

Experimental and Numerical Studies of Shock Wave – Boundary-Layer Interactions over a Cone-Cylinder-Flare in Hypersonic Flows

Benjamin W. Arfi*, Erwan A. B. Corcoral†, Spyros E. Tsentis‡ and Simon A. Prince§
School of Aerospace, Transport and Manufacturing, Cranfield University, UK, MK43 0AL

Zeeshan A. Rana¶
Department of Mechanical Engineering, Prince Mohammad Bin Fahd University, Al-Khobar, KSA

The present study provides experimental and numerical results for turbulent shockwave boundary layer interactions (SBLIs) over a blunt cone-cylinder-flare geometry, on which there is a lack of available data. Experiments were conducted in the Cranfield hypersonic gun tunnel, at Mach 8.2. Two configurations with varying nose length were studied using experimental testing and computational fluid dynamics (CFD). Three dimensionality has been generated by placing the model at incidence. The objective was to extend the range of data available for hypersonic flows and contribute to the design of re-entry vehicles. Experimental results cover forces and moments, that the numerical study aimed to replicate with Reynolds Averaged Navier-Stokes (RANS) modeling, and extended by analyzing the separated flow regions over the model. Additionally, the effect of the nose length on the aerodynamic forces and boundary layer transition were presented.

I. Nomenclature

Symbols

A_{base}	=	Base area of the flare (m^2)
A_{sting}	=	Area of the sting (m^2)
A_{wet}	=	Base wetted area (m^2)
C_A	=	Axial force coefficient
C_B	=	Base pressure coefficient
C_N	=	Normal force coefficient
D_c	=	Cylinder diameter (mm)
L_c	=	Cylinder length (mm)
k	=	Turbulent kinetic energy ($m^2 s^{-2}$)
M_∞	=	Freestream Mach number
P_∞	=	Freestream pressure (bar)
P_0	=	Stagnation pressure (bar)
P_1	=	Barrel pressure (bar)
P_4	=	Driver vessel pressure (bar)
Re_∞	=	Freestream Reynolds number
T_∞	=	Freestream temperature (K)
T_0	=	Stagnation temperature (K)
T_w	=	Wall temperature (K)
y^+	=	Non-dimensional wall distance

*Researcher, Center for Aeronautics, Cranfield University, UK.

†Researcher, Center for Aeronautics, Cranfield University, UK.

‡Research Assistant, Center for Propulsion and Thermal Power Engineering, Cranfield University, UK.

§Professor of Aerodynamics, Center for Aeronautics, Cranfield University, UK; Senior Member AIAA.

¶Associate Professor of Mechanical Engineering, Prince Mohammad Bin Fahd University, Al-Khobar, KSA; Member AIAA.

Greek Letters

γ	=	Heat capacity ratio
ε	=	Turbulent dissipation rate ($\text{J kg}^{-1} \text{s}^{-1}$)
ξ	=	Distance from the cylinder-flare junction (mm)
ρ	=	Density (kg m^{-3})
ω	=	Specific turbulent dissipation rate (s^{-1})

Abbreviations

AUSM	=	Advection Upstream Splitting Method
CFD	=	Computational Fluid Dynamics
GCI	=	Grid Convergence Index
MUSCL	=	Monotonic Upstream-centered Scheme for Conservation Laws
QCR	=	Quadratic Constitutive Relation
RANS	=	Reynolds Averaged Navier-Stokes
SBLI	=	Shockwave Boundary Layer Interaction
SST	=	Shear Stress Transport

II. Introduction

HYPERSONIC flight is a burgeoning technology, for both military and commercial industry. Initially launched during the cold war, the arms race and the space race gave birth to the first manned hypersonic flight in April 1961, with Yuri Gagarin. Both Eastern and Western Blocs acknowledged the significant difficulties posed by hypersonic flows. One of the most challenging aspects with these flows is the interaction between shock waves and the boundary layer along the body. These phenomena can significantly degrade vehicle performance, as they result in substantial aerodynamic loads, heating, and additional drag. Moreover, shock waves represent the most extreme adverse pressure gradient generators, and can cause the separation of the boundary layer, along with complex flow features. Investigating SBLIs on complex geometries is a highly challenging task. Consequently, many studies focused on canonical configurations, such as two-dimensional compression ramp, hollow-cylinder flares, and double-cones, in order to gain a better understanding of the flow field. A series of experiments investigated laminar flows involving compression ramps [1–3], and double-cone and hollow-cylinder flare models [4, 5]. Additionally, it is relatively easier to simulate laminar flows than transitional or turbulent ones [6] when it comes to CFD. Therefore, these experimental results were critical in assessing the accuracy of CFD codes to predict key parameters acting on the body of hypersonic vehicles, namely the pressure distribution and heat transfer.

Turbulent boundary layers, on the other hand, are less prone to separation, but impose considerably more severe conditions than laminar boundary layers in terms of temperature and aerodynamic loads. However, it is difficult to produce fully turbulent boundary-layers experimentally, as few hypersonic wind tunnels can achieve very high Reynolds numbers. Holden [7] analysed the unsteadiness of a fully turbulent boundary layer on an axisymmetric configuration, with the model size chosen to ensure that the transition occurs well ahead of the interaction region. Holden demonstrated in a previous study [8] that the turbulent separation region is highly unsteady, with a length of a fraction of the boundary layer thickness. It is of critical importance to ensure that the flow is fully turbulent to prevent the instabilities caused by the transition from interfering with the observations of the unsteady effects in the separated region[9]. In hypersonic wind tunnels such as the Cranfield Gun Tunnel, where the flow is in the transitional regime [10], the transition must be triggered by means of tripping in order to investigate a fully turbulent boundary layer. The turbulent boundary layer created by forcing transition develops strong vortices, and is thicker than a natural boundary-layer [11]. Hoffman et al. [12] investigated the effects of surface roughness on SBLIs, and the flow structure using modal analysis techniques. They found that the boundary-layer thickness was increased by up to three times its initial height due to surface roughness. Axisymmetric configurations, such as the cone-cylinder-flare, are optimal for the investigation of two-dimensional flows. In contrast to flat plates, these configurations are not susceptible to sidewall effects. However, the time-averaged two-dimensionality is compromised by the appearance of Görtler vortices, especially when there is flow separation [13]. This mechanism is a three-dimensional system of streamwise alternating vortices. Cao et al. [14] observed that prior to reattachment, the Görtler instability is the dominant phenomenon, but it rapidly dissipates after reattachment.

Consequently, it is challenging to identify the various unsteady mechanisms that govern three-dimensional SBLIs, with the streamwise and spanwise structures influencing those present in the separation region. Vanstone and Clemens [15] investigated the instabilities present in a three-dimensional SBLI using a swept ramp at Mach 2. Their findings indicated that spanwise structures driven by cross flows interact with the flow structures present in the boundary layer.

The present paper focuses on the cone-cylinder-flare geometry, that has been previously considered for space and defense programs. A parametric study was carried out by Ashby and Cary [16] in 1965, and analyzed the effect of nose bluntness, cylinder length, flare angle and length using experiments and low order methods. This paper aims at expanding the previous study by providing new data points and a new understanding of the flow physics using RANS CFD. Even though RANS modeling is known to provide less accurate predictions than higher order methods such as Large Eddy Simulation, Zheltovodov [17] demonstrated that RANS could capture the main flow features on simple geometries for weak SBLIs. However, stronger ones are less accurately captured. Knight *et al.* related this issue to a poor prediction of the separated flow region in the case of a 2D compression ramp [18], but it can be assumed to be also valid for simple 3D cases. Several studies aimed at improving RANS turbulence models to solve this issue. Among them, Thivet showed that introducing non-linearities by accounting for secondary flows improved the prediction of pressure and skin friction [19], and therefore the prediction of the separated flow regions. Several studies compared the different RANS turbulence models: Zheltovodov [17] demonstrated that $k - \varepsilon$ models are not suited for hypersonic external flows, that are better predicted by $k - \omega$ models. Holden *et al.* compared experimental results on separated flow regions on the HIFIRE 1 geometry with CFD predictions obtained using Spalart-Allmaras (SA) and $k - \omega$ SST turbulence models [20], and demonstrated that SA underestimates the separation region whereas SST tends to overpredict it. In a more recent study, Cross and West compared results given by Spalart-Allmaras and $k - \omega$ SST with experiments carried out at CUBRC on simple geometries [21]. Different gas models were evaluated, and this study confirmed the analysis of Holden *et al.* [20] on separated flow regions. However, SST demonstrated an overall better prediction of the pressure and heat flux trends than SA, thus making SST a better choice for the type of flows studied in this paper.

This study presents the experimental and RANS CFD results for the axial and normal forces exerted on an axisymmetric cone-cylinder-flare with forced transition and two different cylinder lengths. High-speed schlieren imaging is provided for a qualitative analysis of the flow features, and compared with CFD to give an insight on the complex flow physics. A CFD study of the separated flow region completes this analysis.

III. Experimental Procedure

A. Wind Tunnel Facility

The experiments were conducted in the Mach 8.2 Cranfield Hypersonic Gun Tunnel. A detailed description of the design, performance and operation of the wind tunnel is provided by Needham [22]. This is a short duration wind tunnel in which the driver gas, initially at 138 bar, is heated and compressed by a free piston. The diaphragm section is composed of two 2.5 mm thick diaphragms made of a light alloy H14 Grade, pressurized at 69 bar. The barrel is at atmospheric pressure (1 bar), and possesses a length of 6.1 m and a diameter of 0.08 m. The piston is placed in the barrel just downstream the diaphragm which separates the regions of high and low pressures. The pressure in the dump tank is set to 266 Pa. Firing the tunnel vents the diaphragm section to atmospheric pressure and causes instantaneous rupture of the diaphragms; the piston is accelerated through the barrel and compresses the driver gas. The gas is then expanded through the nozzle and enters the working section at Mach 8.2. The facility allows to reach a stagnation temperature T_0 of 1290 K and the test duration is between 20 ms and 80 ms.

Table 1 Cranfield Hypersonic Gun Tunnel. Flow conditions used in the present study.

