UDC 532.5:621.65.01

A. L. SHUDRIK

USING OPEN SOFTWARE APPLICATION PACKAGES FOR SIMULATION OF VISCOUS INCOMPRESSIBLE FLUID

This paper analyzes the basic principles, advantages and disadvantages of an open package OpenFOAM. Class hydrodynamics tasks are listed. Possibilities and prospects of further use of OpenFOAM accessible using a number of turbulence models in the study of the working process in hydraulic machines are shown. The calculations of test tasks of turbulent flow of viscous fluid in Newton's channels of variable section, the sudden expansion in the impeller vane pump. Results in the form of pressure fields (velocities), the integral defined by their characteristics are presented. An experiment on the stand of the Department "Hydraulic Machines." The comparison of results obtained in packages OpenFOAM, ANSYS CFX and experiments.

Keywords: OpenFOAM, computational fluid dynamics, viscous fluid flow, turbulence model, mathematical model, mesh.

Introduction. The using of computer modeling allows creating high-tech products to improve performance, while reducing the time and cost of development and testing of new products.

Currently, the computational fluid dynamics software (CFD) has received immense popularity and widespread in scientific research. All CFD packages use a typical procedure for constructing an engineering problem (creating geometry, meshing, definition of boundary conditions) and its solution (the choice of solver, start the calculation and processing of the results).

At the moment, there are a number of universal commercial software products for solving CFD tasks, for example, **ANSYS CFX**, **Simulia**, **Fluent**, **Star-CD** and others. License costs of these products is very high, but there are alternative to commercial packages – open source software.

Open source software is software, which is available to view, explore and change. The use of such software requires more user skill but allows you to use the code to correct the most open programs, and to create new programs.

Among the packages available for CFD tasks are best known: OpenFOAM, Elmer [1]. The main disadvantages of these packages can be noted unexamined interface and lack of full documentation to the user. However, the possibility of the use of such packages on any number of processors without any financial cost for the purchase of these packages makes very attractive for small businesses and schools. In connection with the above, it is possible to note the relevance of this work.

OpenFOAM (Open Source Field Operation And Manipulation CFD ToolBox) is an open platform for integrable numerical modeling of continuum mechanics tasks. Numerical method, embedded in the code based on the finite volume method for unstructured meshes. The package is a separate, independent modules. The visual development environment and a library implemented in object-oriented programming language C ++. Very large in terms of the package, it is designed primarily for hydrodynamics tasks. Today is one of the «finished» and well-known applications for CFD tasks.

This package allows to solve hydrodynamics tasks for Newtonian and non-Newtonian viscous fluids, the compressibility and incompressibility, the convective heat transfer and the force of gravity. For the simulation of turbulent flows can be used RANS-models, LES- and DNS-methods. Perhaps the solution subsonic, transonic and supersonic tasks.

Objective. To analyze the basic principles and possibilities of working with **OpenFOAM** for the simulation of turbulent flow of viscous fluid in the channels of hydraulic machines, compare the solution of some taskss with the test solution in **ANSYS CFX** [2] and the experimental data.

Mathematical model. The hydrodynamic test calculations made using one of the included in the package **OpenFOAM** solvers. In this paper, we tested solver simpleFoam, various numerical approximation schemes and integration, and a variety of turbulence models.

The mathematical model (MM), laid the foundation solver simpleFoam, based on solving Reynolds-averaged Navier-Stokes equations, which are closed by means of a turbulence model.

Currently, a large number of different models for the calculation of turbulent flows. They differ in complexity and solution accuracy of the description of the flow. However, the most successful in the calculation of incompressible viscous fluid has proved to k-ε turbulence model. Using this model, the system of fluid motion equations (1) is complemented by a system of differential equations (2), respectively, describing the transfer of turbulent kinetic energy k and the dissipation rate ε [4–6]:

$$\begin{cases} \dfrac{\partial \rho}{\partial t} + \dfrac{\partial}{\partial x_j}\left(\rho u_j\right) = 0; \\ \dfrac{\partial}{\partial t}\left(\rho u_i\right) + \dfrac{\partial}{\partial x_j}\left(\rho u_i u_j\right) + \dfrac{\partial}{\partial x_j}\left(\rho u_i' u_j'\right) = \\ = -\dfrac{\partial p}{\partial x_i} + \dfrac{\partial}{\partial x_j}\left[\mu\left(\dfrac{\partial u_i}{\partial x_j} + \dfrac{\partial u_j}{\partial x_i}\right)\right] + f_i, \end{cases} \quad (1)$$

