



СӨҮЛЕТ ЖӘНЕ ҚҰРЫЛЫС  
АРХИТЕКТУРА И СТРОИТЕЛЬСТВО  
ARCHITECTURE AND CONSTRUCTION

DOI 10.51885/1561-4212\_2022\_4\_198  
MPHTI 05.23.04

I.G. Rumanovski<sup>1</sup>, M.N. Shevtsov<sup>1</sup>, Yu.I. Vakulchik<sup>1</sup>, Ye.T. Togabaev<sup>2</sup>,  
L.M. Utepbergenova<sup>2</sup>, Zh.A. Shakhmov<sup>2</sup>

<sup>1</sup>Civil Engineering Institute, Pacific State University, Khabarovsk, Russia

E-mail: 001776@pnu.edu.ru\*

E-mail: 001776@pnu.edu.ru

E-mail: 000458@pnu.edu.ru

E-mail: 006192@pnu.edu.ru

<sup>2</sup>L.N. Gumilyov Eurasian National University, Astana, Kazakhstan

E-mail: utepber78@mail.ru\*

E-mail: zhanbolat8624@mail.ru\*

APPLICATION OF THE SOLIDWORKS FLOW SIMULATION TOOL ENVIRONMENT TO  
CREATE A VIRTUAL LABORATORY WORKSHOP ON HYDRAULICS

ГИДРАВЛИКА БОЙЫНША ВИРТУАЛДЫ ЗЕРТХАНАЛЫҚ ПРАКТИКУМ ЖАСАУ ҮШІН  
SOLIDWORKS FLOW SIMULATION АСПАПТЫҚ ОРТАСЫН ҚОЛДАНУ

ПРИМЕНЕНИЕ ИНСТРУМЕНТАЛЬНОЙ СРЕДЫ SOLID WORKS FLOW  
SIMULATION ДЛЯ СОЗДАНИЯ ВИРТУАЛЬНОГО ЛАБОРАТОРНОГО ПРАКТИКУМА  
ПО ГИДРАВЛИКЕ

**Abstract.** The article analyzes modern instrumental CAD/CAE systems used for numerical simulation by the finite element method of liquid or air flow, it is shown that the most optimal is the use of the SolidWorks Flow Simulation tool environment. A virtual 3D model of a laboratory hydraulic system was created for a virtual laboratory workshop on the course «Hydraulics», numerical modeling of water flow was carried out, the expediency of using the SolidWorks Flow Simulation tool environment for performing virtual laboratory work was shown.

**Keywords:** instrumental environment, numerical modeling, water flow, virtual model, hydraulic system.

**Аңдатпа.** Мақалада сұйық немесе ауа ортасы ағынының соңғы элементтерін сандық модельдеу үшін қолданылатын қазіргі заманғы аспаптық CAD/CAE жүйелеріне талдау жасалды, SolidWorks flow Simulation аспаптық ортасын қолдану ең оңтайлы екендігі көрсетілген. "Гидравлика" курсы бойынша виртуалды зертханалық практикумға арналған зертханалық гидравликалық жүйенің виртуалды 3D моделі жасалды, су ағынын сандық модельдеу жүргізілді, виртуалды зертханалық жұмыстарды орындау үшін SolidWorks Flow Simulation аспаптық ортасын қолданудың орындылығы көрсетілді.

**Түйін сөздер:** аспаптық орта, сандық модельдеу, су ағымы, виртуалды модель, гидравликалық жүйе.

**Аннотация.** В статье проведен анализ современных инструментальных CAD/CAE систем, применяемых для численного моделирования методом конечных элементов течения жидкой или воздушной среды, показано, что наиболее оптимальным является применение инструментальной среды SolidWorks Flow Simulation. Создана виртуальная 3D модель лабораторной гидравлической системы для виртуального лабораторного практикума по курсу «Гидравлика» проведено

численное моделирование течения воды, показана целесообразность применения инструментальной среды SolidWorks Flow Simulation для выполнения виртуальных лабораторных работ.

**Ключевые слова:** инструментальная среда, численное моделирование, течение воды, виртуальная модель, гидравлическая система.

*Introduction.* Currently, specially equipped laboratories equipped with stands for laboratory work are used for carrying out laboratory work on the course «Hydraulics» for students of construction areas of training. However, the real needs of modern educational programs require the use of modern software – systems CAE (Computed Aided Engineering) to create virtual laboratory workshops. As a result of the use of such tools, it becomes possible to simulate the behavior of the fluid in various real hydraulic structures represented by their virtual counterparts. The process consists in 3D modeling of the hydraulic system and software calculation of the kinematic and dynamic parameters of the fluid using the finite element method, finite differences or finite volumes. As a result of the calculation, it is possible to visualize the flow of a liquid and obtain various computational and graphical information about its characteristics, while saving significant financial and time resources for the development and installation of laboratory equipment, attracting support staff to carry out work, and saving the classroom fund.

