Benchmark numerical simulations of rarefied non-reacting gas flows using an open-source DSMC code

Citation for published version:

Digital Object Identifier (DOI):
10.1016/j.compfluid.2015.07.021

Link:
Link to publication record in Edinburgh Research Explorer

Document Version:
Peer reviewed version

Published In:
Computers and Fluids

General rights
Copyright for the publications made accessible via the Edinburgh Research Explorer is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy
The University of Edinburgh has made every reasonable effort to ensure that Edinburgh Research Explorer content complies with UK legislation. If you believe that the public display of this file breaches copyright please contact openaccess@ed.ac.uk providing details, and we will remove access to the work immediately and investigate your claim.
Benchmark numerical simulations of rarefied non-reacting gas flows using an open-source DSMC code

Rodrigo C. Palharini
Instituto de Aeronáutica e Espaço, 12228-904 São José dos Campos, SP, Brazil

Craig White
School of Engineering, University of Glasgow, Glasgow G12 8QQ, UK

Thomas J. Scanlon, Richard E. Brown
Department of Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK

Matthew K. Borg, Jason M. Reese
School of Engineering, University of Edinburgh, Edinburgh EH9 3JL, UK

Abstract

Validation and verification represent an important element in the development of a computational code. The aim is establish both confidence in the algorithm and its suitability for the intended purpose. In this paper, a direct simulation Monte Carlo solver, called dsmcFoam, is carefully investigated for its ability to solve low and high speed non-reacting gas flows in simple and complex geometries. The test cases are: flow over sharp and truncated flat plates, the Mars Pathfinder probe, a micro-channel with heated internal steps, and a simple micro-channel. For all the cases investigated, dsmcFoam demonstrates very good agreement with experimental and numerical data available in the literature.

Keywords: DSMC, Benchmark, Open-source, Rarefied gas dynamics, Aerodynamics, Low/high speed flows.

1Postdoctoral Research Fellow, Division of Aerodynamics.
2Lecturer, University of Glasgow.
3Senior Lecturer, James Weir Fluids Laboratory.
4Professor, Centre for Future Air-Space Transportation Technology.
5Lecturer in Mechanical Engineering.
6Regius Professor of Engineering, School of Engineering.
1. Introduction

The accuracy and reliability of computer predictions is the focus of much study and debate in the fluid dynamics community. Computational codes can only be considered reliable if they pass a through rigorous process of verification and validation (V&V). In an effort to standardize the V&V process, a significant amount of literature has been produced on the subject, e.g., [1–8]. The present study adopts the V&V definition stated in Ref. [5], i.e.,

Verification: the process of determining that an implemented model is capable of correctly performing the task it was designed for.

Validation: the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended use of the model.

In other words, verification deals with mathematics and numerics; the conceptual model that relates to the real world is not an issue. Validation deals with the actual physics and addresses the accuracy of the conceptual model with respect to the real world, i.e., as measured experimentally [4, 6].

In this paper, high and low speed inert gas flows are investigated in simple and complex geometries using the direct simulation Monte Carlo (DSMC) method [9]. DSMC is the dominant computational technique for numerical investigations of gas flows that fall within the transition-continuum Knudsen number ($Kn$) range; where

$$Kn = \frac{\lambda}{L},$$ (1)

and $\lambda$ is the mean free path of the gas, and $L$ is a characteristic length scale of the system. When the Knudsen number is small ($Kn < 0.01$), non-equilibrium effects are insignificant and the standard Navier-Stokes-Fourier (NSF) equations can accurately predict the gas behavior. As $Kn$ increases ($0.01 < Kn < 0.1$), regions of non-equilibrium begin to appear near surfaces as the molecule-surface interaction frequency is reduced; the most recognizable effect of this is velocity
slip and temperature jump, and the NSF equations with slip and jump boundary conditions can still be used effectively. However, once the Knudsen number increases into the transition-continuum \((0.1 < Kn < 10)\) and free-molecular \((Kn > 10)\) regimes, the NSF equations cannot predict the gas behavior. Recourse to solutions of the Boltzmann equation must be made, and DSMC has proven to be the most reliable method for this purpose in the transition regime, where non-equilibrium effects dominate the gas behavior but inter-molecular collisions are still important. Different forms of Knudsen number can be required to predict different types of continuum breakdown, e.g., a Knudsen number based on local flow gradient lengths can be used across shock waves \([10–12]\).

This paper is intended to be an extension of the DSMC code and results published by Scanlon et al. \([13]\), and demonstrates new developments and capabilities of the \textit{dsmcFoam} code.

\section{Code development and new features}

DSMC is a stochastic particle-based method that provides a solution to the Boltzmann equation by emulating the physics of a real gas. A discrete set of simulator particles are tracked in time and space as they interact with each other and the boundaries of the simulation domain. Particle movements are handled deterministically according to the local time step and their velocity vectors. Once all movements have been completed, inter-molecular collisions are calculated in a stochastic manner in numerical cells. The first key assumption of the method is that a single DSMC simulator particle can represent any number of real atoms or molecules. This can drastically reduce the computational expense of a simulation. Second, it is assumed that particle movements and collisions can be decoupled, which increases the allowable time-step size by several orders of magnitude in comparison with fully-deterministic particle methods, such as molecular dynamics.

The \textit{dsmcFoam} code is employed in the current paper to solve rarefied non-reacting gas flows over flat plates, the aerothermodynamics of the Mars
Pathfinder probe, and pressure-driven flow in micro-channels. This new freeware, based on Bird’s algorithms, has been developed within the framework of the open-source computational fluid dynamics toolbox OpenFOAM [14], in conjunction with researchers at the University of Strathclyde, as described in Ref. [13]. Recent dsmcFoam code improvements [15, 16] not described in Ref. [13] include: a robust measurement framework, vibrational molecular energy, the quantum-kinetic (QK) chemistry model [17], and new boundary conditions, such as implicit, prescribed pressure inlets and outlets for low speed flows [18].

