ANALYSIS OF REAL FLUID FLOW IN PLANAR NETWORKS PROFILES

Mariana VODA (TODIRUTA)

"Politehnica" University of Timişoara

REZUMAT. Lucrarea de faţă,are ca scop analiza curgerii în reţele de profile plane, pentru determinarea mărimilor specifice primare (viteză, presiune) şi globale (forţă, cădere, randament). Curgerea studiată, este o curgere într-o reţea mobilă de profile hidrodinamice (turbină). Pentru acest studiu s-a ales profilul hidrodinamic NACA 8410 dispus într-o reţea dreaptă de profile, cu un unghi de instalare de 40º şi unghiul de intrare de 50º. Pasul relativ al reţelei este de 1,2 m iar domeniul de analiză este bidimensional 2D, cu profilul înglobat în banda de periodicitate. Pentru discretizarea domeniului de analiză s-a ales o reţea structurată cu 36.626 elemente patrulatere. S-a efectuat simularea numerică pentru curgerea fluidului incompresibil vâscos, în regim nestaţionar pentru o viteza de transport U de 8,2 m/s. Se analizează fenomenul de curgere şi se pun în evidenţă parametrii ce influenţează curgerea, prin soluţionarea numerică a ecuaţiilor ce guvernează curgerea fluidului în echipamente hidraulice utilizând programul pentru simularea curgerii fluidelor FLUENT6.3. Pentru definirea domeniului de analiză (2D), a condiţiilor de periodicitate, discretizarea şi definirea condiţiilor de frontieră, s-a utilizat programul de preprocesare GAMBIT. Rezultatele obţinute au fost sub formă numerică, şi au fost postprocesate grafic şi calculate mărimi derivate din cele obţinute în etapa de soluţionare numerica. În urma interpretării rezultatelor obţinute s-a ajuns la concluzia de bază că mărimile hidrodinamice specifice, *primare* şi *globale* determinate prin metoda simulării numerice, rezultatele numerice obtinute sunt valide in concordanta cu fenomenul real studiat. Din distribuţia de presiune statică, rezultă că incidenţa curentului pe profil este bună. De aceea randamentul este ridicat.

Cuvinte cheie: fluid real, retea de profile, coeficient de presiune, simulare numerica.

ABSTRACT: This paper aims to analyze the flow in planar networks profiles to determine the specific primary quantities (velocity, pressure) and global (force, dropping, efficiency). Studied flow is a flow in a mobile network of hydrodynamic profiles (turbine). For this study we chose the NACA 8410 hydrodynamic profile arranged in a straight network, with an installation angle of 40 ° and 50° entrance angle. Relatively step of the network is 1.2 m and the analysis domain is 2D, with the profile embedded in the periodicity band. For meshing domain analysis was chosen a structured grid with 36.626 quadrilateral elements. Numerical simulation was performed for incompressible viscous fluid flow in unsteady regime for U transport velocity of 8.2 m / sec. It analyzes the phenomenon of flow, and highlight parameters which influencing the flow by numerically solving of the equations that govern the fluid flow in hydraulic equipment using the numerical simulation program FLUENT6.3. To define the 2D analysis domain, periodicity conditions, meshing and defining boundary conditions was used GAMBIT preprocessing program. The results obtained were in numerical form, and were post-processed graphically and calculated quantities derived from the numerical solution. Following the interpretation of the results obtained we concluded that specific hydrodynamic sizes, global and primary, determined by the numerical simulation, the results are valid according to the real phenomenon studied. From the static pressure distribution, it is clear that the incidence of the current profile is good. Therefore the efficiency is high.

Keywords: real fluid, profiles network, pressure coefficient, numerical simulation.

1. INTRODUCTION

In the team of Hydraulic Machines department of Mechanical Engineering Faculty Timişoara, Prof. Iuliu Carte was the first which in 1987, in his doctoral thesis addressed the numerical methods in the study of radial-axial profiles for Francis turbines, and later the plane networks profiles. Prof. O. Popa throughout the Laboratory of Hydraulic Machines was first who performed an analytical calculation for thin profiles. Also in the team of Hydraulic Machines, Prof. Susan R. Resiga properly implemented periodicity conditions in Cascade Flow Analysis (CFA) [1]. Analysis of flow in planar networks of profiles is an important problem in the design, optimization and performance evaluation of turbomachinery [4], [5]. A flat network profile consists of a regular arrangement of profiles that are aligned with leading edges on a line or a circle. Hydrodynamic profiles arranged in the network are the basic elements in the construction of runner blades, guide vanes and stator of hydraulic machinery - turbines, pumps, reversible machinery.

