Transactions on Ecology and the Environment vol 19, © 1998 WIT Press, www.witpress.com, ISSN 1743-3541

CFD – a useful tool in spillway capacity determination

James Yang & Bengt Hemström
Vattenfall Utveckling AB, S-814 26 Älvkarleby, Sweden
Email: james.yang@utveckling.vattenfall.se

Abstract

Physical model tests are traditionally used in engineering practice to study hydraulic problems related to dam and spillway structures. Thanks to its rapid development during the past few years, numerical technique has been adopted to replace model tests in solving some of the hydraulic problems. In a R&D project, the CFD program STAR-CD is employed to simulate the discharge capacity of a standard WES spillway. Experiments are made separately to verify the numerical results. The agreement between the numerical simulations and experiments proves to be generally good. STAR-CD can reach the same order of accuracy as a model test usually can. With the technique Embedded Mesh Refinement, the CPU time can be significantly shortened. The study has shown that CFD has potential and is a useful tool in solving such problems as flow through spillways.

1 Introduction

The Swedish hydropower sector has abandoned the frequency analysis approach due to its limitations and introduced in 1990 new guidelines for determination of design floods (Flödeskommittén[1]). These guidelines have led to the fact that the spillway discharge capacity in many of the existing Swedish dams needs to be increased. One example is the Bergeforsen power station on the river Indalsälven, where the spillway capacity has to be extended from 2300 to 3400 m^3/s.

Determination of spillway flow capacity is an important issue concerning dam safety. This concerns not only the existing spillways to be upgraded, but also new spillways to be constructed. Physical model tests are traditionally used to investigate hydraulic problems associated with dams and spillways. Thanks to their successful development during the past 10 - 15 years, CFD calculations have provided an alternative to model tests. Their applications are favoured as the time taken to perform a study is usually shorter and the costs are lower. However, it should be realised that differences exist between different CFD codes. It is therefore decisive to use reliable codes as bases for decision-making. To this end, evaluation should be made of the program to be used for tackling practical problems.

The main purpose of the project is to evaluate the suitability of the CFD code STAR-CD™ for modelling flow discharge through an ungated spillway. The evaluation is made in terms of grid generation, CPU time, accuracy of results, etc. A number of issues concerning the use of CFD are discussed, and simple modelling guidelines are proposed. The verification of the numerical results is done against experiments.

2 Problem Formulation

The layout of the ungated spillway investigated in this study is illustrated in Figure 1. The spillway has a width of 0.30 m and is placed centrally in the flume, whose width is 1.20 m. Its crest is a standard WES spillway profile defined at a design head of $H_d = 0.20$ m and is connected to the downstream bed with a 45° bucket having a radius of 0.20 m. The spillway threshold is 0.30 m above the bed of the flume. The pier of the spillway is sharp-edged and sticks out 5 cm upstream from the vertical front of the spillway, thus resulting in a sharp lateral contraction.

Numerical modelling is made at Vattenfall Utveckling AB with STAR-CD 3.0 (Yang[2]). Three flow cases are examined, i.e. low flow 0.019 m^3/s, design flow 0.050 m^3/s and high flow 0.085 m^3/s. The flow regime at the bucket location is supercritical and not affected by the tailwater.

The experiments independently conducted at the Royal Institute of Technology (KTH) (Dargahi[3]) are used to validate the reliability of the CFD modelling. The test flume is 38.5 m long. The spillway, made of