3. Code sensitivity

The accuracy of a DSMC simulation relies principally on four main constraints: (i) the computational cell size must be smaller than the local mean free path if possible collision partners are restricted to a particle’s current cell, which is the case in dsmcFoam; (ii) the simulation time step must be chosen so that particles only cross a fraction of the average cell length in each time step, and the time step must also be smaller than the local mean collision time; (iii) the number of particles per cell must be large enough to preserve collision statistics; and (iv) the statistical scatter is determined by the number of samples, and for steady state problems sampling must not be started until a sufficient transient period has elapsed.

In this section we examine whether the DSMC requirements described above are rigorously respected. For this purpose, rarefied flow over a zero-thickness flat plate was chosen as a test case.

The freestream conditions are the same to those investigated by Lengrand et al. [19]. In this experimental study, a sharp flat plate of 0.1 m streamwise length and 0.1 m width was positioned at a distance from a nozzle producing a nitrogen flow with a freestream Mach number of 20.2, temperature of 13.32 K and pressure of $6.831 \times 10^{-2}$ N/m$^2$. 
In the computational solution, the geometry was constructed as a 3D flat plate, 0.1 m long and 0.1 m wide, positioned 0.005 m downstream of the uniform nitrogen stream that is parallel to the plate itself. Further details of the freestream conditions are given in Table 1. Based on these properties, and considering the flat-plate length as the characteristic length, the Knudsen number (Kn$_L$) and Reynolds number (Re$_L$) were 0.0235 and 2790, respectively.

Table 1: Freestream conditions for flat-plate simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity ($V_\infty$)</td>
<td>1503</td>
<td>m/s</td>
</tr>
<tr>
<td>Temperature ($T_\infty$)</td>
<td>13.32</td>
<td>K</td>
</tr>
<tr>
<td>Number density ($n_\infty$)</td>
<td>3.719×10$^{20}$</td>
<td>m$^{-3}$</td>
</tr>
<tr>
<td>Density ($\rho_\infty$)</td>
<td>1.729×10$^{-5}$</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Pressure ($p_\infty$)</td>
<td>6.831×10$^{-2}$</td>
<td>Pa</td>
</tr>
<tr>
<td>Dynamic viscosity ($\mu_\infty$)</td>
<td>9.314×10$^{-7}$</td>
<td>N.s/m$^2$</td>
</tr>
<tr>
<td>Mean free path ($\lambda_\infty$)</td>
<td>2.350×10$^{-3}$</td>
<td>m</td>
</tr>
<tr>
<td>Overall Knudsen (Kn$_{Lp}$)</td>
<td>0.0235</td>
<td></td>
</tr>
<tr>
<td>Overall Reynolds (Re$_{Lp}$)</td>
<td>2790</td>
<td></td>
</tr>
</tbody>
</table>

The computational domain used for the calculation was made large enough such that flow disturbances did not reach the upstream and side boundaries, where freestream conditions were specified. A schematic of the computational domain and boundary conditions is given in Fig. 1. Side I-A represents the flat-plate surface, and diffuse reflection with complete thermal accommodation to the surface temperature is the boundary condition applied to this surface. Side I-B represents a plane of symmetry. Sides II and III are boundaries with the specified freestream conditions; particles crossing into the computational domain are generated at these boundaries. Finally, side IV is defined as a vacuum boundary condition; the option for vacuum is suitable for an outflowing gas as there are no particles moving upstream if the Mach number is greater than 3.0 [9].
In order to examine the effect of the grid resolution on the wall heat transfer and pressure coefficients, a set of simulations using standard, fine, and coarse meshes were performed. Grid independence was investigated by performing calculations for different numbers of cells in the $x$- and $y$-directions, and then comparing with a solution calculated on the standard grid. Figure 1 shows the standard computational domain which was divided into two regions. Region 1 consists of 10 cells along side I-B and 80 cells along side II, while region 2 consists of 200 cells distributed along side I-A and 80 cells normal to the plate surface, i.e., along side IV. In this way, the effect of altering the cell size in the $x$-direction may be analyzed for coarse and fine grids by halving or doubling the number of cells with respect to the standard grid, while the number of cells in the $y$-direction is kept constant. The same procedure is adopted for the $y$-direction, i.e., the cell size is altered keeping the number of cells in the $x$-direction constant. According to Figure 2(a), the grid sensitivity analysis shows good agreement for the three mesh sizes investigated indicating that the results were essentially grid-independent.

In a similar manner to the grid independence study, the influence of the time step size on the aerodynamic properties was examined. The time step is chosen to be smaller than both the mean collision time (MCT) and the cell residence time ($\Delta t_{res}$), with the latter being the time taken by a DSMC particle to cross a typical computational cell in freestream conditions. Based on these conditions,
the reference time step ($\Delta t_{ref}$) was set to be $6.28 \times 10^{-8}$ s. Then, two time steps different from the $\Delta t_{ref}$ were investigated ($\Delta t_{ref}/4$ and $\Delta t_{ref} \times 4$). As shown in Fig. 2(b), the resulting simulations are essentially independent of the time step size, so long as the time step and cell size requirements are respected, in conjunction with the other good DSMC practice conditions described above.

In DSMC simulations the intermolecular collisions are the principal driver in the flow-field development. These intermolecular collisions occur in each cell, and sufficient particles should be employed not only to reduce the statistical error during the sampling process, but also to ensure the accuracy of the simulated collision rate. However, the use of a large number of particles greatly increases the computational effort. The balance between computational expense and accuracy has been studied by many authors [20–23], and 30-40 particles per cell is commonly employed [24–28]. However, there are some DSMC simulations [29, 30] that employed as few as 10 particles per cell, and some computations [31] as many as 50 to 120. The number of particles required is heavily influenced by the choice of collision model, and it is well-known that the majorant frequency scheme can use fewer particles than the no time-counter-method (NTC). Recent work has focused on reducing the number of particles required even further [32] using novel collision partner selection schemes. dsmcFoam uses the NTC method, so requires a reasonably large number of particles in order to recover the collision statistics.