The energy characteristics and cavitations operating dependent the behavior of hydraulic machinery. A fluid flow is characterized both by the distribution of velocities (vector field) and the pressure distribution (scalar field). The stream lines are fluid particle trajectories over which the velocity vector is tangent to the line. The purpose of network sections is to change the direction of the fluid that flows through [5]. The hydrodynamic profile is the contour section made by an aircraft wing or propeller blade in a perpendicular plane to the wing surface and parallel to the plane axis. The hydrodynamic profile goal is to create a load-bearing capacity as possible, while having a smaller drag as. The bearing force is the sum of the forces generated by the movement of the fluid around a body designed perpendicular to the main direction of flow of the fluid in which is immersed the body.

Bearing is a mechanical force generated by the interaction and contact between fluid and solid. In order to lift the solid body in necessary to be in direct contact with the fluid. If there is no movement cannot speak of bearing. The property of the fluids to resist to the flow is called viscosity. Normal stress (on the surface of separation) is called pressure. All real fluids are viscous, except superfluid. An ideal fluid is considered inviscid fluid. The purpose of this paper is to analyze the flow in planar networks profiles to determine the specific primary quantities (velocity, pressure) and global (force, dropping, efficiency), in a real fluid flow in a network flat turbine (Fig. 1).

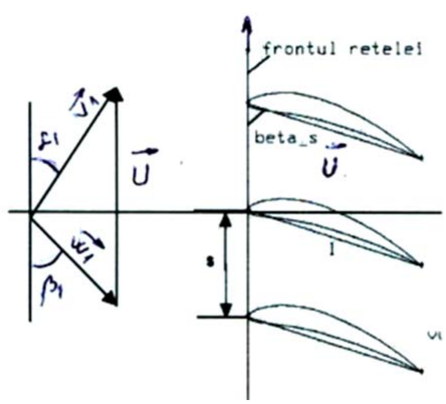

Fig. 1. Velocity triangle of the flow in a planar profiles network.

The flow studied in this paper, is a viscous flow in a mobile network hydrodynamic profiles (turbine). For this study we chose the straight network profiles, a two-dimensional 2D domain analysis with embedded profile periodicity band and result a double-domain analysis. To meshing the analysis domain it was used a structured mesh with 36.626 quadrilateral elements. The numerical simulation was performed for incompressible viscous fluid flow in unsteady regime for U transport velocity of 8.2 m / sec using the network characteristic parameters in Table 1. The boundary conditions imposed are shown in Table 2. This paper is structured as follows: in Section 2 are presented methods and means of investigation used. Section 3 presents the results of analysis. Section 4 reports the findings of this study.

2. METHODOLOGY

Investigation of the manner in which the phenomenon of flow is realized, and highlight the parameters that influence the flow, was performed using methods and techniques for solving numerical equations governing fluid flow in hydraulic equipment. Any flow problem that has been solved using numerical simulation techniques the following the steps below:

a) defined the analysis domain;

b) meshing the analysis domain;

c) define the boundary conditions and operating conditions;

d) Solver selection used depending on the nature of the problem;

e) postprocessing numerical results.

Following the steps outlined above and applying the correct numerical analysis method, is obtain numerical results consistent with the real phenomena studied.

a. *Defining the analysis domain*. Defining the analysis domain for the studied problem was performed using GAMBIT preprocessing program [2]. The vertices of the profile which are next to been studied were imported from CASCADExpert program to GAMBIT program. By joining the vertices which are defining the profile shape Fig. 2, was constructed the profile curve. Then the profile was turned at the installation angle 40°, were put the periodicity conditions, achieving the defined network profiles and was defined the analysis domain. The number of profiles which make up the network is given by the runner blades number. The network profiles were building by defining the analysis domain. In this case we use a straight network profiles with embedded profile in the periodicity band, resulting a double analysis domain. Sizes characteristic of the network profiles studied are presented in Table 1.

