ANNALS of the ORADEA UNIVERSITY.
Fascicle of Management and Technological Engineering, Volume IX (XIX), 2010, NR3

FEM MODELING OF FLUID FLOW FOR KAPLAN TURBINE RUNNER

HORA Cristina, DUMITRESCU Dănuț, HORA Horea
University of Oradea
chora@uoradea.ro, dumitrescudan70@yahoo.com, hora_horea@yahoo.com

Keywords: Kaplan turbine, runner blades, modeling, Finite Element Method (FEM)

Abstract: Within this paper is given the importance and methodological issues specific FEM modeling of fluid flow. In the first part are presented the general considerations viewing main steps in the finite element model of fluid flow along the Kaplan turbine runner blades. In order to exemplify the way in which this method is applied, in the second part are given the results realized in the diagrams of velocities, pressures and vorticity distribution in 6 typical sections of runner and, also a tabular presentation of the peak values of these dimensionless parameters. The last part presents the conclusions.

1. INTRODUCTION

The complex hydrodynamic processes that occur in hydraulic turbines foster and create conditions for the emergence and development of the wear phenomenon. As a result, the need for modeling and simulation of these processes is justified.
The analysis by finite element method (FEM) of turbine runner fluid flow is based on the differential and partial differential equations. They shape incompressible non-viscous flow phenomena. The considered flows are studied in a two-dimensional space domain, the formulation of mathematical phrases being realized with auxiliary variables (current function and speed potential). A key element is the concept of vorticity, which is part of fluid particle rotation during flow. In order to achieve the finite element modeling of flow over the runner blades of Kaplan turbines are susceptible FEMLAB program, which has a specific method of fluid dynamics calculating. The flow phenomena of viscous incompressible fluids are described by Navier-Stokes equation in relation to current function and vorticity. Navier-Stokes time function equations are the base for modeling in FEMLAB program, and they are described by [2]:

$$\begin{cases} \rho\dfrac{\partial v}{\partial t} - \nabla\eta(\nabla v + (\nabla v)^T) + \rho(v\nabla)v + \nabla p = F \\ \nabla v = 0 \end{cases} \tag{1}$$

where η is dynamic viscosity, ρ is fluid density, p pressure, v is velocity and F a potential field of forces, e.g. gravitational forces field. In these relations v and F are vectors as speed and power are values oriented in a three-dimensional space. For the stationary flow, equation (1) becomes:

$$\begin{cases} -\nabla\eta(\nabla v + (\nabla v)^T) + \rho(v\nabla)v + \nabla p = F \\ \nabla v = 0 \end{cases} \tag{2}$$

Generally Navier-Stokes equations are used as dimensionless. Introducing the dimensionless variable $v^* = v/u$, a ladder of length, dimensionless pressure p*, and appropriately scaled forces F*, the equation can be written:

$$\begin{cases} -\nabla(\frac{1}{Re}(\nabla v^* + (\nabla v^*)^T)) + (v^* \nabla)v^* + \nabla p^* = F^* \\ \nabla v^* = 0 \end{cases} \tag{3}$$

where the effect of viscosity is introduced by the dimensionless Reynolds number:

$$Re = \frac{UL\rho}{\eta} \tag{4}$$

The relation between pressure, dimensionless and physical forces is:

$$\begin{cases} p^* = \frac{p}{\rho U^2} \\ F^* = F\frac{L}{\rho U^2} \end{cases} \tag{5}$$

Reynolds number value has a major influence on the flow. Turbulence generally increases when increasing the Reynolds number.
For specific problems of fluid dynamics, there are the following options regarding boundary conditions in FEMLAB program:

- Input on field boundary, specified fluid velocity;
- Exit on the boundary area, the speed is specified by a Neuman-type condition and the pressure is constant (p = 0);
- Slipping condition, where the normal component of velocity is 0;
- Without slipping, the fluid velocity is equal to the boundary (usually 0);
- Direct output, where the pressure and the tangential component of velocity are 0;
- Neutral border.

After solving the problem, you can display (in the form of color-coded diagrams):

- velocity components in x and y directions;
- pressure,
- vorticity.

