NUMERICAL SIMULATION OF THE VISCOUS FLUID FLOW THROUGH LABYRINTHS

Mircea CAZACU[1], Moutasem ALKHATIB[2], Yahia ZAKARIA[3]

This paper presents the mathematical modeling and the numerical simulation of the three dimensional viscous flows through the labyrinths of the hydraulic machines which can be linear or with grooves. The research was performed using three different programs of simulation. Finally, the authors present a comparison between the numerical results and the classical experimental ones, here including the forms of Taylor's whirls.

Keywords: modeling and simulation, CFD, labyrinths, slow flows, hydraulic machines, sealing systems.

1. Introduction

The functioning of several fluid flow systems is affected by the leakage at particular locations in these systems. For instance, some applications have to operate in a leakage-free environment. In multi-stage turbo-machines, leakage flow between stages needs to be reduced to achieve a good efficiency. In other applications the leakage of fuel or chemical liquids has to be avoided to prevent accidents. In all these cases for a given pressure difference the leakage can be reduced, by increasing the flow resistance. This can be simply done by reducing the clearance of the flow passage. But such reduction would lead to practical difficulties such as the removal of accumulated abrasive particles, the occurrence of thermal and mechanical instabilities and difficulties in the assembling and disassembling the components. In practice, it is useful to keep the clearances above a certain minimum value. Other parameters that influence the performance of the labyrinth sealing systems are the following: the shape, the size of the cavities and the material of the labyrinth walls. [1], [2]

The subject in discussion presents a special importance for: a) the improvement of the volume efficiency of turbo machines, especially the hydraulic pumps; b) the issues related to the projection of the labyrinth seals between the rotor and the encasing; c) a special theoretical importance regarding the laminar

[1] Prof., University POLITEHNICA of Bucharest, Romania
[2] PhD Student, University POLITEHNICA of Bucharest, Romania, e-mail: moutasemalkhatib@yahoo.com
[3] PhD Student, University POLITEHNICA of Bucharest, Romania

flow of viscous fluids through these micro clearances which have at the same time complicated shapes.

Due to the small dimensions of the labyrinth seal, which range from tenths of a millimeter in the case of high pressure pumps to few millimeters in the case of hydraulic turbines and big fans, the flow of the real fluid can be in many cases a laminar one. This flow takes place at small Reynolds numbers. [3]

The purpose of this work paper is to determine the parameters which affect the flow pattern within this micro clearance. The obtained results will allow us to control the appearance of the whirls and, therefore, to control the flow resistance within the micro clearances. The volumetric efficiency, accordingly, will be optimized to reach the highest values. Another methods implemented in [4] aimed also to increase the labyrinth seal efficiency by shape optimization.

2. The mathematical model

The new hypotheses considered for the flow equations of motion and mass conservation equation of the incompressible fluid, are:

- The flow has a permanent character, therefore $\partial/\partial t = 0$;
- The motion has an axial symmetry, therefore $\partial/\partial\theta = 0$;
- We neglected the gravity forces in comparison with that of pressure, friction and inertia, which predominate in this phenomenon of fluid flow.

Navier and Stokes equations system [5], written in cylindrical coordinates, take the former hypotheses in consideration and they are as the following:

- Three equations of permanent motion, suitable for the heavy and viscous liquid:

$$V_R \frac{\partial V_R}{\partial R} + V_Z \frac{\partial V_R}{\partial Z} - \frac{V_\theta^2}{R} + \frac{1}{\rho}\frac{\partial P}{\partial R} = \upsilon\left(\frac{\partial^2 V_R}{\partial R^2} + \frac{1}{R}\frac{\partial V_R}{\partial R} + \frac{\partial^2 V_R}{\partial Z^2} - \frac{V_R}{R^2}\right), \tag{1}$$

$$V_R \frac{\partial V_\theta}{\partial R} + V_Z \frac{\partial V_\theta}{\partial Z} + \frac{V_R V_\theta}{R} = \upsilon\left(\frac{\partial^2 V_\theta}{\partial R^2} + \frac{1}{R}\frac{\partial V_\theta}{\partial R} + \frac{\partial^2 V_\theta}{\partial Z^2} - \frac{V_\theta}{R^2}\right), \tag{2}$$

$$V_R \frac{\partial V_Z}{\partial R} + V_Z \frac{\partial V_Z}{\partial Z} + \frac{1}{\rho}\frac{\partial P}{\partial Z} = \upsilon\left(\frac{\partial^2 V_Z}{\partial R^2} + \frac{1}{R}\frac{\partial V_Z}{\partial R} + \frac{\partial^2 V_Z}{\partial Z^2}\right), \tag{3}$$