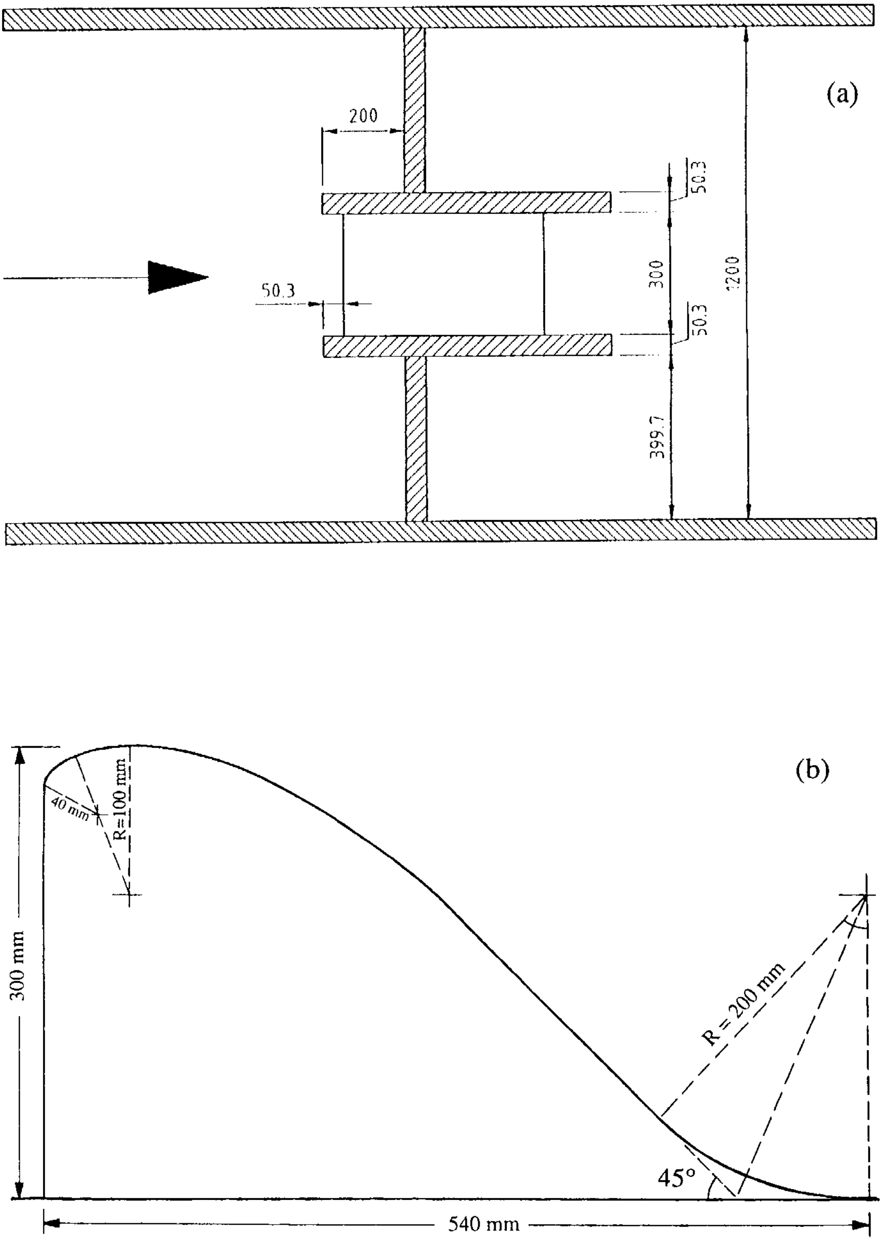

Figure 1: Layout of the ungated spillway modelled with STAR-CD and experiments. (a) plan view, (b) standard WES spillway profile

stainless steel, is located some 28 m from the upstream inlet, so that the flow approaching the spillway is fully developed. The water level is measured with point gauge and the flow rate with a combination of magnetic flow meter and rectangular weir.

3 STAR-CD

STAR-CD is a general-purpose CFD code developed at Computational Dynamics Ltd, London. By general-purpose it is meant that it is similar to other CFD programs like e.g. CFX, FLUENT and PHOENICS and can be used to simulate flow, heat transfer as well as chemical reactions.

STAR-CD is a finite-volume program based on unstructured numerical grid. The grid can be flexibly generated and refined depending on the problem to be solved and the purpose of simulation. Most of the commonly used turbulence models in industrial applications are available in the code (Computational Dynamics Ltd[4]).

In STAR-CD flows with free water surface are modelled with the "Free-Surface Model", which is based on the concept Volume of Fluid (VOF). The model is in nature a two-phase flow algorithm in a sense that both water and air flows need to be simulated when calculating e.g. flow through a spillway.

4 Model Set-up

As both the spillway and the flow are symmetrical about the flume centre plane, only half of the spillway is modelled. The model starts 5 m upstream from the vertical edge of the spillway and ends where the cylindrical bucket begins.

The grid is generated in two steps. First, a coarse grid in created (Figure 2a). A simulation is made with this grid and an approximate solution is thus obtained. The grid is then refined based on the location of the water surface. The refinement is made in those cells in the vicinity of the water surface and above the spillway (Figure 2b). How many new cells one wishes to have in a cell of the coarse mesh depends on, among other