In order to clarify this issue, we executed an additional study to consider the influence of the number of simulated particles on the dsmcFoam solution of a hypersonic flow over a flat plate. Considering that the standard mesh corresponded to a total of 43.7 million particles (or 13 particles per cell on average), two new cases were investigated using the same mesh. These cases corresponded, on average, to 21.8 and 87.4 million particles in the entire computational domain. The effects of such variations on the heat transfer and pressure are shown in Fig. 2(c). According to these results, the standard grid with a total of 43.7 million particles is considered sufficient for the present computations.

The accuracy of the DSMC method may also be influenced by the number
of time steps that results are sampled over \((N_s)\) [24–30]. Since the macroscopic properties of the flow are obtained by sampling all particles within a cell, the number of samples must be sufficient to minimize the statistical error. The magnitude of the statistical error reduces with the square root of the sample size, and it is important to determine the value of \(N_s\) that provides acceptable data scattering. For this purpose, the standard grid with approximately 43.7 million particles was run for 50,000, 100,000, 200,000, and 300,000 sampling time steps. Figure 2(d) shows very good agreement across the range of number of samples considered. Based on these plots, an \(N_s\) of 300,000 was considered as providing an acceptable fluctuation level for the case investigated.

In this section, hypersonic non-reacting gas flow simulations over a zero thickness flat plate were performed. Grid spacing, time step size, number of particles per cell, and number of computational samples were examined in order to test that the assumptions adopted as standard would lead to results independent of the grid, time step and number of statistical samples. On examining these results, no appreciable changes were observed; however, altering the parameters mentioned above, significantly impacted on the computational efficiency of the simulations. In the next section, we adopted the standard procedure for all of the simulations, and the results obtained using \textit{dsmcFoam} are compared to other numerical and experimental data.
Figure 2: (a) Effect of varying the number of cells, (b) the time step, (c) the number of samples, and (d) number of DSMC particles per cell on the heat transfer (left column) and pressure (right column) coefficients in the zero-thickness flat-plate case.
4. Benchmark test cases for *dsmcFoam*

The validation strategy consists of comparing the results obtained using *dsmcFoam* with other numerical, analytical, or experimental results available in the literature. In the following sections, the validation process for *dsmcFoam* is discussed in detail.

4.1. Benchmark Case A: Flow over sharp and truncated flat plates

Rarefied hypersonic flow over flat plates has been studied theoretically, experimentally, and numerically by many authors, e.g., [33–40]. The extremely simple geometry makes the flat plate one of the most useful test cases for numerical validation purposes.

The test cases we choose to validate *dsmcFoam* for non-reacting flows are based on the experimental-numerical study conducted by Lengrand *et al.* [19] and Allègre *et al.* [37]. In their experimental work, sharp and truncated flat plates of 0.1 m length ($L_p$), 0.1 m width ($W_p$), and 0.005 m thick ($T_p$) were positioned in a flow of nitrogen at two angles of incidence, 0° and 10°. The physical model was supplied with an internal water cooling system which maintained the wall temperature at 290 K. Wall pressure and heat flux measurements were made by placing pressure transducers and chromel-alumel (Ch/Al) thermocouples along the longitudinal symmetry axis of the flat plates. In addition, density flowfield measurements were carried out by employing an electron beam fluorescent technique. The uncertainties in the experimental pressure, heat flux and density measurements were estimated to be 15%, 10%, and 10%, respectively.

In addition to the experimental work, numerical simulations were performed using the NSF equations [19, 37] and the DSMC method [19, 37, 39]. The NSF results were obtained at ONERA using an implicit finite-volume method taking into account velocity slip and temperature jump at the wall. The DSMC in-house code were developed by the Laboratoire d’Aérothermique of the Centre National de la Recherche Scientifique (CNRS) [19] and the Institute of Space and Aeronautical Science (ISAS) [39]. In the DSMC computations performed
by Lengrand et al. [19], vibrational molecular energy was neglected and the Larsen-Borgnakke model [41] was employed for rotational-translational energy exchange. Particle collisions and collision sampling were performed using the variable hard sphere (VHS) model and the time-counter technique (TC) [9], respectively. However, the diatomic molecular collision (DMC) model [42] and the null-collision technique (NCT) [43] were adopted by Tsuboi et al. [39]. Since the data and assumptions employed in each method are available in the literature, the discussions below are limited only to details considered necessary.

In order to validate dsmcFoam, 3D sharp and truncated flat plates, as illustrated in Fig. 3, with the same dimensions as in Lengrand et al. [19] and Allègre et al. [37], were modeled. In the present computational solution, the two plates were immersed in nitrogen gas with an inlet imposed 0.005 m upstream of the plate. The freestream conditions (Table 1) and the computational domains are similar to those presented in Section 3. The computational mesh was composed of 4.7 million and 3.4 million cells for the sharp and truncated cases, respectively. On average, 13 DSMC particles per cell were employed in the simulations; the VHS collision model was applied, and the energy exchange between the translational and rotational modes was modeled using the Larsen-Borgnakke algorithm [41]. The NTC [44] technique was used to control the molecular collision sampling. The value of rotational collision number ($Z_{rot}$) was set to be 1 for the sharp plate to match that used by Lengrand et al. [19]. No information for $Z_{rot}$ in the truncated flat-plate case was given by Allègre et al. [37], therefore we used $Z_{rot} = 1$ and $Z_{rot} = 5$ to compare with their results. Additional simulation parameters are given in Table 2.

The resulting normalized density ($\rho/\rho_\infty$) contours for zero-thickness, sharp, and truncated flat plates are shown in Fig. 4, compared with other numerical and experimental results. Despite the different energy redistribution models and collision techniques used in each of the simulations, a very good qualitative agreement is evident between the dsmcFoam results and the numerical and experimental studies presented by Allègre et al. [37] and Tsuboi et al. [39].
Figure 3: 2D schematic of sharp (a) and truncated (b) flat plates.

