9th International Congress on Civil Engineering, May 8-10, 2012
Isfahan University of Technology (IUT), Isfahan, Iran

Transient analysis of sea water intake using CFD model

Mehdi Assadi Niazi [1], Sadegh Barzegar [2], Alireza Elhami Amiri[3], Ebrahim Mokallaf[4]

1- MS of water structures Engineering, Ardabil Regional Water Company.
2- MS of Mechanics Engineering, Iran marine Industrial Company.
3- PHD of mechanics, Sharif University.
4- MS of water engineering Ardabil regional water company.

M.asadiniazi@gmail.com
s.barzegar@sadragroup.com
elhamiam@mech.sharif.edu
emokallaf@yahoo.com

Abstract

Computational Fluid Dynamics is a mathematical tool, used to simulate fluid flow problems. The simulation results, are then used to consider kinematic and thermodynamics of flow particles within a specified geometry using Euler and Navier-Stoks solvers to simulate the most complex geometries in two and three dimensions. Sea water intake is one of complex geometry. In order to understand flow behavior incorporating control volume definition, it is essential to study all the forces (both internal and external).The transient analysis of the seawater intake in case of a power failure of Seawater pumps has been performed in this model. In this study, we have used Finite Volume Method to solve Navier-Stokes equations along with standard K-ε turbulent equations as well as Volume of Fluid (VOF) equations which are governing the free surface fluid motion.

Keywords: Transient, CFD(computational fluid dynamics)Sea water,intake, Navier-Stokes flow.

1. INTRODUCTION

Computational Fluid Dynamics, which referred to more often as CFD, is a mathematical tool, used to simulate a fluid flow problems. The simulation results, are then used to consider kinematic and thermodynamics of flow particles within a specified geometry. This would give useful hints to optimize the existing design and relevant facilities. As it is obvious, all flow fields can be described by appropriate partial differential equations which are referred as government equations. In fact, CFD is an approach to solve these equations numerically and consequently obtain flow variables. The number of unknown variables e.g. velocity, pressure, temperature … are required to completely define a flow fields variables case by case. Nevertheless, it can be shown mathematically that a unique numerical solution exists for most cases even for more complicated flows such as turbulent and free surface motion flows. However, the number of complicated problems, which can be calculated and evaluated by CFD, has been increasing by developing more advanced numerical techniques. The main matter which limits the engineering application of CFD is difficulty in describing complex geometries. In recent years, many researchers have been conducted to overcome this shortcoming. Developing and using of unstructured grids is the most appropriate one. This gives more ability to Euler and Navier-Stoks (flow governing equations) solvers to simulate the most complex geometries in two and three dimensions.

generally, the performance of generating unstructured grids is so high and it does not require the user skills. Figure 1 shows the complex geometry of Bandar Abbas sea water intake facility. For such a complex geometry, the flow field can be claimed to be known whenever velocity and some static quantities are determined in every point of the model at every time. The number of static quantities, used for particular problems, depends on the nature of flow. Incompressible flows may be described by only one quantity such as pressure. However, for compressible flows one additional quantity e.g. density or temperature is required to define the state of fluid in every point of flow domain. This is not the case for the current project where the fluid (Sea Water) is definitely incompressible.

In order to understand flow behavior incorporating control volume definition, it is essential to study all the forces (both internal and external) that can be applied by flow on it. Conservative laws will describe the interaction between internal and external forces with fluid and flow characteristic. In fact, conservative governing equations, define that all flow properties such as energy and mass will remain constant under the action of internal and boundary forces.