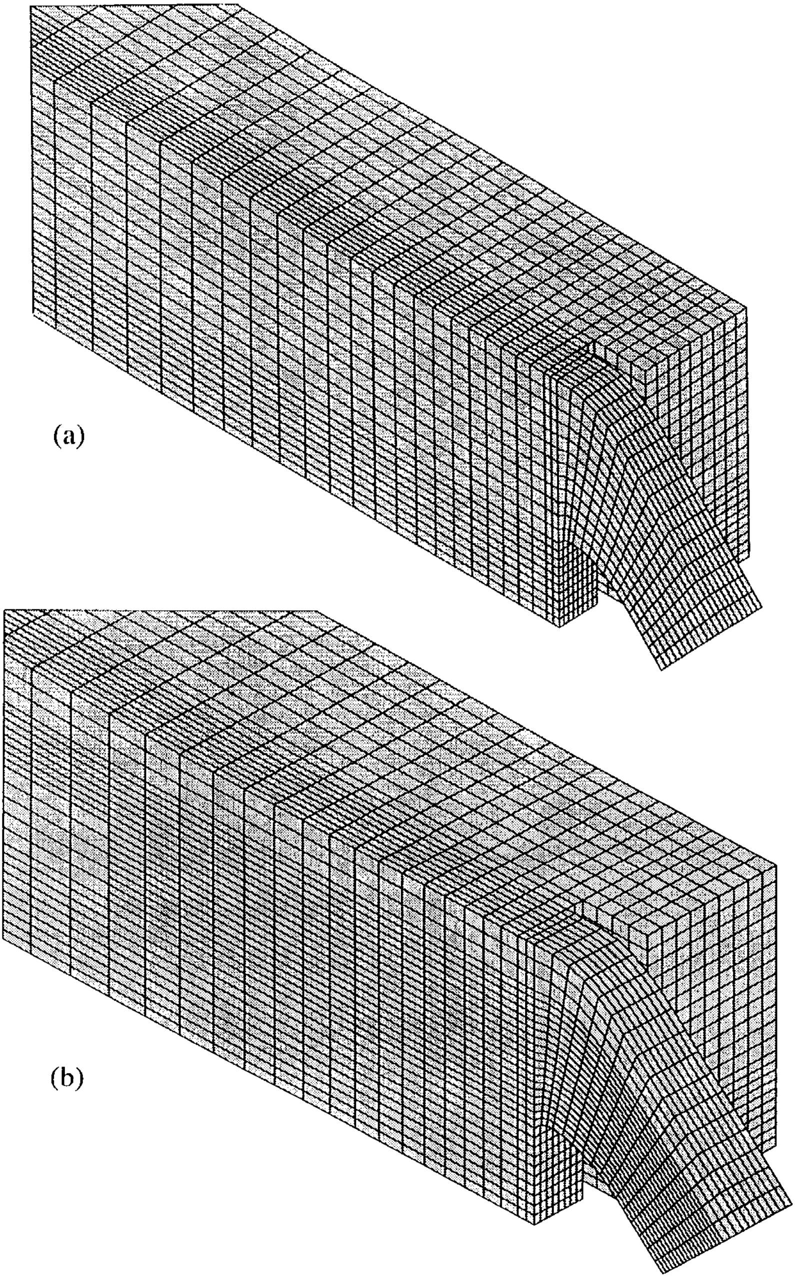

Figure 2: Grid generated with embedded mesh refinement. (a) coarse mesh, (b) refined mesh based on the coarse mesh solution

things, the modelling purpose. The simulation with the fine grid starts from the coarse grid solution.

The initial coarse mesh has about 17 300 hexahedral cells and this mesh is common for all flow cases. How large domain needs to be refined depends mainly on the magnitude of the flow. The mesh after refinement consists of 20 500, 21 400 and 22 250 cells for the low, design and high flow cases, respectively.

The way the final mesh is generated is called "Embedded Mesh Refinement" in STAR-CD and it is a practical method that can be used in many applications. The coarse mesh is refined only in those areas where higher space resolution is required and the majority of the cells with air remain coarse.

The k-ε turbulence model is used, together with the default law-of-the-wall for turbulence. Starting from an initial flow condition, the calculations are performed time dependent. The simulation for a flow case is considered converged when stable flow conditions have been achieved in time in the model.

All the simulations are made using one processor on a Silicon Graphics Power Challenge /L 4xR10000 workstation.

5 Results and Discussions

5.1 Discharge capacity

Examples of the results from STAR-CD are given in Figure 3. Figure 3a shows the water-surface profile through the spillway for the high-flow case (0.085 m^3/s). Figure 3b illustrates the velocity vector on the mesh surfaces for both the water and the air for and this corresponds to the design-flow case (0.050 m^3/s).

