17th Australasian Fluid Mechanics Conference
Auckland, New Zealand
5-9 December 2010

Hypersonic Drop-Tests of a Sphere: CFD Comparison to Experimental Results

R. Choudhury[1], G. Doig[2], A. Neely[1] and T. Barber[2]

[1]School of Engineering and IT
University of New South Wales at the Australian Defence Force Academy, ACT 2600, Australia

[2]School of Mechanical and Manufacturing
University of New South Wales, NSW 2052, Australia

Abstract

Drop tests were performed in the T-ADFA Shock Tunnel in Canberra to determine the aerodynamic coefficients of a 37.5mm diameter polystyrene sphere experiencing short-duration (approx. 1 millisecond) Mach 10 flow. The aerodynamic forces acting on the sphere were determined from a trajectory analysis based on high-speed schlieren video footage. In order to gain a clearer understanding of the flow-field generated around the models during the tests, transient, moving-mesh computational fluid dynamics (CFD) has been used to perform simulations which approximate the conditions experienced in the shock tunnel as closely as possible. For the assumption of steady free-stream conditions the simulations predicted a sphere displacement of 13mm during the nominal 1ms of test time. The numerical displacement predictions were in good agreement with those obtained from the experiment.

Introduction

It is difficult to obtain quantitative data from drop-test shock tunnel experiments due to the short duration of useful test times, CFD simulations can be used to gain additional insight into the flow physics and in this process also useful in validating the applicability of the commercial code to external hypersonic flow. Time dependent solutions can be obtained from CFD simulations enabling better visualization of the unsteady behaviour of the flow and also to analyse the trajectory of the test geometry during the free flying phase of the experiment. CFD allows the easy variation of free-stream conditions enabling parametric studies of a problem, and thus having both programmes complementing each other provides the researcher with the maximum possible insight into the flow.

Experimental techniques used for short duration hypersonic flows such as the direct acceleration measurement technique [4] where the models are weakly restrained with low stiffness support such as thin wires, so that the effect of the restorative force can be neglected, show oscillations caused by the mechanical vibration due to the insufficient rigidity of the test model. Although a signal recovery process can be applied to this technique to remove the oscillations from the acquired force response data [11]. The drop-test technique, where the model is in "free flight" during the test time removes the need for the complex force measurements and the signal analysis. It is potentially a simple and cost effective method. The aerodynamic data is gathered from automated image processing algorithms which track the in-plane motion of the model from the captured image sequences. These algorithms can also be used to extract the angle of attack of slender models [9].

The numerical analysis of the shock tunnel drop tests was performed using the commercial solver ANSYS CFX 12.1 which is a fully implicit, finite volume, Reynolds-Averaged Navier-Stokes CFD code. It was used to conduct the moving mesh simulations and to analyse the trajectory of a polystyrene sphere during the free-flight experiment. Numerical time-dependent solutions involving dynamic meshes have been used to study store-separation at transonic speeds [7]. Validation work has been performed with the commercial code for hypersonic flow conditions [5].

Experimental Setup

To demonstrate the practicality of the free flight technique in the T-ADFA shock tunnel, the simplest test case was selected as the model; a commercially sourced, expanded polystyrene foam sphere of 37.5 mm in diameter. The model was "flown" in the nominal test flow condition as described in Table 1.

Free-stream conditions		**Uncertainties (±%)**
u_∞ (m/s)	2588	1.0
M_∞	10.30	1.8
ρ_∞ (g/m^3)	6.0	10.9
P_∞ (Pa)	291	14.7
T_∞ (K)	168	5.3
H_0 (MJ/kg)	3.94	1.8
Re_∞ (1/m)	1.17×10^6	8.8

Table 1: T-ADFA shock tunnel free-stream conditions

A magnetic solenoid suspension system was used to release the model just before the arrival of the core test flow. The extracted trajectory data was converted from position-time data to velocity-time data and then to acceleration-time data. The average value of acceleration was calculated to determine the drag force and the value of the free-stream dynamic pressure was used to calculate the drag coefficient [9].

The T-ADFA shock tunnel which is short duration impulse facility has a test time of approximately 1 millisecond. The free-piston shock tunnel is capable of producing high levels of nozzle reservoir pressure and enthalpy. The Mach 10 hypervelocity nozzle used for the shock tunnel experiments is a 12.7 mm diameter inlet, 305 mm diameter exit, 7.5^{o} half-angle conical nozzle. For more detail on the T-ADFA shock tunnel, the reader is directed to existing literature [8].

