

## Effects of mesh resolution on hypersonic heating prediction

Quanhua Sun,<sup>a)</sup> Huiyu Zhu, Gang Wang, and Jing Fan

*Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China*

(Received 03 November 2010; accepted 26 January 2011; published online 10 March 2011)

**Abstract** Aeroheating prediction is a challenging and critical problem for the design and optimization of hypersonic vehicles. One challenge is that the solution of the Navier-Stokes equations strongly depends on the computational mesh. In this letter, the effect of mesh resolution on heat flux prediction is studied. It is found that mesh-independent solutions can be obtained using fine mesh, whose accuracy is confirmed by results from kinetic particle simulation. It is analyzed that mesh-induced numerical error comes mainly from the flux calculation in the boundary layer whereas the temperature gradient on the surface can be evaluated using a wall function. Numerical schemes having strong capability of boundary layer capture are therefore recommended for hypersonic heating prediction.

© 2011 The Chinese Society of Theoretical and Applied Mechanics. [doi:10.1063/2.1102201]

**Keywords** hypersonic flow, aeroheating, CFD, mesh resolution

Hypersonic flight vehicles typically refer to aircrafts that can fly in the atmosphere with more than five times the speed of sound. With the increase of the flight speed, vehicles could suffer terrible thermal loading due to shock heating and skin friction from the ambient atmosphere, which is based on the fact that the surface heat flux or aeroheating on an aircraft is proportional to the cube of the flight speed, whereas the drag force is proportional to the square of the speed. However, prediction of aeroheating is a hard task both experimentally and numerically. For instance, Bertin and Cummings listed the aeroheating prediction as one of the most challenging problems in computational fluid dynamics (CFD).<sup>1</sup>

In the literature, issues on aeroheating prediction have been widely studied,<sup>2</sup> including various physical models, numerical scheme, and mesh resolution. Among these issues, physical models are probably the most challenging one, which addresses the flow physics such as real gas effects, wall catalysis, radiation and cooling, transition and turbulence. Effects of physical models on aeroheating, however, are already qualitatively known thanks to tremendous efforts.<sup>1</sup> The mesh resolution, on the other hand, is purely numerical. It is well-known that numerical results of aeroheating depend strongly on the employed mesh.<sup>3-5</sup> In CFD practices, mesh independence study is usually performed by refining meshes until numerical results agree with experimental or flight data. The bad thing is that the solution becomes inaccurate when the mesh is further refined. In other words, an improper mesh could predict larger or smaller heating for a hypersonic flow. Therefore, analysis of detailed effect of mesh resolution on heat flux is beneficial to the design of effective mesh for hypersonic heating prediction.

To start the analysis, a simple case, supersonic flow over a front step, is first simulated. The governing equa-

tions are the Navier-Stokes equations. The simulated gas is argon, which is preferred for kinetic particle simulation that is adopted for the purpose of validation. The numerical scheme employed is the muscl type finite volume method with Roe's FDS scheme and the minmod limiter for the convective flux evaluation and the central difference scheme for the viscous flux calculation. Other specification of the case is: step height  $2h$ , wall temperature 1 000 K, free stream gas temperature 200 K, free stream Mach number 10, free stream Knudsen number ( $\lambda/h$ ) 0.001, free stream Reynolds number 13 000.

The mesh independence is studied by employing two sets of meshes. The first set is equally-spaced mesh whose mesh size is set at  $1/5$ ,  $1/10$ ,  $1/20$ ,  $1/40$ ,  $1/80$  of the semi-height of the step, respectively. The second set employs clustered mesh where the mesh near the surface is smoothly refined based on the equally-space mesh whose size is  $1/80h$ , so that the smallest mesh size in the normal direction becomes  $1/2$ ,  $1/4$ ,  $1/8$ ,  $1/16$ ,  $1/32$ ,  $1/64$ ,  $1/128$  of the base mesh size  $1/80h$ , respectively. Simulations show that the mesh-independent results can be obtained.