- Mass conservation equation of the incompressible fluid:

$$\frac{1}{R}\frac{\partial\left(R V_R\right)}{\partial R} + \frac{\partial V_Z}{\partial Z} = \frac{V_R}{R} + \frac{\partial V_R}{\partial R} + \frac{\partial V_Z}{\partial Z} = 0, \tag{4}$$

Through the finite differences method [6] we solved these equations and we obtained the algebraic relation of the streamline in the meridian plane for two cases one of them without cavity and the other with cavity.

$$\psi_0 \cong \frac{1}{160}\left\{\begin{array}{l}
64\sum_1^4\psi_i - 16\sum_5^8\psi_i - 8\sum_9^{12}\psi_i + \\
+\frac{2\chi}{r_0}\left[8(\psi_4-\psi_2)+2(\psi_5+\psi_6-\psi_7-\psi_8+\psi_{10}-\psi_{12})\right]- \\
-\frac{3\chi^2}{r_0^2}\left[4(\psi_2+\psi_4)-\frac{1}{4}(\psi_{10}+\psi_{12})\right]+\frac{3\chi^3}{r_0^3}\left[2(\psi_2-\psi_4)+\frac{1}{4}(\psi_{12}-\psi_{10})\right]+ \\
+\mathrm{Re}\chi\left\{\begin{array}{l}
v_0\left\{\begin{array}{l} 2(\psi_5+\psi_6-\psi_7-\psi_8+\psi_{10}-\psi_{12})+8(\psi_4-\psi_2)- \\ -\frac{3\chi}{r_0}\left[4(\psi_2+\psi_4)-\frac{1}{4}(\psi_{10}+\psi_{12})-\frac{15}{2}\psi_0\right]- \\ -\frac{2\chi}{r_0}\left[4(\psi_1+\psi_3)-\frac{15}{2}\psi_0-\frac{1}{4}(\psi_9+\psi_{11})\right]- \end{array}\right\}+ \\
+w_0\left\{\begin{array}{l} 2(\psi_5-\psi_6-\psi_7+\psi_8+\psi_9-\psi_{11})+8(\psi_3-\psi_1)- \\ -\frac{\chi}{r_0}\left[(\psi_5+\psi_7-\psi_6-\psi_8)-\frac{1}{16}(\psi_{13}+\psi_{15}-\psi_{14}-\psi_{16})\right]- \\ -\frac{3\chi^2}{r_0^2}\left[2(\psi_1-\psi_3)+\frac{1}{4}(\psi_{11}-\psi_9)\right]\end{array}\right\}- \\
-3u_0\chi^2(u_1-u_3)
\end{array}\right\}
\end{array}\right\} \tag{5}$$

We consider that the network step is constant, so $\chi = \delta z = \delta r$, and the Reynolds number: $\mathrm{Re} = R_i U_i / \nu$. Fig.1 presents the labyrinth seal with straight clearance without cavity (a) and with cavity (b)

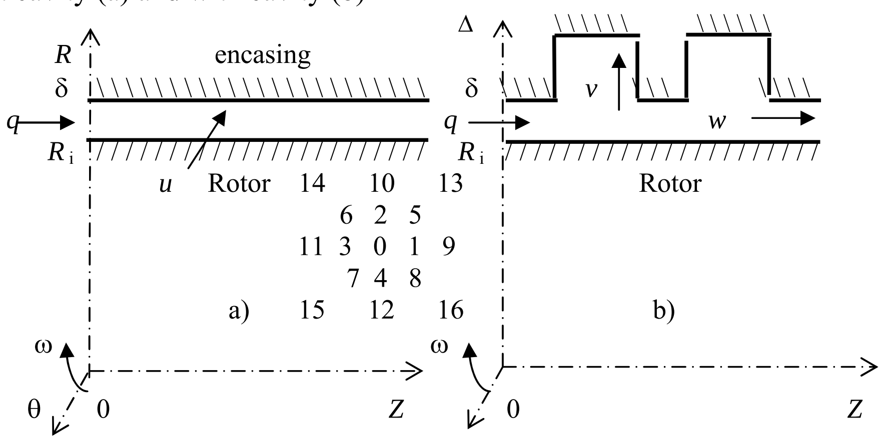

Fig.1. a) Labyrinth seal without cavity; b) Labyrinth seal with cavities.

