ON THE VALIDATION OF THE PARTICLE FINITE ELEMENTS METHOD (PFEM) FOR COMPLEX ENGINEERING FLUID FLOW PROBLEMS

A. Larese De Tetto, R. Rossi, S.R. Idelsohn and E.Oñate

International Center for Numerical Methods in Engineering (CIMNE)
Campus Norte UPC, Edifici C1, Gran Capitan, 08034, Barcelona
e-mail: onate@cimne.upc.edu
web page: http://www.cimne.com

Key words: PFEM, Fluid Dynamics

Abstract. Several comparisons between experiments and computational models are presented in the following pages. The objective is to verify the ability of Particle Finite Elements Methods (PFEM) [1] [2] to reproduce hydraulic phenomena involving large deformation of the fluid domain [4].

1 INTRODUCTION

The simulation of complex fluid flows involving large variations of the computational domain, constitutes an open challenge using most numerical techniques. The Particle Finite Element Method allows to merge the advantages of the “standard” FEM with the ones of meshless methods and it is naturally well suited to address this category of phenomena [3], [6]. At the current stage it still remains open the aspect of its validation in application to real problems. Current work aims to fill this gap by providing some experimental comparison to real flow cases.

First of all the behavior of a jet after a flip bucket is analyzed both for a 2D and a 3D case. The parameters compared are in this case the trajectory and the values of pressure on the “invert”. It follows the analysis of the opposite phenomenon: the under seal flow under a planar sluice gate. Finally the flux over a stepped spillway is briefly analyzed.

2 FLIP BUCKET

Flip buckets are energy dissipators used at the end of ski jump spillway of big dam: the purpose of this structure is to throw water well clear of the dam. The jet of a ski jump spillway leaves horizontally whereas the jet of a flip bucket is deflected upwards to induce disintegration in the air.

Both a 2D and a 3D model are considered in order to reproduce the experimental setup used by W. H. Hager [7] and R. Juon at the Zurich University. The original aim of the
A. Lares De Tetto, R. Rossi, S.R. Idelsohn and E.Oñate

An investigation was to propose a simple theory for the behavior of a flux over flip buckets. This included the creation of fitting curves for the experimental data which can be taken for a comparison with the PFEM numerical solution.

Figure 1: Photo of the experimental set-up at the Zurich University [7]

(a) Geometry of the experimental setting  (b) Schematic representation of a flip bucket

Figure 2: 2D model

Figure 3: 2D model

A simple 2D model which reproduced the geometry was built. All the fluid used in the analysis is progressively inserted in a Lagrangian way imposing the water depth at the inlet and the discharge [5].
The two parameters analyzed are the jet trajectory and the pressure along the “invert” (the reversed curve which makes the fluid to “jump”) of the incoming channel. Different scenarios are considered by varying the discharge and consequently the Froude Number while preserving the geometry of the invert and the depth at the inlet. For each case different meshes are used to verify the convergence to the real solution.

All the details on the theoretical and empirical functions used in the comparison can be found in [7].
The solutions are improving as expected when the mesh is refined as can be seen in fig. 9.

The results seem also to improve for the increasing of Froude Numbers. This can be explained by the reduced importance of the viscosity effects which cannot be resolved on the coarse meshes used.

![Figure 7: Comparison between theoretical and computational output: velocity variation](image)

Analogous considerations can be made in the case of the analysis of pressure head on the invert that can be compared with an empirical function given in [7].

The 3D model was then built to reproduce the effect of the introduction of a deflector as shown in fig 10 and 11, the planar and side developments of this wave were compared with photos of the experiment and it is qualitatively respected as shown in fig.12 and 13.
Figure 8: Empirical pressure head development above the invert

Figure 9: Comparison between empirical values and computational output: mesh variation

(a) 1cm mesh

(b) 0.5cm mesh
Figure 10: Effect of the insertion of a deflector

Figure 11: 3D Model
Figure 12: Fr5-Side

Figure 13: Fr5-Plane
3 SLUICE GATE

The behavior of an under seal flow is the second analyzed hydraulic phenomenon. A planar sluice gate creates a regular and controlled discharge of fluid: this is controlled only by the geometrical characteristics and by the depth of water of the upstream tank. The data are taken from an experiment made at the hydraulic division of the University of Padua. The under seal flow is governed by:

\[ Q = a \cdot C_c \sqrt{2gh} \]  

where \( C_c \) is the contraction coefficient that, for a planar thin gate is 0.611, \( a \) is the sluice gate elevation from the bottom of the channel and \( h \) is the water level in the upstream tank.

The parameters controlled in this case are:
- The pressure along the gate;
- The outing discharge;
- The analysis of the free surface of the downstream water;

![Graphs showing pressure distribution over time](image1)

Figure 14: Comparison between empirical functions, computational output and hydrostatic distribution

Different models have been built for the creation of a system that represented the real setting of an upstream tank with constant level of water. The inflow is given again in a Lagrangian way which originates a perturbation in the level of the reservoir [5]. The pressure head value in function of the vertical coordinate is compared with the hydrostatic distribution and the experimental values, as can be seen in fig. 14.

The discharge, obtained by the integration of the velocity diagram viewed in different sections for a same time instance, presents oscillations with an error that arrives at the
10% of the expected values. This can be explained by the oscillations in the level of the upstream tank.

The contraction of the under seal flow, on the contrary, is well reproduced in fact the oscillation which is present is of the same order than the dimension of the mesh, as can be seen in fig.15.

Inserting an high step, a slow downstream discharge is created. From the clash between an upstream fast discharge and a downstream slow one, an hydraulic jump has to occur, this is what is shown in fig. 16; the development of the free surface is the parameter which is compared with experimental data: the manual measurement of the free surface in a dissipation phenomenon like the hydraulic jump can only be qualitative and can be made with a low precision in some points whereas the computational datum is a continuum one.
Figure 17: Hydraulic jump. Blue line: free surface in the computational model. Pink line: interpolation between experimental measurements. Pink points: experimental measurements.
4 STEPPED SPILLWAY

The last problem considered, which is currently under investigation, is the flux over a stepped spillway. This is a category of structure which is nowadays becoming common because of the introduction of roller compacted concrete (RCC) that made it to become economically competitive with traditional spillway with dissipation pools. Basically, what is analyzed is the flux over a stair. The experimental data are taken from a Phd thesis done by Prof M.Sánchez-Juni and A.Táboas Amador at the Universitat Politecnica de Catalunya [8], [9].

Many and precise informations are available to reproduce accurate simulations of the development of the phenomenon. 2D models are right now created to analyze the development of velocity and pressure over the steps in the upper part of the stair where air is not present.
Figure 20: Velocity distribution after 3.4 sec

(a) Computational results

(b) Experimental results
REFERENCES