M_∞	P_4 (bar)	P_1 (bar)	P_0 (bar)	P_∞ (bar)	T_0 (K)	T_∞ (K)	T_w (K)	Re_∞ (m^{-1})
8.2	138	1.0	108.9	0.0095	1290	89.3	293 ± 1.1	9.44×10^6

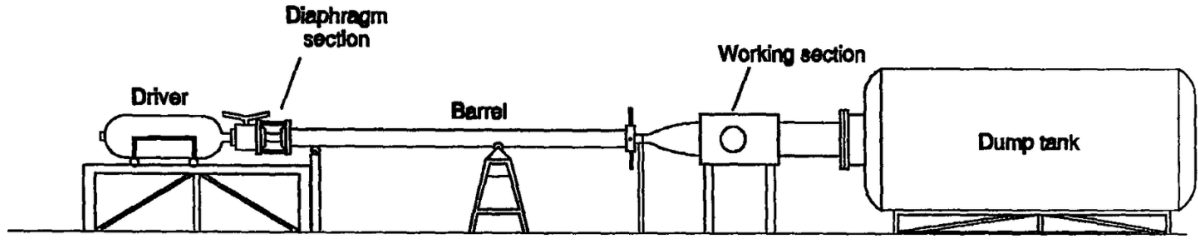


Fig. 1 Schematic of the Cranfield Hypersonic Gun Tunnel [10].

B. Model Geometry

Murray et al. [13] argued that the best candidate to generate two-dimensional flows is a body of revolution at zero incidence. This allows the elimination of parasitic three-dimensional interactions due to end plates. The configuration used for the experiments consists of a cylinder with a flare with a deflection of 20° . The cylinder is prolonged by a blunt nose. Two models were investigated, with the only variable being the length of the cylinder, as shown in Fig. 2. The models are described by the ratio of their cylinder length, L_c , over their cylinder diameter, D_c . Three-dimensionality was generated by pitching the model at different angles of attack. According to Hillier et al. [23], it is "a very interesting case [...] but exceptionally difficult to document". This constituted an additional source of motivation for the authors to provide data on three-dimensional flows with complex boundary layers.

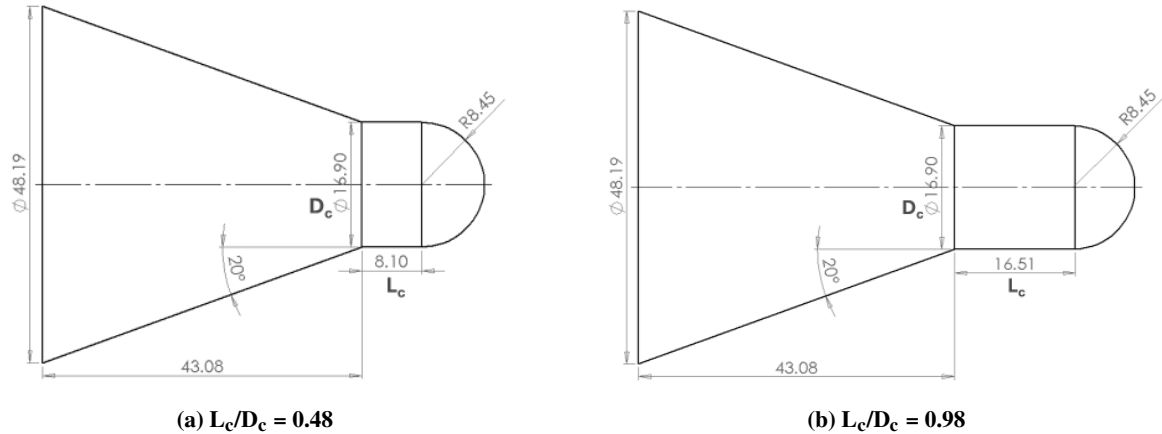


Fig. 2 Blunted nose cone-cylinder-flare geometries used in the experiments.

Transition was forced using a 0.1 mm thick trip wire placed on the nose at half radius to create turbulent flow, as presented in Fig. 3.

C. Experimental Methodology

The angles of attack investigated in the study were 0° , -2.5° , -5° and -7.5° . However, the -7.5° case was not investigated with the $L_c/D_c = 0.98$ model as the bending moment could break the setup. Fig. 4 illustrates the experimental rig used to measure the forces and moments. The tests used the MK3 to measure the normal and axial forces, a three component strain gauge balance, which is described extensively by Opatowski [24]. Additionally, the laboratory was equipped with a schlieren system to visualize the density variations during the runs. The system comprises a horizontal knife edge and a polarizer. The camera allowed to capture images in real time at 4000 fps. Both data acquisition and schlieren videos are triggered with the noise made by the piston hitting the entrance of the nozzle when firing the tunnel.

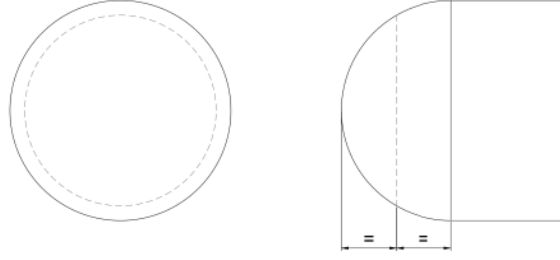


Fig. 3 Trip wire location on the nose (dashed lines).

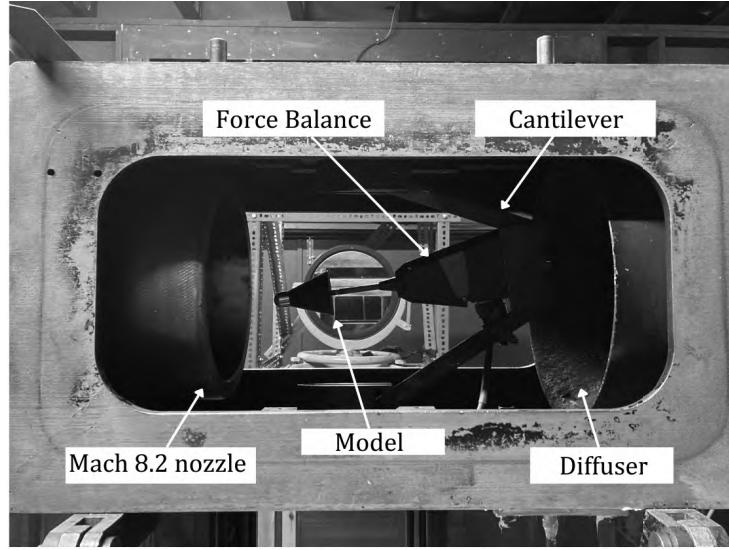


Fig. 4 Model mounted in the test section for forces and moments study.

IV. Numerical Procedure

A. Computational Domain

A structured grid has been generated for each model using Fidelity Pointwise. For each case, assuming that the flow is symmetric at low incidence, only half of the model has been meshed. Additionally, as the focus of this study is on the cylinder-flare junction, the wake has not been simulated, and the computational domain ends at the base of the model. The computational domain is represented in Fig. 5, as well as the boundary conditions: free stream with the static temperature T_∞ , static pressure p_∞ and Mach number M_∞ , a turbulence intensity of 0.01 and a turbulence viscosity ratio of 10. The freestream conditions are given in Tab. 1. The pressure outlet is defined with the static pressure p_∞ , and the same values of turbulence intensity and viscosity ratio. The wall has a no-slip condition and a wall temperature T_w equal to that of the room during the experiments.

Views of the mesh are provided in Fig. 6. The grid has especially been clustered on the nose, to correctly capture the stagnation point and development of the boundary layer, and at the cylinder-flare junction, to capture the separation and reattachment lines. The wall distance was chosen to maintain a non-dimensional value of $y^+ = 1$.

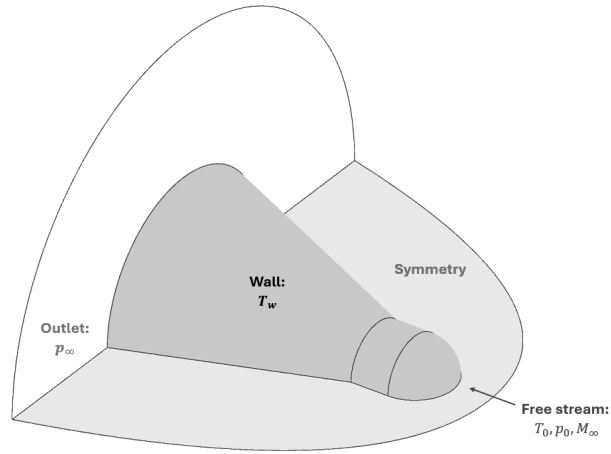


Fig. 5 Computational Domain.

