USE GID AND TDYN (CFD) TO MODEL FLOW AT PUMP INTAKES IN FREIGHT PLATFORMS

Carlos Bacolla[1] and Gerardo Franck[2]

[1] *Facultad de Ingeniería y Ciencias Hídricas, Universidad Nacional del Litoral, cc 217. Ruta Nacional 168, km 472.4, (3000) Santa Fe, Argentina.*
[2] *Aula CIMNE – FICH. FICH, UNL. Ruta Nacional 168 Km 472,4, (3000) Santa Fe, Argentina.*

SUMMARY: This work presents a series of numerical simulations intended to determine whether the use of commercially available computational fluid dynamics software (e.g. GiD and Tdyn code) can provide a viable alternative to physical models for predicting the ocurrence of vortices and swirl in pumps intakes.
The large magnitude of stations pumping and water intakes at freight platforms on river, the power and the expensive conservation justify this research aiming at design improvement. Once the physical experiments were performed , GiD with Tdyn code was used to simulate the circulation patterns in the vicinity of a pumps intake.
Significant cost savings could be obtained if the designs of pump intakes and other structures could be developed using CFD models. We intend showing that Tdyn code can successfully predict vortices and swirl and enhance the design of pumps intakes.

KEYWORDS: pumps intake, flow water intake , vortices, swirl, freight platform

INTRODUCTION

The most widely used system for installing river water intakes in the Litoral Region is the disposition of pumps on docks. In this kind of intake the pumps remain submerged in the natural flow i.e. in a free flow with variable speeds and depths, which means the adduction is produced freely until it reaches the pumps intake section.
A number of hydraulic problems were detected in the pumps, such as the formation of vortexes and asymmetries in the intake flow, which causes serious mechanical problems and a great downfall in the pumps output. The present study aims at attempting solutions to this problems and setting design specifications for this kind of installations.

FEATURES OF THE INSTALLATIONS

These installations have a platform which is usually made of concrete and is placed above the maximum levels of predictable swells supported by piles in the river bed.

The fixing shield for the motor pump is placed on the platform and on top of it is the main body of the pump, which has the drive bearing, superior conducting bearing, water outlet pipe and ignition engine. The pump column is placed below this level; it consists of the water impulsion pipe, which bears the control axis and the intermediate drive bearings. The actual pump is at the lower end of the column, which is composed of a framework, the horn-shaped tulip that leads the threads as they enter and the drum wheel or drum wheels if it is multi-phase.The length of the pump column is a function of the variation of the river levels, and the pump has to be submerged at a minimum depth, higher than that required to avoid air intake and cavitation in the drum wheel.

HYDRAULIC MODELATION

The models reproducing flow systems to free surfaces, such as canals, rivers, and estuaries, and used to solve problems of level variation and changes in flow patterns due to emplacement of structures within the flow, work well in turbulent regime.

EXPERIMENTAL STUDIES ON THE MODEL

The construction of big pumping stations and water intakes require deep serious study and research so as to optimize their design and aim at reducing investment and operative cost and achieve proper hydraulic performance.
In order to satisfy these requirements it is important to get as close as possible to the ideal flow conditions mentioned below:

- **Uniformity**. On a conveniently defined transversal section of the installation, the magnitude and direction of the speeds must be equal at all points in t the section.
- **Permanence.** The magnitude and direction of the speeds must not vary with time.
- **Flow type.** The flow must be of the one phase type .i.e. there must not be dragging of air or vapor.
- **Vorticity**. There shouldn't be superficial or submerged vortexes.
- **Flow pre-rotation**. The flow reaching the pump impeller shouldn't present excessive rotation.

Following the analysis of the results of trials on a physical model, it should be pointed out that: **Under no circumstances will a pump simply submerged in the river be able to work properly; it will always have an asymmetric flow with horizontal axis vorticity that will affect its performance and disturb its operation.**
To avoid the consequences mentioned above, pump adduction is always required to uniform the flow before it goes into the pump. The trials on a physical model also show that the distribution of the flow is strongly asymmetrical at the intake section of the pump in the direction of the canal flow but symmetric in perpendicular direction. This fact gave rise to the idea that the subdivision of the intake section of the pump in various sections perpendicular to the direction of the canal flow with flow orienting elements could improve distribution. In spite of the advancements achieved in the trial method on physical models by means of shreds indicating flow orientation, the following aspects remain uncertain:

- *at low speeds they go in directions that do not allow to discern whether they are caused by gravity or by the flow.*
- *the length of trial within the suction tube is limited by the level of water in the canal, which is not high enough; when the shreds come close to the tube the pump represents, they are attracted by this speeds field, the limit being somewhat imprecise.*
- *the lower the suction speed, the better the flow conditions; and it is under those conditions when the observation system losses sensitivity.*
- *the shreds define only the orientation of the flow lines, not the speeds. This might lead to mistakes since asymmetries are not detected. These limitations in the trial system with physical models have stopped us from moving on towards the final objectives of the project, that is the reason why we decided to use numerical simulation.*

PROJECT REFORMULATION.

"Integrating Numerical Simulation and Physical Modeling in the Design Optimization Process of Water Intakes on Docks". Although the experimental developments performed so far have not given a complete answer to the objectives proposed, they constitute a base for the analysis which allows a first approach to the problem. The results obtained support further analysis tools whose power and versatility allow the formulation of the desired design lines. On this basis, a stage not planned in the original project has been conceived in which the results come from the integration of physical modeling and numerical simulation.