2. CASE STUDY. FLUID FLOW MODELING FOR KAPLAN TURBINE RUNNER OF HPP TILEAGD

2.1. Kaplan turbine runner geometry

Kaplan turbine runner of HPP Tileagd has a number of $z_r = 6$ blades and was represented by a number of 6 sections. Those characteristic elements are presented in table 1, in which the angle β_s is measured between front flat network profiles and wing chord (fig.1). The sizes x_d, x_f, x_c, y_c are measured from the origin coordinate system for hydrodynamic profile placed horizontally, origin of the profile is the leading wing edge (the leading wing edge has coordinates (0,0)) [1]. Geometric elements used in table 1 and figure 1 are as follows:

- R – cylinder radius on which profile network is structured;
- β_s - installation angle of profiles in the network;
- L – wing chord length;

- $t = \frac{2\pi R}{z_r}$ - pitch
- x_d – the position of profile maximum thickness;
- d – the profile maximum thickness. In this case, the thickness is calculated as the vertical distance between back and soffit of blade;
- x_f – the position of maximum profile height;
- f – the relative maximum profile height, that is, the maximum distance between wing chord and its skeleton curve;
- x_c, y_c - stacking axis position of profiles.

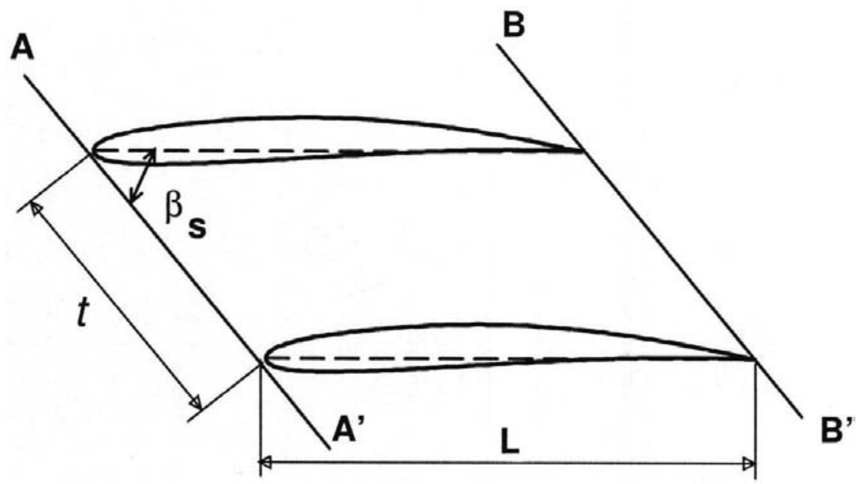


Fig. 1 - Network profile geometric elements

Table 1 - Elements of geometry and angle of the rotor blade Kaplan turbines from HPP Tileagd

Profile no.	r [mm]	β_s [°]	L [mm]	x_d [mm]	d [mm]	x_f [mm]	f [mm]	x_c [mm]	y_c [mm]
1.	1500,0	6	1300,0	333,45	28,73	333,45	28,73	444,60	14,95
2.	1333,7	8	1231,9	333,21	42,25	333,21	42,25	464,40	13,30
3.	1166,9	10	1147,7	315,04	57,15	315,04	57,15	467,11	12,62
4.	1000,6	12	1038,9	291,51	70,43	291,51	70,43	444,64	14,85
5.	833,7	14	911,9	274,11	83,98	274,11	83,98	402,14	21,52
6.	766,9	16	863,9	266,59	88,54	266,59	88,54	387,02	23,06

2.2. Model development

Generating 3D model of the rotor is particularly difficult and requires information systems with significant computing power. Therefore, FEM analysis was conducted in a 2D field set to 2.5 x 2 m. Were considered 6 sections of the rotor blade, numbered from 1 to 6 (fig. 2) and having geometric and angular characteristic data, shown in table 1.

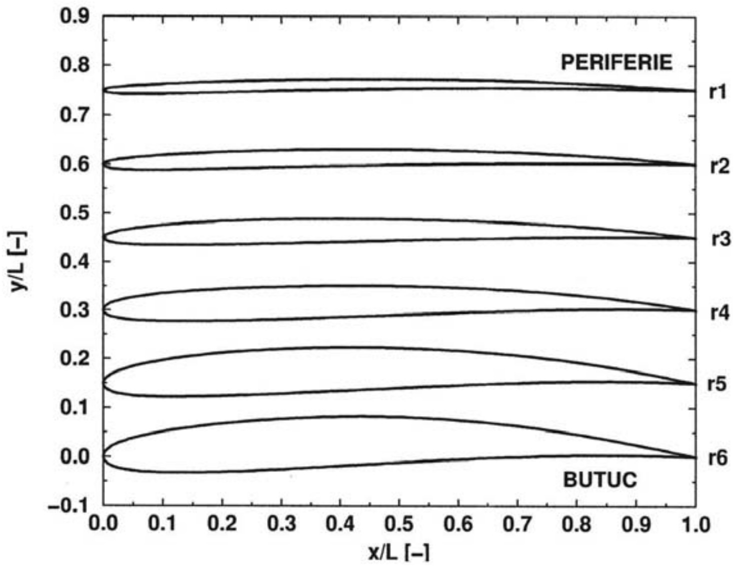


Fig. 2 - Hydrodynamic profiles used for Kaplan runner

Based on these, geometric models were made in all 6 sections. The profile 1 model 1 is shown in figure 3. The 8 circles distributed along the profile are necessary to define the control points of the profile.