$$\begin{cases} \dfrac{\partial}{\partial t}\left(\rho k\right) + \dfrac{\partial}{\partial x_j}\left(\rho \overline{u_j} k\right) = \dfrac{\partial}{\partial x_j}\left(\Gamma_k \dfrac{\partial k}{\partial x_j}\right) + P_k - \rho\varepsilon; \\ \dfrac{\partial}{\partial t}\left(\rho \varepsilon\right) + \dfrac{\partial}{\partial x_j}\left(\rho \overline{u_j} \varepsilon\right) = \dfrac{\partial}{\partial x_j}\left(\Gamma_\varepsilon \dfrac{\partial \varepsilon}{\partial x_j}\right) + \\ + \dfrac{\varepsilon}{k}\left(C_{\varepsilon 1} P_k - \rho C_{\varepsilon 2}\varepsilon\right), \end{cases} \quad (2)$$

where $P_k = -\rho\overline{u_i' u_j'}\dfrac{\partial \overline{u_i}}{\partial x_j}$ – term expressing the generation of energy k, $\Gamma_k = \mu + \dfrac{\mu_t}{\sigma_k}$, $\Gamma_\varepsilon = \mu + \dfrac{\mu_t}{\sigma_\varepsilon}$.

Dissipation rate parameters ε. and turbulent viscosity μ_t determined by the expressions: $\varepsilon = \dfrac{\mu}{\rho}\overline{\left(\dfrac{\partial u_i'}{\partial x_j}\right)^2}$, $\mu_t = \rho C_\mu \dfrac{k^2}{\varepsilon}$, $C_\mu = 0{,}09$, $C_{\varepsilon 1} = 1{,}44$, $C_{\varepsilon 2} = 1{,}92$, $\sigma_k = 1{,}0$, $\sigma_\varepsilon = 1{,}3$.

Also it is necessary to note, that for realization of package OpenFOAM it is necessary all boundary conditions and parameters of turbulence on all surfaces to write down manually. On the one hand this inconvenience, with another – universality for various MM.

For example, for calculations of values k and ε it is necessary to use the following formulae [1]:

$$k = \frac{3}{2}(UI)^2, \tag{3}$$

$$\varepsilon = \frac{C_\mu^{0,75}\cdot k^{1,5}}{l}, \tag{4}$$

where U – velocity vector;

I – turbulence intensity;

l – scale of turbulence.

Realization of MM in OpenFOAM.

1) Construction of settlement area, the task of boundary conditions.

At the first stage the three-dimensional model of a liquid of considered area is under construction. Further it is exported to format **PARASOLID** for construction of a finite-element mesh in generator ICEM CFD. Discretization of system of the equation of movement in given computing package OpenFOAM is made on finite-volume method (FVM). Localization of discrete values of speed and pressure is carried out in the centers of cells of the constructed settlement mesh. Then the mesh is kept in a format *.msh.

The next is an import of a mesh in OpenFOAM using a command **«fluent3DMeshToFoam»**.

After that, set the initial boundary conditions. In OpenFOAM they are in the directory «0». At the inlet the normal speed is defined in the computational domain, the output – static pressure. For rotate computational domain is defined by the speed in **MRFProperies** file.

In the process of calculating the automatically generated temporary directory where the results of the iterative calculations are stored. The general structure and purpose of the main settlement OpenFOAM file is presented in table 1.

Table 1 – The structure and purpose of calculation of basic files in OpenFOAM

Calculated task		
0	Directory with the initial boundary conditions	
	epsilon	initial condition ε
	k	initial condition k
	p	pressure initial value
	U	velocity initial value
constant	The directory containing the computational mesh, the parameters of the environment and turbulence	
	polyMesh	directory with computational mesh
	MRFProperies	rotating fields settings
	transportProperties	physical properties of the liquid
	turbulenceProperties	turbulence model
system	Directory with parameters of solver and results processing file	
	controlDict	solver and specify its parameters
	fvSchemes	setting numerical integration schemes
	fvSolution	monitoring the progress of solving the task
	sampleDict	sections of profiles that define the necessary parameters

2) Run the calculation. For the tasks of the turbulent flow of viscous incompressible fluid in steady formulation used solver **simpleFoam**. The computational process occurs before the set value discrepancies.

3) Processing and visualization of results.

For processing the values of the calculated values (pressure, velocity) used sample utility. The directory system is placed sampleDict file. It contains the profiles of cross sections or points where it is necessary to define these parameters.

For visualization of the fluid flow is used free package ParaView (fig. 1). This package provides the user with the possibility of interactive visualization and analysis of large amounts of data for qualitative and quantitative analysis.