Figure 1 displays the temperature field  $(T - T_\infty)/(T_{\text{shock}} - T_\infty)$  of the flow where a bow shock can be easily observed. The gas temperature jumps across the shock and keeps increasing until it reaches the surface that is relatively cool. Simulations show that the flow field can be captured correctly even with a coarse mesh having the size of  $1/80h$ . The heat flux on the surface, however, depends strongly on the mesh resolution. Figure 2 shows the heat flux coefficient on surface located at  $0.5h$  height, where  $dx$  is the smallest mesh size. Clearly, the value of simulated heat flux increases quickly with mesh refinement at early stage. After it reaches its maximum, the heat flux converges gradually to a constant that is the mesh-independent value. For this case, a mesh size around  $1/1000$  of the semi step height is needed to reach the mesh-independent value. If experimental data are available and assumed to be

<sup>a)</sup>Corresponding author. Email: qsun@imech.ac.cn.

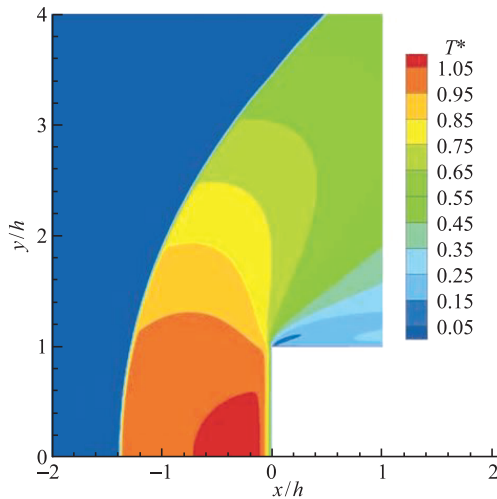


Fig. 1. Temperature field  $T^* = (T - T_\infty)/(T_{\text{shock}} - T_\infty)$  of argon flow over a front step where step height =  $2h$ ,  $T_w = 1\,000\text{ K}$ ,  $M_\infty = 10$ ,  $T_\infty = 200\text{ K}$ ,  $Kn_\infty (= \lambda/h) = 0.001$ ,  $Re_\infty = 13\,000$ .

consistent with the constant value, then a simulation could give good prediction even with a relatively coarse mesh (mesh size of  $1/40h$ ), but bad prediction will then show up with finer meshes, which explains the observation in some studies.<sup>5</sup>

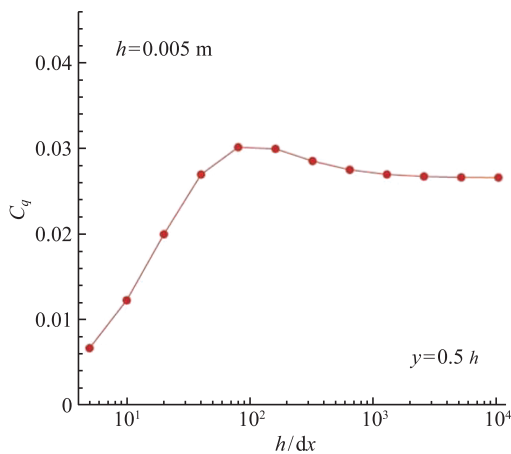


Fig. 2. Predicted heat flux coefficient obtained from a set of meshes. The simulation is for the flow over a front step case. The mesh is denoted by the smallest mesh size  $dx$ . The flux is on the surface located at  $0.5h$  height.

It is necessary to validate whether the converged heat flux is the physical solution of the flow. A totally different numerical technique, the direct simulation Monte Carlo (DSMC) method,<sup>6</sup> is employed to simulate the same flow. The DSMC method is a particle approach that simulates a large number of microscopic molecules by tracking their motions and collisions. It is numerically expensive to simulate the current flow where the global Knudsen number is only 0.001. Figure 3 shows the heat flux coefficient obtained from both

CFD and DSMC simulations where the CFD solution is the mesh-independent data. The overall agreement between the two solutions is very good. Difference is observed only in a very small region near the step corner, which can be attributed to two factors. One is that it is hard for CFD to capture the flow gradients near the corner. Another is that the local Knudsen number near the corner is large and the Navier-Stokes equations become invalid in this small region. Therefore, it can be concluded that the mesh-independent result is the physical solution of the Navier-Stokes equations.