Numerical model and setup

A quarter-model was used in this case to save computational time and resources. A full 3d simulation was not necessary since the simulation was modelled as a one degree of freedom problem; movement is only in the axial direction. This assumption was based on the fact that during the experiments the sphere was observed to fall a distance of 1.04 mm after the arrival of the flow, which is only 4% of the observed axial displacement and thus can be ignored in this simulation [9]. A fully-structured hexahedral mesh was generated in ANSYS ICEM, Figure 2.

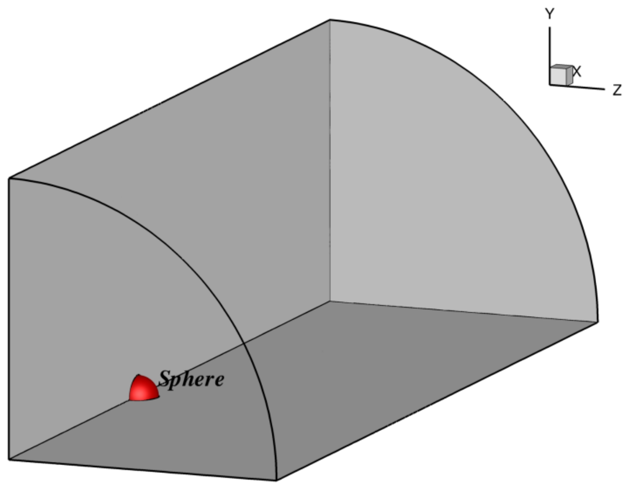

Figure 1: Computational domain used for the 3d transient simulation

The inlet of the domain was defined using a specific normal speed and pressure. The value of the free-stream velocity and pressure outlined in Table 1 was used as the inlet condition for the simulations.

An appropriate turbulence intensity percentage and a length scale were chosen for the simulation. The turbulent intensity has an influence on the shock location as well as resolution [6]. A turbulent intensity of 0.1% was specified, which allowed for an enhanced representation of the shock in the flow-field. The outlet was defined using a static pressure with a value equivalent to the free-stream pressure during the shock tunnel experiment (Table 1). The rest of the domain was defined as zero-shear walls other than the sphere where the "no-slip" boundary condition was employed

The initial grid used had a total of approximately 6×10^5 elements. Additionally simulations were conducted with grid elements of 1.3 million and 1.9 million for a mesh sensitivity study. The simulations with higher number of grid elements over-predicted the displacement of the sphere after a time of around 0.6ms subsequent to the initial shock. A possible explanation could be that the fine meshes tend to degrade faster than coarser meshes due to the increased displacement of the sphere and thus over-predict the displacement values.

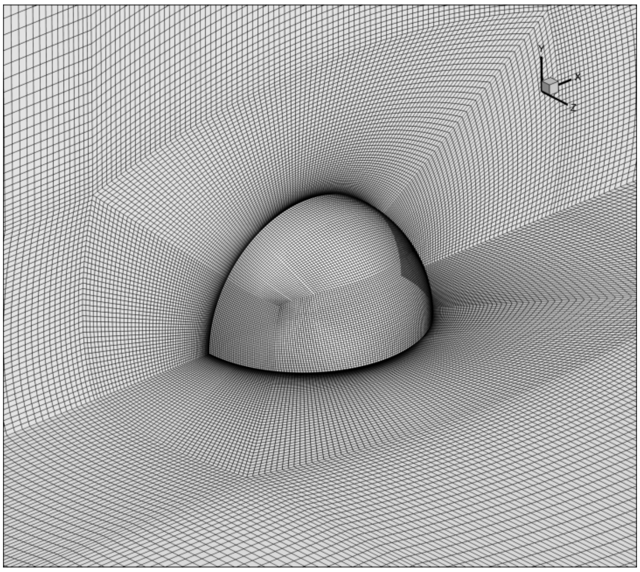
Figure 2: Mesh around the sphere for 3d simulation

The test gas, air was modelled as an ideal gas and all simulations were run as time-dependent following an initial steady-state case to test the mesh and discretisation schemes. Mesh deformation was enabled to incorporate the sphere movement. The mesh movement is achieved by specifying the motion of points on particular mesh regions i.e. the boundary of the domain with the help of the CFX Expression Language (CEL). CEL expressions are mathematical, logical and operational functions. These expressions were used to define the motion of the sphere according to 1-DOF physics. This enabled the calculation of displacement by the solver every consecutive time step according to the aerodynamic forces experienced by the sphere.