Table 2: Numerical parameters for the flat-plate simulations.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>$Z_{rot}$</th>
<th>$\omega$</th>
<th>$d_{ref}$ [m]</th>
<th>$\Delta t$ [s]</th>
<th>$MCT$ [s]</th>
<th>$\lambda_\infty$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sharp</td>
<td>1</td>
<td>0.74</td>
<td>$4.17 \times 10^{-10}$</td>
<td>$6.28 \times 10^{-8}$</td>
<td>$1.90 \times 10^{-5}$</td>
<td>$3.35 \times 10^{-3}$</td>
</tr>
<tr>
<td>Truncated</td>
<td>1 and 5</td>
<td>0.74</td>
<td>$4.17 \times 10^{-10}$</td>
<td>$6.28 \times 10^{-8}$</td>
<td>$1.90 \times 10^{-5}$</td>
<td>$3.35 \times 10^{-3}$</td>
</tr>
</tbody>
</table>
Figure 4: Density ratio ($\rho/\rho_\infty$) contours around zero thickness (top left), sharp (top right), and truncated (bottom) flat plates.
The normalized density ($\frac{\rho}{\rho_{\infty}}$) and temperature ($\frac{T}{T_{\infty}}$) distributions normal to the sharp flat-plate surface at the non-dimensional streamwise location $X/L_p = 0.75$ are shown in Fig. 5. Good agreement is found between the DSMC calculation and the experimental results. The density peak is captured well by the present simulation, and the normalized density profile follows the same trend of the numerical and experimental results performed by Tsuboi et al. [39] and Lengrand et al. [19], respectively. The NSF simulations of Lengrand et al. [19] were not able to predict correctly the density profile at the position considered.

Analyzing the translational and rotational temperature profiles in Fig. 5(b), a difference between the rotational and translational temperatures is observed, which indicates thermally non-equilibrium conditions. The normalized temperature is low close to the surface, increases to a maximum value inside the shock layer at $Y = 0.05$ and then declines to the freestream temperature at the upper boundary condition. In general, there is very close agreement of translational and rotational temperature profiles from *dsmcFoam* and the CNRS DSMC code [19].

![Normalized density and temperature profiles](image_url)  

(a) Normalized density ($\frac{\rho}{\rho_{\infty}}$) and (b) normalized temperature ($\frac{T}{T_{\infty}}$) profiles normal to the sharp flat plate at streamwise position $X/L_p = 0.75$. 

---

14
Figure 6 shows the heat transfer ($C_h$), pressure ($C_p$) and skin friction ($C_f$) coefficients along the flat plates. For the sharp flat plate case (left column) the comparison of the $dsmcFoam$ results with the experimental data is better than found by Lengrand et al. [19] and Tsuboi et al. [39]. The skin friction coefficient shows good agreement with the Lengrand et al. [19] DSMC results at the leading edge and from position $X/L_p \cong 0.4$ to 1.0. When the NSF calculations for $C_h$ and $C_p$ for a rarefied flow over a sharp flat plate are compared with experimental and DSMC results, it is clear that the CNRS NSF simulations were unable to capture the surface quantities for the conditions investigated.

According to Lengrand et al. [19], possible sources of experimental error are related to uncertainties in the freestream conditions, measurement procedures, and the influence of the leading edge bluntness or bevel angle. In order to investigate the impact of the leading edge bluntness, Fig. 6 (right column) shows comparisons of the $dsmcFoam$ results with experimental data from Allègre et al. [37] and previous DSMC simulations. From the heat transfer ($C_h$) plot, excellent agreement is seen between the DSMC simulations apart from at the leading edge ($X/L_p = 0$) where the $dsmcFoam$ results do not tend to zero. Both computations demonstrated significant difference when compared with experimental data.

In contrast with $C_h$ results, the pressure coefficient ($C_p$) shows very good agreement between numerical and experimental data. However, the numerical results show slightly higher values for $C_p$ at the flat-plate leading edge. For the skin friction coefficient along the truncated flat plate, no numerical results were available in the literature. Since the value of $Z_{rot}$ was not specified in Ref. [37], $dsmcFoam$ computations for rarefied gas flow over the truncated flat plate were performed with $Z_{rot} = 1$ and 5; however, no significant differences in the surface quantities were observed.

In summary, hypersonic non-reacting gas flow over three-dimensional zero-thickness, sharp, and truncated flat plates was simulated using $dsmcFoam$. Excellent agreement between numerical and experimental data for the density contours was found. The results also demonstrated that the shape of the leading
edge can affect the flow and shock structure over the plate. For the aerodynamic properties on a sharp flat plate, satisfactory agreement was found from the leading edge up to \( X/L_p = 0.2 \); however, certain discrepancies were observed further along the plate. In contrast, the truncated case exhibited differences between the numerical and experimental data in the leading edge region, while better agreement was evident towards the trailing edge. Comparisons between the
DSMC and NSF results demonstrate that the continuum approach, even when using slip velocity and temperature jump boundary conditions, cannot be used with confidence to predict these types of thermodynamically non-equilibrium flows.

4.2. Benchmark Case B: Flow over a 70° blunted cone

The flow over blunt bodies at high speeds and high altitudes displays complex flow interactions, and requires a precise determination of the heating rate, aerodynamic forces, and the flowfield surrounding the body. The characterization of the wake region is also a key factor for the success of re-entry missions.

In an experimental set-up, a 70° blunted cone, identical in geometric proportions to that of the Mars Pathfinder probe, was chosen by the AGARD Working Group 18 [45]. Rarefied flow experiments were performed in five different facilities: the SR3 wind tunnel at CNRS-Meudom, the V2G, V3G and HEG wind tunnels at DRL-Göttingen, and the LENS wind tunnel at the Buffalo Research Center (Calspan, University of Buffalo, USA). The experimental test conditions used in each of these experimental facilities are available in Ref. [45].

Allègre et al. [46–48] provided detailed information regarding experiments conducted at CNRS-Meudon. The CNRS group employed three freestream flow conditions, representative of different levels of rarefaction, and three probe models, each one having a base and afterbody sting diameter of 0.05 and 0.0125 m, respectively.

