THE PHYSICAL AND COMPUTATIONAL MODELLING OF DILUTE GAS-PARTICLE FLOWS

John Young

Hopkinson and ICI Professor of Applied Thermodynamics
Cambridge University Engineering Department

Gas-particle flows are called dilute when the particle motion is driven by the gas but not *vice versa*, and the particles do not interact with each other. Applications abound in environmental engineering (transport and fallout of pollutants), mechanical engineering (deposition in turbo-machinery and heat transfer equipment), chemical engineering (transport of chemically created aerosols), nuclear engineering (deposition of radioactive aerosols), medical engineering (drug delivery to the lungs) and agricultural engineering (crop spraying).

The apparent simplicity of this two-phase flow belies its physical and computational difficulties. Both Lagrangian and Eulerian methods for solving the particle equations have their place. When turbulence is unimportant, the deterministic Lagrangian particle tracking method has long been favoured for calculating the particle velocity field. The main drawback has been the difficulty of obtaining the particle concentration field other than by an expensive ‘sledgehammer’ calculation. An elegant and much cheaper method to overcome this problem will be described. Two-fluid Eulerian methods also have their difficulties including the numerical capture of particle-free zones, the accurate representation of concentration discontinuities, and the correct application of solid surface boundary conditions. Examples of a scheme which overcomes some of these problems will be presented.

When the flow is turbulent, the computational difficulties are compounded by incomplete physical understanding. Work over the past two decades has disclosed the crucial rôle of ‘turbophoresis’ in transporting particles to solid surfaces. This mechanism dominates in the buffer layer close to the surface and can be responsible for increasing the particle deposition rate by several orders of magnitude. The modelling of turbophoresis will be discussed and its effect in channel and boundary layer flows illustrated. Finally, some calculations illustrating the interaction of all these mechanisms near solid surfaces will be presented.