FOSSEE, IIT Bombay
OpenFOAM Case Study Project
January 31, 2020

DSMC simulations of Mach 10 hypersonic rarefied flow over a 2D cylinder

Mrinal Mahato

M.Tech., Aerodynamics

Department of Aerospace Engineering

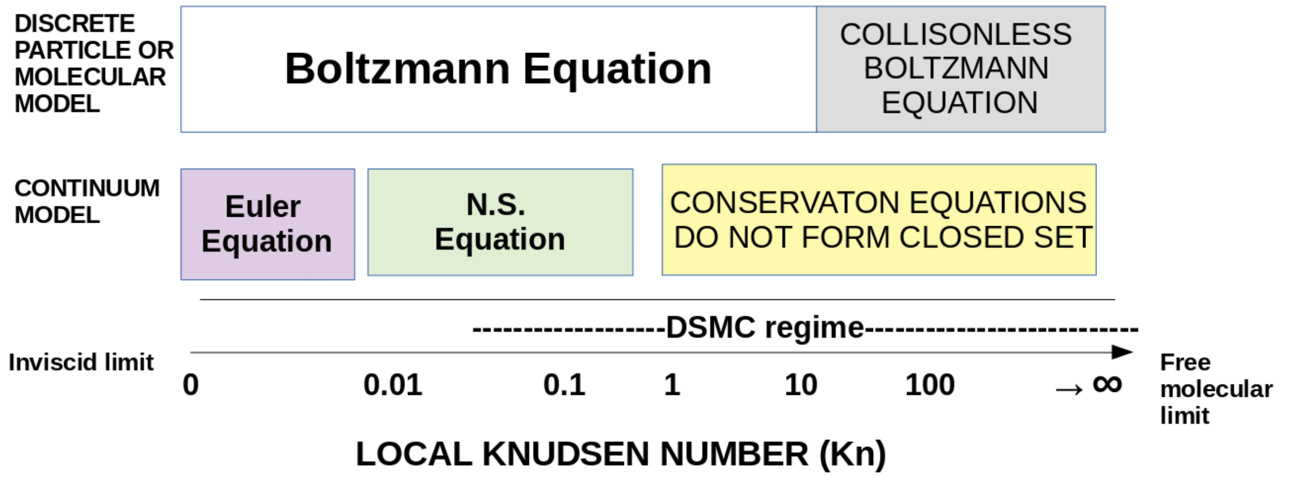
IIT Bombay, Powai, Mumbai-76

Abstract

Rarefied hypersonic flow around a 2D cylinder has been simulated using dsmcFoam. The flow is characterised by single species Ar and a Kn 0.25 as well as Mach 10. The ambient temperature is 200 K alongwith the wall temperature is provided as 500 K. Different grids are simulated and the extracted data is compared with available literature. These results are discussed. Rarefied flows are characterised by velocity and temperature slip close to the wall. These observations are discussed.

1. Introduction:

Rarefied flows are characterised by flow Knudsen number or Kn. Low density or rarefied conditions can encounter non-conservative quantities. Conventional FVM solvers work on an assumption that the fluid is in continuum. As the flow conditions become rarer and the density reduces, the continuum assumption starts to break down.



***reproduced from original book, ''Molecular gas dynamics' Page 20 by G.A. Bird,**

Fig. 1. Simulation methods versus Knudsen number (Kn)

This can lead to non-conservation of flow properties which conventional FVM solvers require additional modifications to simulate. DSMC solver however is categorically easier to implement for these conditions. DSMC or Direct Simulation Monte Carlo is a lagrangian solver based on Boltzmann Equation which solves on DSMC particle who themselves are collection of real flow molecules. The basic assumption is to separate the movement of DSMC molecule and the collision physics for an appropriately small time step. Collision momentum transfer is solved using statistical method and hence Monte Carlo. There are many commercial DSMC solvers as well as solvers available under GPU licenses. One such solver is dsmcFoam which is part of the OpenFoam suite.

A test case of flow over a 2D cylinder is chosen to validate dsmcFoam solver. This case is chosen to ease in into DSMC simulations. Simulations are carried out for different grids and different simulation parameters are compared. All results are compiled and discussed.