The water head, H (m), between the spillway threshold and the undisturbed water surface upstream the spillway is an important measure of the flow capacity, Q (m^3/s), of a spillway, which is usually written as

$$Q = C \cdot B \cdot H^{3/2}$$

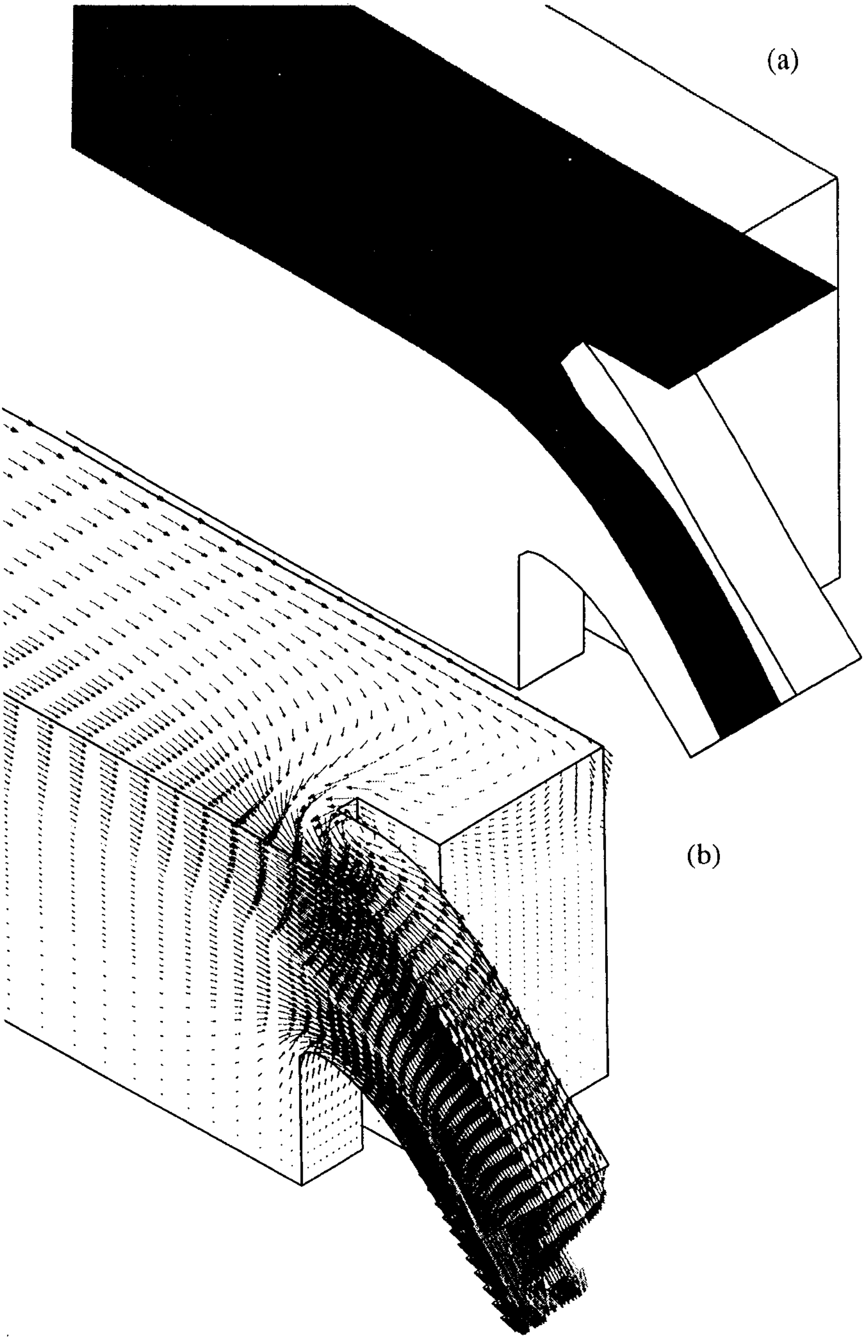

Figure 3: (a) Water-surface profile through spillway for high flow case, (b) Velocity vector on the mesh surface for design flow case.

in which B = spillway width (m) and C = discharge coefficient ($m^{1/2}/s$). Together with the experimental data, the numerical results are summarised in the following table (Table 1). δ_H and δ_Q are errors in the water head H and flow discharge Q of the STAR-CD results relative to the experimental ones. The corresponding values of the discharge coefficient are also given for reference.