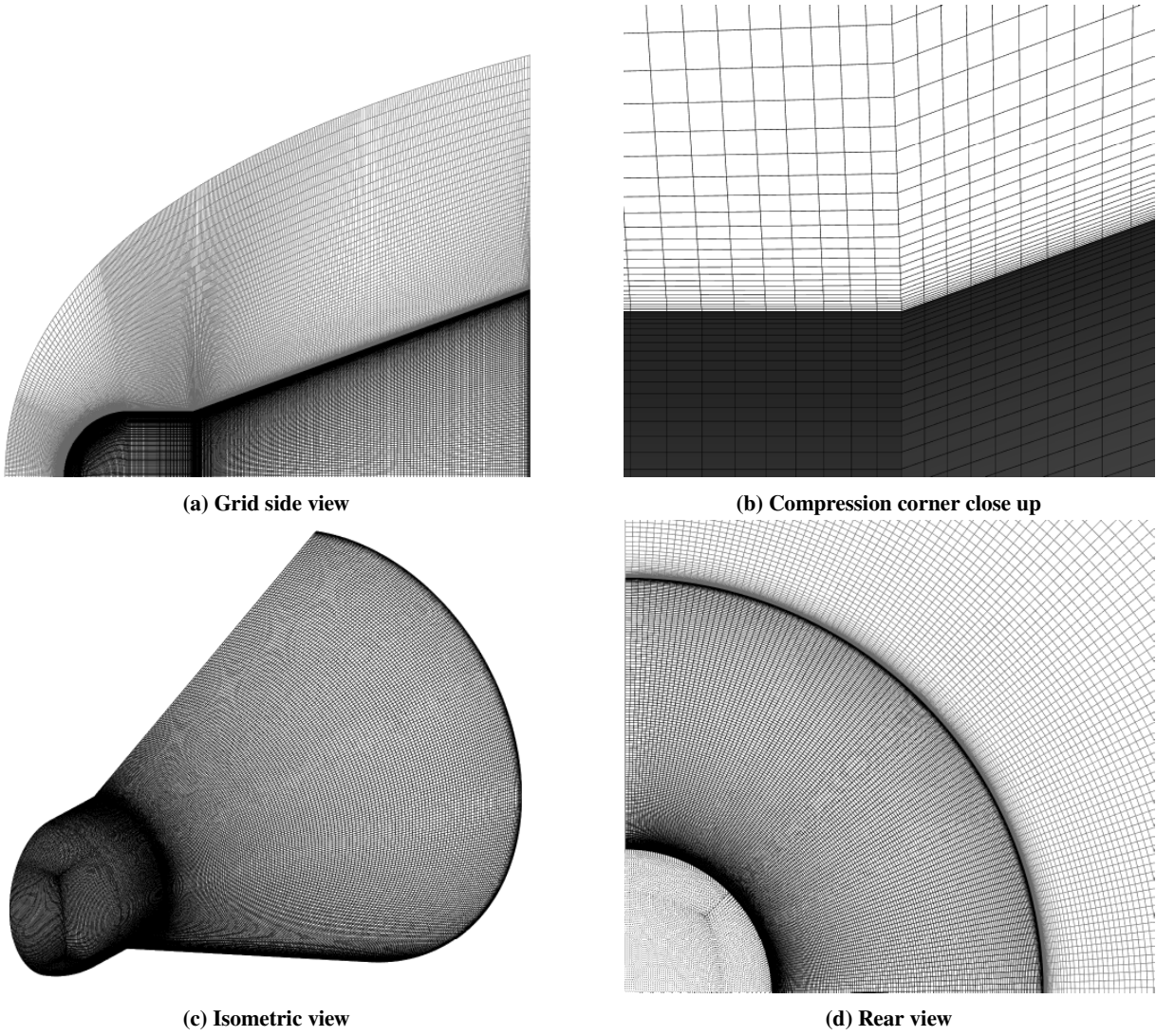


Fig. 6 Different views of the mesh.

B. Solver Setup

To simulate the present cases, the flow is assumed to be fully turbulent, and the forced transition due to the tripping is not modelled. The computational model is based on the Reynolds-averaged Navier-Stokes (RANS) equations, describing the conservation of mass, momentum and energy, completed with the $k - \omega$ Shear Stress Transport (SST) turbulence model. The latter is corrected for hypersonic flow by changing the a_1 coefficient to $a_1 = 0.355$ [25]. The Quadratic Constitutive Relation (QCR) is selected to account for secondary flows, which improves the match with experimental data in terms of pressure and skin friction distributions [19]. Another option could have been the Spalart-Allmaras turbulence model, but it was proven to underpredict separated flow regions, and provide a less accurate prediction of pressure and heat transfer profiles compared to the $k - \omega$ SST model [21].

The AUSM+ FVS inviscid flux is selected, as it performs better than Roe based fluxes at such regimes, and is more robust to the carbuncle phenomenon [26, 27]. The air and its properties are modeled using the equilibrium air real gas model from STAR-CCM+. It is based on curve fits by Gupta [28], and calculates the thermodynamic properties using pressure and temperature. The model is valid up to 30,000 K and 10^2 atm. The experimental temperature is within the model range of validity and the experimental pressure is slightly above the limit, but this model is considered sufficient to describe the air behaviour. The space discretization is ramped up from 1st order (7,500 iterations) to 2nd (7,500 iterations) and then 3rd order *Monotonic Upstream-centered Scheme for Conservation Laws* (MUSCL) (15,000 iterations).

To assess the discretization errors, a grid convergence index (GCI) [29] study has been conducted. For each grid, 3 levels of refinement have been created, and at each level the mesh has been uniformly refined. Thus, grids of 2.5M, 5M and 10M cells were created for the model $L_c/D_c = 0.48$, and grids of 3M, 6M and 12M for the model $L_c/D_c = 0.98$. The parameters studied were the axial and normal forces, separation and reattachment point location in the symmetry plane, on the windward surface of the model. The outcome of this preliminary study is presented in Table 2. GCI_{21} for the model $L_c/D_c = 0.48$ is systematically below 0.2%, which is deemed low enough to choose a medium mesh of 5M cells. However, higher discrepancies between the medium and fine grids for the model $L_c/D_c = 0.98$, especially for the separation and reattachment locations, result in a GCI_{21} higher than 0.5% and therefore require a finer mesh of 12M cells.

Table 2 Results of the GCI study for both models.

	Axial force	Normal force	Separation location	Reattachment location
$GCI_{21}^{L_c/D_c=0.98}$	0.091%	0.123%	0.588%	0.523%
$GCI_{21}^{L_c/D_c=0.48}$	0.00004%	0.195%	0.005%	0.113%

V. Methodology

A. Corrections to Data

1. Base Pressure Corrections

The contribution from the base pressure can affect the overall pressure distribution and therefore the measured axial force. To compensate for the presence of the sting during the experiments, or the absence of wake in the numerical simulations, the base pressure is calculated using the semi-empirical method of Gabeaud [30], which is adequate for bodies with no fin and no boat-tailing [31] in Mach number ≥ 3 flows.

$$C_B = \frac{2}{\gamma M_\infty^2} \frac{2\gamma M_\infty^2 - (\gamma - 1)}{\gamma + 1} \left(\frac{2}{\gamma + 1} \right)^{1.4} \frac{1}{M_\infty^{2.8}} - \frac{2}{\gamma M_\infty^2} \quad (1)$$

With γ the heat capacity ratio and M_∞ the free stream Mach number. When calculating C_B using the free stream conditions ($\gamma = 1.4$ and $M_\infty = 8.2$), $C_B = -0.0177$. This base pressure coefficient is then used to correct the axial force coefficient $C_{A,corrected}$ according to Eq. 2.

$$C_{A,corrected} = C_A - C_B \frac{A_{base} - A_{wet}}{A_{base}} \quad (2)$$

Where A_{base} is the larger base area of the flare, and A_{wet} is the base wetted area. Introducing the area of the sting A_{sting} , necessary to conduct the experiments:

$$A_{wet} = A_{base} - A_{sting} \quad (3)$$

$$C_{A,corrected}^{exp} = C_A - C_B \frac{A_{sting}}{A_{base}} \quad (4)$$

Regarding CFD base corrections:

$$A_{wet} = 0 \quad (5)$$

$$C_{A,corrected}^{CFD} = C_A - C_B \quad (6)$$

B. Numerical schlieren visualization

To compare the numerical flow visualization with the experimental schlieren images, the non-dimensional Eq. 7 from Wu and Martin [32], based on density gradients $\nabla\rho$, has been used.

$$NS = c_1 \cdot \exp \left(-c_2 \cdot \frac{|\nabla\rho| - |\nabla\rho_{min}|}{|\nabla\rho_{max}| - |\nabla\rho_{min}|} \right) \quad (7)$$

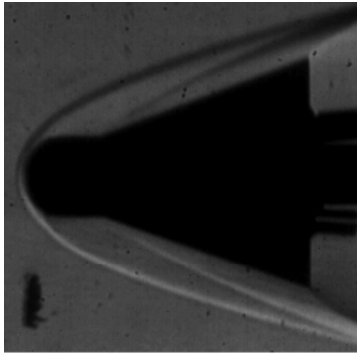
c_1 and c_2 are constants that can be tuned to visually match the experimental results. The chosen values are $c_1 = 0.8$ and $c_2 = 300$. This method enhances the small density gradients and makes them more visible. Bigger density gradients tend toward 0 (dark shades of the schlieren images), whereas low to no density gradients are represented in lighter shades. This method however does not take into account the walls of the gun tunnel.

VI. Results

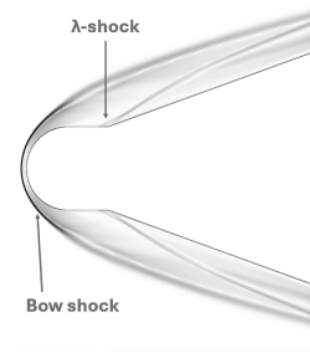
A. Flow visualisations

The experimental and numerical schlieren images are compared for both models at every angle of incidence in Figs. 7 and 8. The experimental schlieren images reveal the presence of a dark spot located beneath the model. This phenomenon can be attributed to the oxidation of the mirror and does not exert any influence on the flow dynamics. The strong bow shock ahead of the nose is accurately captured by the computations. A conical shock emanates from the cylinder-flare junction due to the compression induced by the flare. At 0° incidence, both the bow and flare-induced shocks interact downstream the flare. As the angle of attack is decreased, the windward surface's interaction line shifts in an upstream direction, while the leeward surface's interaction line moves downstream, to a point where the shocks become almost parallel on the leeward surface. These interactions are delayed with a longer cylinder. The numerical schlieren images provide clear evidence of the λ -shock shape in all cases, whereas the shock features in the experimental schlieren close to the junction are too blurred to allow any meaningful conclusions to be drawn. As the angle of attack is decreased towards -7.5° , the junction shock becomes stronger on the windward surface, shifts closer to the body, and the separated flow region predicted by CFD shrinks. In contrast, on the leeward surface, the λ -shock seems to weaken, shifts away from the body and the separated flow region expands both upstream and downstream of the junction.

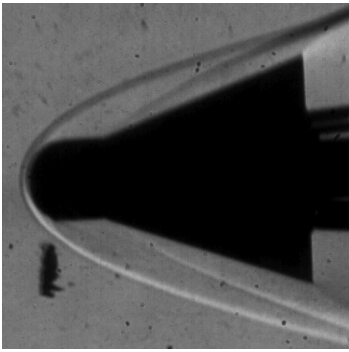
It is difficult to assess whether the separation predicted by CFD is a physical phenomenon, or if $k - \omega$ SST is overpredicting the flow separation, as it could be expected from this turbulence model [20, 21]. As mentioned previously, tripping the boundary layer during the experiments might also have strengthened it, thus conferring it more resistance against the adverse pressure gradient encountered at the cylinder-flare junction. This could explain why the flow separation is not visible on the experimental schlieren images.