After writing the initial and boundary conditions, we obtained the following results by using C++ program.

3. Results gained by using a C++ program:

a) For the clearance without cavities

We created a program in C++ environment to achieve the numerical integration of Navier-Stokes equations of motion for a viscous fluid. It can be noticed from analyzing the results the generation of Taylor's whirls, as it is illustrated in fig.2. However, the fact that the rotor is rotating, while the case is fixed, should be also taken in consideration. The fig.2 represents two whirls, each one of them has a different direction of rotation. [7]

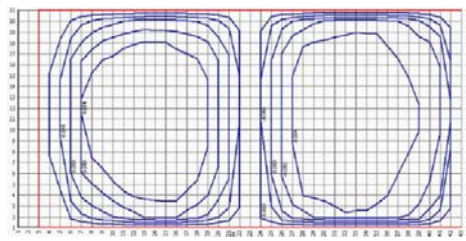

Fig. 2. The schematic representation of two Taylor's whirls between the two components: the rotor and the encasing (sectional view).

Fig.3 shows the schematic representations of a constant rotational velocity of the fluid (the component u), caused by the rotational motion of the rotor.

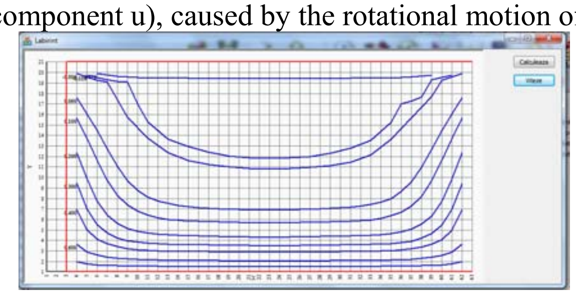

Fig. 3. The schematic representation of *u*.

b) For the clearance with cavities

The created program allows us to modify the geometric dimensions of the cavities of the labyrinth seal. The figures 4 and 5 show the streamline shapes for different widths and heights of the cavities (the cases a, b and c). Reynolds's number was considered Re = 1 and Re = 2 for Fig.4 and Fig.5, respectively.

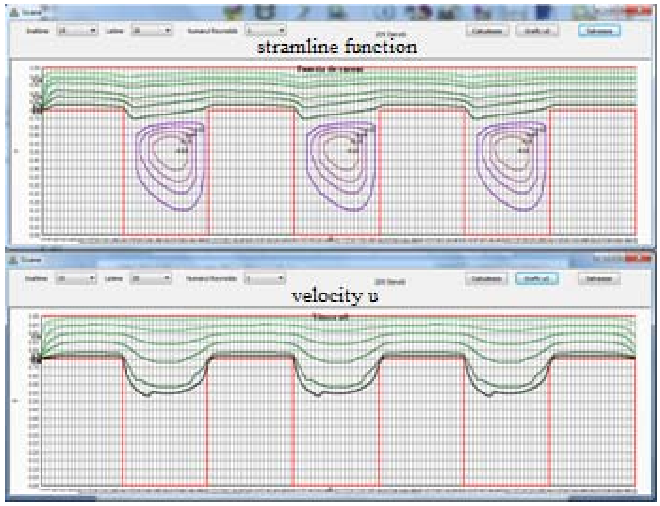

Fig. 4.a. Streamline illustration and velocity distribution when the clearance height equals 1/3 of the cavity height. (Re=1).