The CNRS model utilized for the flowfield density measurements was made of brass, water cooled, with a wall temperature remaining close to 290 K during all measurements. An electron beam fluorescent technique was used to measure the density field around the blunted cone [46]. For the aerodynamic force measurements, the model was made of aluminum, uncooled, with the wall temperature estimated to be close to 350 K. The model was directly attached to an external balance providing direct measurements of drag, lift, and pitching moment, and indirect determinations of the center of pressure at different angles of attack [47]. For heat transfer measurements, a steel model was used
in which the wall temperature was kept close to 300 K [48]. Chrome-alumel (Ch/Al) thermocouples were embedded through the wall thickness at nine locations along the forebody, base plane, and sting, and the thin-wall technique was applied to measure the heat fluxes on the steel probe.

An extensive set of simulations at these experimental test conditions were completed using both DSMC [49–58] and Navier-Stokes [59–62] methods prior to the release of the experimental data. In this way, it was possible to perform a blind validation test of the computational codes.

In the present work, the simulated freestream conditions are the same as those used in the SR3 low-density wind tunnel (case 1) [46–48]. The Mars Pathfinder probe was immersed in a non-reacting uniform nitrogen flow of velocity, mass density, and temperature equal to 13.316 m/s, $1.73 \times 10^{-5}$ kg/m$^3$, and 13.316 K, respectively. Energy exchange was allowed between the translational and rotational modes and was controlled by the Larsen-Borgnakke phenomenological model [41]. Molecular collisions were modeled using the variable hard sphere (VHS) model [63], and the no-time-counter (NTC) collision sampling technique [44]. In addition, simulation parameters for N$_2$ are: reference diameter ($d_{ref}$), rotational collision number ($Z_{rot}$) and viscosity index ($\omega$) set equal to $4.17 \times 10^{-10}$ m, 5, and 0.74, respectively [9].

Figure 7(a) shows the experimental model configuration and Fig 7(b) gives an amplified view of the dsmcFoam computational grid. The computational grid was composed of a mixture of 7.1 million hexa- and polyhedral cells with, on average, 10.5 simulated particles per cell. A uniform hexahedral mesh, with cell sizes smaller than the freestream mean free path, is used for most of the domain, with some polyhedral cells to capture the surface geometry. Each simulation was performed using 240 processors on the parallel machine at the University of Strathclyde, and 10 days were required to fully resolve each of the cases.

The computational domain was large enough so that the upstream, downstream, and upper boundary conditions could be specified as freestream. In order to minimize computational effort, quarter symmetry was employed for $0^\circ$ angle of attack. Undisturbed freestream conditions were imposed 0.02 m
upstream of the probe, and the computational domain normal to the probe extended a distance 0.08 m in the y- and z-directions. The surface temperature was set at 290 K, 300 K, and 350 K for the density, heat transfer and aerodynamic force measurements, respectively. The surface boundary condition assumed the gas-surface interaction to be diffuse, with full thermal accommodation at the specified surface temperature.

![Experimental 70° blunted cone model for heat transfer measurements with thermocouple locations](image1)
![Schematic of the corresponding dsmcFoam 3D computational mesh](image2)

Figure 7: (a) Experimental 70° blunted cone model for heat transfer measurements with thermocouple locations from [48], and (b) schematic of the corresponding dsmcFoam 3D computational mesh.

In Fig. 8 experimental density flowfields at different angles of attack [46] are compared with the results from the dsmcFoam calculations. Qualitatively, the results show a good level of agreement between the experimental and dsmcFoam results. According to Allègre et al. [46] the flowfield density measurement accuracy is estimated to be 10%, except in the region encompassing the forward shock wave, which is characterized by high density gradients and has a higher uncertainty.

Comparison is also made with the DAC (DSMC Analysis Code) simulations, developed at the NASA Johnson Space Flight Center [64] and available
in Ref. [49]. In Fig. 9, excellent agreement of the density ratio \((\rho/\rho_\infty)\), overall temperature \((T_{ov})\), and Mach number \((M)\) contours at 0° degree angle of attack is found between the codes; where

\[
T_{ov} = \frac{T_{trans}\xi_{trans} + T_{rot}\xi_{rot}}{\xi_{trans} + \xi_{rot}},
\]  

(2)

with \(T_{trans}\) and \(T_{rot}\) the translational and rotational temperatures of the gas, respectively, and \(\xi_{trans}\) and \(\xi_{rot}\) the number of degrees of freedom in the translational and rotational modes, respectively.
Figure 8: Density ratio ($\rho/\rho_\infty$) distributions from *dsmcFoam*, and from the SR3 experiments [46] at different angles of attack ($\alpha$), and Mach number 20.2.
Figure 9: (top left) Density ratio ($\rho/\rho_\infty$), (top right) overall temperature ($T_{ov}$), and (bottom) Mach number ($M$) distributions from `dsmcFoam` and from the DAC simulations [49] at 0° degree of attack, and Mach number 20.2.
The aerodynamic forces and moments have also been experimentally and numerically investigated [47, 51, 53] for different angles of attack. In Figs. 10 and 11 a satisfactory concurrence is found between the experimental data and dsmeFoam simulations. According to Allègre et al. [47], the global uncertainty in the aerodynamic coefficients and forces did not exceed ±3%, and the maximum difference between measured and simulated results was 8.6% on the normal force at 30° angle of attack. Table 3 shows the drag and lift coefficients, and the axial and normal forces coefficients, from the experimental measurements and numerical predictions using the dsmeFoam code.