The development of a virtual laboratory workshop also makes it possible to use it in electronic training courses hosted in the electronic educational environment of the university. This solution allows the educational process to be carried out via the Internet, which significantly increases the availability of educational resources, regardless of the place of residence of the trainees. It also becomes possible to use CAE systems in related training courses of the educational program, the development of project training technology for the development of skills for optimal design of real hydraulic systems and obtaining appropriate professional skills, skills and competencies.

*Materials and methods of research.* It is customary to characterize computing complexes for hydraulic calculations by the level of complexity of the tasks to be solved (the supported number of nodes of the computational grid, the degree of consideration of nonlinearities), by the number of models of behavior of liquids and gases. Currently, there are a significant number of software solutions in the field of numerical modeling of gas and hydrodynamics problems on the CAE solutions market [1, 2, 3, 4, 5, 6, 7, 8, 10] To date, CFD packages are conditionally divided into the following classes: [10]

1. «Heavy» – high-class complexes suitable for both scientific and engineering calculations, capable of solving the most complex problems taking into account a large number of effects and using a wide range of mathematical approaches, including specific ones. The «heavy» class includes the leaders among commercial CFD packages – ANSYSCFX (ANSYS, Inc.), Star-CD (CDadapco), FLUENT (ANSYS, Inc. together with Fluent, Inc.). All of them contain a large number of turbulence models, are capable of solving problems of varying complexity taking into account gorenje, chemical reactions, multiphase flows, support various types of grids, etc.

2. Middle class. Designed mainly for calculations of the engineering level of complexity. The set of models used can also be quite wide. This category includes COSMOSFloWorks (Solidworks.), STAR-CM+ (CD-adapco), ANSYS FLOTRAN (ANSYS, Inc.), SolidWorks Flow Simulation, COMSOL Multiphysics.

3. «Light» – CFD complexes using algorithms of low accuracy (used, for example, as educational and methodological), or having a narrow focus of calculation (specially created for a specific problem).

After analyzing the above solutions in the field of modeling the behavior of liquid and gaseous media, the following conclusions can be drawn: CAE systems such as: Ansys Fluent, COMSOL Multiphysics, SolidWorks Flow Simulation are currently most widely used [1, 2, 3] In research

practice, the instrumental media Ansys Fluent [4, 5, 6, 7, 8] and COMSOL Multiphysics [9] are widely used for modeling complex hydrodynamic processes. In applied problems of optimization of engineering hydraulic systems, the tool environment of SolidWorks Flow Simulation has been more widely used. The analysis of the capabilities of the numerical modeling systems described above has shown that the use of the SolidWorks Flow Simulation tool environment is most appropriate in educational practice. This environment has a number of advantages: [1, 2]

1. Built-in 3D design system with advanced tools for designing parts and assemblies.
2. Developed and convenient tools for the physical description of the modeling problem.
3. Databases of solids and fluids with sets of physical characteristics.
4. Availability of services for visualization and analysis of calculation results.
5. Availability of a software assistant to set initial and boundary conditions during modeling.
6. The possibility of a joint integrated solution of problems of hydraulics, thermodynamics and strength calculations in a single engineering environment.

Fluid flow modeling in SolidWorks Flow Simulation is based on 2 basic principles:[2]

- Direct use of embedded CAD as a source of geometric information;
- Combination of full 3D CFD modeling with simpler analytical methods in cases where the resolution of the finite element grid is insufficient for full 3D modeling.

Flow Simulation technology integrated with the CAD system SolidWorks includes a number of technologies:[3]

- CAD Data management;
- Generation of a finite volume grid;
- CFD Solvers;
- Engineering modeling technologies;
- Processing of results

The physical model of the fluid flow process is described by the well-known Navier-Stokes differential equations for Newtonian incompressible fluids: [7]

Equation of motion:

$$\rho \left[ \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] = -\text{grad } p + \eta \Delta \vec{v} + \vec{f}$$

The continuity equation:

$$\text{div } \vec{v} = 0$$

Heat transfer equation (energy balance):

$$\frac{\partial T}{\partial t} + (\vec{v} \cdot \nabla) T = a \Delta T$$

where:  $\rho$  – density;  $v$  – speed;  $t$  – time;  $\eta$  – viscosity coefficient;  $a$  – coefficient of thermal conductivity;  $\vec{f}$  – the field of mass forces of a different nature acting on a liquid;  $\nabla$  – is the Hamilton operator, the partial derivative of coordinates.