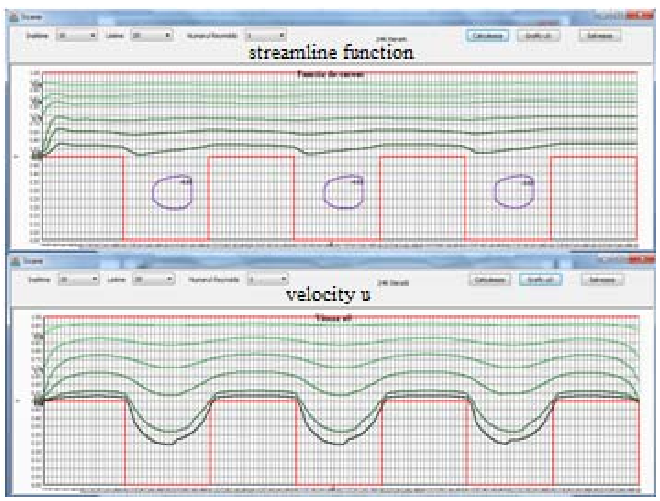

Fig. 4.b. Streamline illustration and velocity distribution when the clearance height equals the cavity height. (Re=1).

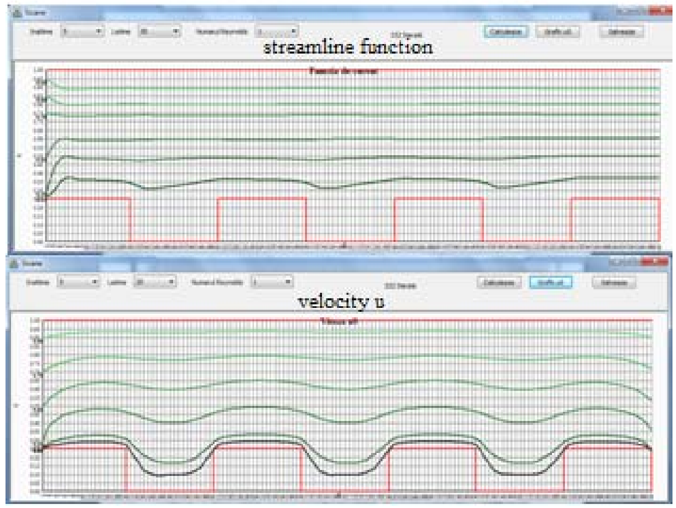

Fig. 4.c. Streamline illustration and velocity distribution when the clearance height equals three times the cavity height. (Re=1).

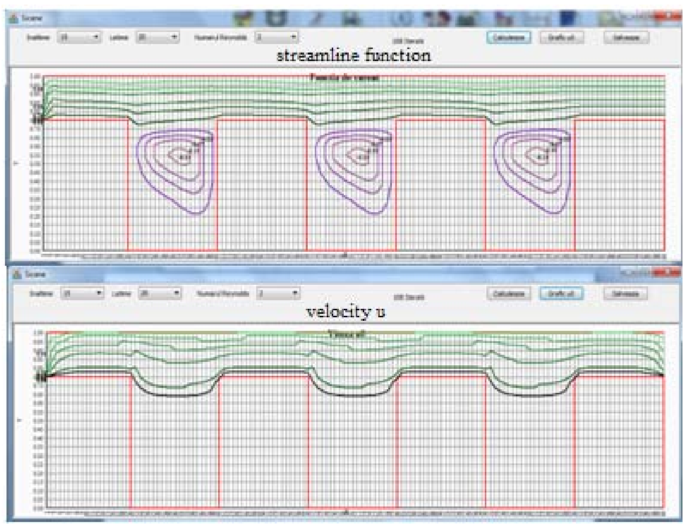

Fig. 5.a. Streamline illustration and velocity distribution when the clearance height equals 1/3 of the cavity height.(Re=2).