A displacement diffusion equation was solved to determine the mesh displacements throughout the volume of the mesh. The mesh stiffness was increased in the region around the sphere, which is a useful approach to avoid mesh folding as it is a region of large deformation. An upwind finite-volume scheme [3] was used for the spatial discretisation and a second order backward Euler method was employed for the transient scheme.

The turbulence scheme was set to first-order accuracy for a stable solution. A time step of 2×10^{-7} seconds was used for the current simulation based on the smallest cell size of the grid and calculated for a Courant number of less than one which was set as a stability criterion for the numerical solution [1]. A two-equation $k - \omega\, SST$ model was used for the turbulence modelling. The choice of the turbulence model was based on results obtained from steady-state simulation of the sphere in hypersonic flow with the shock tunnel free-stream conditions outlined in Table 1 used as the operating conditions. The $k - \omega\, SST$ turbulence model has previously been shown to yield good results for hypersonic flow [10].

Results

<u>Steady-State Results</u>

A steady-state analysis was conducted to validate the numerical approach before running the transient simulations. Drag coefficient obtained from both the experiment and the simulation was examined for comparisons. Figure 3 shows the Mach number contour for the sphere calculated by the steady-state analysis.

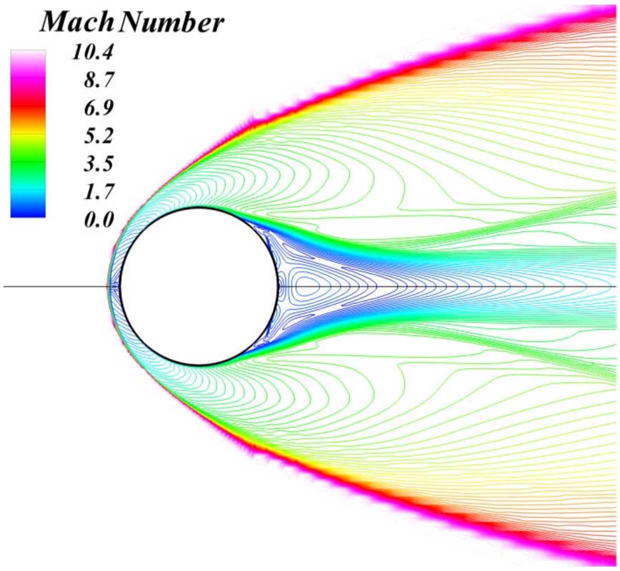

Figure 3: Mach number plot, steady-state

The value of the coefficient of drag obtained from the steady-state simulation was 0.88. The drag coefficient deduced from the free flight experiments in the T-ADFA shock tunnel gave a value of 0.95. The difference in the drag coefficient can attributed to the acknowledged uncertainty resulting from image processing of the low resolution experimental video, which was used to analyse the sphere displacement and thereby derive the aerodynamic coefficients. The numerical simulation did not take into account the high temperature gas effects, which may be another source for the difference in the drag coefficient values although the free-stream values presented in Table 1 correspond to a low enthalpy condition.

Transient & Moving Mesh Results

The displacement history of the sphere obtained from the numerical analysis was compared to that of the experiments. Figure 4 shows the mesh deformation after a time of 1.0 millisecond has elapsed.

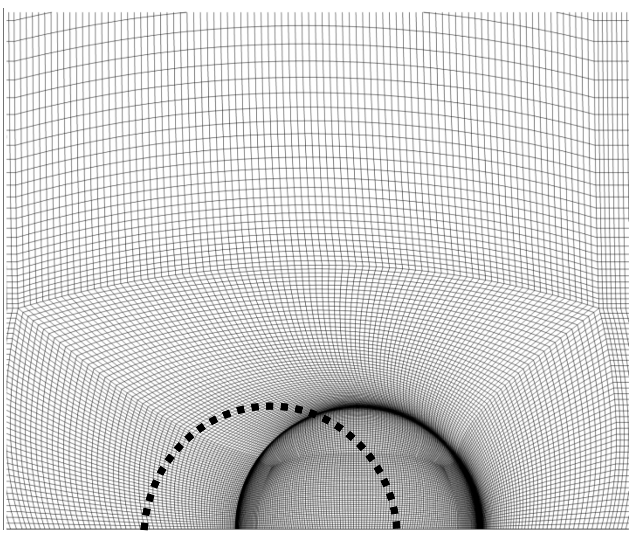

Figure 4: Mesh around the sphere after displacement, t=1.0ms, approximate position at t=0 shown as dotted curve

Figure 5 and Figure 6 show numerical schlieren images obtained from the analysis at a time of 0.6ms and 1.0ms respectively. The reference line drawn shows the displacement of the sphere at those specified times. A difference in the structure of the wake of the body is noticeable when the schlieren images for these specific simulation times are compared.