![Figure 10: Drag (C_d) and lift (C_l) coefficients at different angles of attack (α).](image)

![Figure 11: Axial (C_A) and normal (C_N) force coefficients at different angles of attack (α).](image)
Table 3: Experimental and *dsmcFoam*-calculated aerodynamic and force coefficients.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>1.657</td>
<td>1.652</td>
<td>0.302</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>1.657</td>
<td>1.652</td>
<td>-0.302</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>5°</td>
<td>1.629</td>
<td>1.642</td>
<td>-0.798</td>
<td>-0.057</td>
<td>-0.062</td>
<td>8.772</td>
<td>1.628</td>
<td>1.642</td>
<td>0.860</td>
<td>0.084</td>
<td>0.080</td>
<td>-4.762</td>
</tr>
<tr>
<td>10°</td>
<td>1.615</td>
<td>1.611</td>
<td>-0.248</td>
<td>-0.133</td>
<td>-0.143</td>
<td>7.519</td>
<td>1.614</td>
<td>1.615</td>
<td>0.0620</td>
<td>0.148</td>
<td>0.140</td>
<td>-5.405</td>
</tr>
<tr>
<td>15°</td>
<td>1.569</td>
<td>1.561</td>
<td>-0.510</td>
<td>-0.200</td>
<td>-0.213</td>
<td>6.500</td>
<td>1.568</td>
<td>1.568</td>
<td>0.000</td>
<td>0.213</td>
<td>0.200</td>
<td>-6.103</td>
</tr>
<tr>
<td>20°</td>
<td>1.538</td>
<td>1.496</td>
<td>-2.731</td>
<td>-0.249</td>
<td>-0.269</td>
<td>8.032</td>
<td>1.539</td>
<td>1.564</td>
<td>-0.700</td>
<td>0.291</td>
<td>0.266</td>
<td>-8.591</td>
</tr>
<tr>
<td>30°</td>
<td>1.432</td>
<td>1.356</td>
<td>-5.726</td>
<td>-0.324</td>
<td>-0.351</td>
<td>8.333</td>
<td>1.462</td>
<td>1.344</td>
<td>-4.137</td>
<td>0.434</td>
<td>0.398</td>
<td>-8.294</td>
</tr>
</tbody>
</table>

The effect of the angle of attack on the heat transfer ($C_h$) and pressure ($C_p$) coefficients is shown in Figs. 12 and 13. In this set of plots, the results are presented as a function of the normalized arc distance ($s/R_n$) measured from the forebody stagnation point to the end of the sting. Here, the *dsmcFoam* results are compared with DSMC computations provided from the DAC [50] and molecular gas dynamics simulator (MGDS) [40] codes, as well as with the experiments performed at the CNRS facilities [48].

According to Fig. 12, *dsmcFoam* shows a good agreement with DAC and MGDS for all angles of attack considered. The heat transfer coefficient is captured well by the numerical codes at the forebody and sting regions; however, a significant difference in $C_h$ is seen between computations and experiments at the probe shoulder ($S/R_n \approx 2$). This difference is even higher when the angle of attack is increased, as shown in Fig. 14. At 30° angle of attack, the flow is compressed against the probe shoulder generating high heating rates in this region. Nevertheless, there is no thermocouple at this position; the last thermocouple on the forebody region is located at $S/R_n = 1.56$ but the simulated heat transfer peak occurs at $S/R_n \approx 2.0$. For this reason, the peak in the heat transfer is not captured by the CNRS experiments. Even at locations where thermocouples are present, both *dsmcFoam* and MGDS predict higher heat fluxes than the experiments. Again, at 30° angle of attack, MGDS predicts a slightly lower heat flux at the stagnation point, compared to *dsmcFoam*. MGDS used a mesh refinement algorithm to ensure cells stay smaller than the local mean free path, whereas *dsmcFoam* uses a fixed grid, so the cells in the high density region near
the stagnation point may be larger than the local mean free path.

Figure 12: Heat transfer coefficients ($C_h$) along the Mars Pathfinder surface from $dsmcFoam$, DAC, MGDS and CNRS experiments at different angles of attack ($\alpha$).

Figure 13 shows the pressure coefficient along the Mars Pathfinder probe surface. Excellent agreement is found between the $dsmcFoam$ and DAC codes for $0^\circ$ angle of attack. As the angle of attack is increased, the pressure coefficient increases at the probe shoulder, following the same trend as the heat transfer coefficient.
Figure 13: Pressure coefficient ($C_p$) along the Mars Pathfinder surface for *dsmeFoam* and DAC simulations at different angles of attack ($\alpha$): full geometry (left), and forebody surface (right).

Figure 14: (top) Heat transfer ($C_h$) and (bottom) pressure ($C_p$) coefficient contours at $0^\circ$ (left) and $30^\circ$ (right) angles of attack ($\alpha$), and Mach number 20.2.
When a probe enters a planetary atmosphere at high velocities, the forebody flowfield is dominated by a strong shock wave that causes the excitation, dissociation and possibly ionization of the gas surrounding the vehicle. This highly thermochemically non-equilibrium flow rapidly expands around the probe shoulder into the near wake region with a significant increase in rarefaction [49, 58, 65]. The flowfield complexity for the Mars Pathfinder probe is shown in Fig. 15(a). Due to this complexity, the aerothermodynamics of the wake may not be measured accurately; according to Wright and Milos [66] the uncertainty in the aeroheating measurements in this region is typically assumed to be in the range of 50-300%. This level of uncertainty plays a significant role in the vehicle design and the correct selection of a thermal protection system (TPS).

In order to compare the results obtained using the dsmcFoam code with those from DAC simulations provided by Moss et al. [49], normalised density, velocity, and temperature profiles are presented at four different locations in the probe afterbody region as depicted in Fig. 15(b).

![Figure 15: (a) Schematic of the planetary probe flow structure [58], and (b) macroscopic properties measurement locations.](image)

From Figs. 16 to 18, it is clear that there is very good agreement between the DAC and dsmcFoam simulations. However, for the density and temperature profiles at location $X_1 = 0.0095$, some slight discordance is observed. In this region a very strong flow expansion occurs and the different mesh densities...
between the two simulations may have some influence on the flowfield structure.

