Numerical Modelling of Hypersonic Flow of Spacecrafts

M. Mačák¹, P. Vyroubal¹

¹Department of Electrical and Electronic Technology, Brno University of Technology, Brno 616 00, Czech Republic

E-mail: martin.macak@vut.cz, vyroubal@vut.cz

Abstract—Numerical simulations of rarefied gas flows are an important area in the design of aerospace vehicles. They can provide a suitable alternative to experiments, which can be often too complicated due to extreme conditions. Navier-Stokes equations, which describe continuum fluid flow, generally lose their applicability in low pressures. As a result, it is necessary to adjust these equations or use alternative models. This article studies the possibility of using a continuum numerical model for a simulation of a hypersonic flow of a reentry module. Based on the Knudsen number, it was shown that it is still possible to use Navier-Stokes equations either without any adjustments or with the addition of a slip flow boundary condition.

Keywords—rarefied flow, Knudsen number, Ansys Fluent, hypersonic flow, spacecraft

1. INTRODUCTION

Numerical modelling of continuum-rarefied gas flow is a research area of great interest as it can provide a convenient tool to describe and predict aerodynamic behavior of hypersonic vehicles. With rapid development of recyclable spacecrafts, the prediction of hypervelocity flow became very important. During reentry, flow around spacecraft can vary from free molecular in space up to hypersonic in the atmosphere [1, 2].

Such wide range of flows can be described by a kinetic theory governed by the Boltzmann equation [3]. However, it is known that solving the Boltzmann equation directly is very difficult and time consuming due to its complexity of the collision integral term. An alternative approach to solve the Boltzmann equation is the particle-based Direct Simulation Monte Carlo method, which tracks a large number of particles and their collisions are described in a stochastic way [4]. This method is widely used for the investigation of rarefied flows. However, as the number of molecules increases, this method starts to have unacceptable memory demands near continuum regimes, especially caused by large number of collisions [2, 4]. To estimate whether interactions on the molecular scale will affect transport of momentum and energy on the continuum scale, it is possible to calculate the ratio of the molecular mean free path to the characteristic length scale of the system. This ratio is called the Knudsen number (Kn). Different flow regimes can be identified based on this parameter. Kn<0.001 describes the continuum regime, 0.001<Kn<0.1 describes the slip flow regime, 0.1<Kn<10 describes the transition regime and Kn>10 describes the free molecular flow [5].

Well known Navier-Stokes equations are only focused on the description of a continuum flow and they show limitations in capturing nonequilibrium conditions in rarefied gas flows. There have been few attempts to increase the applicability of Navier-Stokes equations, namely implementing the effects of self-diffusion [6] and adjusting velocity boundary conditions to include the slip flow [4]. There have been attempts to provide unified models, which could be able to describe all flow regimes (Fokker-Planck [7] or Hybrid methods [8]). These models are usually very complex and cannot be easily transported into common Computational Fluid dynamics software, which makes their use somewhat problematic.

This article focuses on the applicability of common continuum solvers for the description of hypersonic flow. A simulation of a hypersonic flow around a reentry module [6] was carried out. Ansys Fluent software was used for the simulations and the performance of pressure-based and density-based solvers was compared. Additionally, the effects of slip flow were included in the study.

DOI: 10.13164/eeict.2022.238

238


2. NUMERICAL MODEL

Numerical model used in this work is based on the solution of Navier Stokes equations for compressible
gases. They consist of a set of nonlinear partial differential equations, which describe conservation of
mass, momentum, and energy (eqs. 1-3) [9]. The simulations were carried out in Ansys Fluent software,
which is a commercially available software. The software offers two numerical methods: pressure-based
and a density-based solver [10].

Equation of continuity [9]:
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  
(1)

Conservation of momentum [9]:
\[
\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p - \nabla \cdot \mathbf{t} + \mathbf{F}
\]  
(2)

Conservation of energy [9]:
\[
\rho \left( \frac{\partial E_{kt}}{\partial t} + \nabla \cdot (\mathbf{v}(E_{kt} + p)) \right) - \nabla \cdot (K \nabla T) = S
\]  
(3)

Where \( \rho \) denotes density, \( \mathbf{v} \) denotes velocity vector, \( p \) denotes pressure, \( \mathbf{t} \) denotes shear friction
tensor, \( \mathbf{F} \) denotes a general source term vector, \( E_{kt} \) denotes internal thermodynamic temperature, \( K \)
denotes thermal conductivity, \( T \) denotes thermodynamic temperature and \( S \) denotes a general energy
source term.

