CONTRIBUTION TO THE FLUID PROPERTIES ANALYSIS OF HEZ-HYDROMOTOR - COMPUTER MODELLING

Bohumil TARABA

Author: ***Bohumil Taraba, Assoc. Prof., Ph.D.***
Workplace: ***Slovak University of Technology in Bratislava, Faculty of Materials Science and Technology in Trnava, Institute of Production Systems and Applied Mechanics***
Address: ***Rázusova 2, 917 24 Trnava, Slovak Republic***
E-mail: ***bohumil.taraba@stuba.sk***

Abstract

The article is focused on the analysis of parameters of the water flowing through the space between the rotor and stator in hydraulic motor - hybrid energy source (HEZ). Two geometrical pairs were analyzed: half sphere rotor in the conical stator and exponential shape rotor in the combination with graduated conical stator. For both cases, fluid simulation models were developed. Analyses were targeted at the prediction of velocity and pressure fields in water stream inside the pair. Interpretative computing software used was the ANSYS -Flotran CFD. The evaluation of obtained results can help understand the context with function of hydraulic motor and formulate the motor function hypothesis.

Key words

ANSYS CFD, computer modelling, fluid analysis, hydromotor, SETUR

Introduction

HEZ-hydromotor is classified as SETUR turbine [1]. Bladeless rotor is inserted in a bladeless stator. The flowing fluid is a liquid - water. The interaction effect of wrapping water on rotor and stator produces rotational movement of the rotor. The torsion moment originates on the rotation axis that is utilized as an energy source for selected appliances. The analysed bladeless hydromotor has been patented in several countries in the world [2]. An accurate analysis of the phenomena in water-flow cross-section rotor-stator pair with the principles of physics has not been published yet. The aim of the article is to obtain applied results of computer modelling for the formulation of a hypothesis of the rotation movement formation in the rotor axis.

Theoretical background

Generally, isothermal fluid problems require solving the equation of mass conservation, Navier-Stokes equation and Bernoulli's equation. To meet the objectives set out in the article, only the Bernoulli's equation is applied. The total specific energy balance for steady-state flow of an incompressible fluid in two places of calculated area 1-2 is described by the Bernoulli's equation for real liquids [3]

$$gz_1 + \frac{p_1}{\rho} + \frac{w_1^2}{2} = gz_2 + \frac{p_2}{\rho} + \frac{w_2^2}{2} + e_z, \qquad [\mathrm{J.kg^{-1}}] \qquad (1)$$

where g is the acceleration due to gravity [m.s^{-2}], z is the elevation [m], p is the pressure [Pa], w is the velocity of fluid [m.s^{-1}], ρ is the density [kg.m^{-3}] and e_z is the specific energy loss [J.kg^{-1}].

Specific energy loss exists on a local resistance in the place of water discharge from the hydromotor. It shall be determined from the relationship

$$e_{z\,loc} = \xi \frac{w^2}{2}, \qquad [\mathrm{J.kg^{-1}}] \qquad (2)$$

where ξ [-] is the local loss coefficient and w is the fluid velocity in a flow-through section.

Numerical simulations contain no specified data of the loss coefficient. The local losses in numerical fluid calculations represent a part of the calculation procedure in the form of dissipation energy.

The Reynolds number for the fluid flow in a flow-through section is calculated by the formula

$$Re = \frac{w\,L}{\nu}, \qquad [-] \qquad (3)$$

where ν [m^2.s^{-1}] is the coefficient of kinematic viscosity of water and L is the characteristic dimension. For the laminar flow, Re < 2320. The value of Re higher than 2320 is the reason for setting the turbulent calculation procedure.

Simulation model

Computer modelling of fluid flow in the hydromotor is an informational experiment. The fluid task problem is analysed deterministically using the numerical method of finite element. Model solution is implemented through the ANSYS-Flotran CFD program code [4]. The simulation model is considered as a dynamic fluid system in which variables vary in space and time. Stochastic phenomena are neglected.