Table 1. **Characteristic quantity of profiles**

The hydrodynamic shape	NACA8410
Installation angle β_s	40^o
Relative step s/c	1,2
Entrance angle β_1	50^o
Tangential velocity U	8,2 m/sec

In Figure 2 are presented the profile vertices NACA 8410 given by CASCADExpert program. On the surfaces which delimiting the periodicity band we have periodicity conditions for velocity and pressure.

Fig. 2. The profile vertices NACA 8410.

b. *Meshing the analysis domain.* Achieving a proper meshing of the analysis domain is important to obtain an accurate numerical solution. The mesh is structured with quadrilateral elements and 36.626 cells as shown in Fig. 3. On the profile curve was made a finer mesh to capture phenomena that may occur around the profile.

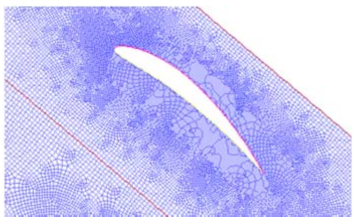

Fig. 3. Meshing the analysis domain.

c. *Defining the boundary conditions*. Here are defined physical properties of substances and boundary conditions. At the entrance the boundary condition imposed is velocity, and at the outlet pressure conditions. It is provided use of an incompressible fluid and impervious to the profile surface. The solid surface is a border profile impervious, and therefore the normal component of the velocity is zero in the profile.

$$(v_n \equiv v \cdot n = 0 + v_\tau = 0)\ \vec{v}$$

Table 2. **Boundary conditions**

Layer	Imposed condition	Value
Inlet	Velocity	8.2 m/s
Outlet	Pressure	0 Pa
Solid boundary	Wall	
Periodic boundary	Periodicity conditions	

From experimental observations it appears that the real fluid flow (viscous) around the hydrodynamic profiles and the flow not avoid the trailing edge of the profile. The boundary conditions imposed in solving the flow analysis are presented in Table 2.

d. *Selection methodology for solving flow equations.* Solving the equations which governing the fluid motion was made for a 2D unsteady regime, tackling the Navier-Sokes flow equations [7]:

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t} + \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

The characteristic movement of the viscous fluid is carried out by Reynolds number. This parameter is the ratio of inertial forces and friction forces.

$$\mathrm{Re} = \frac{v \cdot d}{\nu} > 2300$$

These flow equations were solved using FLUENT 6.3 program for simulating fluid flow [3]. To solve the real fluid flow network profiles k-epsilon turbulence model is used, implemented in FLUENT. The boundary conditions used are shown in Table 2. The profile surface velocity was zero, so v = 0. Incompressible fluid condition was used, and the impervious surface on the profile.

e. *Postprocessing numerical results.* Postprocessing is to present qualitative and quantitative numerical results based on numerical simulation. Introducing qualitative results were as color map for different sizes of interest to study. For this purpose we obtained results that describe the static pressure, dynamic pressure, total pressure, fluid velocity profiles in the network. A quantitative analysis involves the development of graphics, accessing or calculating characteristic values specific to the physical phenomenon studied. We calculated the hydrodynamic field, dropping (H), mechanical power (P_m), hydraulic power (P_h) turbine efficiency (η).

3 RESULTS

Quantitative analysis of the results involves calculating characteristic values specific to the physical phenomenon studied. Thus was obtaining the following sizes:

– the force on the profile F = 1.414,64 N;
– total pressure at inlet P_{in}= 2.943,2 Pa;
– total pressure at outlet P_{out}= 32,028839 Pa;
– discharge Q = 4,33 m^3/sec;
– dropping (H):

$$H = \frac{p_1 - p_2}{\rho g} + \frac{v_1^2 - v_2^2}{2g} + z_1 - z_2 \left[\frac{\mathrm{J}}{\mathrm{N}} = \mathrm{m}\right] \tag{4}$$

$$H = 0,29675\ \mathrm{m}$$

– mechanical power (P_m)

$$P_m = U \cdot F_u \tag{5}$$

where: U is the transport velocity; F_u – the force on the profile; $P_m = 11.600,11934$ [W].