Table 1. Comparison of numerical and experimental results

Flow Case	Experiment		STAR-CD Modelling			
	H (m)	C	H (m)	C	δ_H (%)	δ_Q (%)
Low Flow	0.1022	1.938	0.106	1.835	3.7	5.6
Design Flow	0.1948	1.938	0.196	1.892	0.6	0.9
High Flow	0.2718	2.000	0.278	1.933	2.3	3.4

As seen in the table, the calculated water level and flow discharge agree relatively well with the test values. For the design and high flow cases, the error is less than 2.3% for the water head and 3.4% for the discharge, which is considered satisfactory. For the low flow case, the error is somewhat larger, which is mainly due to the insufficient cell concentration. The mesh, although refined, is still relatively coarse in relation to the small water depth over the spillway.

It has been found in this study that the mesh in the vicinity of the water surface plays a vital role in the modelling accuracy. As mentioned above, the "Free-Surface Model" in STAR-CD is based on the Volume-of-Fluid (VOF) concept. If the fraction of water in a cell is denoted as ϕ, $0 < \phi < 1$ applies in those cells at the water-air interface. This means that the interface can cross several cells in the direction perpendicular to it due to numerical diffusion. To reduce the diffusion and obtain a relatively sharp surface, the only way is probably to have tighter cells.

The method Embedded Mesh Refinement in STAR-CD can be easily used to achieve desired grid concentration in domains of interest. It leads to an optimum usage of cells and can be effectively adapted to spillway flow modelling.

5.2 CPU time

In another study financed by ELFORSK AB, this spillway is modelled with three other CFD programs, i.e. CFX (version 4.1), PHOENICS (2.1) and FLUENT (4.4) (Yang et al.[5], Cederström et al.[6]). The simulations are made by different users and independent of each other.

One important conclusion from this study is that the dominating weakness of CFD modelling is the long computation time. To make a free-surface simulation with 50 000 - 80 000 cells, the CPU time required on the same workstation ranges from one to three weeks, depending on the program. This sets the limit for many practical applications.

Embedded Mesh Refinement makes it possible to perform a simulation with fewer cells without losing the modelling accuracy. It is obvious that fewer cells lead to shorter computer time. Thanks to this technique the CPU time can be significantly shortened. The spillway simulation for a flow case takes only two days to accomplish.

6 Concluding Remarks

The discharge capacity of the WES spillway has been modelled with the model Volume of Fluid (VOF) in the commercial code STAR-CD and the verification is made against experiments.

In order to reduce numerical diffusion and achieve satisfactory modelling accuracy it is desired that the grid at the water-air interface should have higher concentration in the direction perpendicular to the interface. The grid generation technique Embedded Mesh Refinement in STAR-CD can be effectively used to serve this purpose and it also leads to substantial reduction in the CPU time.

The findings of the study have shown that STAR-CD can reproduce the flume tests with acceptable accuracy. Higher accuracy is acquired for design and high flow situations. The CFD modelling agrees relatively well with the experiments in terms of water stage, flow discharge capacity and water-surface profile through the spillway.

7 Acknowledgements

This R&D project and also the preparation of the paper are funded by Vattenfall AB, Vattenkraft (PH). The writing of this paper would not have been possible without the support from Mr. Per-Gunnar Karlsson at Vattenkraft (PH). The STAR-CD license used for this study is provided by Computational Dynamics Ltd. The technical support from its staff is gratefully acknowledged.

References

[1] Flödeskommittén, *Riktlinjer för bestämning av dimensionerande flöden för dammanläggningar* (Guidelines for Calculation of Design Floods for Dams, in Swedish). Final report, Stockholm and Norrköping, Sweden, 1990.

[2] Yang, J., *Utskovsberäkningar – matematiska modeller med CFD-koden STAR-CD* (Simulations of Spillway Discharge Capacity – Numerical Models with STAR-CD, in Swedish), Report US 97:37, Vattenfall Utveckling AB, Älvkarleby, Sweden.

[3] Dargahi, B., *An Experimental Study of Discharge Characteristics of Spillways,* Report No. TRITA-AMI 3032, Division of Hydraulic Engineering, the Royal Institute of Technology, Stockholm, 1997.

[4] Computational Dynamics Ltd, *STAR-CD - Manuals, Commands and System, Tutorials*, 1996, London.

[5] Yang, J., Jansson, S., Hemström, B., Alavyoon, F. & Johansson, N., *Utveckling och verifiering av beräkningsteknik för utskovskapacitet* (Development and Verification of Numerical Technique for Spillway Capacity Determination, in Swedish), Report US 97:15, Vattenfall Utveckling AB, Älvkarleby, Sweden.

[6] Cederström, M., Dargahi, B., Hammar, L., Johansson, N. and Rundqvist, J., *CFD Modelling of Spillway Capacity – a comparison between hydraulic and mathematical models,* Int. Symposium on New Trends and Guidelines on Dam safety, June 17-19, 1998, Barcelona.