To summarize this section, simulations have been performed using the *dsmcFoam* code for non-reacting flows over both flat plates and the Mars Pathfinder probe. The present data are compared with experimental and numerical solutions available in the open literature. Assuming the average uncertainty in the experimental data to be approximately 10% [19, 37, 46–48], a satisfactory level of agreement between the measurements and computations has been achieved.

Figure 16: Density profiles \( \left( \frac{\rho}{\rho_{\infty}} \right) \) computed by *dsmcFoam* and DAC.
Figure 17: Velocity profiles ($V_x/V_\infty$) computed by *dsmcFoam* and DAC.

Figure 18: Temperature profiles ($T_{\text{ov}}/T_\infty$) computed by *dsmcFoam* and DAC.
4.3. Benchmark Case C: flow in patterned 2D microchannels

The previous test cases have been for hypersonic flows, but progress has also been made in the extension of the *dsmcFoam* code to subsonic, pressure-driven flows in micro- or nano-scale geometries. In contrast to the previous cases where the Knudsen number is high because the gas density is low, micro-scale devices often operate in standard atmospheric conditions. The Knudsen number is high in these types of problems because the characteristic length scale $L$ is small.

*dsmcFoam* has previously been benchmarked for planar Poiseuille flow with defined pressure inlets and outlets [67], where comparison to analytical solutions for the non-linear pressure profile were presented. The general starting point for the treatment of an inlet or outlet boundary condition in DSMC is to impose a particle flux. The rate of particle insertion, $\dot{N}$, can be computed from the equilibrium Maxwell-Boltzmann distribution, which requires boundary values of temperature, density, and velocity. The streaming velocity profiles for internal micro-scale flows at the inlet and outlet boundaries are generally not known *a priori*, so the boundary conditions described below use the theory of characteristics to calculate the local streaming velocity as the simulation proceeds.

Wang and Li [18] proposed an inlet boundary condition with target gas properties of pressure $p_{in}$ and temperature $T_{in}$, prescribed at the inflow boundary. The perfect gas law is used to calculate the inlet number density $n_{in}$,

$$n_{in} = \frac{p_{in}}{k_B T_{in}}. \quad (3)$$

Based on the theory of characteristics, the stream-wise $u_{in}$ and tangential $v_{in}$ velocities at two-dimensional inlet boundary faces $f$, using values from the boundary cell centres $j$, are calculated as

$$\left( u_{in} \right)_f = u_j + \frac{p_{in} - p_j}{\rho_j a_j}, \quad (4)$$

and

$$\left( v_{in} \right)_f = v_j, \quad (5)$$
where \( u_j \) and \( v_j \) are first order extrapolations from the cells attached to the relevant boundary face, \( \rho \) is mass density and \( a \) is the local speed of sound. The pressure \( p_j \) is calculated in these boundary conditions from the overall temperature as:

\[
p_j = \rho_j R \left[ \frac{3T_{tr} + \bar{\zeta}_{rot}T_{rot}}{3 + \zeta_{rot}} \right]_j.
\]  

(6)

At the exit boundaries, only the pressure is defined and the boundary conditions are those first proposed by Nance et al. [68]:

\[
(\rho_{out})_f = \rho_j + \frac{p_{out} - p_j}{(a_j)^2},
\]  

(7)

\[
(u_{out})_f = u_j + \frac{p_j - p_{out}}{\rho_j a_j},
\]  

(8)

\[
(v_{out})_f = v_j.
\]  

(9)

\[
(T_{out})_f = p_{out} / \left[ R \left( \rho_{out} \right)_f \right].
\]  

(10)

The pressure \( p_j \) is again calculated from Eq. (6). The process for selecting the required translational and rotational energies for particles at the boundaries is standard in DSMC, and details can be found in Ref. [9].

Here, we investigate a pressure-driven flow through a micro-channel with two heated steps on its lower surface, as shown in Fig. 19, which was first considered using DSMC by Fang and Liou [69]. The inlet pressure is 0.73 atm and the inlet temperature is 300 K, giving an inlet Knudsen number, based on the channel height \( H \) and the VHS mean free path, of around 0.08. Cases with inlet-to-outlet pressure ratios \( (p_{in}/p_{out}) \) of 2.5 (Case 1) and 4 (Case 2) are investigated here, using the inlet and outlet boundary algorithms described above. All surfaces are considered to be fully diffuse, with temperatures of 323 K and 523 K for \( T_1 \) and \( T_2 \), respectively. The non-uniform, non-isothermal geometry greatly increases the complexity of the channel flow problem.

The channel height \( H \) is 0.9 \( \mu \)m and the aspect ratio is 6.7. The steps inside the channel have a height \( h \) of 0.3 \( \mu \)m, a length of 1.0287 \( \mu \)m, and respective
leading edge positions of 1.4787 µm and 3.4713 µm. The working gas is nitrogen, with standard VHS parameters of \( \omega = 0.77 \) and \( d_{ref} = 4.17 \times 10^{-10} \) m at a reference temperature of 273 K, and Larsen-Borgnakke energy exchange performed on a ‘single molecule’ basis, where each collision partner is considered in turn for relaxation with a constant rotational relaxation number of 5. Vibrational energy is excluded from the calculations because of the relatively low temperatures involved. Many of these parameters are not defined in Fang and Liou [69], so there may be some uncertainty in the results. 7656 rectangular computational cells, and a constant time step of \( 1 \times 10^{-11} \) s were used in the \textit{dsmcFoam} simulations; post-processing of the results confirmed that these parameters met good DSMC practice throughout the entire domain. The \textit{dsmcFoam} results that follow have been sampled for 200,000 time steps after steady state was achieved. Case 1 comprised around 220,000 DSMC particles, and Case 2 had 300,000. The simulations were performed in parallel on two cores of a desktop PC equipped with an i7 processor, and took around 24 hours for each simulation. Figure 20 shows the contours of overall temperature \( T_{ov} \) for Case 2.
Figure 21 shows a comparison of the overall temperature profiles for the $p_{in}/p_{out} = 2.5$ case, along two lines for the length of the channel. These two locations are illustrated by the dashed lines in Fig. 19; the first location is at the top of the steps, while the second one is mid-way between the top of the steps and the upper surface. Excellent agreement between the independent DSMC results can be seen here, with the peaks in the temperature profiles corresponding to the locations of the steps. The results for Case 2 also show excellent agreement, but have been omitted for conciseness.