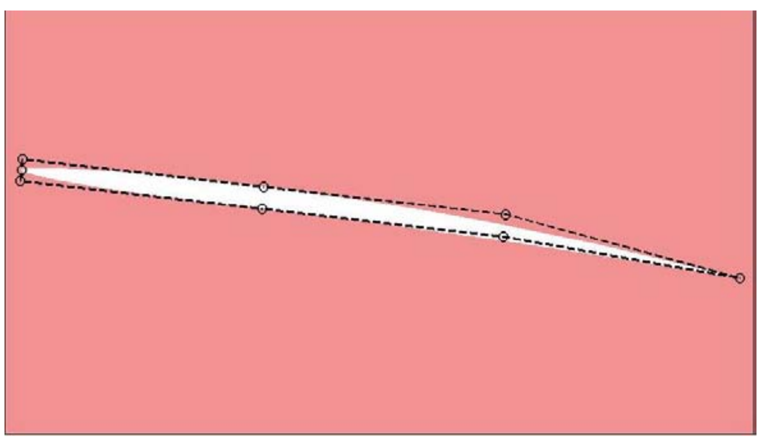


Fig. 3. – Profile 1 geometric model

2.3. Setting border and boundary conditions

After performing the geometrical model, border conditions are defined according to figure 4. Rectangle representing the profile outer border. The left side (segment (a)) is the border of the jet fluid entry on which, the speed v_z = 1,8 m/s is specified. The other remaining 3 sides (segments marked with (b)) of the outer border, will be defined as border exit with the condition: p = 0. Finally the whole shape of the blade profile is defined as the border without slip (curves (c)), where the fluid velocity: $v_x = 0$, $v_y = 0$.

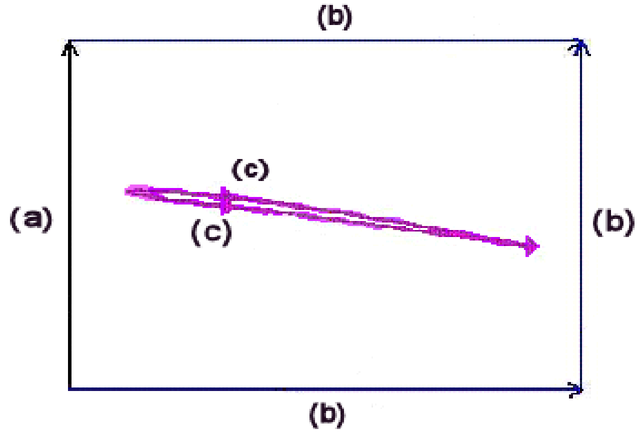


Fig. 4. - Explanatory border conditions

2.4. Model meshing

Model meshing is realized with the triangular elements described in paragraph [2,3] (figure 5). In profile 1 the number of nods is N = 2214 and the number of elements is E = 4192.