The problems are identified as an isothermal discharge of liquid from the reservoir through the space between the rotor and stator (liquid discharge through a small hole into a free environment). The water flows due to the gravity effect. Water level above the inlet to the space between the rotor and stator is 1 m. The water level is stable. Flowing water is considered as the Newtonian fluid with the temperature of 18 °C, the density of 998.5 kg.m^{-3} and the dynamic viscosity coefficient of 1.06.10^{-3} Pa.s [3].

Geometrical model

Two different geometrical pairs of stator and rotor were analysed and compared. In Figure 1, the pair where the rotor has a shape of a half sphere and the stator is made as a cone (model 1) is shown. Figure 2 illustrates an alternative model. The rotor diameter varies exponentially according the formula R= 0.07097−0.05874exp(−y/0.04522) [m] and the stator is a cone with gradual change of diameter (model 2). The basic dimensions are shown in F1 and 2.

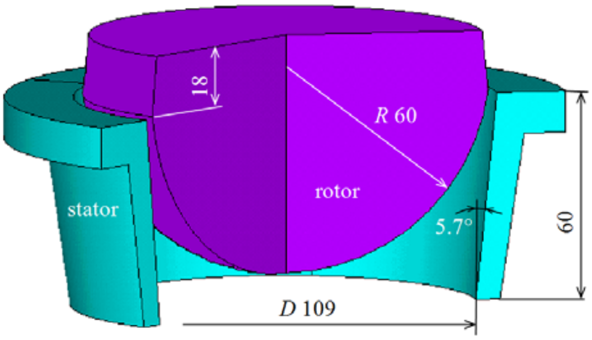

Fig. 1 *Hydromotor with the shape of half sphere rotor and cone stator*

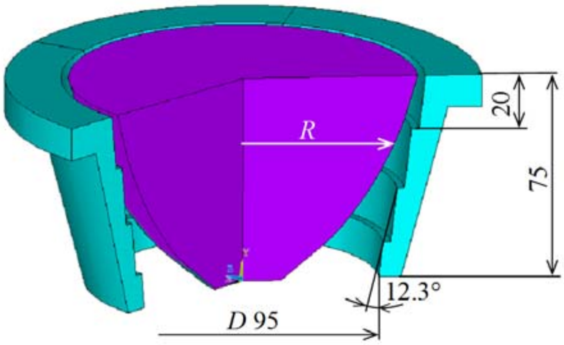

Fig. 2 *Hydromotor with exponential rotor and gradual stator*

Model of hydromotor with a reservoir of water is shown in Figure 3. Water reservoir is a cylindrical vessel with the diameter of 0.3 m and the height of 1 m. Model was simplified into a two-dimensional task, since the presented computer modelling was focused only on the prediction of velocity and pressure fields. The higher level of problem solution represents the development of 3D models in the future.