2. Flow Conditions:

For this study, the flow is characterised by single species Ar and the Knudsen number for the flow is 0.25. The cylinder diameter is 10 inches or 0.3048 m, correspondingly the mean free path ‘λ’ of the flow is 0.0762.

$$Kn\,(Knudsen\,number)=\frac{\lambda\,(Mean\,free\,path)}{l\,(characteristic\,length)}$$

The particle density for this condition is calculated to be 1.699x10^{19} m^{-3}. These equations are used to calculate 'n'.

$$\lambda = \frac{K_B T}{\sqrt{2}\pi d^2 P} \qquad\qquad P = n K_B T$$

Grids are generally created with thickness of the order of λ. Time steps for the simulation is calculated using following relations.

$$U_{RMS} = \sqrt{\frac{8 K_B T}{\pi m}} \qquad\qquad \Delta t < \frac{\lambda/3}{U_{RMS}}$$

It is found out that the Δt is of the order of 4 x 10^{-6} secs. Geometry and grid details are discussed in next section. Flow velocity is 1347.6 m/s which corresponds to Mach 10 and the ambient temperature of the flow is 200 K. The wall is kept at a constant temperature of 500 K.

3. Geometry and Mesh Details:

This case comprises of a simple geometry, i.e. a 2D cylinder and appropriate boundary faces upto 2D from the cylindrical surface. As the geometry shown in Fig. 2 is symmetric, only half of the problem setup is considered for grid generation. Open source software Gmsh is used for grid generation.

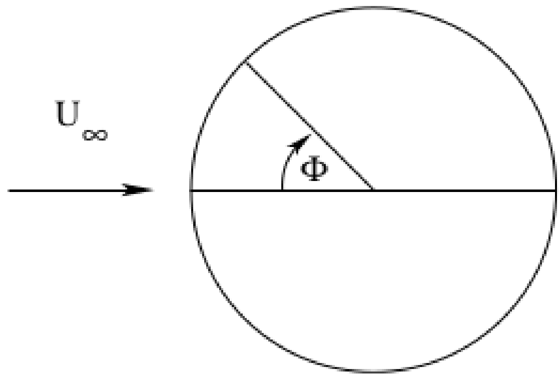


Fig. 2. 2D cylinder geometry

Grids generated in Gmsh are essentially 2D grids with a single unit element defined in z-axis. The front and back surfaces are defined as symmetry surfaces (in OpenFoam, these surfaces can be defined by 'symmetry' or 'empty'). The first attempt of grid generation is shown in Fig.3 which consist of simple blocks. Fig. 4 shows a grid with adjusted multi-blocks to reduce cell skewness.

Fig. 5 shows an un-structured grid to validate the solver for unstructured meshes. Fig. 6 shows a modified structured grid with changed boundaries, which helps in simplifying the mesh domain further and can reduce skewness.