(a) Experiment, 0° AoA



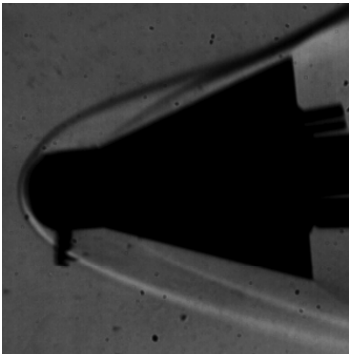
(b) RANS, 0° AoA



(c) Experiment, -2.5° AoA



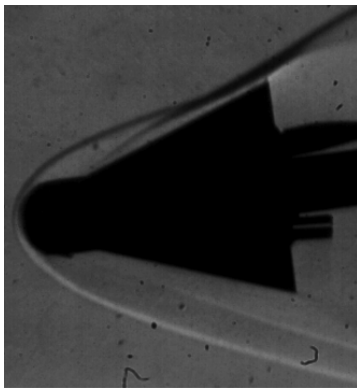
(d) RANS, -2.5° AoA



(e) Experiment, -5° AoA



(f) RANS, -5° AoA

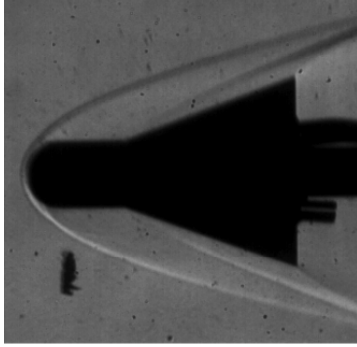


(g) Experiment, -7.5° AoA



(h) RANS, -7.5° AoA

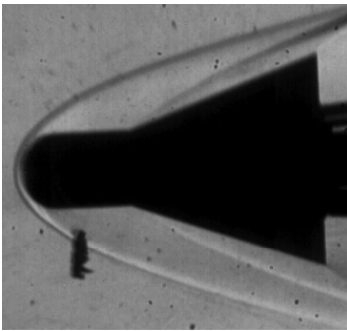
Fig. 7 Experimental and numerical schlieren images at various AoA, $L_c/D_c = 0.48$ model.



(a) Experiment, 0° AoA



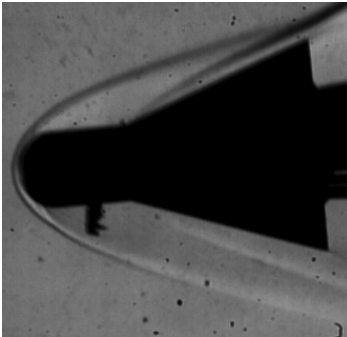
(b) RANS, 0° AoA



(c) Experiment, -2.5° AoA



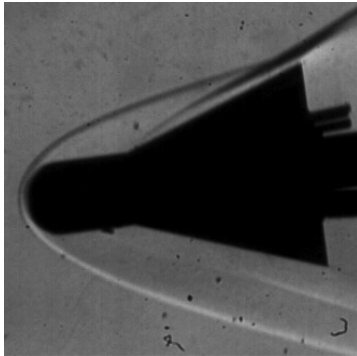
(d) RANS, -2.5° AoA



(e) Experiment, -5° AoA



(f) RANS, -5° AoA



(g) Experiment, -7.5° AoA



(h) RANS, -7.5° AoA

Fig. 8 Experimental and numerical schlieren images at various AoA, model $L_c/D_c = 0.98$.

B. Separated flow region

A numerical study of the separated flow region was carried out to assess the effect of the cylinder length at the cylinder-flare junction. The data was observed in the symmetry plane, thus creating 2 pairs of separation and reattachment points, on the leeward and windward surface respectively, presented in Fig. 10 and Table 3, which also gives the width as a fraction of L_c . The simulations reveal that the leeward flow separation expands further upstream and downstream in the case of the longer cylinder $L_c/D_c=0.98$. Fig. 10a shows that the leeward separation location is quite stable at low incidence, but significantly moves upstream as the angle of attack is further decreased. A similar behavior is observed for the leeward reattachment location in Fig. 10b. As a logical result, the leeward separation width, presented in Fig. 10c, follows the same trend as the two previous parameters: a higher increase in separation width with a decrease in angle of incidence, and a larger separation for the case $L_c/D_c=0.98$.

The windward surface however shows different behaviors for the two cases:

- $L_c/D_c=0.48$: both separation and reattachment locations are little affected at low incidence. The separation location moves downstream at higher incidence, and the reattachment location moves upstream, thus reducing the separation width.
- $L_c/D_c=0.98$: the separation location moves downstream as soon as incidence is introduced, and seem to reach an asymptote between -5° and -7.5° . Similarly for the reattachment location, which moves upstream, and for the separation width: both reach an asymptote between -5° and -7.5° . The flow reattaches further downstream than for the short cylinder, thus causing overall wider separation regions on the windward surface, except at -5° .

This larger extent of the separation for the case $L_c/D_c=0.98$ was found to have several reasons. The analysis was carried out at -7.5° but can be generalised to every angle of attack. At fixed streamwise positions along the cylinder, the boundary layer profiles, presented in Fig. 11, are more developed for the shorter model on both windward and leeward surfaces. As the flow was not slowed down as much as on the longer cylinder, the boundary layer carries more momentum and is therefore less likely to separate due to the adverse pressure gradient encountered at the cylinder-flare junction. Correlated to this first reason is the second: the flow carries less energy on the longer cylinder, as presented in Fig. 12, and is therefore more likely to separate due to an adverse pressure gradient.

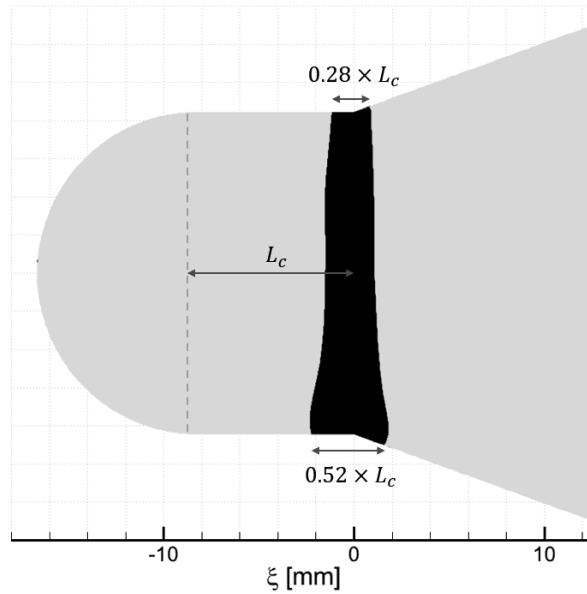
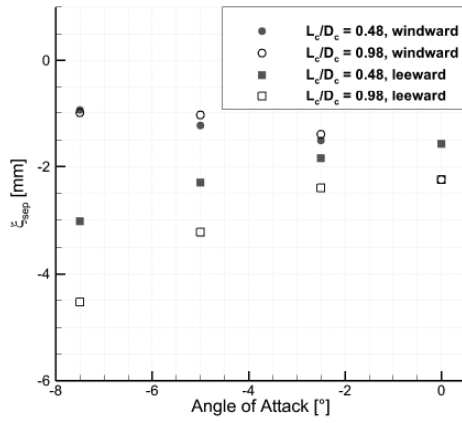


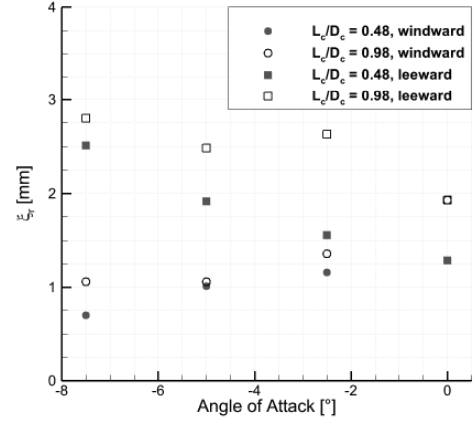
Fig. 9 Separation bubble shape for the model $L_c/D_c = 0.48$ at -5° incidence.

Table 3 Separation bubble width variation with the angle of attack.

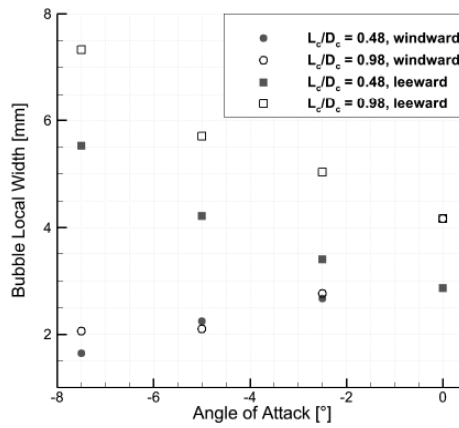
	AoA	$L_c/D_c = 0.48$			$L_c/D_c = 0.98$		
		ξ_{sep} [mm]	ξ_r [mm]	Width [mm]	ξ_{sep} [mm]	ξ_r [mm]	Width [mm]
Leeward surface	0°	-1.57	1.29	2.86 [0.35× L_c]	-2.24	1.93	4.17 [0.25× L_c]
	-2.5°	-1.84	1.56	3.40 [0.42× L_c]	-2.40	2.63	5.03 [0.30× L_c]
	-5°	-2.30	1.92	4.22 [0.52× L_c]	-3.22	2.48	5.70 [0.35× L_c]
	-7.5°	-3.02	2.51	5.53 [0.68× L_c]	-4.53	2.80	7.33 [0.44× L_c]
Windward surface	0°	-1.57	1.29	2.86 [0.35× L_c]	-2.24	1.93	4.17 [0.25× L_c]
	-2.5°	-1.51	1.16	2.67 [0.33× L_c]	-1.40	1.36	2.76 [0.17× L_c]
	-5°	-1.23	1.01	2.24 [0.28× L_c]	-1.04	1.06	2.10 [0.13× L_c]
	-7.5°	-0.95	0.7	1.65 [0.20× L_c]	-1.00	1.06	2.06 [0.12× L_c]