NUMERICAL RESOLUTION

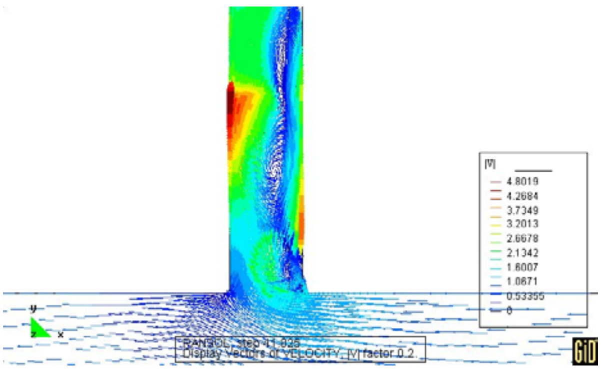
First, some of the situations previously tried with the physical model were modeled again, this time using the GID interface and the Tdyn simulation environment, and with the objectives listed below:

- *to compare the results to those obtained with the physical model so as to revalidate the mathematical modeling.*
- *to simplify the mathematical modeling from the physical model results and obtain results in the short term.*
- *to model situations with relevant characteristics from the point of view of the potential results in order to define design norms for the Dock Intakes and compare them to the physical model results.*

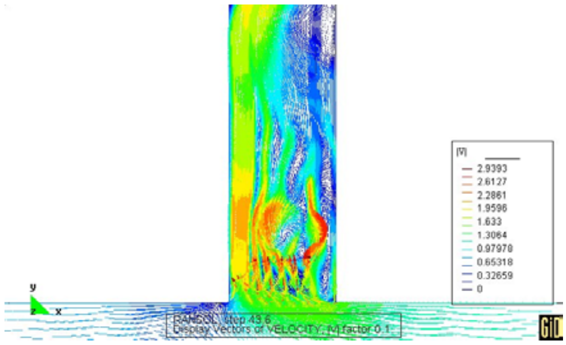
Accordingly, we decided to perform the first mathematical modeling in 2D, which is seen as valid after the results obtained with the physical model. Besides, execution is faster. Starting from these results, some 3D modeling will be performed to justify conclusions. The adopted magnitudes coincide with those typically found at the established intakes.

MODELED CASES

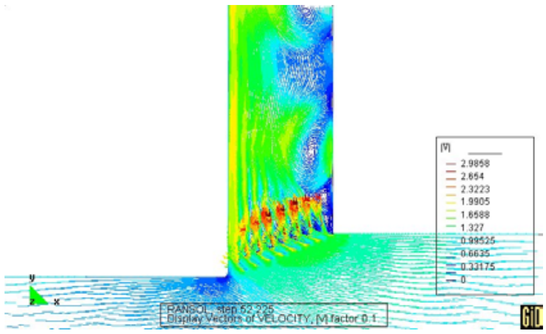
The cases below were modeled with GID and Tydn.
Case 1: cylindrical case with horizontal suction section, with no deflectors. The circulation speeds in river and case equal 1m/s.
Case 2: cylindrical case with horizontal suction section, with deflectors. The circulation speeds in river and case equal 1m/s.
Case 3: cylindrical case with 22,5° leaning suction section, with deflectors. The circulation speeds in river and case equal 1m/s.
The results are:
Case 1: A strong vorticity of horizontal axis on the edge of the waters above the case is observed, which is spread all along the case, generating high speeds upwards on the water wall below the case which diminish up waters and becomes negative in some cases. They generally match the results of the physical modeling.



Case 2: Vorticity along the first section of the case is observed, which is weak compared to the previous case and tends to diminish upwards. Speeds remain high on the water wall above the case. Again, they generally match the results of the physical modeling.



Case 3: Vorticity even weaker than case 2 is observed, tending to become uniform about 3 diameters away from the suction section. Speeds remain high on the water wall above the case/shield.



CONCLUSIONS:

The results obtained with the mathematical modeling using the GID interface and the Tydn simulation environment are highly satisfactory and open a wide perspective for the present study. On the one hand, the intakes of the mathematical modeling give a much more detailed series of magnitudes than the physical modeling, which although not entirely accurate, indicate a very similar general conformation in relation to the hydraulic modeling. This makes them reliable enough when it comes to reaching general conclusions and defining convenient geometric forms of the approximation pipe leading to the pumps suction. On the other hand, the mathematical modeling makes it possible to study the numerous variables to be taken into account because it allows reducing costs and time.

REFERENCES

1. Roberge, J.A., “Use of computational fluid dynamics (cfd) to model flow at pump intakes” Thesis Worcester Polytechnic Institute, Environmental Engineering May 1999.

2. Constantinescu, G.S. and V.C. Patel, 1997, Role of turbulence model in prediction of pump intake vortices, 10th International Conference on Numerical Methods, Swansea, UK.

3. Constantinescu, G.S., and V.C. Patel, 1998, “Numerical model for simulation of pump-intake flow and vortices”, *Journal of Hydraulic Engineering*, Vol. 124, No. 2, Pp 123-134.

4. Dhillon, G.S., Sakhuja, V.S. and Paul, T.C., “Modelling Criteria for Vortex Formation at Pipe Intakes”, Vol. V.XIX Congress IAHR Nueva Delhi, India, 1981.

5. GiD, A pre/postprocessing enviroment for generation of data and visualisation in finite element analysis. CIMNE, Barcelona, 1996.