Numerical Simulations of Flow around Bluff Bodies ANGEL BETHANCOURT, KUNIO KUWAHARA, Institute of Computational Fluid Dynamics, SATOKO KOMURASAKI, Nihon University — Simulations of flow around bluff bodies are carried out using Cartesian coordinates. A binary function is used to mark the position of the body in the grid system; therefore, a close-up near the boundary of the body will resemble a staircase configuration. Presently, a multi-directional finite ifference method is incorporated into the flow solver, it helps smooth the solution near the boundary, but still for applications at high Reynolds number resolution is still insufficient. Separation for test cases is still greater that expected due to the numerical diffusion caused by the roughness on the boundary. In order to control the behavior of the flow, we introduce a negative viscosity coefficient along surface of the body to compensate for the numerical diffusion. Its effects are confined to the points next to the surface body. It acts as a boundary layer velocity correction. Simulations show that fine-tuning this parameter can control the size of the wake behind bluff bodies.

Kunio Kuwahara
Institute of Computational Fluid Dynamics