(a) Separation location, ξ_{sep}

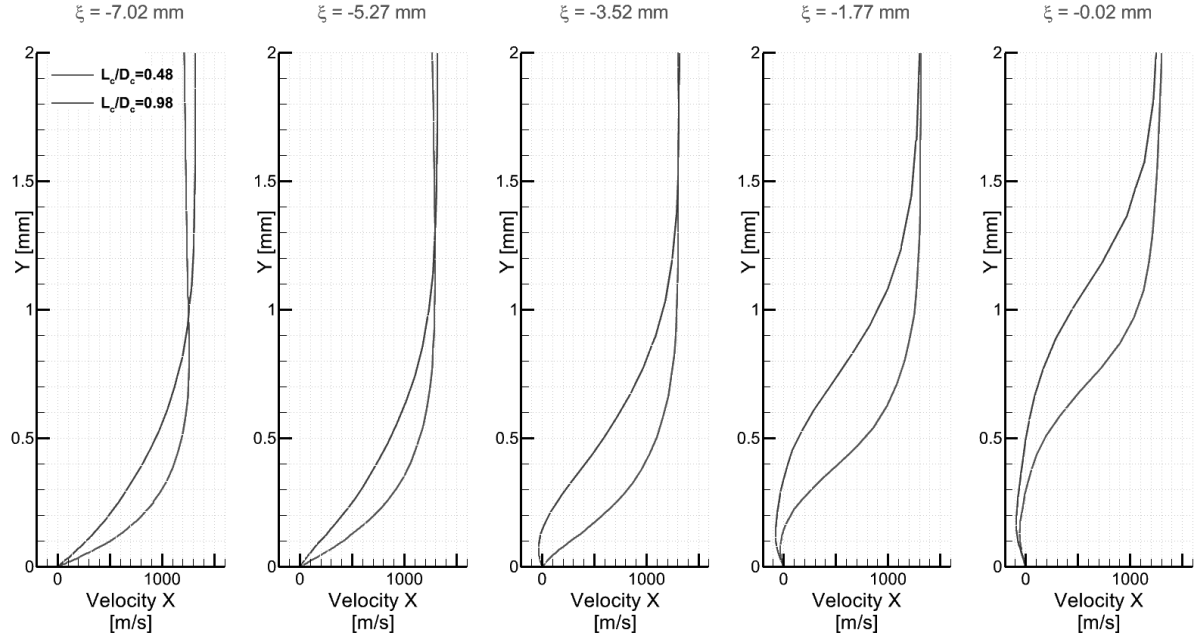


(b) Reattachment location, ξ_r

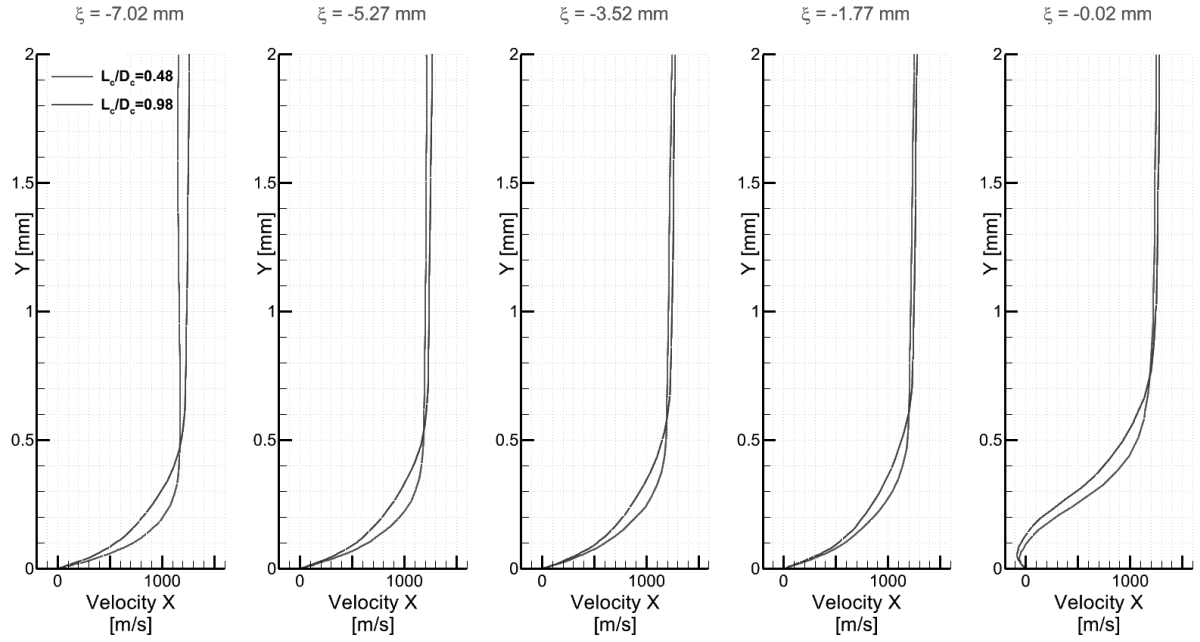


(c) Bubble width

Fig. 10 Comparison of separation location, reattachment location and bubble local width for both leeward and windward surfaces.



(a) Leeward surface



(b) Windward surface

Fig. 11 Streamwise velocity profiles at discrete locations on the cylinder of both short and long models at -7.5° .

C. Pressure distributions

The position of the separated flow region on the model greatly alters the pressure prediction close to the cylinder-flare junction. As there are only CFD data to analyse the flow separation and reattachment, it is not possible to assess whether

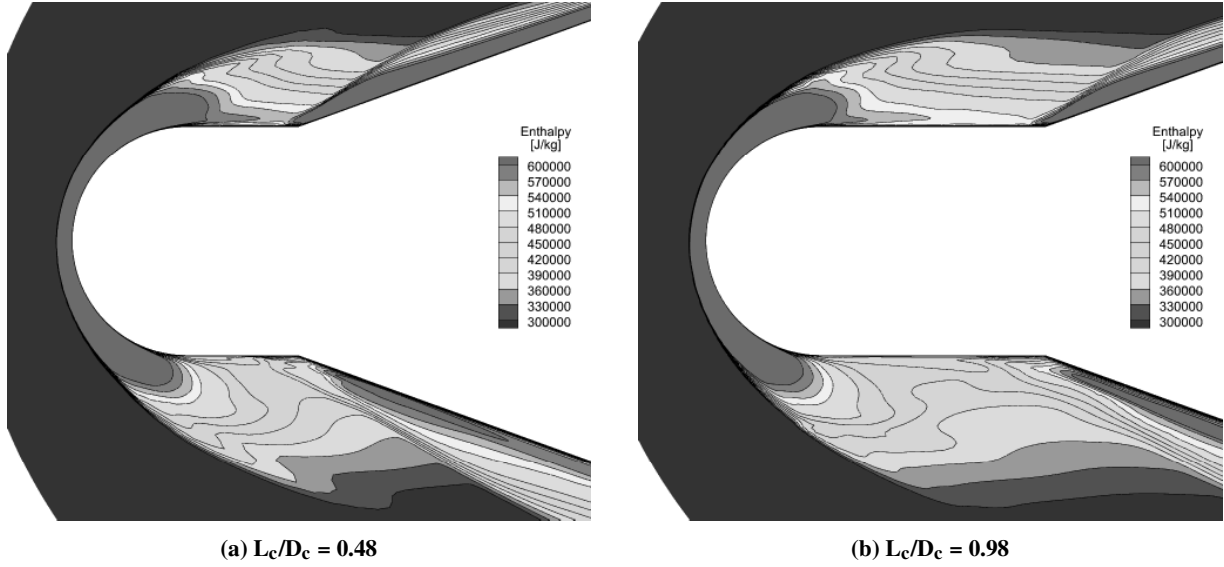


Fig. 12 Enthalpy contours at -7.5° AoA.

the separation is correctly predicted. However, the pressure contours can be used qualitatively to explain the flow features and their effects on the body. At 0° incidence, the flow structure is axisymmetrical, as can be expected. To compare the evolution of the pressure distribution with the angle of attack, 2 cases were selected for the long model and are presented in Fig. 13: -2.5° and -7.5° . As the angle of attack is decreased, the windward λ -shock becomes stronger and the interaction between the λ -shock and the strong bow shock emanating from the nose takes place closer to the body, resulting in higher pressures on the windward region of the flare. The separated flow region also shrinks, as presented in Section VI.B, thus causing the higher pressures closer to the cylinder-flare junction. The opposite happens on the windward surface: the shock moves away from the body, resulting in lower pressures and a larger separated flow region. The pressure isosurfaces are much more clustered at -7.5° , indicating more abrupt changes in pressure than at -2.5° .

The effect of the cylinder length is highlighted in Fig 14 for -7.5° . A shorter cylinder causes a stronger λ -shock at the cylinder-flare junction, resulting in a higher pressure jump at this location. Additionally, as the strong bow shock is closer to the body on the windward surface, the interaction with the λ -shock takes place slightly upstream compared to the longer cylinder. As a consequence, the higher pressure region is moved upstream as well. The pressure is also higher on the leeward surface of the shorter cylinder.

D. Aerodynamic forces comparison

As explained throughout this work, the cylinder length affects the flow at the cylinder-flare junction, altering the flow separation and therefore the pressure distribution. Consequently, the axial and normal forces are also impacted. The forces were acquired experimentally using the force balance described in section III.C. The acquisition of data is triggered by the burst of the diaphragms, with 2000 samples at 10 kHz.

Table 4 Experimental and CFD axial and normal force values, $L_c/D_c = 0.48$.