For a three-dimensional space in the field of gravity for a viscous incompressible fluid, the formulas are correct: [7]

$$\left\{ \begin{array}{l} \rho \left[ \frac{\partial v_x}{\partial t} + v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} + v_z \frac{\partial v_x}{\partial z} \right] = -\frac{\partial p}{\partial x} + \eta \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2} \right) + \rho g_x \\ \rho \left[ \frac{\partial v_y}{\partial t} + v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y} + v_z \frac{\partial v_y}{\partial z} \right] = -\frac{\partial p}{\partial y} + \eta \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right) + \rho g_y \\ \rho \left[ \frac{\partial v_z}{\partial t} + v_x \frac{\partial v_z}{\partial x} + v_y \frac{\partial v_z}{\partial y} + v_z \frac{\partial v_z}{\partial z} \right] = -\frac{\partial p}{\partial z} + \eta \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right) + \rho g_z \\ \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0 \\ \frac{\partial T}{\partial t} + v_x \frac{\partial T}{\partial x} + v_y \frac{\partial T}{\partial y} + v_z \frac{\partial T}{\partial z} = a \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \end{array} \right.$$

For the numerical solution of this system of equations, the finite difference method (MD) and the finite volume method (MCO) have become widely used.

The finite difference method is based on the approximation of derivatives included in the original differential equations by their discrete (difference) analogues. The undoubted advantages of the MCR are high efficiency and ease of implementation, as well as the visibility of the sampling procedure, which makes it possible to build high-order accuracy schemes. However, these advantages are realized only when using a sufficiently «good» regular (structured) grid – almost orthogonal and with smoothly changing cell sizes. As a consequence, the vast majority of MD applications are limited to cases of relatively simple geometry computational domains. To a certain extent, a solution to the problem of modeling flows in areas of complex geometry was found in the 1970s, when the finite volume method was widely used to discretize the equations of hydrodynamics. The starting point of the MCO is the integral formulation of the laws of conservation of mass, momentum, energy, etc. Balance ratios are recorded for a small control volume; their discrete analog is obtained by summing mass, momentum, etc. flows calculated by some quadrature formulas over all the faces of the allocated volume. Since the integral formulation of conservation laws does not impose restrictions on the shape of the control volume, the MCO is suitable for discretization of hydro-gas dynamics equations on both structured and unstructured grids with different cell shapes, which, in principle, completely solves the problem of complex geometry of the computational domain. [11, 12, 13, 14, 15]

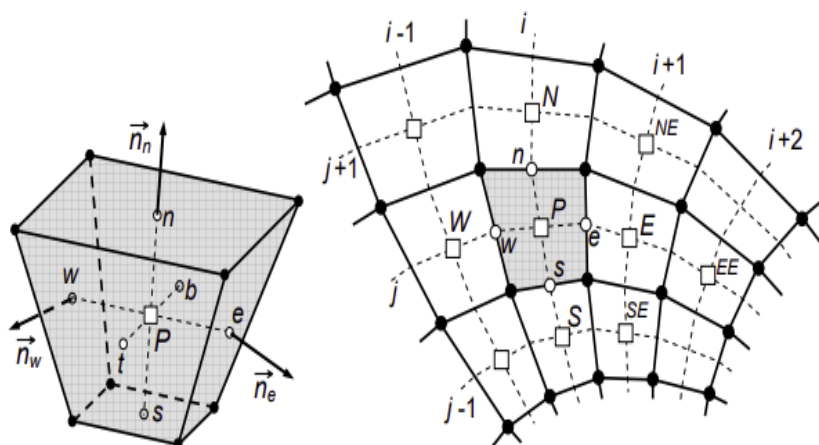
It should be noted, however, that the use of unstructured grids is quite algorithmically complex, time-consuming to implement and resource-intensive to perform calculations, especially when solving three-dimensional problems. This is due both to the variety of possible cell shapes of the computational grid, and to the need to use more complex methods to solve a system of algebraic equations that does not have a specific structure. The practice of recent years shows that the developed development of computing tools based on the use of unstructured grids is only possible for sufficiently large companies with the appropriate human and financial resources. It turns out to be much more economical to use block-structured grids, which involves dividing the flow area into several subdomains (blocks) of a relatively simple shape, each of which has its own computational grid. In general, such a composite grid is not structured, however, the usual index numbering of nodes is preserved inside each block, which allows the use of effective algorithms developed for structured grids. In fact, in order to move from a single-block grid to a multi-block one, it is only necessary to organize the docking of blocks, i.e. the exchange of data between adjacent subdomains to take into account their mutual influence. Note

also that splitting the task into separate relatively independent blocks naturally fits into the concept of parallel computing on cluster systems with processing of individual blocks on different processors (computers). All this makes the use of block-structured grids in combination with MKO a relatively simple, but extremely effective means of expanding the geometry of the problems being solved, which is extremely important for small university groups developing their own programs in the field of hydro-gas dynamics. The finite element method, especially widely used in deformable solid mechanics, relies on the variational problem of the minimum error of approximation of the desired solution by basic functions, and not directly on the original «physical» equations. Apparently, it is this additional mathematical load, which makes the FEM more difficult to understand and sometimes necessary modifications, along with the lack of obvious advantages over the MCO and difficulties in ensuring the necessary accuracy of the description of thin boundary layers, that is the reason for the relatively low popularity of the FEM in computational fluid dynamics.