Solving typical tasks.

The paper discusses three test items:

1) The flow in the channel of variable section (D_{max}/d_{min} = 2,1; L = 7,25D_{max}).

2) The flow in the channel with sudden expansion of the pipeline (D/d = 2; L = 9D). Analytically and numerically determined coefficients of local resistance.

3) Calculation of the impeller (D_1/D_2 = 0,46; b_2/D_2 = 0,0685).

In tasks 1, 2, 3, test results were compared with calculations obtained in Ansys CFX package [2]. For task 1 is further carried out an experiment on the stand of the Department "Hydraulic Machines" Results.

Task 1. Fig. 2 matched visually flow channel of variable section with packages – Ansys CFX and OpenFOAM.

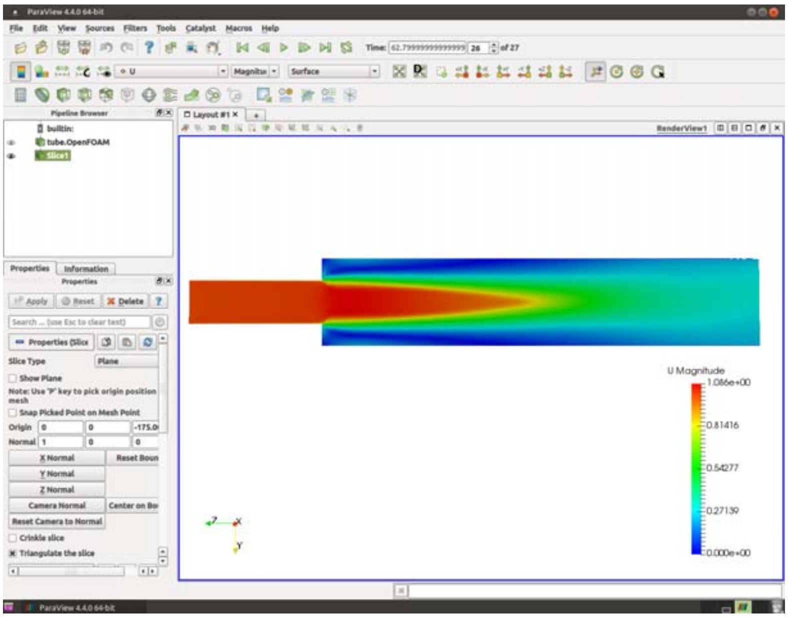
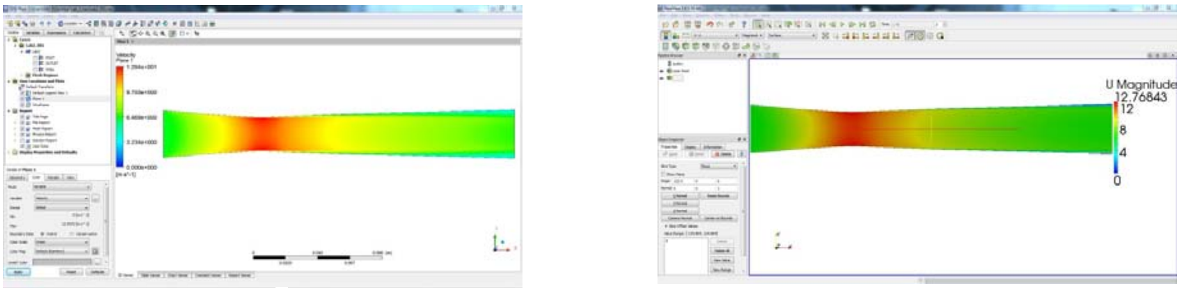


Fig. 1 – Interface ParaView package (Task 2 – sudden expansion)



a *b*

Fig. 2 – Flow visualization when using packages:
a – Ansys CFX; *b* – OpenFOAM

Fig. 3 shows a comparison of the distribution of static pressure $H_{st} = P_{i,j}/\rho g$ along the channel resulting from the physical experiment and packages – Ansys CFX and OpenFOAM.