Figure 6 represents the steady-state flow structure. The bow shock is fully formed along with the recompression shock. The recompression shock is visible in Figure 6, which shows that the flow in the wake of the body is fully developed after 1.0ms.

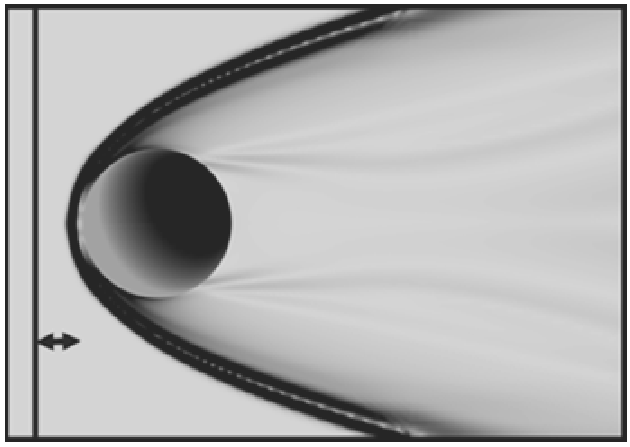

Figure 5: Numerical schlieren of the sphere at t=0.6ms, a reference line is drawn to indicate the displacement

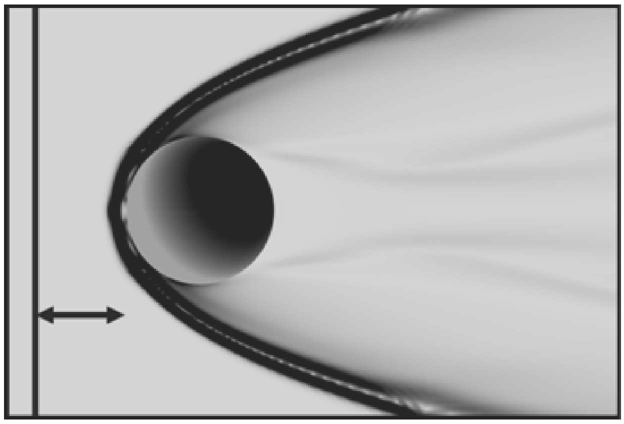

Figure 6: Numerical schlieren of the sphere at t=1.0ms, distance from the reference line showing its displacement.

Figure 7 shows the bow shock and the wake profile of the sphere at approximately 0.27ms after the initial shock. It shows the initial period of the flow being established over the sphere and shows the formation of the bow shock. It suggests that the flow in the wake of the body is initially highly transient and the bow shock is slowly approaching its steady-state shape.

The displacement history obtained from the CFD analysis is compared to the experimental data in Figure 8. There is an excellent agreement between the numerical results with the experimental values as the CFD predicted displacement graph is well within the scatter of the experimental uncertainty. A displacement of 13mm is observed for the sphere after 1 millisecond of test flow.

It should also be noted that the inlet was defined as a constant pressure inlet during the CFD simulations whereas during the experiments there is a continuous decrease in the nozzle reservoir pressure during the test time for the condition employed in the shock tunnel. These factors may lead to over-prediction of the displacement by the constant pressure inlet CFD simulations.