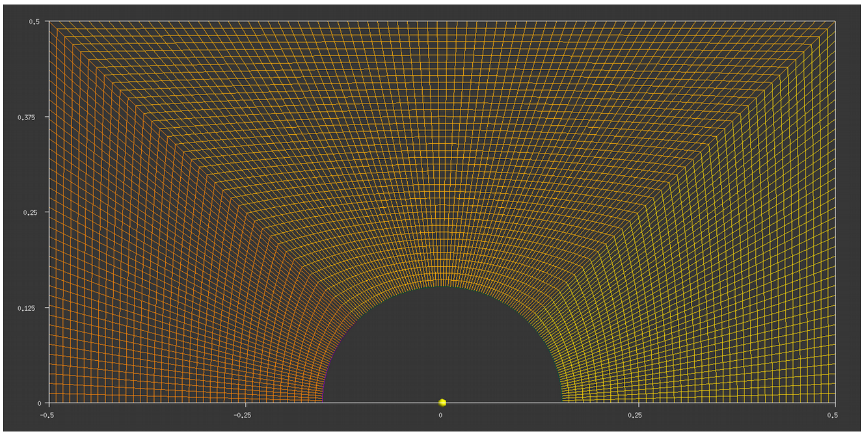


Fig. 3. Simple strctured grid for Kn = 0.25

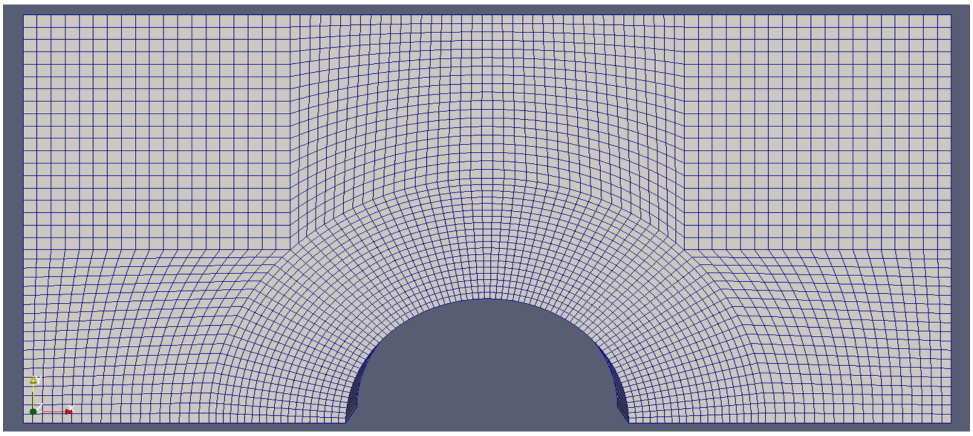


Fig. 4. Multi-block strctured grid for Kn = 0.25

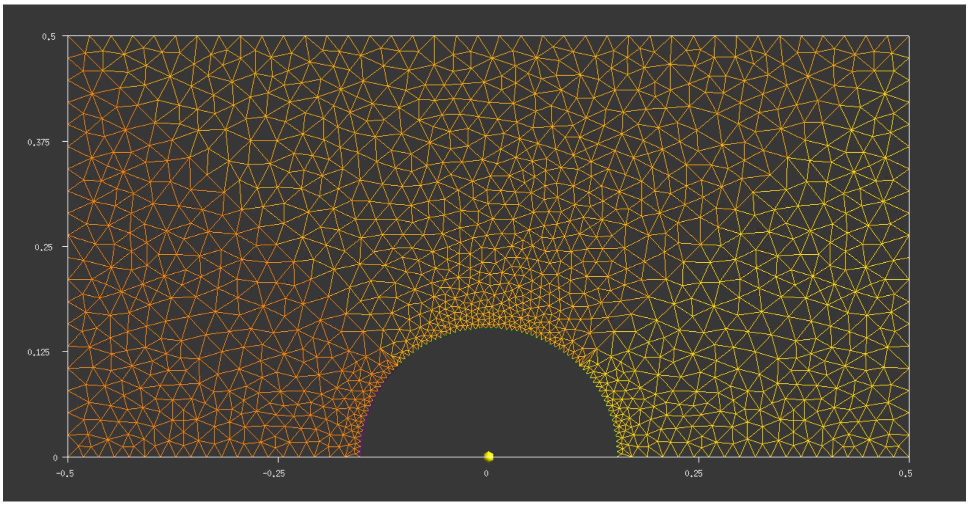


Fig. 5. Unstructured grid for Kn = 0.25

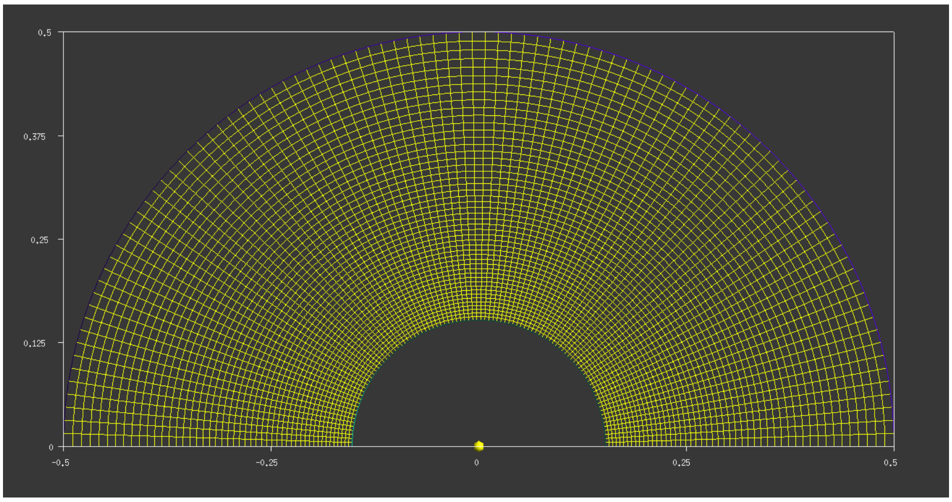


Fig. 6. Semi-circular structured grid for Kn=0.25

4. Problem Set-up:

A typical problem folder in dsmcFoam environment consists of three subfolders, mesh file and some monitoring scripts. These subfolders are ‘system’, ‘constant’ and ‘0’. The first one contains dictionary files for various aspects of simulations such as controlDict, decomposeParDict, dsmcInitialiseDict etc. Similarly ‘constant’ folder has files such as dsmcProperties. ‘0’ folder contains files with different variable names which essentially is used to provide initial conditions for the flow simulation. Boundary conditions are provided here too. Steps to start a DSMC simulation using dsmcFoam are as follows:

