ARCHIVES OF CIVIL AND MECHANICAL ENGINEERING

Vol. VIII 2008 No. 3

Numerical modelling of flow through moving water-control gates by vortex method. Part I – problem formulation

S. W. KOSTECKI

Wrocław University of Technology, Wybrzeże Wyspiańskiego 25, 50-370 Wrocław, Poland

A vortex method for simulating a flat flow within moving complex boundaries is presented. Thanks to the use of Lagrange variables (the trajectories of vortex particles) to determine the evolution of vorticity and velocity fields the method offers the possibility of modelling high Reynolds number flows. A procedure for formulating boundary conditions for flows confined by a moving impervious boundary, consisting in the superposition of three velocity fields: the first originating from the vortex particles, the second being a potential field satisfying the no-through-flow boundary condition and the third one resulting from the fact that the vortex sheet is modelled along the impervious boundaries and satisfying the no-slip-flow condition, is described. The original derivation of a formula for vortex sheet intensity, based on the single layer potential theory and leading to the formulation a second-kind Fredholm equation for vortex sheet intensity, is presented. This paper is the first part of a work covering the theoretical foundations and general description of the vortex method algorithms and boundary conditions. An example illustrating the computation of the vorticity and velocity fields of the flow through a moving hydraulic gate will be provided in the second part of the work.

Keywords: *hydraulic gates, turbulence flow, vorticity, vortex method, moving boundary*

1. Introduction

One of the principal hydraulic engineering problems is the calculation of the velocity of the water flow through moving hydraulic gates. As the hydraulic gate moves, conditions conducive to boundary layer separation and considerable pressure fluctuations arise. The flow around the gate may become very unstable, producing a highly variable load and so vibrations.

Similar problems as above also occur for the flow through hydraulic machines (turbines), in off-shore drilling (e.g. the water flow around riser tubes transporting oil from the seabed to the surface) and in aerodynamics (during the turbulent flow over the wings and fuselage of airplanes and in the wind engineering of bridges and high buildings).

The flow around moving hydraulic gates has been the subject of several experimental studies in recent decades. Their primary aim was to identify the mechanism causing gate vibrations. Thang [15] and Billeter [4] carried out model studies of the flow under a vibrating lift gate and found that the instability of the single shear layer separating from the gate lip results in sharp fluctuations in the pressure acting on the gate's bottom and rear edge. The fluctuations are the most common cause of gate vibrations. Aydin et al. [1] carried out studies on a hydraulic model of the gate as it was moving in a pressure conduit to determine the hydrodynamic force averaged over time.

The solution of the above problem is beset with numerous difficulties due to the high Reynolds number (usually exceeding 10^5) and the geometric complexity of the boundary, which may move or deform under the action of the liquid and this motion, in turn, affects the flow. For this reason, an analytical solution cannot be obtained, except for very simple cases of flow.

This paper presents a numerical method of computing the velocity and vorticity within a flow region bounded by a moving boundary with boundary velocity-liquid velocity consistency conditions imposed on it. In order to determine the dynamics of the liquid in such a region one must solve the Navier-Stokes equation (fully combined with the boundary elastic deformation equation) in three-dimensional space. The solution of the problem simplified to a flat flow with a moving inelastic boundary, constituting the first significant step towards the full problem solution, is derived here. The method will be extended to deformable boundary and three-dimensional problems in later work.

The discrete vortex method in a stochastic formulation, which is effective for high turbulence flows, is applied to solve the flow problem.

The vortex method formulation is based on vorticity field discretization into a finite number of particles (vorticity carriers) and the description of their motion in the Lagrangian coordinates. The problem solution consists in tracking the evolution of the particles (their position and intensity), resulting from advection, and simulating vorticity diffusion by the stochastic (random walk) method. The flow velocity field is explicitly recovered from the vorticity distribution by means of the Biot-Savart integral. The boundary conditions for the flow problem, including the moving boundary condition, stem from the friction forces produced by viscosity and are expressed by the equality between the velocity of the moving boundary and that of the liquid contiguous to it.

2. Governing equations

The flow of an incompressible viscous liquid in region D confined by boundary S is described by the Navier-Stokes Equation (1), mass conservation Equation (2) Dirichlet boundary condition (3) and initial condition (4):

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot (\nabla \mathbf{u}) = -\nabla P + \nu \nabla^2 \mathbf{u}, \quad \mathbf{x} = [x_1, x_2, x_3] \in D,\ t > 0, \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0, \tag{2}$$

$$\mathbf{u} = \mathbf{U}_b, \ \mathbf{x} \in S, \tag{3}$$