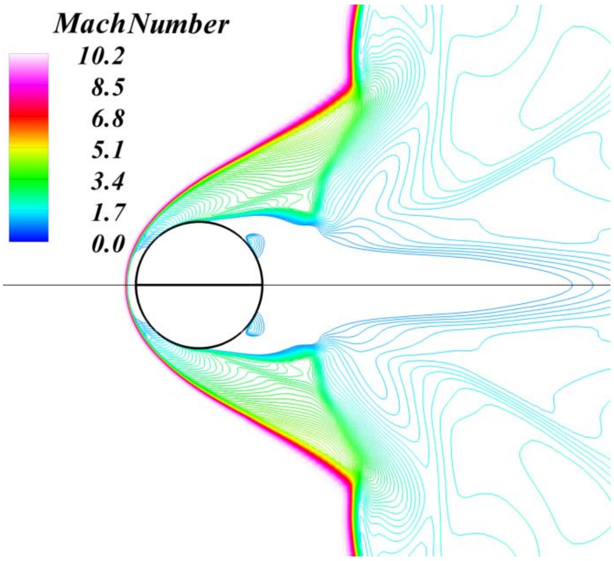

Figure 7: Mach number contour plot at t = 0.27 ms

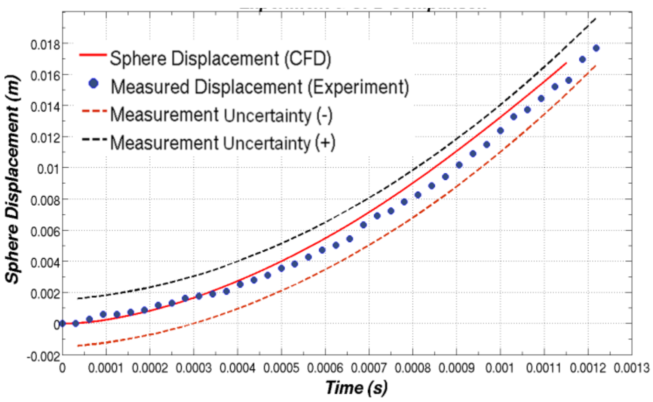

Figure 8: Displacement plot obtained from the numerical analysis compared to the experimental readings

Conclusions

The transient behaviour of a 37.5mm diameter polystyrene sphere in hypersonic flow at the T-ADFA shock tunnel has been analysed using the results from CFD simulation. The numerical results are in good correlation with the shock tunnel experimental data. The results obtained are for a one degree of freedom displacement of the sphere in hypersonic flow. The coefficient of drag of the sphere was derived to be 0.88 which is in good agreement with values obtained from literature [2]. The simulations show that the flow achieves a steady-state condition during the nominal test time of 1ms. This shows that the commercial code is capable of producing useful results, and was able to provide a wealth of additional information about the flow-field which could not have been obtained from the experiment alone.

Acknowledgments

The authors would like to thank the CFD Lab at UNSW for providing access to the high performance computing cluster to run the simulations presented in this paper.

References

[1] Anderson, J., Computational Fluid Dynamics: The Basics with Application, *McGraw-Hill, Inc,* 1995.

[2] Bailey, A.B. and Hiatt, J., 'Free-Flight Measurements of Sphere Drag at Subsonic, Transonic, Supersonic and Hypersonic Speeds for Continuum, Transition and Near-Free-Molecular Flow Conditions', *Von Karman Gas Dynamics Facility, Arnold Engineering Development Center, Arnold Air Force Station, 1971*

[3] Barth, J., Jespersen, C, 'The design and application of upwind schemes on unstructured meshes' *AIAA Aerospace Sciences Meeting, 27th, Reno, NV,* Jan 9-12, 1989.

[4] Bernstein, L., "Force measurement in short-duration hypersonic facilities", *AGARDograph No.214, Department of Aeronautical Engineering, Queen Mary College, University of London*, 1975.

[5] Menezes, V., Saravanan, S., Jagadeesh, G., and Reddy, K. P. J., 'Experimental Investigations of Hypersonic Flow over Highly Blunted Cones with Aerospikes,' *AIAA Journal, Vol. 41, No. 10*, 2003.

[6] Doig, G, 'Compressible Ground Effect Aerodynamics', PhD thesis, *School of Mechanical and Manufacturing Engineering, The University of New South Wales*, 2009.

[7] Lijewski, L., Suhs, N., 'Time –Accurate Computational Fluid Dynamics Approach to Transonic Store Separation Trajectory Prediction' *Journal of Aircraft, Vol. 31, No. 4*, July-August 1994.

[8] Mallinson, S.G., Gai, S.L. and Mudford, N.R., 'An Experimental Investigation of Hypervelocity Flow in a Conical Nozzle', *Applied Scientific Research*, vol. 57, March 1997, pp. 81-93.

[9] Neely, A.J., West, I., Hruschka, R., Park, G. and Mudford, N.R., 'Determing aerodynamic coefficients from a high speed video of a free-flying model in a shock tunnel', *ICHSIP*, Canberra, 2008.

[10] Roy, C.J. & Blottner, F.G., 'Assessment of One- and Two-Equation Turbulence Models for Hypersonic Transitional Flows', *Journal of Spacecrafts and Rocket*s, vol. 38, No. 5, September-October 2001, 699-710.

[11] Tanno, H., Komuro, T., Sato, K., Itoh, K., 'Free-flight force measurement technique in the impulsive facility in HIEST', *22nd International Congress on Instrumentation in Aerospace Simulation Facilities,* 2007