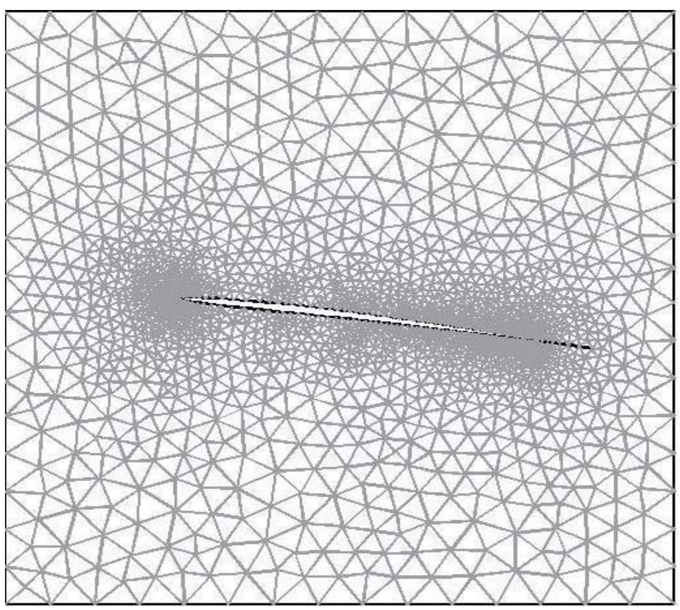


Fig.5 - Triangular finite element grid

2.5. Representation and interpretation of modeling results

Resolving application results show the color-coded two-dimensional diagrams in Figures 6, 7 and 8. Modeling program source code and the diagrams resulted for profiles 2, 3, 4, 5, 6 are presented in [1].

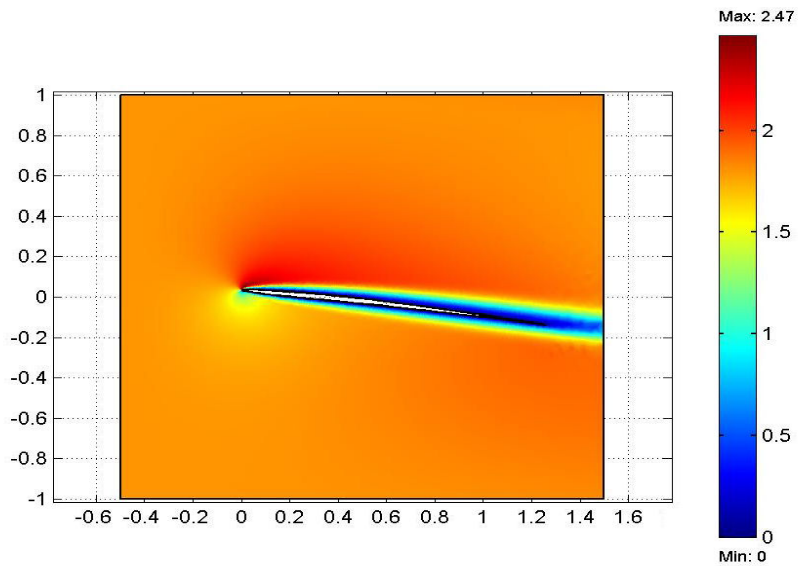


Fig.6 – Speed diagram – profile 1.

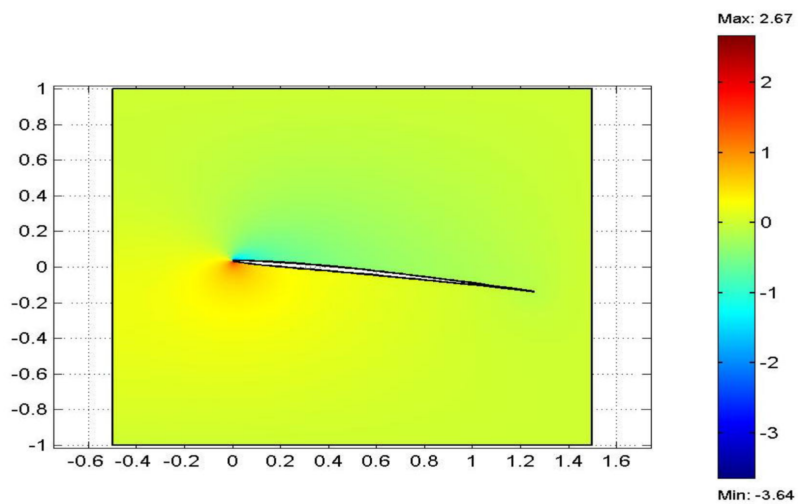
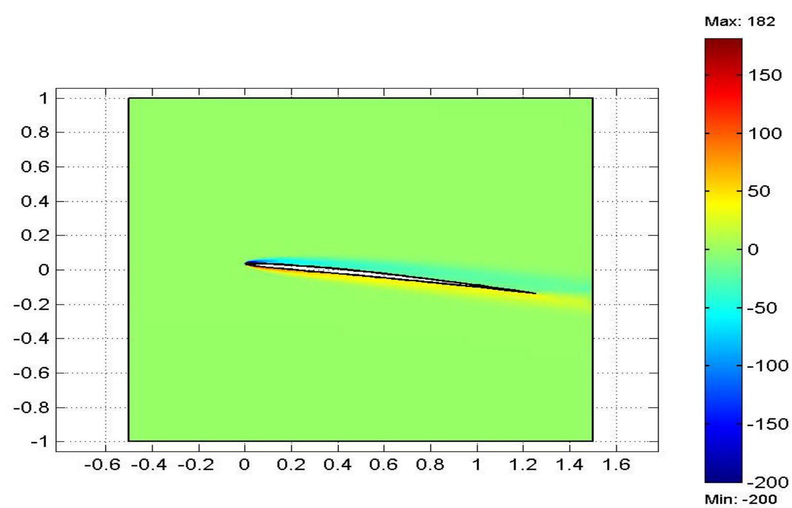
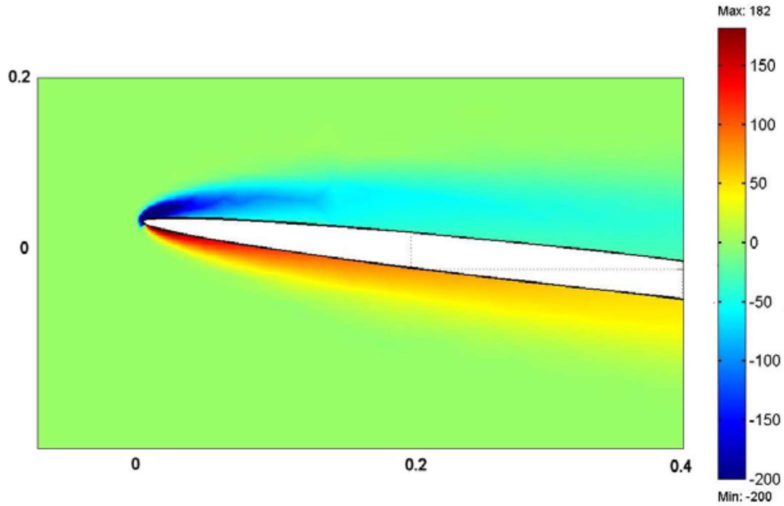