![Figure 21: Comparison of temperature distribution results from Fang and Liou [69], and dsmcFoam for $p_{in}/p_{out} = 2.5$.](image)

Figure 22 shows the heat transfer at the upper surface for both $p_{in}/p_{out} = 2.5$ and 4. In general, the agreement between the DSMC results is very good, but dsmcFoam predicts a slightly higher heat transfer from $x/h = 2.4$ to 3.4 for both pressure ratios. The peak heat transfer around the step locations are lower for the $p_{in}/p_{out} = 4$ case, particularly at the second step, because the gas is more rarefied and so heat transfer from the gas to the surface is reduced.
4.4. Benchmark Case D: Knudsen minimum

In order to compare micro-scale results from *dsmcFoam* to available experimental data, a series of 2D isothermal pressure-driven Poiseuille flows of nitrogen gas are solved over a large range of Knudsen number. Figure 23 is a sketch of the simple geometry.

![Figure 23: 2D micro-channel geometry for pressure-driven Poiseuille flow.](image)

The variable hard sphere collision model is used with the standard nitrogen properties at a reference temperature of 273 K, i.e. viscosity coefficient \( \omega = 0.74 \) and reference diameter \( d_{ref} = 4.17 \times 10^{-10} \) m. The mass flux \( \dot{m} \) is measured...
and normalized as follows:

\[ Q = \frac{nL\sqrt{2RT}}{h (p_{\text{in}} - p_{\text{out}})} \]  

(11)

where \( L \) and \( h \) are the length and height of the 2D planar Poiseuille flow channel, respectively, and \( R \) is the specific gas constant; \( T \) is the isothermal temperature that the simulations were performed at, which is 273 K. This value is used for the boundary condition of the inlet gas temperature, and is also the temperature assigned to the fully diffuse surfaces of the channel walls. The inlet pressure \( p_{\text{in}} \) and outlet pressure \( p_{\text{out}} \) are set using the boundary conditions procedure of §4.3. A rarefaction parameter \( \delta_m \) is defined as the average Knudsen number of the inlet and outlet Knudsen numbers (based on the VHS mean free path and the channel height \( h \)) in each case:

\[ \delta_m = \frac{\sqrt{\pi}}{2Kn_m} \]  

(12)

The inlet to outlet pressure ratio in all cases is 3, and the aspect ratio of the planar Poiseuille geometries is 20. The gas densities are varied to achieve the different Knudsen numbers. Our \textit{dsmcFoam} results are compared in Fig. 24 to

Figure 24: Normalised mass flow rate, showing the Knudsen minimum phenomenon.
previous DSMC results \cite{70} using nitrogen gas in a channel of the same aspect ratio. Experimental results from Ewart et al. \cite{71}, which were obtained with helium gas, are also plotted for comparison. The two sets of DSMC results are in good agreement, and the agreement with experimental data is excellent at low Kn and reasonable at high Kn. It has previously been noted \cite{72} that the asymptotic value that $Q$ obtains is proportional to ln ($L/h$); since the experimental work was performed on geometries with very large aspect ratio ($L/h = 1000$), it is expected that the DSMC results for an aspect ratio of 20 will not match exactly. Unfortunately, it is not possible to simulate an aspect ratio of 1000 using DSMC, as the velocities would be too low to obtain a converged solution in a practical time scale. The famous Knudsen minimum \cite{73} can clearly be observed in Fig. 24, where the normalized mass flow rate has a minimum at about $Kn = 1$.

5. Conclusions

The verification and validation of new developments and features in the \textit{dsmcFoam} code have been presented for high and speed non-reacting flows in different geometries. First, sensitivity analyses were carried out for mesh, time step, number of samples and particles was carried out for a flow over a zero-thickness flat plate. Choosing cell sizes, time steps, number of particles, and number of samples within the ranges dictated by good DSMC practice, led to solutions that were independent of these simulation parameters. The validation procedure aimed to compare computed \textit{dsmcFoam} results with other numerical and experimental data available in the literature. Four different geometries were employed in the investigation: sharp and truncated flat plates, the Mars Pathfinder probe, a micro-channel with heated steps, and a simple micro-channel. In the flat plate cases, the density contours and temperature profiles showed a good concurrence between numerical and experimental data. The leading edge shape was shown clearly to influence the surface quantities. In these results, good agreement was found at the leading and trailing
edges of the sharp and truncated flat plates, respectively. In addition, conventional CFD results showed marked differences from both the DSMC simulations and experimental data, demonstrating that rarefied gas effects are not captured well by a continuum-based solver.

Hypersonic rarefied non-reacting gas flows over the Mars Pathfinder probe were also investigated. The *dsmcFoam* solver demonstrated its capabilities to successfully resolve hypersonic flows over such complex geometries. Aerodynamic surface quantities, the flow structure in the shock and wake regions, the drag, lift, and axial and normal forces acting on the probe all show a high level of agreement with CNRS experiments as well as numerical results from the DAC and MGDS codes.

In addition to the high speed benchmark cases, low speed gas flow through a micro-channel with two heated steps was considered in order to further validate the new pressure-driven *dsmcFoam* boundary conditions. The results were compared with published DSMC simulations, and an excellent level of agreement was found. In order also to compare with available micro-scale experimental data, normalized mass fluxes were calculated over a range of Knudsen numbers to demonstrate that the Knudsen minimum in Poiseuille channel flow can be captured. The results of these cases further validates the work reported in Refs. [15, 67] on subsonic, prescribed pressure inlets and outlets.

6. Acknowledgements

The authors acknowledge the financial support provided by Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq/Brazil) under Grant No. 200473/2010-7. Our calculations were performed on the 1100 core High Performance Computing facility of the Faculty of Engineering at the University of Strathclyde.
References


39