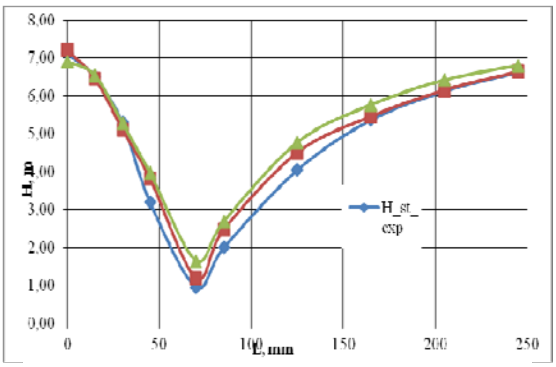


Fig. 3 – Diagram of static pressure along the channel
(blue – experiment; red – Ansys CFX; green – OpenFOAM)

Task 2. To determine the loss ratios used the classic formula of hydraulics:

$$h_{se} = \xi_{se} \frac{(V_1 - V_2)^2}{2g}, \tag{5}$$

$$\xi_{se} = \left(1 - \frac{F_1}{F_2}\right)^2, \tag{6}$$

$$\xi_{se} = \frac{2gH_{se}}{V_1^2}. \tag{7}$$

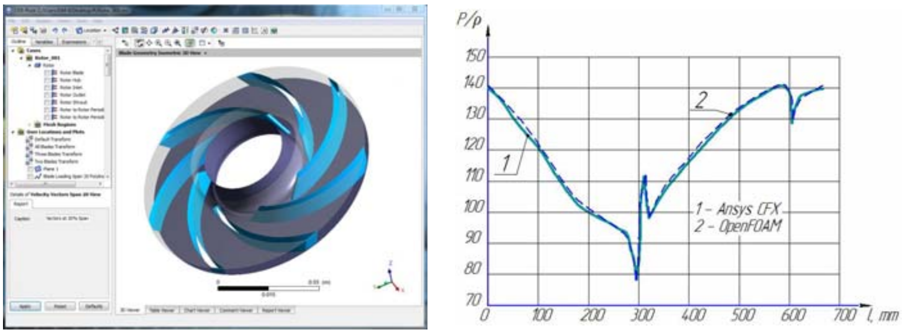
The following values (table 2) were obtained by numerical calculations.

Table 2 – Results of calculation of loss and the coefficient of resistance

Method of calculation	h_{se}, m	ξ_{se}
Theoretical	0,0287	0,5625
Ansys CFX	0,0292	0,5728
OpenFOAM	0,2905	0,5733

The error between accrued and experimental values is not more than 3 %.

Task 3. Visualization of flow in the impeller in packages Ansys CFX and OpenFOAM is presented in fig. 4.



a *b*

Fig. 4 – The pressure distribution along the blade:
a – solid model; *b* – pressure distribution

Conclusions. This paper analyzes the basic principles, advantages and disadvantages of an open package **OpenFOAM**. The calculations of test tasks of turbulent flow of viscous Newton's fluid in channels of variable section, the sudden expansion in the impeller vane pump. In consequence of fairly good agreement of the experimental results, the calculation in **Ansys CFX** package (which proved itself quite well) and an open packet of **OpenFOAM**, you can make a conclusion about the adequacy of CFD solutions of tasks in the software product.

In view of the fact that the software is open and it is possible to create your own solver, in perspektiveetot package can be used for research of hydraulic machines with their work on the gas-liquid mixtures, to take into account the surface roughness of the working bodies.

References: **1.** *Greenshields Christopher J.* OpenFOAM The Open Source CFD Toolbox. User Guide / *Christopher J. Greenshields.* – Режим доступа : www.openfoam.com. – Дата обращения : 21 января 2016. **2.** *Академическая версия программной системы конечно-элементного анализа ANSYS.* – Режим доступа : www.ansys.com/Student. – Дата обращения : 20 сентября 2015. **3.** *Пугачев П. В.* Математическое моделирование рабочих процессов лопастных гидромашин. Расчет вязкого течения в лопастных гидромашинах с использованием пакета ANSYS CFX // *П. В. Пугачев, Д. Г. Свобода, А. А. Жарковский.* – СПб : Политехн. ун-т, 2015. – 116 с. **4.** *Шипенко О. Н.* Моделирование вязкого турбулентного трехмерного потока в гидродинамическом трансформаторе / *О. Н. Шипенко, В. Г. Солодов* // Автомобильный транспорт. – Х. : ХНАДУ. – 2011. – Вып. 29. – С. 98–104. **5.** *Кочевский А. Н.* Современный подход к моделированию и расчету течений жидкости в лопастных гидромашинах / *А. Н. Кочевский, В. Г. Неня* // Вісник СумДУ. – 2003. – С. 15. **6.** *Гарбарук А. В.* Моделирование турбулентности в расчетах сложных течений : учебн. пособие // *А. В. Гарбарук, М. Х. Стрелец.* – СПб : Политехн. ун-т, 2012. – 88 с. **7.** Каталог продукции. ООО Производственная компания «Борец». – Москва, 2014. – 495 с. **8.** *Раухман Б. С.* Расчет обтекания несжимаемой жидкостью решеток профилей на осесимметричной поверхности в слое переменной толщины / *Б. С. Раухман* // Изд. АН СССР, МЖГ. – 1971. – № 1. – С. 83–89. **9.** *Жарковский А. А.* Математическое моделирование рабочих процессов в центробежных насосах низкой и средней быстроходности для решения задач автоматизированного проектирования : автореф. дис. … д-ра техн. наук : 05.04.13 / *Жарковский Александр Аркадьевич* ; Санкт-Петербургский государственный политехнический университет. – Санкт-Петербург, 2003. – 32 с. **10.** *Хитрых Д.* Ansys Turbo : Сквозная технология проектирования лопаточных машин : рус. ред. *Д. Хитрых* // Ansys Solution. – 2007. – № 6. – С. 31–37. **11.** *Хитрых Д.* Ansys Turbo : Обзор моделей турбулентности : рус. ред. *Д. Хитрых* // Ansys Solution. – 2005. – № 1. – С. 9–11. **12.** *Барашков С. А.* FlowVision – современный инженерный инструмент в исследовании газодинамических характеристик компрессоров / *С. А. Барашков* // САПР и Графика. – 2005. – № 1. – С. 44–48.