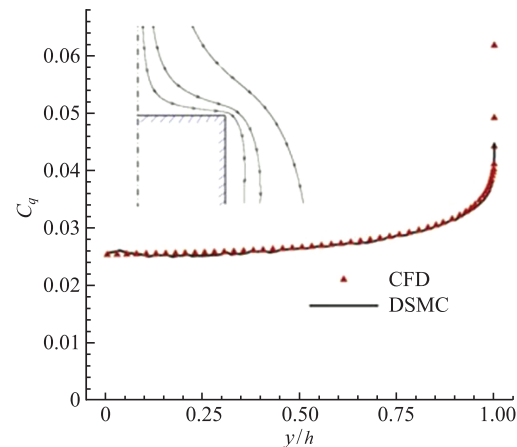


Fig. 3. Comparison of heat flux coefficient along the front step obtained using CFD and DSMC simulations for the flow over a front step case.

The effect in Fig. 2 of mesh size on the heating prediction is observed in many flow simulations. Figure 4 presents CFD results for several problems including flow over front-step, cylinder, and sphere under different flow conditions. The simulated gas includes argon and air. In the plot, the mesh size is represented by the surface grid Reynolds number ( $Re_g = \rho_c a_c \Delta y / \mu_c$ ) where the length scale is the mesh size in the normal direction and all values are at the near surface cell. Clearly, mesh-independent results are obtained for all the cases when  $Re_g$  is less than 5, which agrees well with findings in the Ref. 1. Of course, many factors will affect the behavior of mesh dependence for aeroheating. For instance, different numerical scheme will have slight different converging process of mesh resolution as shown in Fig. 5 where the flow over a cylinder is simulated.

It should be mentioned that the surface grid Reynolds number is only one measure of the mesh size. A very important fact is that  $Re_g$  specifies only the size of the surface mesh. The size of other part of mesh will also affect heat flux evaluation, which is often overlooked by some researchers. In practice, larger cells are used in domain away from the aircraft surface in order to save computational cost. Then the flow may not be resolved locally with the large cells. In addition, the non-uniform mesh will produce additional numerical error. This may be the reason why meshes having very small surface cells may predict bad results sometime.

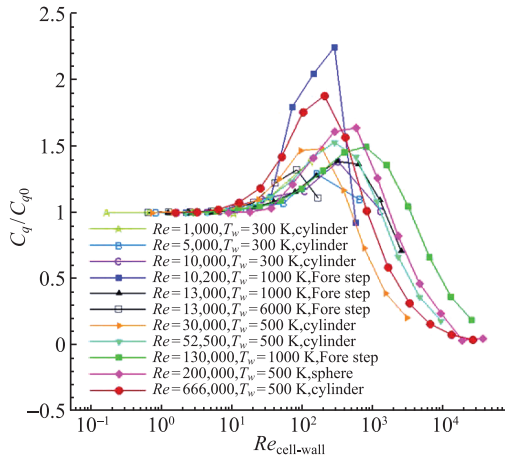


Fig. 4. Effects of surface grid Reynolds number on heat flux predicted at the stagnant point where the heat flux is normalized by the grid-independent value for several flow cases.

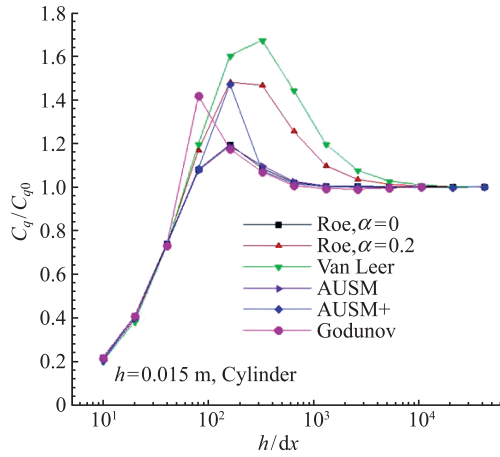


Fig. 5. Heat flux coefficient predicted by typical numerical schemes for flow over a cylinder. Schemes employed are: Roe's FDS scheme, Roe's scheme with entropy fix, van Leer's FVS scheme, AUSM scheme, AUSM+ scheme, and exact Riemann (Godunov) solver.