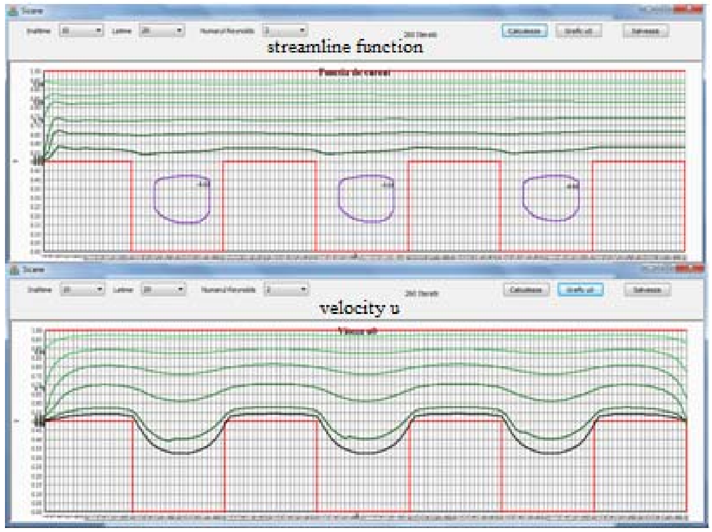

Fig. 5.b. Streamline illustration and velocity distribution when the clearance height equals the cavity height. (Re=2).

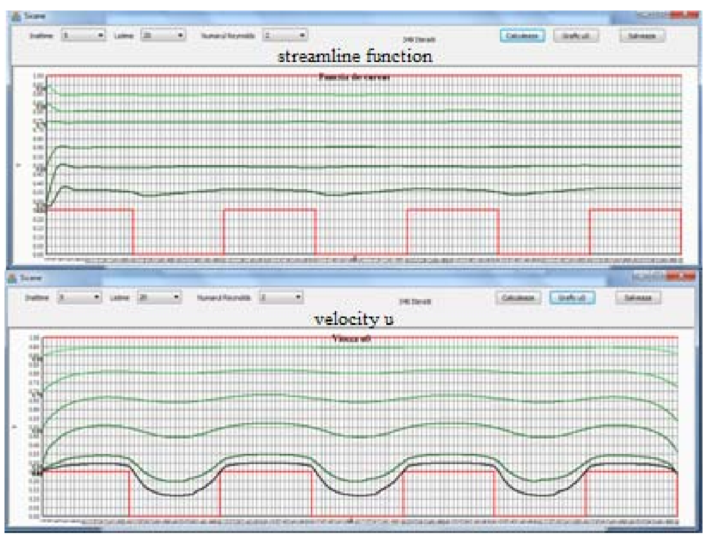

Fig. 5.c. Streamline illustration and velocity distribution when the clearance height equals three times the cavity height. (Re=2).

By studying and comparing the previous figures we can draw the following observations:

- The case (a) from fig.4 and fig.5 presents the best configuration of the labyrinth seal dimensions because it gives the opportunity for stronger vortexes to be generated. These vortexes, in their turn, resist the fluid flow and, consequently, they increase the volume efficiency of the centrifugal pumps;

- By increasing the clearance height and decreasing the cavities height, the generated vortexes diminished. Which lead to a smaller resistance of the fluid flow;
- When Reynolds's number is increased, the fluid flow velocity increases and the vortexes get a higher opportunity of appearance.

4. Results gained by using Ansys

For the same boundary conditions and hypothesis used in former stages of the study, the Ansys program was used for three different geometric models. Simulation stages depend on the type of the simulation performed, but they all have the same main processes. The simulation stages include the building of the geometric model, choosing the meshing method and details, determination of the boundary conditions and finally finding the solution and visualizing the gained results.[8], [9]

The dimensions of the three different geometric models used during this simulation can be seen in Fig.6. The depth and width of the cavity were modified to check the influence of these dimensions on the fluid flow shape. For the experiment 2 two different type of liquid were studied. Fig.8 a, and fig.8 b show the effect of the liquid properties on the fluid flow shape.

Geometry of the Experiment 1

Geometry of the Experiment 2

Geometry of the Experiment 3

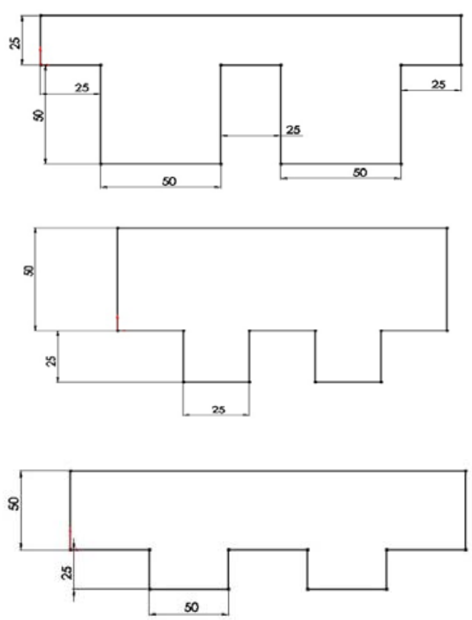

Fig. 6. Dimensions of three different geometric models.