AoA [$^\circ$]	C_A^{Exp}	C_A^{CFD}	C_N^{Exp}	C_N^{CFD}
0	0.223 ± 0.002	0.283	-0.011 ± 0.011	0.000
-2.5	0.238 ± 0.008	0.287	-0.089 ± 0.050	-0.065
-5	0.262 ± 0.012	0.300	-0.164 ± 0.094	-0.138
-7.5	0.301 ± 0.022	0.322	-0.251 ± 0.152	-0.218

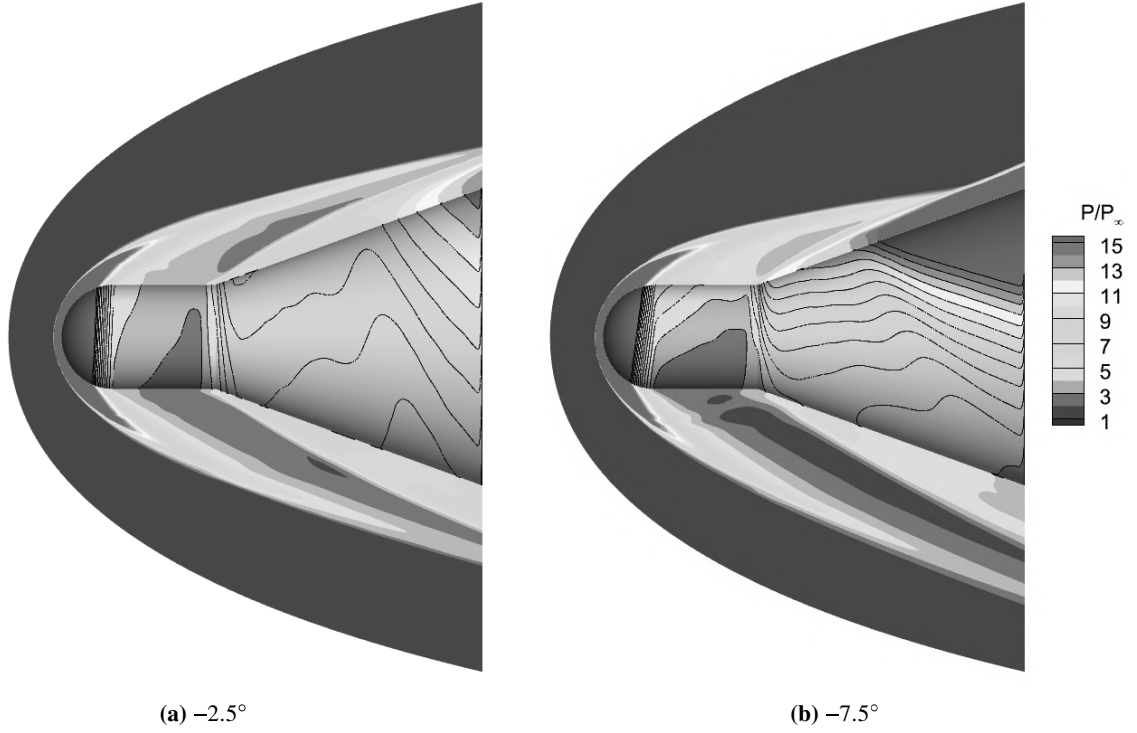


Fig. 13 Pressure contours variation with the angle of attack.

Table 5 Experimental and CFD axial and normal force values, $L_c/D_c = 0.98$.

AoA [$^\circ$]	C_A^{Exp}	C_A^{CFD}	C_N^{Exp}	C_N^{CFD}
0	0.213 ± 0.010	0.265	-0.010 ± 0.030	0.000
-2.5	0.228 ± 0.003	0.270	-0.097 ± 0.013	-0.060
-5	0.250 ± 0.018	0.285	-0.157 ± 0.123	-0.130
-7.5	N/A	0.310	N/A	-0.216

1. Axial force

As the increase in the cylinder length reduces the overall pressure on the model, the axial force is also lessened. This is illustrated in Fig. 15, which presents the results of both experiments and simulations using the $k - \omega$ SST model. The agreement between the two sets of data is satisfactory, with similar trends, although the numerical simulations tend to overpredict the axial force. The trip wire mounted on the model resulted in the boundary layer transitioning on the cylinder for the experiments. A forced turbulent boundary layer is thicker than a natural turbulent boundary layer and this phenomenon was not replicated by the $k - \omega$ SST model. The greater discrepancy between experiments and CFD at lower angles of attack, such as 0° and -2.5° , can be explained by the CFD predicting separation for these angles, whereas it was not evidenced experimentally whether separation occurred at low angles of incidence.

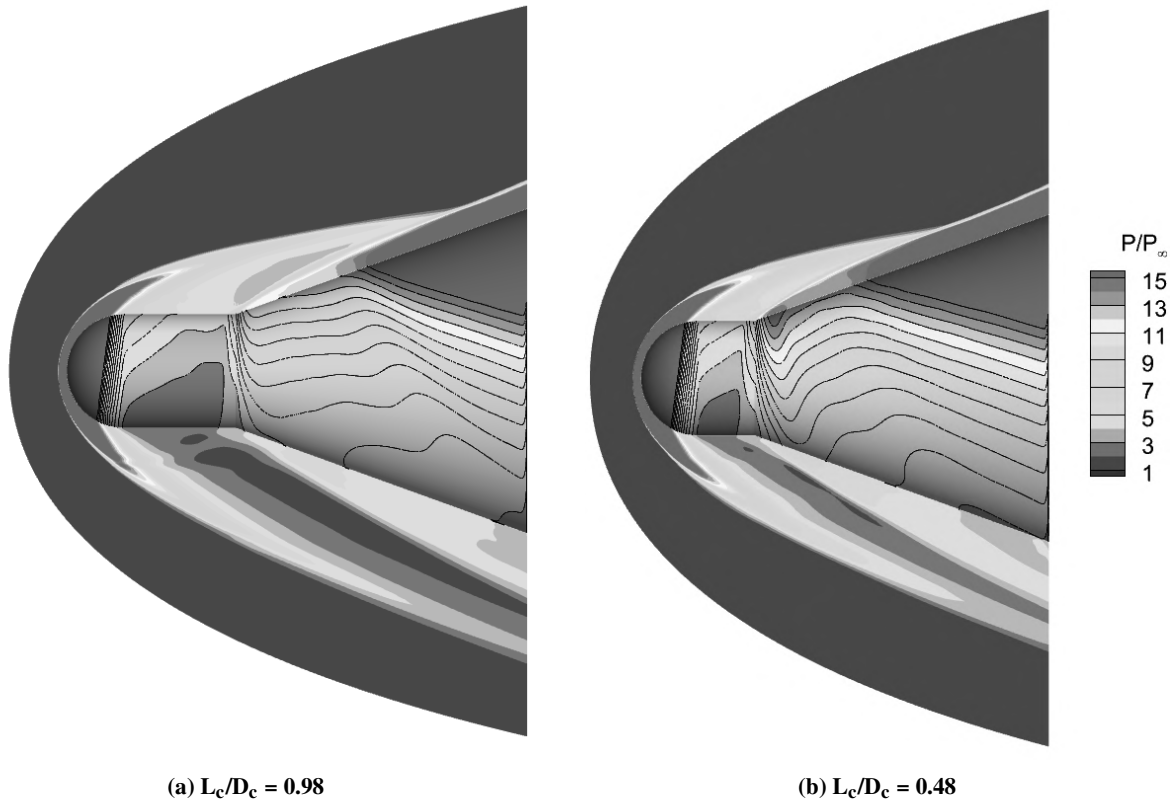


Fig. 14 Pressure contours differences between the two models at -7.5° incidence.

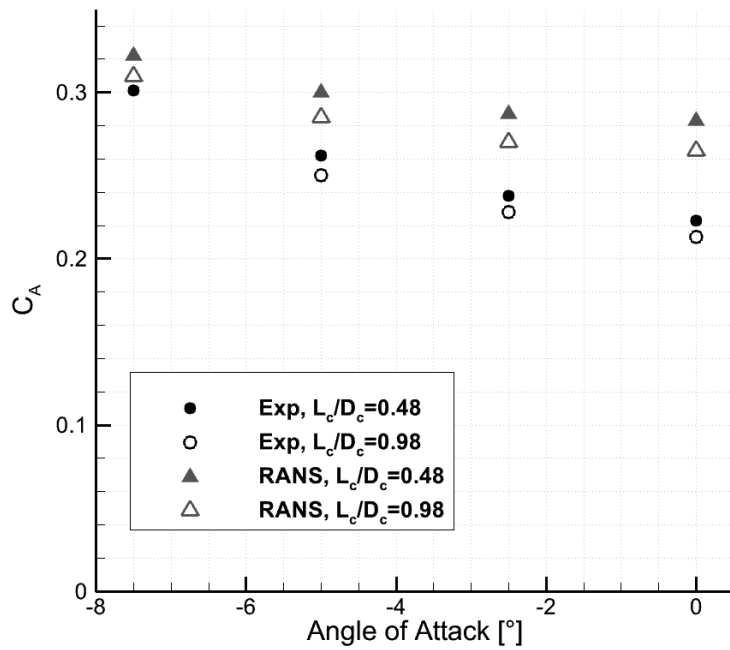


Fig. 15 Comparison of the axial force coefficient of both models.

Although it is difficult to isolate the contribution of each component to the axial force through experimental measurements, numerical simulations provide a clearer understanding of the force distribution across the model components. Figure 16 details the individual contributions of the cone, cylinder, and flare to the total axial force for the long cylinder configuration. The results indicate that the cylinder contributes negligibly to the overall axial force, meaning the short cylinder model exhibits a nearly identical distribution of forces. As the angle of incidence increases, the flare's contribution becomes more prominent, driven by the larger wetted surface area exposed to the flow. This increased exposure amplifies pressure forces acting on the flare, particularly at higher angles. In contrast, the axial force contribution from the cone remains nearly constant, dominated by the presence of the detached bow shock at the nose. This shock pattern significantly influences the axial force generated by the cone but does not vary substantially with changes in incidence angle.