Our simulations have indicated that the size ratio between neighboring cells should be less than 1.2 to avoid obvious numerical error.

From numerical simulation, it is clear that the mesh size should be very small near the surface to predict correctly the heat flux. The main reason may be that the gradients of flow properties such as temperature are very large near the surface or in the boundary layer. Figure 6 shows a typical temperature profile in the boundary layer where results from several meshes are plotted. It is found that the temperature distribution is nonlinear, thus numerical error could occur during the space discretization. Flux evaluation is then the main source for the error, especially the evaluation of the temperature gradient on the surface.

In fact, the temperature distribution near the sur-

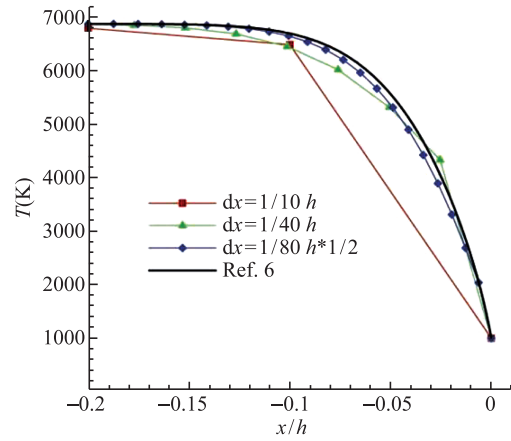


Fig. 6. Typical temperature profile along the normal direction in the boundary layer for the case of flow over a front step.

face can be estimated using wall function. Wall function has been employed for bounded turbulent flow simulations. For hypersonic laminar flow, we can also derive wall function for temperature and velocity. In hypersonic boundary layer, quasi one dimensional flow can be assumed. Then the temperature along the surface normal direction can be approximated as

$$T(y) = T_w (cy + 1)^{\frac{1}{\omega+1}}, \quad (1)$$

when the viscosity employs the VHS model<sup>6</sup> that  $\mu \propto T^\omega$ , where  $c = [(T_c/T_w)^{\omega+1} - 1]/y_c$ . The surface heat transfer is calculated as

$$q = \mu_w \cdot \frac{1}{\omega + 1} \frac{(T_c/T_w)^{\omega+1} - 1}{T_c/T_w - 1} \cdot \frac{T_c - T_w}{y_c}. \quad (2)$$

Expression (1) is quite accurate for hypersonic flows. Figure 7 shows the temperature gradients evaluated at different locations for the front-step case. The wall function can give good results even when the mesh is large, which illustrates the benefit of the wall function. However, the mesh independence of heat flux evaluation is only slightly improved using the wall function, which indicates that the numerical error comes from the flux evaluation in the boundary layer.

To verify the error source, the flux on a mesh face is calculated using different mesh sizes based on analytic expressions of flow properties within the boundary layer. The temperature is approximated using Eq. (1). The velocity profile can also be derived. For instance, the tangent velocity component in the boundary layer can be approximated as

$$u = \frac{a}{c} (\omega + 1) \left[ \frac{T}{T_w} - 1 \right] + \frac{b}{c} (\omega + 1) \frac{T}{T_w} \left[ \frac{\left( \frac{T}{T_w} \right)^{\omega+1}}{\frac{\omega+2}{\omega+1} c} - y \right] - \frac{b}{(\omega + 2) c^2}, \quad (3)$$