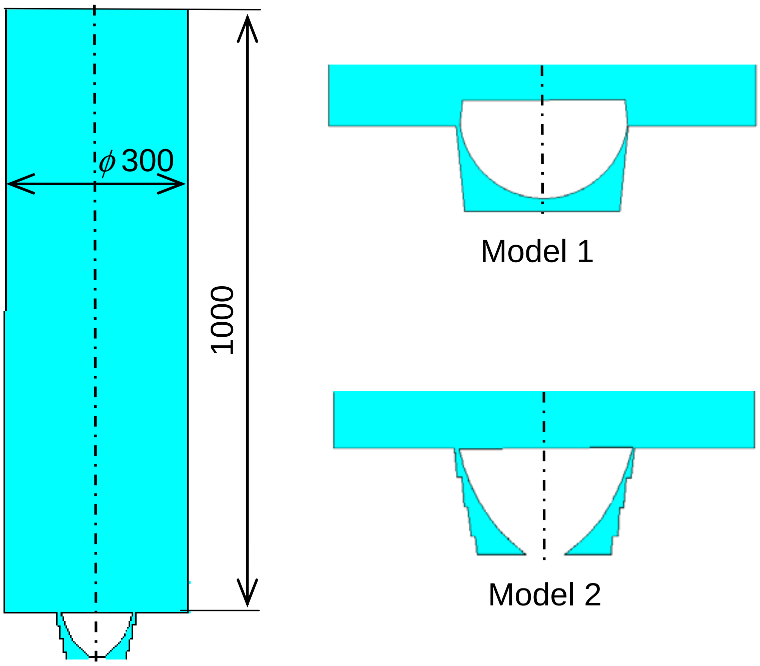

Fig. 3 *Geometrical model of hydromotor*

The mesh was generated from the elements of FLUID141 type. Surface of the reservoir was meshed using the elements with the length 0.025 m, mesh surface of flow area under the reservoir bottom was generated with the default element size 0.001 m and in the area of water discharge between the rotor and the stator. It was necessary to refine the mesh at least twice.

Calculating procedure

The investigated process is a quasi-steady. This means that the process could be modelled as a steady (steady-state) phenomenon. It is not possible to consider the fact that water outlet of the hydromotor fills the entire outlet section and the analysis cannot have the character of flow in classical piping systems.

Finally, the process analysis applies the computational procedure of VOF ANSYS-Flotran-CFD type (Volume Of Fluid). This means that the reservoir above the rotor at the time t = 0 seconds is fully filled and, with increasing time, the water flows through the space between rotor and stator into free enviroment. The modelling of the water flow through the hydromotor was accomplished by the computer procedure: Transient-Adiabatic-Turbulent-Flow-Incompressible and with the activation of VOF. The Reynolds value was calculated to be Re = 12600.
The boundary conditions for the calculation were defined at hydromotor's walls: $w_x = w_y = 0$ m.s^{-1}. The free surface under rotor and stator is loaded with the pressure p = 0 Pa. The upper surface of the reservoir was not loaded.

Obtained results

The results of analyses were selected in order to understand the causes of rotor movement - velocity vector fields and the particles flow traces for the chosen outlet time of 3 seconds. The obtained results are the graphical dependencies of pressure on the rotor surface. Figures 4 and 5 show the vector of the velocity fields [m.s^{-1}] for model 1 and model 2.
It is typical for both analysed pairs that the water flow keeps to the rotor surface and is detached at the rotor bottom. This means that the water flowing like a viscous liquid is able to pass its momentum to the rotor (deviated from the vertical axis y), and the torque moment is created on the rotor shaft.