References: **1.** Greenshields, Christopher, J. *OpenFOAM The Open Source CFD Toolbox. User Guide*. Web. 21 January 2016 <www.openfoam.com>. **2.** *Akademicheskaja versija programmnoj sistemy konechno-elementnogo analiza ANSYS.* Web. 20 September 2015 <www.ansys.com/Student>. **3.** Pugachev, P. V., D. G. Svoboda and A. A. Zharkovskij. *Matematicheskoe modelirovanie rabochih processov lopastnyh gidromashin. Raschet vjazkogo techenija v lopastnyh gidromashinah s ispol'zovaniem paketa ANSYS CFX.* Saint Petersburg: Politehn. un-t, 2015. Print. **4.** Shipenko, O. N., and V. G. Solodov. "Modelirovanie vjazkogo turbulentnogo trehmernogo potoka v gidrodinamicheskom transformatore." *Avtomobil'nyj transport.* Kharkov: KhNAHU, 2011. No. 29. 98–104. Print. **5.** Kochevskij, A. N., and V. G. Nenja. "Sovremennyj podhod k modelirovaniju i raschetu techenij zhidkosti v lopastnyh gidromashinah." *Visnik SumDU*. 2003. Print. **6.** Garbaruk, A. V., and M. X. Strelec. *Modelirovanie turbulentnosti v raschetah slozhnyh techenij: uch. posobie*. Saint Petersburg: Politehn. un-t, 2012. Print. **7.** *Katalog produkcii. OOO Proizvodstvennaja kompanija "Borec".* Moscow, 2014. Print. **8.** Rauhman, B. S. "Raschet obtekaniya neszhimaemoy zhidkostyu reshetok profiley na osesimmetrichnoy poverhnosti v sloe peremennoy tolschinyi." *Izd. AN SSSR, MZhG* 1 (1971): 83–89. Print. **9.** Zharkovskiy, A. A. *Matematicheskoe modelirovanie rabochih protsessov v tsentrobezhnyih nasosah nizkoy i sredney byistrohodnosti dlya resheniya zadach avtomatizirovannogo proektirovaniya. Avtoref. dis. na soiskanie nauch. stepeni d-ra tehn. nauk.* Saint Petersburg, 2003. Print. **10.** Hitryih, D. "Ansys Turbo: Skvoznaya tehnologiya proektirovaniya lopatochnyih mashin." *Ansys Solution* 6 (2007): 31–37. Print. **11.** Hitryih, D. "Ansys Turbo: Obzor modeley turbulentnosti." *Ansys Solution* 1 (2005): 9–11. Print. **12.** Barashkov, S. A., et al. "FlowVision – sovremennyiy inzhenernyiy instrument v issledovanii gazodinamicheskih harakteristik kompressorov." *SAPR i Grafika* 1 (2005): 44–48. Print.

Надійшла (received) 05.10.2015

Шудрик Александр Леонидович – аспирант, Национальный технический университет «Харьковский политехнический институт», ассистент кафедры «Гидравлические машины», г. Харьков; тел.: (095) 454-03-87; e-mail: shudral88@gmail.com.

Shudrik Aleksandr Leonidovich – Postgraduate Student, National Technical University "Kharkov Polytechnic Institute", Assistant at the Department of "Hydraulic machines", Kharkov; tel.: (095) 454-09-87; e-mail: shudral88@gmail.com.