this section is to explore potential research topics that is related to supersonic flow separation for slender bodies with emphasis on the relationship of between the flow separation mechanism and the angle of incidence. As determined from this research, the angle of incidence plays a role in the initiation of the nose separation which affects the vortical structures developed downstream of the shoulder of the slender bodies. The angles of incidence that initiated nose flow separation were explored for the cone-cylinder and hemisphere-cylinder, where the cone-cylinder cone deflection angle was constrained to 25° . The geometry configuration and flow conditions that were considered are as follows:

- Hemisphere-cylinder with 15° angle of incidence at M3.2
- Hemisphere-cylinder with 18° angle of incidence at M3.2
- Cone-cylinder with 25° angle of incidence at M3.2

The flow topology results are seen in figures F.1, F.2, and F.3. Each configuration consists of cross flow separation where a distinct flow separation line can be determined. The transition nose flow separation characteristics are seen for figures F.1 and F.3, where surface streamlines are converging and potentially forming separation lines.

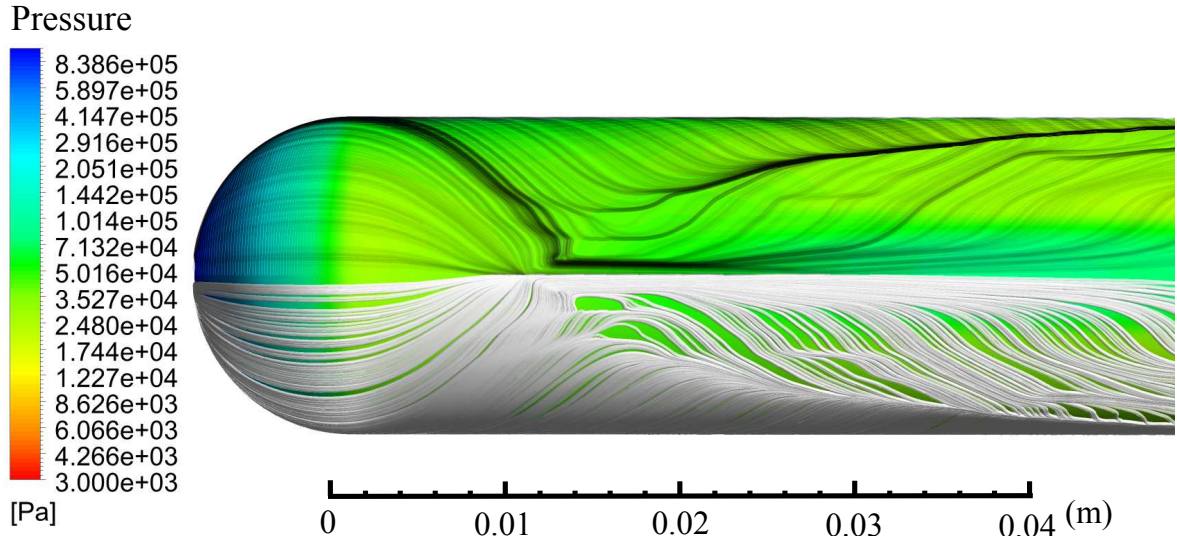


Figure F.1: Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 15° incidence and M3.2 flow conditions

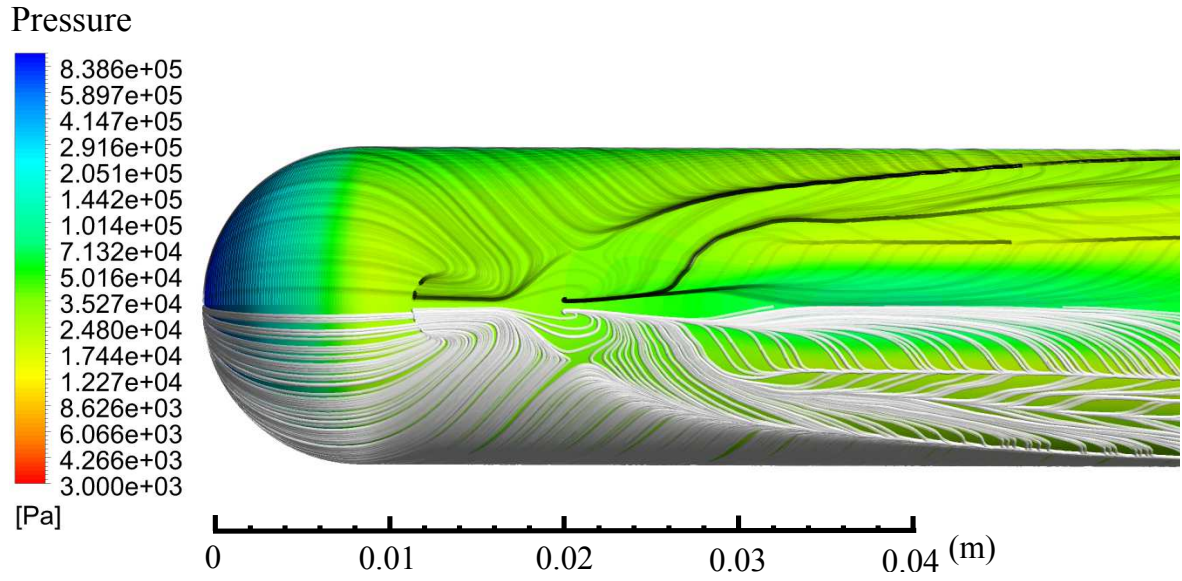


Figure F.2: Wall shear surface streamlines and pressure contours of the hemisphere-cylinder at 18° incidence and M3.2 flow conditions

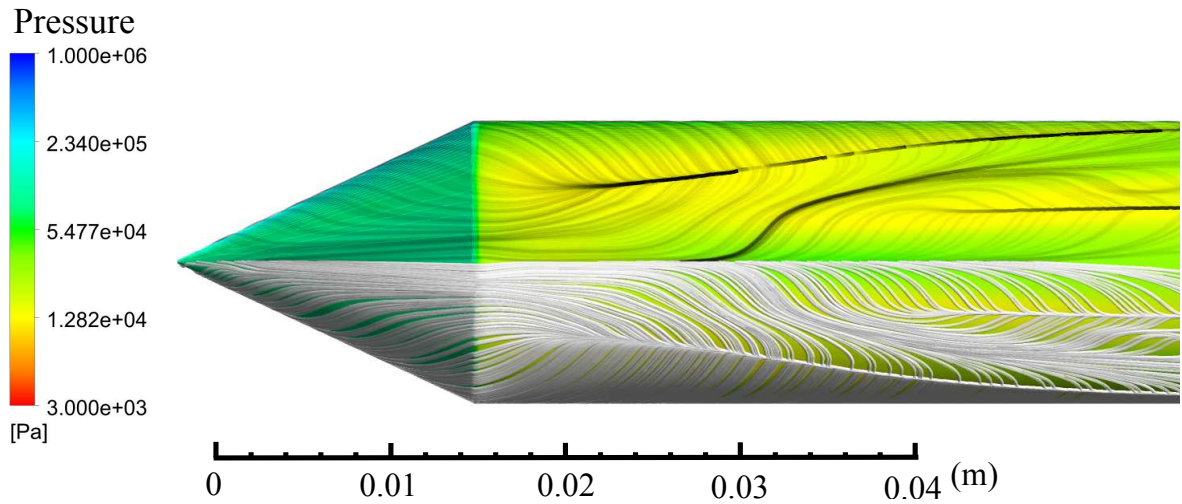


Figure F.3: Wall shear surface streamlines and pressure contours of the cone-cylinder at 25° incidence and M3.2 flow conditions