Fig. 7 – Pressure diagram – profile 1



a).



b).

Fig. 8 – Vorticity diagram a) profile 1, b) detail

After finite element analysis the maximum dimensionless flow parameters are obtained, they will be used as input for cellular automata simulation. These values are presented in table 2.

Table 2 - The maximum values of dimensionless flow parameters

Profile No.	Positon		Velocity v*	Pressure p*	Vorticity ω*
	x	y			
1.	0,0144	0,0285	0,132	2,42	198,4
2.	0,0165	0,0371	0,204	1,78	176,3
3.	0,0204	0,0405	0,174	1,81	144,2
4.	0,0157	0,0475	0,282	1,96	138,4
5.	0,0362	0,0372	0,526	1,32	133,5
6.	0,0228	0,0457	0,556	1,29	124,1

CONCLUSIONS

Based on the results obtained using FEM it is found that:

1. The purpose of flow analysis using the finite element method is to provide the necessary input data simulation program of wear runner blades, which use cellular automata method.

2. Maximum requests are located generally in the leading wing edge, maximum velocity values being obtained on the blade back, while the maximum pressure values are located on the soffit. Vorticity is also higher near the leading wing edge, having positive values on soffit and negative values on blade back.

3. In a comparative analysis of the blades different sections characteristic values there is an increased flow velocity at the proximity of hub. It should be noted that the achieved velocities are "relative" as in this study the turbine speed has not been taken into consideration. The probable cause in speed increasing is given by attack angle increasing and blade profile thickening. At the same time, the maximum pressure decreases with proximity to the hub and the pressure difference between the soffit and back increases around the leading wing edge. Regarding vorticity it is indicated that the soffit area of blade surface seeks the maximum positive values and on blade back the negative values.

4. In profiles near the hub, in the area behind the blade is found a turbulence field that can cause adverse effects in turbine function.

BIBLIOGRAPHY

[1]. Hora C. – Studii şi cercetări privind fiabilitatea sistemelor hidraulice din structura centralelor electrice, Teză de doctorat, Oradea, 2007

[2]. Resiga, R. – *Complemente de mecanica fluidelor şi tehnici de soluționare numerică*, Editura Orizonturi Universitare, Timişoara, 1999

[3]. Zienkiewicz, O.C., Taylor, R.L. - *The Finite Element Method*, 4th edition, McGraw-Hill, London, England, 1994

[4].***ANSYS Workbench, CFX, AUTODYN, FLUENT, ©2006 ANSYS, Inc. ansysinfo@ansys.com

[5].***MATLAB Signal Processing Toolbox User's Guide The Works Inc., 1998