– hydraulic power (P_h)

$$P_h = \rho g Q H = Q \cdot \Delta P_{tot} \tag{6}$$

$$P_{tot} = \rho \cdot \frac{v^2}{2} \tag{7}$$

ΔP_{tot} = 2.911,176, P_h = 1,6 kW

– efficiency η

$$\eta = \frac{P_m}{P_h}, \qquad \eta = 92\,\% \tag{8}$$

– pressure coefficient (C_p)

$$C_p = \frac{p - p_{in}}{\frac{\rho \cdot v_{in}^2}{2}} \quad \rho = 10^3 \tag{9}$$

Qualitative presentation of results is done in the form of map color for different sizes of interest to study (Figs. 4...8).

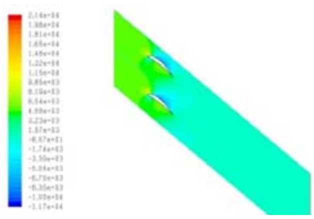

Fig. 4. Static pressure.

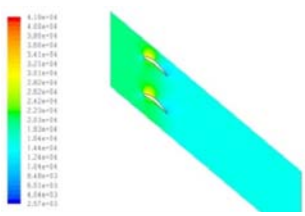

Fig. 5. Dynamic pressure.

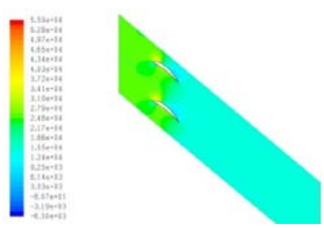

Fig. 6. Total pressure.

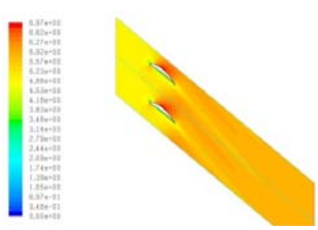

Fig. 7. Relative velocity.

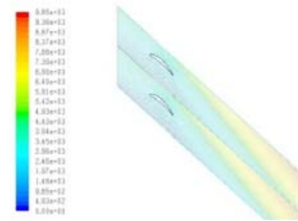

Fig. 8. Streamlines.

4. CONCLUSION REMARKS

The basic conclusion tells us that the studied hydrodynamic sizes global and primary determined by the method of numerical simulation, are valid according to the real phenomenon studied. Static pressure is the pressure inside a fluid. The static pressure distribution shows that the current incidence on the profile is good static pressure is higher on the leading edge of the hydrodynamic profile where profile incident occurs.

Therefore the efficiency is high. By the total pressure distribution, it is clear that the inlet pressure is high and there is power dropping on the grid profiles, and the fluid energy is less at outlet. Dynamic pressure is the additional pressure of a fluid that would strike a surface and would be required to completely consume the kinetic energy. From the distribution of the relative velocity is observed that due to the forces which occur in the interaction between the liquid and solid, the extrados profile relative velocity begins to increase. The analysis of the streamlines there shows a drop in layers parallel to each other. The layers of fluid move at different rates remain parallel to each other, without mixing. This is possible when the external force to maintain the flow is comparable with the resistance force that opposes the fluid. Streamlines are parallel, so the flow is laminar.

REFERENCES

[1] Susan-Resiga R., Muntean S., Anton I., *Analiza numerică a curgerii cu desprinderi în rețele plane de profile,* GAR Nr. 156, Contract Nr. 120/1999.

[2] ***, Gambit 1. User's Guide, Fluent Incorporated, May 1998.

[3] ***, FLUENT 5. User's Guide, Fluent Incorporated, July 1998.

[4] Susan-Resiga R., *Complemente de mecanica fluidelor şi tehnici de soluționare numerică*, Editura Orizonturi Universitare, Timişoara, 1999, ISBN 973-9400-60-4.

[5] Susan Resiga R. M*ecanica fluidelor numerică*, Editura Orizonturi Universitare, Timişoara, 2003, 223 p. ISBN 973-638-014-9.