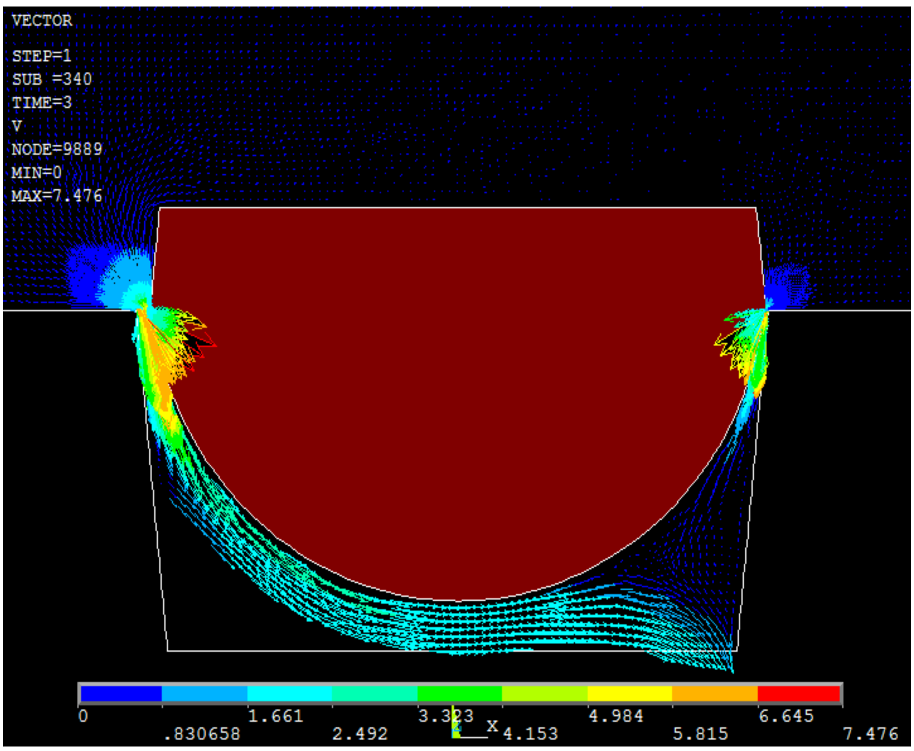

Fig. 4 *Velocity vector field [m.s^{-1}] in the time of 3 sec. Model 1- half sphere rotor*

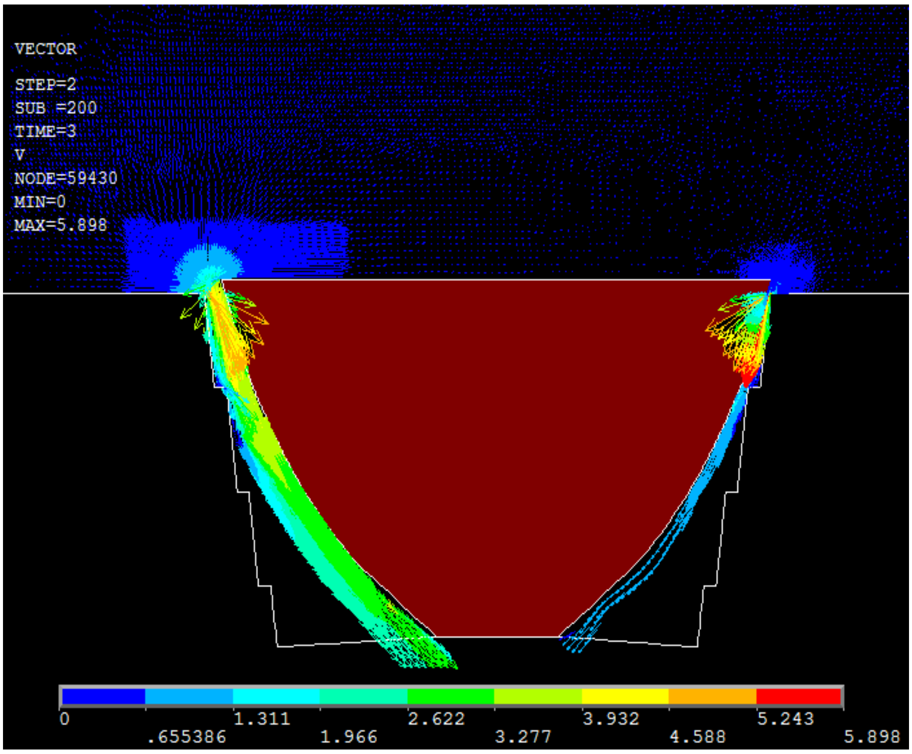

Fig. 5 *Velocity vector field [m.s^{-1}] in the time of 3 sec. Model 2 - exponentially varied rotor*

In Figures 6 and 7, the velocity fields of chosen particles are shown. The velocity fields are of a local character (area with the dimension about 10 mm). Water is accelerating prior to entering the outlet section.