The relation between pressure and density was described by the ideal gas law [9]:
\[
\rho = \frac{p}{RT}
\]  
(4)

Where \( R \) denotes molar gas constant.

Additional parameter such as: viscosity, thermal conductivity and specific heat capacity were described

The Knudsen number was described in a local manner, which allows to study the rarefaction in the
whole domain [5]:
\[
Kn = \frac{\lambda}{\rho \nabla \rho}
\]  
(5)

Where \( \lambda \) denotes mean free molecular path.

For the slip flow regime, when \( Kn \) is between 0.001 and 0.1, a Maxwell velocity boundary condition
was used [5]:
\[
\mathbf{v}_s = \frac{2 - \sigma}{\sigma} \frac{\lambda}{n} \nabla \mathbf{v}
\]  
(6)

This equation can be also extended to include temperature effects.

Turbulence was described by a Reynolds Averaged Navier-Stokes SST k-\( \omega \) model and a laminar flow
model [12]. The simulations were carried out on a geometry of Apollo 6 capsule [6] shown in Fig. 1. The
gas velocity was set to 1500 m/s with a turbulent intensity of 1% to represent a hypersonic flow. It
was assumed that the capsule was at the altitude of approximately 25 km. The computational domain
consisted of 3.3 million fluid cells with a maximum size of 50 mm. A local sizing with a maximum size
of 15 mm was set to the area around the module. An inflation layer consisting of 10 cells was added to
properly describe the boundary layer. For solver settings, a coupled pressure-velocity with a pseudo-
transient approach was used. For spatial discretization a standard second order upwind method was
applied.
3. RESULTS

After initial simulations, an estimation of local Knudsen number described by eq. 5 was carried out. The visualization of Knudsen number is shown in Fig. 2. It is possible to see that overall, Knudsen number is within the range of continuum and slip flow regimes. Increased rarefaction can be seen at the edge of the shockwave. Also edges of the module seem to be critical areas, as the Knudsen number raised to values, which represent early stages of transition flow.

Density and velocity distributions are shown in Fig. 3. Overall, the results obtained with the density-based solver agree with the results from the pressure-based solver. The main difference lied in the compactness of the shock wave. The density-based solver was able to describe steep gradient more accurately than the pressure-based solver. A difference was also visible in the area behind the module. Density-based solver showed a minimum density of 0.13 g/m$^3$ and a maximum velocity of 1725 m/s. Pressure-based solver showed a minimum density of 0.25 g/m$^3$ and a maximum velocity of 1530 m/s.
Figure 3: Density and velocity distributions at a pressure of 2500 Pa and an inflow velocity of 1500 m/s. Density distribution for density-based solver (a). Density distribution for the pressure-based solver (b). Velocity distribution for density-based solver (c). Velocity distribution for the pressure-based solver.

4. CONCLUSION

The presented results show that it is possible to use continuum solvers even for hypersonic flows. Even though Navier-Stokes equations have a limited applicability in the rarefied region, they might still produce reasonable results. The most critical point is the calculation of the Knudsen number, as its value might show that even for high-speed rarefied flow, continuum assumption can be still applied. In most cases it was only necessary to apply the slip flow condition. From comparison of presented simulations, the choice of the solver did not influence the result significantly. Surprisingly, the pressure-based solver seemed to converge much faster than the density solver, which theoretically should be able to describe high velocity compressible flows much better. A reason for this might be that pressure-based solvers are much more widely used, so they are updated more often. This information might be especially important for decreasing the simulation time for large spacecrafts. Additionally, the density-based solver does not support the slip flow boundary condition so its applicability might be slightly limited. Overall, the use of classic continuum models is advantageous due their interconnection with another physics modules in commercially available softwares. This model can be additionally extended to include more detailed slip flow boundary conditions or even the effects of the self-diffusion.

ACKNOWLEDGMENT

This work was supported by the BUT specific research program (project No. FEKT-S-20-6206).
REFERENCES