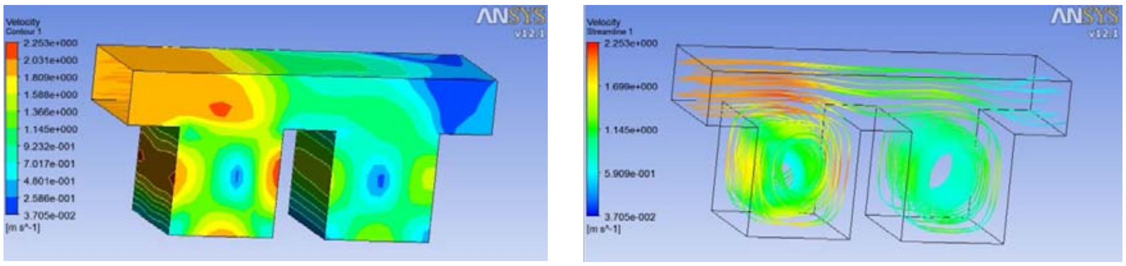

Fig. 7. Results of experiment 1

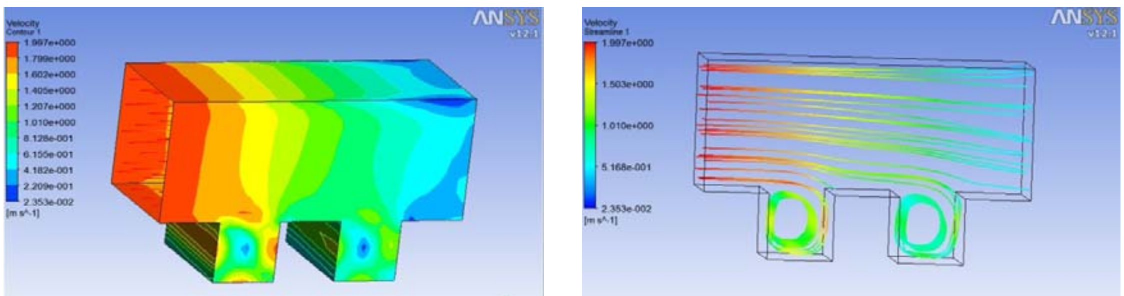

Fig. 8.a. Results of experiment 2 a.

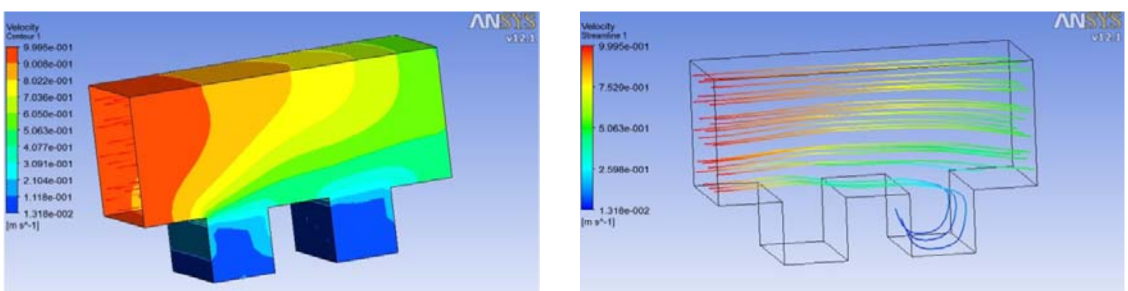

Fig. 8.b. Results of experiment 2 b.