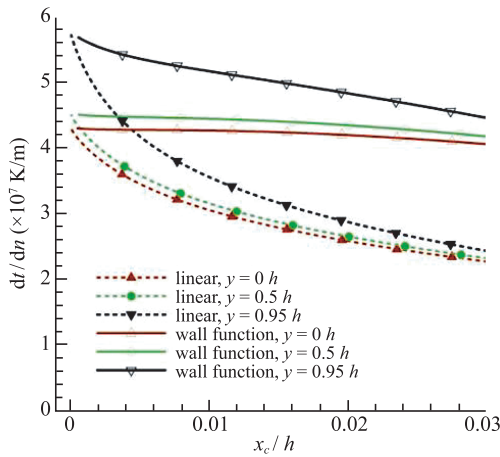


Fig. 7. Temperature gradients evaluated at different surface locations using linear expression and wall function for the case of flow over a front step.

where  $a = (\partial u / \partial y)_0$ ,  $b = -dp / \mu_w dx$ ,  $c = \frac{1}{y_c} \left[ \left( \frac{T_c}{T_w} \right)^{\omega+1} - 1 \right]$ .

The pressure is assumed constant along the normal direction, thus the density distribution is also available. With the known flow properties, the flux at a location of  $1/50 h$  away from the surface is calculated using the corresponding discrete values. It is found that the calculated fluxes have good accuracy when the mesh size is very small. When the mesh size increases, the flux error increases and depends on the numerical scheme. Figure 8 shows the energy flux calculated with different mesh size using typical schemes involving the minmod limiter. It is noticed that the error appears when the mesh size is about 5 microns ( $h/dx=1000$ ) for the Roe scheme with entropy fix (Roe2), which agrees with the results in Fig. 1. Therefore, the flux evaluation is the main source for numerical error when the mesh size is large.

It can be concluded that hypersonic heating can be correctly predicted as long as the governing equations are valid. It turns out that the mesh resolution plays a very important role in numerical simulations. Very fine

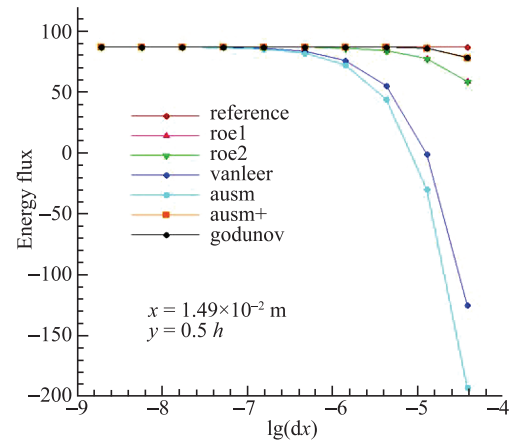


Fig. 8. Energy flux ( $\text{kg}^2/\text{s}^3$ ) through a mesh face calculated using numerical schemes when the discrete values on the mesh are obtained from wall functions for the case of flow over a front step. The mesh face is located at  $x = 0.0149 \text{ m}$  and  $y = 0.5 h$  where the front step is at  $x = 0.015 \text{ m}$  and  $h = 0.005 \text{ m}$ .

mesh is usually required near the surface of hypersonic vehicles in order to resolve the nonlinear distribution of flow properties. Numerical schemes having strong capability in capturing boundary layer are recommended for hypersonic heating prediction.

*This work was supported by the National Natural Science Foundation of China (50836007, 90816012, 10621202.)*

1. J. J. Bertin, and R. M. Cummings, Annual Review of Fluid Mechanics **38**, 129 (2006).
2. J. M. A. Longo, Aerospace Sciences and Technology **7**, 429 (2003).
3. K. A. Hoffmann, AIAA-1991-467 (1991).
4. P. Periklis, V. Ethiraj, P. Dinesh, P. L. Mark, and O. Dave, Applied Mathematical Modelling **23**, 705 (1999).
5. C. Yan, J. Yu, and J. Li, Acta Aerodynamica Sinica **24**, 125 (2006).
6. G. A. Bird, *Molecular Gas Dynamics and the Direct Simulation of Gas Flow* (Clarendon Press, 1994).