- Command *gmshToFoam **.msh* is run in the home terminal.
- 'boundaries' file is modified in folder constant/polymesh/ (if mesh is generated using blockMesh, boundaries are defined in the input file itself, then this step is not necessary)
- *decomposePar* – If the user wants to run the simulation in more than one core. User has to check the decomposeParDict file. The domain will be decomposed based on divisions defined here)
- *dsmcInitialise*-(this is used for populating the domain with DSMC particles) in parallel the command is :// ***mpirun -np X dsmcInitialise -parallel > log &***
- *dsmcFoam* :// the solver can run now. In parallel :// ***mpirun -np X dsmcFoam -parallel > log &***
- *reconstructPar* :// Command to reconstruct the decomposed data from various core folders.

5. Results and discussions:

Individual grids discussed above are populated with DSMC particles where each DSMC particle corresponds to about 10^{14} actual particles. Based on the volume of the domain and the number density provided, DSMC particles are populated inside the grid such that the average is about 20 or more DSMC particle per cell in the domain. Flow simulations were carried out for each mesh at Mach 10 till convergence. Fig. 7 shows a Mach contour comparison for the same geometry for a denser flow condition of Kn=0.01 whereas Fig. 8 shows the Mach contour at Kn=0.25. It is observed that in the denser medium, the shock upfront is crisper whereas in the following case, the shock is very diffused which demonstrates the low density effects of the flow conditions.

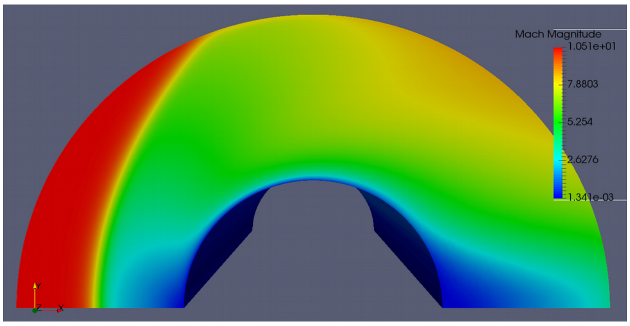


Fig. 7. Mach 10 contours for Kn=0.01

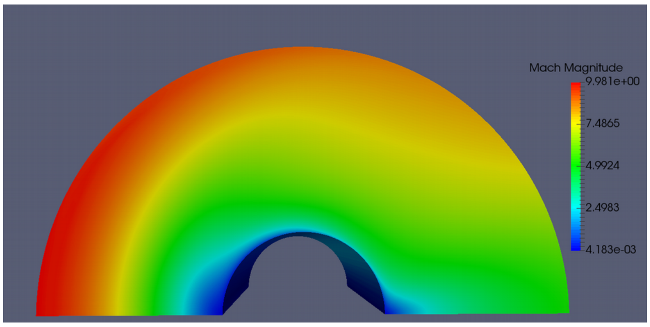


Fig. 8. Mach 10 contours for Kn=0.25

Results extracted from the simulations carried out have been compared with the literature data [Ref. 1]. Fig. 9 shows the comparison of temperature profile, 1D length ahead of the stagnation point. As obvious, the shock is diffused so the after-shock temperature profile is gentle. The temperature profile from each simulation compare well with the data from literature.