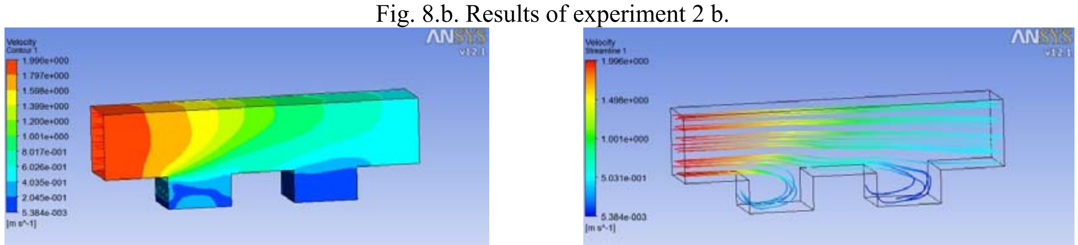

Fig. 9. Results of the experiment 3.

By analyzing the results obtained from the three different experiments in the figures fig.7, fig8.a, fig.8.b, fig.9 and taking the fig.6 into consideration, we can draw the following observations:

- The difference between the experiments 1 and 2 that we reduced the width and depth of the cavity, and increased the height of the clearance in the experiment 2 in comparison to those of the experiment 1. It can be noticed that the vortex in the experiment 2 has been diminished.
- The experiments 2 and 3 are different only in the width of the cavity. It can be noticed that the vortex in the experiment 3 is very weak and almost neglected.
- The only modification introduced in the experiment 2b compared to 2a is the properties of the liquid, where in 2b the liquid has a higher viscosity.
  We noticed that the viscosity of the liquid has an important role in determining the shape and strength of the vortex.

5. Results obtained by using Solid Works

During the simulation process performed in the program Solidworks, we used the same details and dimensions shown in the fig.6. The obtained results of this simulation confirmed the results obtained in the former simulation performed on Ansys (Article 4). Fig.10 presents the results of the experiment 1.

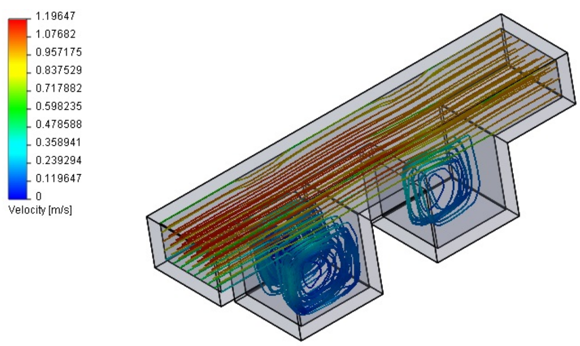

Fig.10. Results obtained by using SolidWorks.

6. Experimental studies on cavity Labyrinth Seals

Assessing the quality of a labyrinth seal is related decisively to the experimental observation of the spectrum of the hydrodynamic fluid flow in a meridian plane, with neglecting, in accepted approximation, the influence of the rotational movement of the rotor wall. The classical experimental researches [3], [5], [6] had the purpose to determine the properties of the flow through different forms of labyrinths in order to establish the conditions which related to the mathematical solving of the problem. These researches were performed in a device built specially for the study of the laminar motion of liquids.

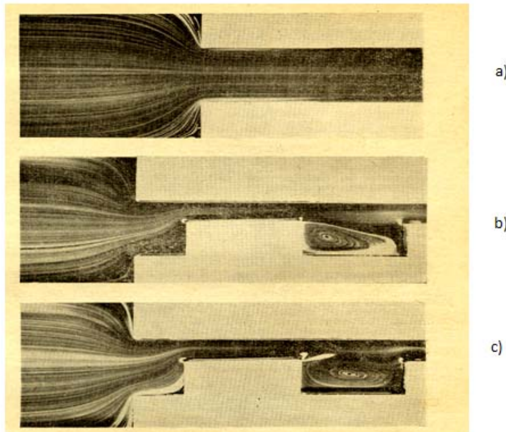

Fig. 11. Flow visualization through labyrinth seal obtained in a device for studying the viscous fluid flow at Re = 3,0 ÷ 33,4 (a), Re = 14,9 (b) and Re = 19,7 (c).

The equipment works with different viscosities of the oil in order to achieve hydrodynamic leakage similar to reality. The equipment has models of labyrinth seal with geometric dimensions higher than the real dimensions in the hydrodynamic machines, $\delta / \delta' = 100 - 1000$, to ease the viewing of the flow. Such labyrinth seals were presented to illustrate the hydrodynamic fluid flow through the straight labyrinth seal for Reynolds's number in range 3.0 - 33.4 (fig. 11.a) and through cavity labyrinth seal for Reynolds's number Re = 14.9 (fig. 11.b) and Re = 19.7 (fig. 11.c).