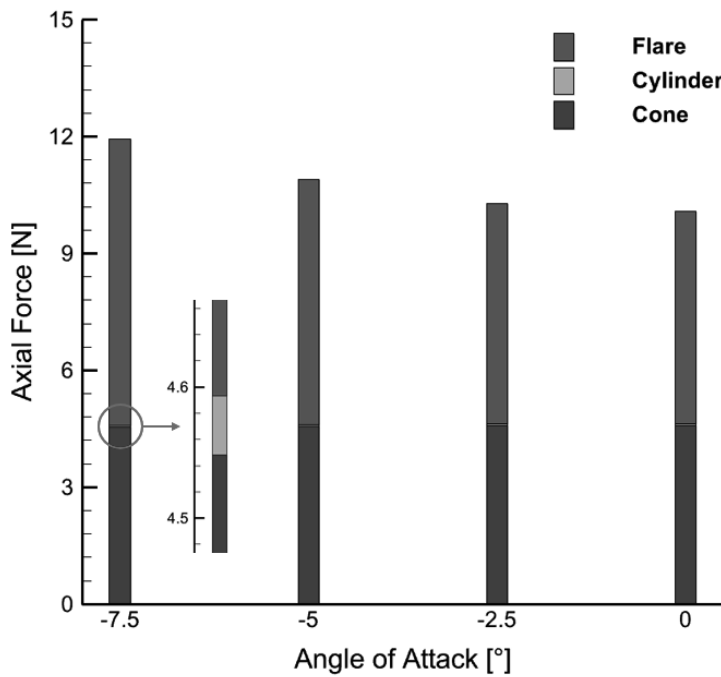


Fig. 16 Contribution from each component of the long model to the axial force at different angles of attack.

2. Normal force

The normal force coefficient (C_N) for both the short and long cylinder models exhibits a high degree of consistency across all angles of attack, as shown in Fig. 17. This indicates that cylinder length has a negligible effect on lift generation. These observations are supported by both numerical simulations and experimental tests, with a discrepancy of less than 20% between the two methods. However, the error between the experimental and computational results increases with the angle of attack, which can be attributed to the increasing complexity of the flow. At higher angles of attack, the separation bubble expands, leading to a more challenging flow structure to model accurately. In the experimental data, the normal force coefficient is found to be greater in absolute value compared to the computational results. This difference can be explained by the behaviour of the separation bubble, which contributes to lift loss. The $k - \omega$ SST turbulence model used in the simulations tends to overpredict the size of the separation bubble, leading to a slight underestimation of the normal force coefficient in comparison to the experimental data.

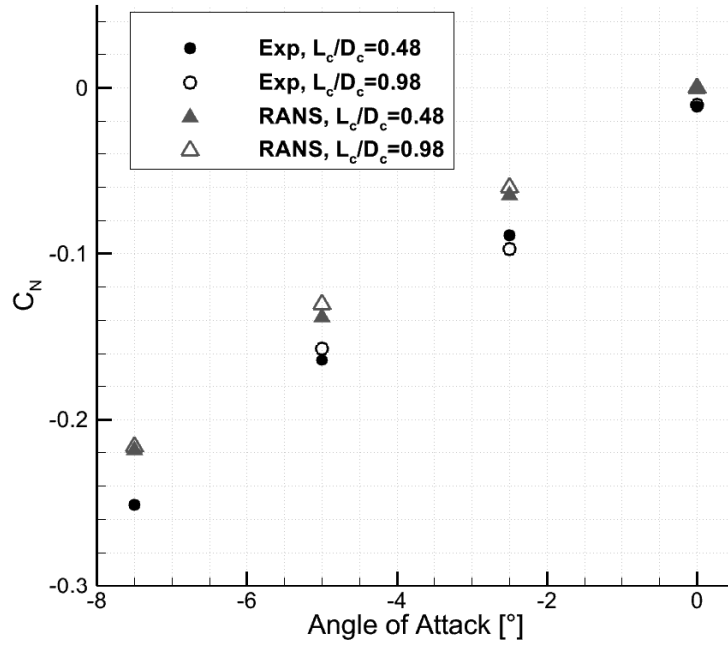


Fig. 17 Comparison of the normal force coefficient of both models.

The contribution from each part of the model is found in Fig. 18. The dominant source of normal force is the flare, whose contribution increases with the angle of incidence due to the larger exposed wetted area. This increase in wetted area effectively enhances the pressure differential across the surface, leading to higher normal force generation. In contrast, the cylinder and nose contribute significantly less to the overall normal force. The cylindrical section, in particular, generates minimal force due to its streamlined geometry and low incidence to the flow direction. The nose, while contributing some lift through its interaction with the incoming flow, produces substantially lower normal force compared to the flare. The increase in normal force with higher angles of attack can be primarily attributed to the geometry of the flare and its interaction with the expanding separation bubble at larger incidence angles, which enhances the pressure recovery downstream. The results align well with the trends observed in both numerical simulations and experimental data, providing a clear understanding of the aerodynamic load distribution across the model.

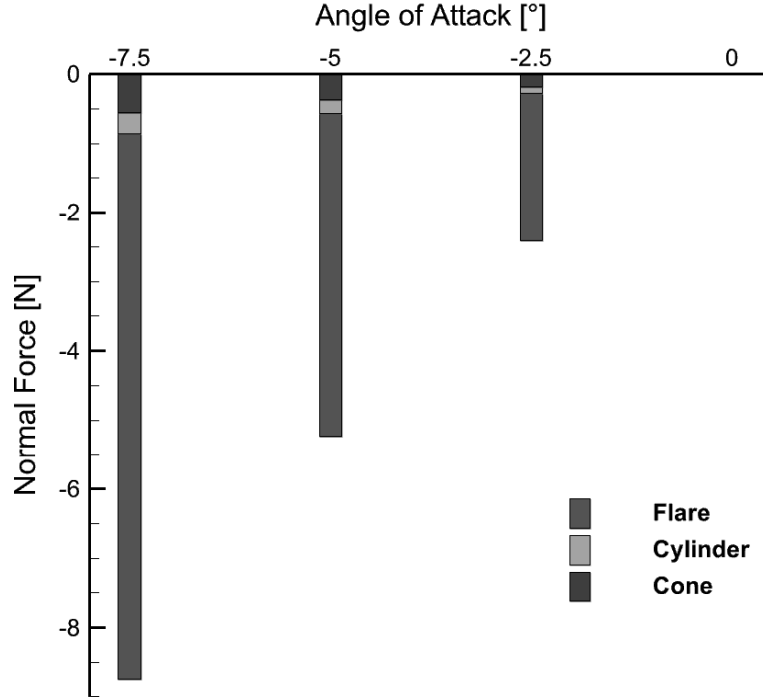


Fig. 18 Contribution from each component of the long model to the normal force at different angles of attack.

VII. Conclusion

Experimental and numerical work were conducted to investigate the effects of turbulent shock-wave boundary-layer interactions on the aerodynamic characteristics of a re-entry vehicle model, specifically focusing on a blunted nose cone-cylinder-flare configuration in Mach 8.2 flow. Key parameters examined included varying angles of incidence (-7.5° , -5° , -2.5° , and 0°) and the impact of different cylinder lengths on flow behavior. The study compared results from experiments and RANS CFD using the $k - \omega$ SST turbulence model. CFD was also used to analyse the separation bubble dynamics.

Introducing an angle of incidence increased both the axial and normal forces acting on the model, primarily due to intensified pressures on the windward side of the flare. This pressure rise stemmed from complex shock interactions between a strong bow shock originating from the cone and a λ -shock formed at the cylinder-flare junction. The normal force increased linearly with the angle of attack, while the axial force seemed to follow a 2nd order polynomial evolution. Data points at higher angles of incidence would be needed to draw any further conclusion.

The comparison between two cylinder lengths showed that the longer cylinder configuration resulted in a reduction in axial force with negligible change in normal force. The extended cylinder led to a weakened boundary layer upstream of the cylinder-flare junction, producing a larger separated flow region and a less intense λ -shock. Consequently, this allowed the shock-shock interaction to shift downstream, significantly altering the pressure distribution along the flare and reducing the axial force by 4.5%.

Results presented in this paper demonstrate that a cylinder length-to-diameter ratio of 0.98 is aerodynamically more efficient than its counterpart with a ratio of 0.48. However, at high angles of attack, the width of the separation bubble is increased and cause a loss of effectiveness of the control surfaces, especially above -5° .

Further studies include the spectral analysis of the SBLI to determine the frequencies associated with the unsteady nature of the interactions at play. Proper Orthogonal Decomposition can be employed to identify the most significant modes in the flow, while the Dynamic Mode Decomposition will allow to determine the associated frequency and determine if there is a phase difference between the oscillations of the bow shock and the separation shock.

Acknowledgments

The authors would like to express their gratitude to Mr Lynton Banks-Davies, Cranfield University's Senior Technical Officer, for operating the Cranfield Hypersonic Gun Tunnel and providing invaluable assistance during the experiments. The simulations were conducted utilising the Cranfield University HPC facility (Crescent), and the authors acknowledge the technical support provided by the Cranfield HPC staff.

