E-29

A RANS solver using a 3D unstructured FVM procedure

○Hao ZHANG, Hajime NAKAGAWA, Taisuke ISHIGAKI and Yasunori MUTO

1. Introduction

Turbulent flows in complex geometries are usually encountered in engineering practices, which pose new challenges and special requirements on the RANS (Reynolds-averaged Navier-Stokes equation) solver for the numerical simulation. An unstructured mesh is able to offer significant flexibility in treating sophisticated boundaries and provide a convenient means of pursing mesh adaptation. Which, combining with an FVM (Finite Volume Method) procedure, can greatly improve the applicability of the turbulence models.

This study summarizes the key problems during the construction of a RANS solver using a 3D unstructured FVM procedure and demonstrates its capabilities of predicting some selected flow phenomena in laboratory flumes.

2. Model description

The RANS solver starts from the continuity equation and the momentum equations. The models from the k-ε family are used for the turbulence closure with the wall function approach.

The governing equations are discretized by a cell-centered FVM on a collocated unstructured mesh. A strict but effective data structure is introduced to store the CV (Control Volume) information. The power law scheme is adopted for the spatial discretization, and the implicit Crank-Nicolson scheme is used for the temporal integral. The surface fluxes are calculated from the Rhie-Chow interpolation to avoid the checkerboard phenomenon. The final algebraic equation system is solved by a preconditioned GMRES (Generalized Minimal Residual Algorithm) solver incorporated with an ILUTP (Incomplete LU Factorization with Threshold and Pivoting) preconditioner. The solution procedure follows the widely used SIMPLE method (Semi-Implicit Method for Pressure-Linked Equations).

3. Model verification

The proposed model has been employed to predict three kinds of flows with different meshing strategies. These include the flow in a rectangular channel, the flow in a square embayment and the flow around a series of spur dykes with local scour holes. Some computational result is given in this page.

4. Conclusions and Future researches

An unstructured mesh based RANS solver has been presented with an FVM procedure. The solver is able to reproduce some selected flows in different domains with a reasonable accuracy and manifests a potential to apply to the actual river conditions.

The near wall area is not well predicted in the test cases. The turbulence kinetic energy in the boundary mesh is found to be slightly over-estimated. Further improvement is needed. The variation of the free surface has been neglected for the computational convenience. This simplicity may lead to a questionable result in the case that the free surface plays an important role.

Unit: Y, Z: cm; velocity:cm/s

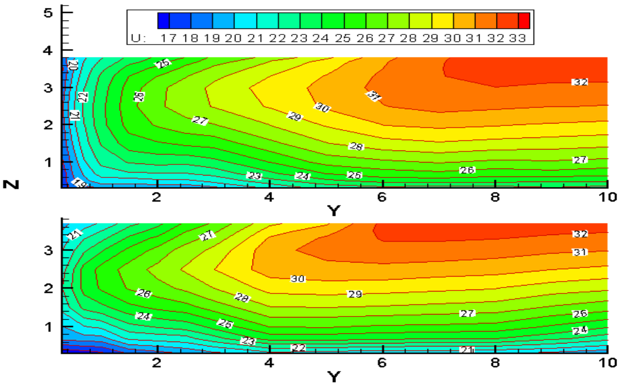

Fig.1 Longitudinal velocity in a rectangular channel (Top: Experiment; Bottom: Computation)

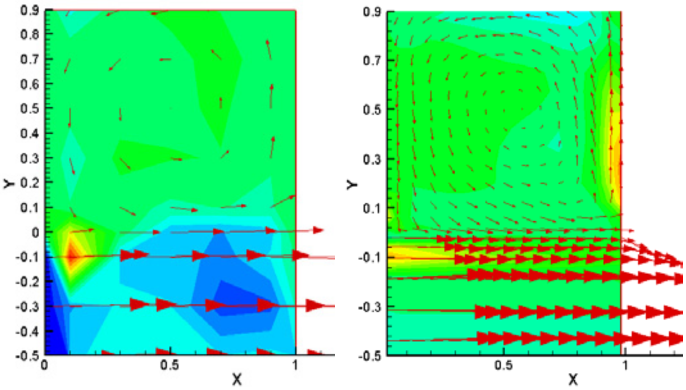

Fig.2 Horizontal vortex in a square embayment (Left: Experiment; Right: Computation)

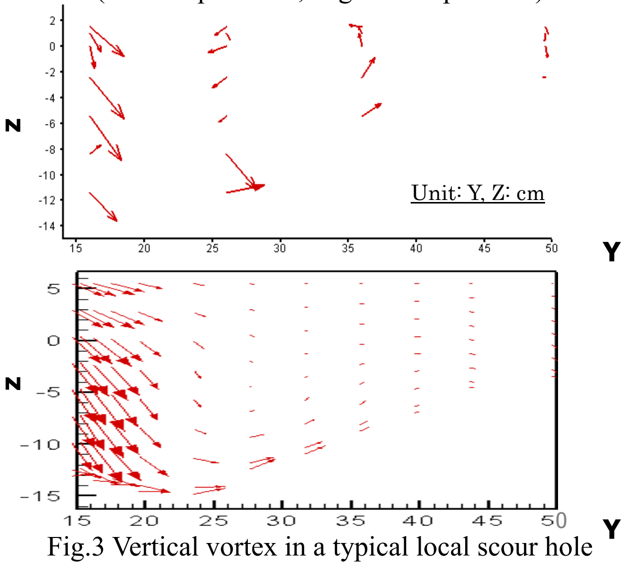

Fig.3 Vertical vortex in a typical local scour hole (Top: Experiment; Bottom: Computation)