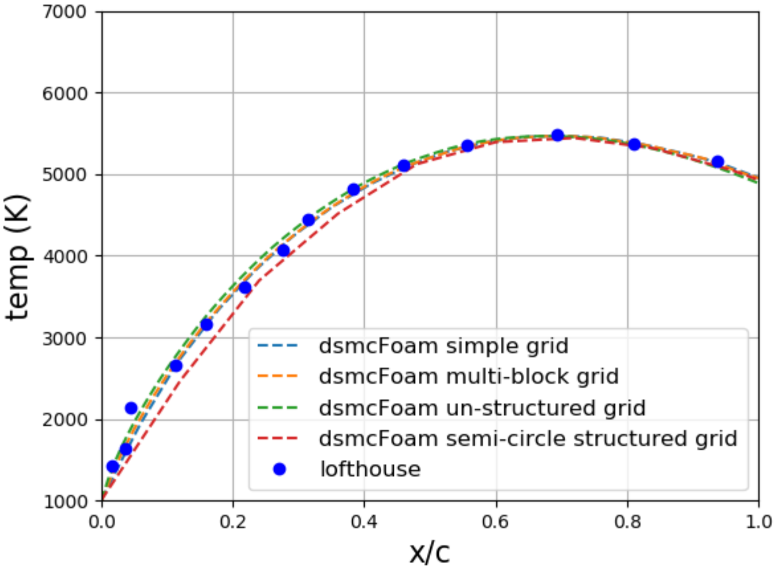


Fig. 9. Temperature distribution upstream of the stagnation point

Fig.10 shows the comparison of temperature along the wall for each simulation with the literature. Present simuations compare well with available data. Similarly, Fig. 11 shows the comparison of velocity along the wall with literature data and it is observed that the simulations compare well.

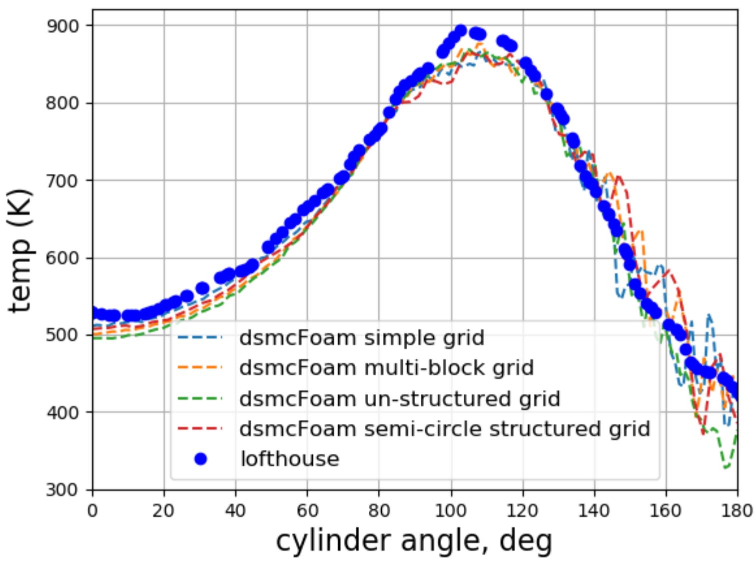


Fig. 10. Temperature distribution along the cylinder surface

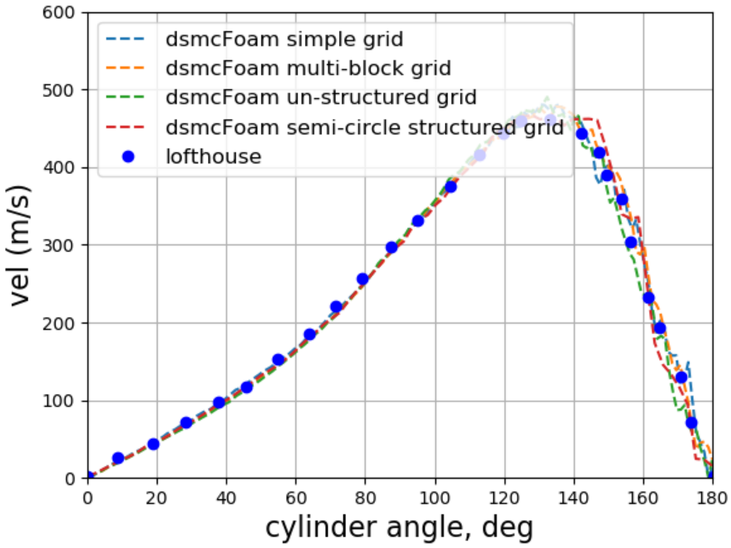


Fig. 11. Velocity distribution along the cylinder surface

6. Conclusions:

Solver dsmcFoam is validated for Kn=0.25 case of Mach 10 hypersonic flow over 2D cylinder. DSMC solvers are extensively used for rarefied flows which includes hypersonic flights in upper atmosphere, re-rentry physics, micro-channels, heat sinks etc. DsmcFoam solver is part of the OpenFoam package and the standard version is very useful to predict rarefied flows for high speed non-reacting cases. The usefulness increases as the flow becomes more rarefied.

References

[1] NONEQUILIBRIUM HYPERSONIC AEROTHERMODYNAMICS USING THE DIRECT SIMULATION MONTE-CARLO AND NAVIER-STOKES MODELS by Andrew J. Lofthouse, phD thesis (Aerospace Engineering) in The University of Michigan,2008