

Abstract Submitted
for the DFD09 Meeting of
The American Physical Society

Numerical Investigation of Base Drag for an Axisymmetric Underwater Vehicle with Bluff Afterbody MIN-JAE KIM, KURNCHUL LEE, Agency for Defense Development(ADD) — The objective of this study is to predict the drag of an axisymmetric underwater vehicle with bluff afterbody. FLUENT, commercial CFD code, is used to simulate high Reynolds number turbulent flows around the vehicle. The computed drag coefficients are compared to available experimental data at various Reynolds numbers. Four widely used two-equation turbulence models are investigated to evaluate their performance of predicting the anisotropic turbulence in a recirculating flow region, which is caused by flow separation arising from the base of the vehicle. The simulations with Realizable $k-\varepsilon$ and $k-\omega$ SST turbulence models predict the anisotropic turbulent flows comparatively well and the drag prediction results with those models show good agreements with the experimental data.

Min-Jae Kim
Agency for Defense Development(ADD)

Date submitted: 04 Aug 2009

Electronic form version 1.4