7. Conclusions

In addition to the observations we mentioned above, some general conclusions of this work paper can be drawn. Firstly, the experimental results and the results obtained by using three different simulation methods were compared. It can be noticed that the four methods used during this research gave similar results. The measured values were compared with those determined analytically. The performed experiments allowed the validation of the mathematical model, assumptions and boundary conditions formulated in C++, ANSYS, and Solid Works programs. Secondly, the final results were then analyzed to study the parameters which affect the fluid flow properties.

The results showed that the width and depth of cavity affect the form of vortex inside it, consequently, affect the performance of the labyrinth seal. Under different labyrinth seal designs, there are different intensities of turbulent whirls within the path. The flow resistance patterns and energy dissipation patterns are also different [6]. The geometric dimensions should be designed to increase the internal cavity turbulence so that the leakage will be reduced. That can be done by choosing the optimal size and shape of the cavity of the labyrinth seal and considering the optimal height in the straight section of the annular passage. On the basis of the conclusions mentioned above, we can anticipate the tendencies of the future research and designs, generally for hydraulic machines, and especially for labyrinth seals.

REFERENCES

[1] *X.H. Hao, Y.L. Ju , Y.J. Lu*, Experimental study on the performance of the labyrinth sealing displacer for 10 K G-M refrigerator, Institute of Refrigeration and Cryogenics, Shanghai Jiao Tong University, Shanghai 200240, China, article accepted 27 January 2011.

[2] S.*P. Asoka, K. Sankaranarayanasamy, T. Sundararajan, G. Vaidyanathand, K. Udhaya Kumare,* Pressure drop and cavitation investigations on static helical-grooved square, triangular and curved cavity liquid labyrinth seals, Elsevier, Dec. 2010.

[3] *D.Dumitrescu, M.D.Cazacu,* Studiul teoretic şi experimental al curgerii lichidelor reale prin labirinţii turbomaşinilor hidraulice. Comunicările Conferinţei de Maşini Hidraulice, Timişoara, Sept.1964, Vol. 3, p. 117 – 131, (în ediţia germanã, Theoretisches und experimentelles Studium der Strömung reeller Flüssigkeiten durch die Abdichtungslabyrinthe der Strömungsmaschinen. Mitteilungen der Konferenz für Wasserkraftmaschinen, B.III, S.127-141), Theoretical and experimental study of the actual liquid flow through hydraulic turbo labyrinths. Communications Conference, Hydraulic Machines (in Romanian)

[4] *V.Schramm; J.Denecke; S.Kim; S. Witting*, Shape optimization of a labyrinth seal applying the simulated annealing method, Taylor&Francis Inc, 2004.

[5] *Th.Oroveanu*, Mecanica fluidelor vascoase. Editura Academiei, Bucureşti, 1967 (p.38 and 25).

[6] *D.Dumitrescu, M.D.Cazacu*, "Theoretische und experimentelle Betrachtungen über die Strömung zäher Flüssigkeiten um eine Platte bei kleinen und mittleren Reynoldszahlen". Zeitschrift für Angewandte Mathematik und Mechanik, **1**, 50, 1970, 257- 280.

[7] *M.D. Cazacu, Moutasem Ibrahim Alkhatib,* Viscous liquid three dimensional flow through the turbo-machines labyrinths, AGIR, Stiinţă si Inginerie a XII Conf. Natională, Science and Engineering, 12th National Conference, Sebes, Vol. 22, pp 1-12, 2012.

[8] *Yahia Zakaria*, A Static Wheel-rail Interaction Analysis by Using ANSYS Program, Sinteze de mecanica teoretica si aplicata, (Synthesis of theoretical and applied mechanics), Matrixrom, Bucharest, 2013. (in Romanian)

[9] *Alexey N.Kochevsky*, Possibilities of simulation of fluid flows using the modern CFD software tools, Sumy, Ukraine., arxiv.org/pdf/physics/0409104