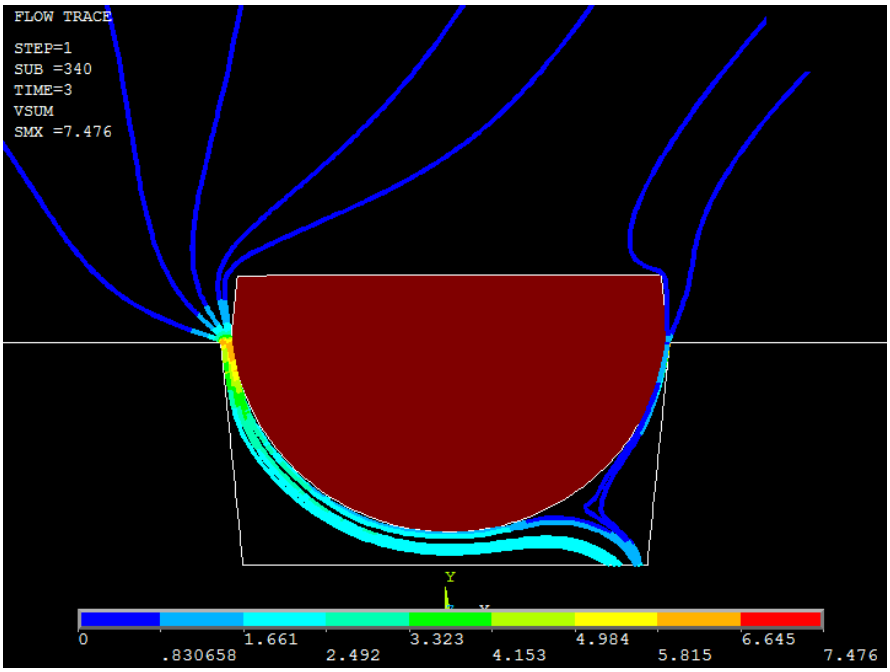

Fig. 6 *Particle flow traces in the time of 3 sec., velocity [m.s^{-1}]. Model 1*

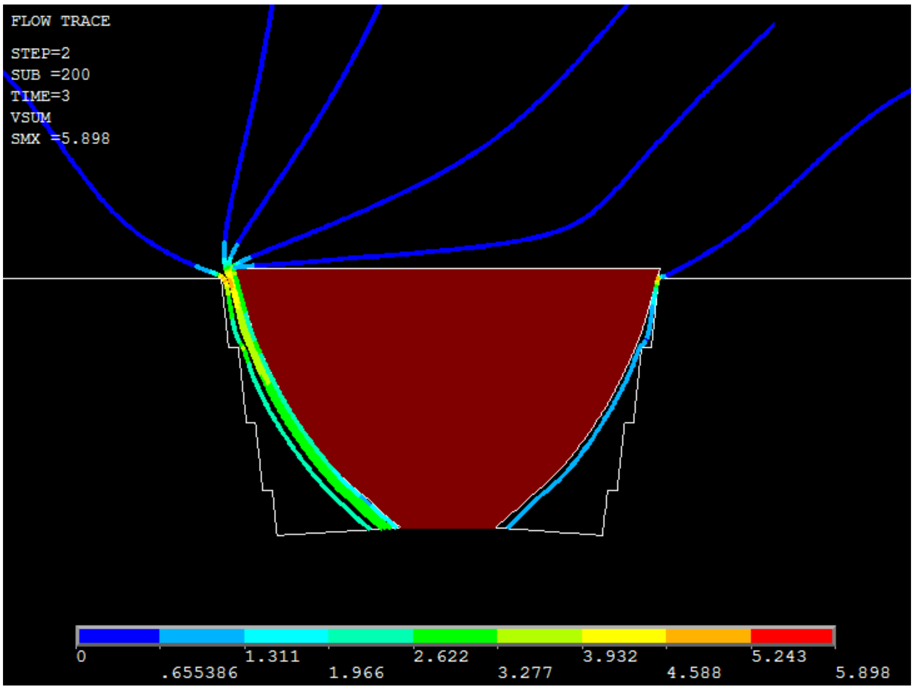

Fig. 7 *Particle flow traces in the time of 3 sec., velocity field [m.s^{-1}]. Model 2*

The shape of the rotor-stator pair affects the velocity field and influences the underpressure formation as shown in Figures 8 and 9. As follows from simulation for the gradated stator the stepped change in diameter has a positive influence on the underpressure development (Figure 9).

According to Bernoulli's equation (1), the underpressure is set on both sides of the rotor. The difference is in value of underpressure and the length (actually surface) on which the effects of pressure changes occur. The result represents the difference of pressure forces. According to the second Newton's law, the resulting force accelerates the rotor towards the lower pressure area. Owing to the inertia, the rotor comes closer to the opposite side of the stator, the pressure value is changed and the contact with the wall of the stator occurs. Consequently, the rotor comes back and along with the rotation.