References

- [1] Needham, D. A., and Stollery, J., "Boundary layer separation in hypersonic flow," *3rd and 4th Aerospace Sciences Meeting*, Aerospace Sciences Meetings, American Institute of Aeronautics and Astronautics, 1966. <https://doi.org/10.2514/6.1966-455>, URL <https://doi.org/10.2514/6.1966-455>.
- [2] Holden, M. S., "A study of flow separation in regions of shock wave-boundary layer interaction in hypersonic flow," *11th Fluid and Plasma Dynamics Conference*, Fluid Dynamics and Co-located Conferences, American Institute of Aeronautics and Astronautics, 1978. <https://doi.org/10.2514/6.1978-1169>, URL <https://doi.org/10.2514/6.1978-1169>.
- [3] Hankey, W., and Holden, M. S., *Two-dimensional Shock Wave-boundary Layer Interactions in High Speed Flows*, AGARDograph, Tech. Editing and Reprod. Limited, 1975. URL <https://books.google.co.uk/books?id=-TfGzweEACAAJ>.
- [4] Holden, M. S., and Wadhams, T. P., "Code Validation Study of Laminar Shock/Boundary Layer and Shock/Shock Interactions in Hypersonic Flow Part A: Experimental Measurements," *39th Aerospace Sciences Meeting and Exhibit*, American Institute of Aeronautics and Astronautics, 2001. <https://doi.org/10.2514/6.2001-1031>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2001-1031>, _eprint: <https://arc.aiaa.org/doi/pdf/10.2514/6.2001-1031>.
- [5] Harvey, J. K., Holden, M. S., and Wadhams, T. P., "Code Validation Study of Laminar Shock/Boundary Layer and Shock/Shock Interactions in Hypersonic Flow Part B: Comparison with Navier-Stokes and DSMC Solutions," *39th Aerospace Sciences Meeting and Exhibit*, American Institute of Aeronautics and Astronautics, 2001. URL <https://arc.aiaa.org/doi/abs/10.2514/6.2001-1031>.
- [6] Knight, D., and Zheltovodov, A., "Ideal-Gas Shock Wave-Turbulent Boundary-Layer Interactions (STBLIs) in Supersonic Flows and Their Modeling: Two-Dimensional Interactions," *Shock Wave-Boundary-Layer Interactions*, Cambridge Aerospace Series, Cambridge University Press, Cambridge, 2011, pp. 137–201. <https://doi.org/10.1017/CBO9780511842757>.
- [7] Holden, M., "Studies of the mean and unsteady structure of turbulent boundary layer separation in hypersonic flow," *22nd Fluid Dynamics, Plasma Dynamics and Lasers Conference*, American Institute of Aeronautics and Astronautics, Honolulu, HI, U.S.A., 1991. <https://doi.org/10.2514/6.1991-1778>, URL <https://arc.aiaa.org/doi/10.2514/6.1991-1778>.
- [8] Holden, M. S., "Shock wave-turbulent boundary layer interaction in hypersonic flow," *10th Aerospace Sciences Meeting*, Aerospace Sciences Meetings, American Institute of Aeronautics and Astronautics, 1972. <https://doi.org/10.2514/6.1972-74>, URL <https://doi.org/10.2514/6.1972-74>.
- [9] Holden, M. S., "Experimental Studies of Shock Wave–Boundary-Layers Interactions," *Shock Wave-Boundary-Layer Interactions*, edited by H. Babinsky and J. K. Harvey, Cambridge Aerospace Series, Cambridge University Press, Cambridge, 2011, pp. 259–313. <https://doi.org/10.1017/CBO9780511842757>, URL <https://www.cambridge.org/core/books/shock-waveboundarylayer-interactions/1EC84B4EBF65B1B20C0CA60806B7DFF5>.
- [10] Babinsky, H., "A Study of Roughness in Turbulent Hypersonic Boundary-Layers," PhD Thesis, Cranfield University College of Aeronautics, 1993.
- [11] Coleman, G. T., "Hypersonic Boundary Layer Studies," PhD Thesis, University of London, 1973.
- [12] Hoffman, E. N., Kendhammer, D. M., LaLonde, E. J., Andrade, A., and Combs, C. S., "Effects of distributed roughness on shock-wave/boundary-layer interactions at Mach 7.2," *AIAA SCITECH 2023 Forum*, American Institute of Aeronautics and Astronautics, National Harbor, MD & Online, 2023. <https://doi.org/10.2514/6.2023-0268>, URL <https://arc.aiaa.org/doi/10.2514/6.2023-0268>.
- [13] Murray, N., Hillier, R., and Williams, S., "Experimental investigation of axisymmetric hypersonic shock-wave/turbulent-boundary-layer interactions," *Journal of Fluid Mechanics*, Vol. 714, 2013, pp. 152–189. <https://doi.org/10.1017/jfm.2012.464>, URL <https://www.cambridge.org/core/product/A21941BCDD16F63479A1F83316D6CE7C>, edition: 2013/01/02 Publisher: Cambridge University Press.
- [14] Cao, S., Klioutchnikov, I., and Olivier, H., "Görtler Vortices in Hypersonic Flow on Compression Ramps," *AIAA Journal*, Vol. Vol. 57, No. No. 9, 2019, pp. 3874–3884. <https://doi.org/10.2514/1.J057975>.

- [15] Vanstone, L., and Clemens, N. T., "Proper Orthogonal Decomposition Analysis of Swept-Ramp Shock-Wave/Boundary-Layer Unsteadiness at Mach 2," *AIAA Journal*, Vol. 57, No. 8, 2019, pp. 3395–3409. <https://doi.org/10.2514/1.J057874>, URL <https://doi.org/10.2514/1.J057874>, publisher: American Institute of Aeronautics and Astronautics.
- [16] Ashby, G. C., Jr., and Cary, A. M., Jr., "A Parametric Study of the Aerodynamic Characteristics of Nose-Cylinder-Flare Bodies at a Mach Number of 6.0," NASA Technical Note NASA-TN-D-2854 / L-4160, NASA, Langley Research Center, Langley Station, Hampton, Va., Jun. 1965.
- [17] Zheltovodov, A., "Shock waves/turbulent boundary-layer interactions - Fundamental studies and applications," American Institute of Aeronautics and Astronautics, New Orleans, 1996. <https://doi.org/10.2514/6.1996-1977>, _eprint: <https://arc.aiaa.org/doi/pdf/10.2514/6.1996-1977>.
- [18] Knight, D., Yan, H., Panaras, A., and Zheltovodov, A., "RTO WG 10 - CFD validation for shock wave turbulent boundary layer interactions," American Institute of Aeronautics and Astronautics, Reno,NV,U.S.A., 2002. <https://doi.org/10.2514/6.2002-437>, _eprint: <https://arc.aiaa.org/doi/pdf/10.2514/6.2002-437>.
- [19] Thivet, F., "Lessons learned from RANS simulations of shock-wave/boundary-layer interactions," American Institute of Aeronautics and Astronautics, Reno,NV,U.S.A., 2002, p. 11. <https://doi.org/10.2514/6.2002-583>.
- [20] Holden, M., Wadhams, T., MacLean, M., and Mundy, E., "Experimental studies of shock wave/turbulent boundary layer interaction in high reynolds number supersonic and hypersonic flows to evaluate the performance of cfd codes," *40th Fluid Dynamics Conference and Exhibit*, Fluid Dynamics and Co-located Conferences, American Institute of Aeronautics and Astronautics, 2010. <https://doi.org/10.2514/6.2010-4468>, URL <https://arc.aiaa.org/doi/10.2514/6.2010-4468>.
- [21] Cross, P. G., and West, M. R., "Simulation of Hypersonic Flowfields Using STAR-CCM+," Tech. rep., NAWCWD TM 8824, Jan. 2019.
- [22] Needham, D. A., "Progress report on the Imperial College hypersonic gun tunnel," 1963.
- [23] Hillier, R., "Gun Tunnel Studies of Shock-Wave/Boundary-Layer Interactions; Laminar, Transitional and Turbulent," *20th AIAA International Space Planes and Hypersonic Systems and Technologies Conference*, International Space Planes and Hypersonic Systems and Technologies Conferences, American Institute of Aeronautics and Astronautics, 2015. <https://doi.org/10.2514/6.2015-3512>, URL <https://doi.org/10.2514/6.2015-3512>.
- [24] Opatowski, T., "A Three Component Gun Tunnel Balance Designed for Testing Thin Delta Wings," Tech. rep., 1971.
- [25] Georgiadis, N., and Yoder, D., "Recalibration of the Shear Stress Transport Model to Improve Calculation of Shock Separated Flows," *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, Aerospace Sciences Meetings, American Institute of Aeronautics and Astronautics, 2013. <https://doi.org/10.2514/6.2013-685>, URL <https://arc.aiaa.org/doi/10.2514/6.2013-685>.
- [26] Liou, M.-S., "A Sequel to AUSM: AUSM+," *Journal of Computational Physics*, Vol. 129, No. 2, 1996, pp. 364–382. <https://doi.org/10.1006/jcph.1996.0256>, URL <https://www.sciencedirect.com/science/article/pii/S0021999196902569>, publisher: Academic Press.
- [27] Kitamura, K., Shima, E., Nakamura, Y., and Roe, P. L., "Evaluation of Euler Fluxes for Hypersonic Heating Computations," *AIAA Journal*, Vol. 48, No. 4, 2010, pp. 763–776. <https://doi.org/10.2514/1.41605>, URL <https://doi.org/10.2514/1.41605>, publisher: American Institute of Aeronautics and Astronautics _eprint: <https://doi.org/10.2514/1.41605>.
- [28] Gupta, R. N., Lee, K.-P., Thompson, R. A., and Yos, J. M., "Calculations and curve fits of thermodynamic and transport properties for equilibrium air to 30000 K," , Oct. 1991. NTRS Author Affiliations: Vigyan Research Associates, Inc., Hampton, NASA Langley Research Center, Textron Defense Systems NTRS Report/Patent Number: NAS 1.61:1260 NTRS Document ID: 19920002067 NTRS Research Center: Legacy CDMS (CDMS).
- [29] Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., and Raad, P. E., "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications," *Journal of Fluids Engineering*, Vol. 130, No. 078001, 2008. <https://doi.org/10.1115/1.2960953>, URL <https://doi.org/10.1115/1.2960953>.
- [30] Gabeaud, A., "Base Pressures at Supersonic Velocities," *Journal of the Aeronautical Sciences (Reader's Forum)*, Vol. 17, No. 8, 1950, pp. 525–526.
- [31] Love, E. S., "The Base Pressure at Supersonic Speeds on Two-Dimensional Airfoils and Bodies of Revolution (With and Without Fins) Having Turbulent Boundary Layers," Tech. Rep. NACA-RM-L53C02, Apr. 1953. NTRS Author Affiliations: National Advisory Committee for Aeronautics. Langley Aeronautical Lab. NTRS Document ID: 20030064135 NTRS Research Center: Langley Research Center (LaRC).

- [32] Wu, M., and Martin, M. P., “Direct Numerical Simulation of Supersonic Turbulent Boundary Layer over a Compression Ramp,” *AIAA Journal*, Vol. 45, No. 4, 2007, pp. 879–889. <https://doi.org/10.2514/1.27021>, URL <https://doi.org/10.2514/1.27021>, publisher: American Institute of Aeronautics and Astronautics _eprint: <https://doi.org/10.2514/1.27021>.



DSPACE

<https://dspace.org/>

Experimental and numerical studies of shock wave -- boundary-layer interactions over a cone-cylinder-flare in hypersonic flows

Arfi, Benjamin W.; Corcoral, Erwan A.; Prince, Simon A.; Rana, Zeeshan A.; Tsentis, Spyros

2025-01-06

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2025-1333>

Arfi BW, Corcoral EA, Prince SA, et al., (2025) Experimental and numerical studies of shock wave -- boundary-layer interactions over a cone-cylinder-flare in hypersonic flows. In: AIAA SCITECH 2025 Forum, 6-10 Jan 2025, Orlando FL, USA. Paper number AIAA 2025-1333