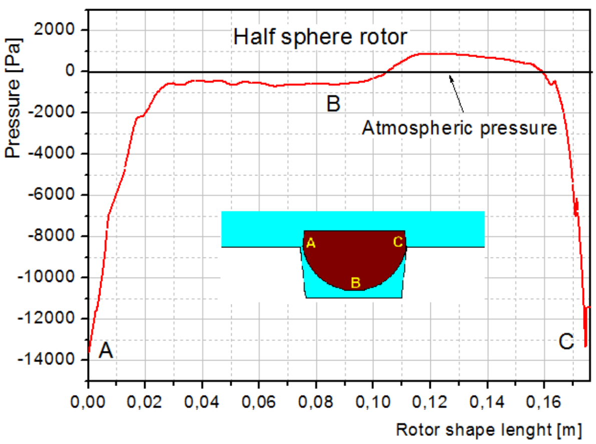

Fig. 8 *Dependence of the pressure on the rotor surface. Model 1*

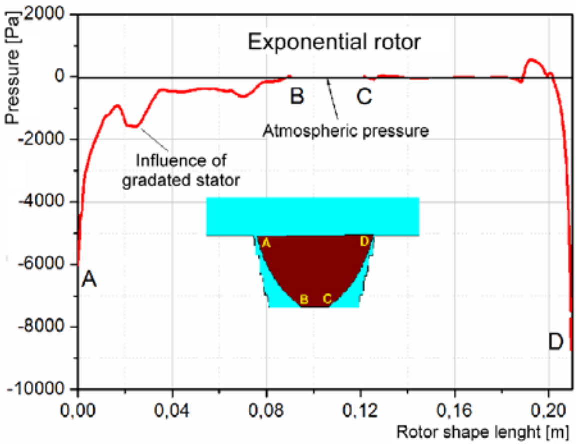

Fig. 9 *Dependence of the pressure on the rotor surface. Model 2*

Conclusion

Evaluation of the results for the presented numerical analyses allows formulating the following hypothesis and conclusions.

The hydraulic motor of a SETUR type is a rotating pendulum moving in a viscous fluid environment within the walls of the stator. The driving rotor vibration force is a periodic change of direction and the size of the force resulting from the pressure field on the rotor surface. Tangential force component arises on the rotor surface through viscous fluid flow braking.
- HEZ-hydromotor is the action and, under pulse flow conditions, a bladeless turbine. It consists of complex fluid-dynamic and mechanical-dynamic systems.

Acknowledgements

This contribution was written with the support of the Operational Programme Research and Development within the for project: “Hybrid power source for technical and consulting laboratory use and promotion of renewable energy sources” (ITMS 26220220056), financed from resources of the European Regional Development Fund.

References:

1. Princip turbiny SETUR a základní možnosti využití. Principle of SETUR turbine and its basic utilisation. Available on the Internet: http://www.setur.cz/
2. Miroslav Sedlacek et al. : Fluid machine. Available on the Internet: http://www.google.com/patents/US6139267
3. TARABA, B., BEHÚLOVÁ, M., KRAVÁRIKOVÁ, H. *Mechanika tekutín. Termomechanika: Zbierka príkladov. Fluid Mechanics. Thermodynamics and Heat transfer: Exercises.* Trnava: AlumniPress, 2007. 242 p. ISBN 978-80-8096-021-6
4. ANSYS Theoretical Manual (2012). Available on the Internet: http://www.pdfqueen.com/pdf/an/ansys-10-users-manual/10/

Reviewers:

Jozef Krchnár, Assoc. Professor, Ph.D. – Faculty of Mechanical Engineering STU in Bratislava

Mária Behúlová, Assoc. Professor, Ph.D. – Institute of Production Systems and Applied Mechanics, Faculty of Materials Science and Technology STU in Trnava