The leading position of the MCO in relation to other methods of discretization of hydro-gas dynamics equations is confirmed by the trends of the modern software market. Suffice it to note that MKO is used in such world-famous hydrodynamic packages as FLUENT, CFX, STAR-CD, FINE (NUMECA), CFD-ACE, etc., and the first versions of all these packages were based on the use of block-structured grids, and only as the experience and financial power of the development companies have moved (are moving) to use unstructured grids. The use of FEM for modeling fluid flows naturally arose in hydrodynamic applications to branched finite element programs, developed for the elastic-thermomechanical analysis of structures; among the most famous are ANSYS Flotran and COSMOS FloWorks. However, even in this area, there is a noticeable interest of developers in MKO as a more flexible and technologically advanced way of discretizing hydro-gas dynamics problems.

According to the MKO, the spatial discretization of the problem is carried out by dividing the computational domain into small contiguous volumes. Inside each control volume there is one (and only one) point of «binding» of the desired grid solution. In most developments focused on solving three-dimensional problems for areas of complex geometry, the cells of the computational grid are used as a control volume: the grid nodes are located at the vertices of a polyhedron (for structured grids, a hexahedron, see Figure 1), the grid lines run along its edges, and the values of the desired values are attributed to the geometric center of the cell.

Alternative options (for example, building a control volume around a grid node or introducing different control volumes for different variables) are less common and will not be considered in this article. In modern computational fluid dynamics, MCO is the most popular method of spatial discretization of equations, which is due to its flexibility in relation to the grids used, the manufacturability of the software implementation, and the physical clarity of the basic principles. The use of block-structured grids is a relatively simple and, at the same time, an effective means of solving the problem of modeling flows in areas of complex geometry. The use of unstructured grids opens up wider opportunities and is more user-friendly, but requires large additional costs for software development, which is available only to fairly large companies. In the SolidWorks Flow Simulation tool environment, the finite volume method is used.



**Figure 1.** Structured grid of control volumes with variables «anchored» to the center of cells:

• – grid node, ... – cell center, o – face center [11]

Alternative options (for example, building a control volume around a grid node or introducing different control volumes for different variables) are less common and will not be considered in this article. In modern computational fluid dynamics, MCO is the most popular method of spatial discretization of equations, which is due to its flexibility in relation to the grids used, the manufacturability of the software implementation, and the physical clarity of the basic principles. The use of block-structured grids is a relatively simple and, at the same time, an effective means of solving the problem of modeling flows in areas of complex geometry. The use of unstructured grids opens up wider opportunities and is more user-friendly, but requires large additional costs for software development, which is available only to fairly large companies. In the SolidWorks Flow Simulation tool environment, the finite volume method is used.

*Results and discussion.* Let's consider the implementation of a laboratory workshop in the SolidWorks Flow Simulation environment.

Let's build a solid-state model of a hydraulic system for virtual simulation of fluid flow. As a prototype, we will use an experimental installation for conducting a course of laboratory work on hydraulics. The installation includes elements such as a Venturi tube, sections of pipe of various diameters, an elbow, and the inner surfaces of pipes have roughness.

The work allows you to simulate fluid flow in real conditions and study phenomena such as laminar and turbulent flow modes, calculate the Reynolds number, determine the coefficients of hydraulic friction (Darcy coefficient), study the effect of local hydraulic resistances, such as: knee, smooth narrowing-expansion, sudden expansion, sudden narrowing and roughness of the inner surface of the pipe on kinematic and dynamic parameters of the fluid.

Source data:

Task type:

The task is internal

Exclusion of cavities without flow conditions

Fluid environment:

The fluid medium of the project is water, Characteristics of the fluid flow:

The type of fluid flow is laminar and turbulent, there is no cavitation

Conditions on the walls:

The thermal condition on the wall by default is an adiabatic wall The roughness of the pipe walls is 20 microns.

Initial conditions:

Ambient pressure -101325 Pa, Temperature – 20 °C

Turbulence parameters - turbulence intensity - 2%, turbulence scale - 0.0022 m.

Boundary conditions:

The volume flow rate at the inlet is 0.005 m<sup>3</sup>/s

Objectives:

Calculation of average pressure; Calculation of total pressure; calculation of velocities

The results of modeling the pressure field and velocity field are shown in Figures 2 and 3, respectively.

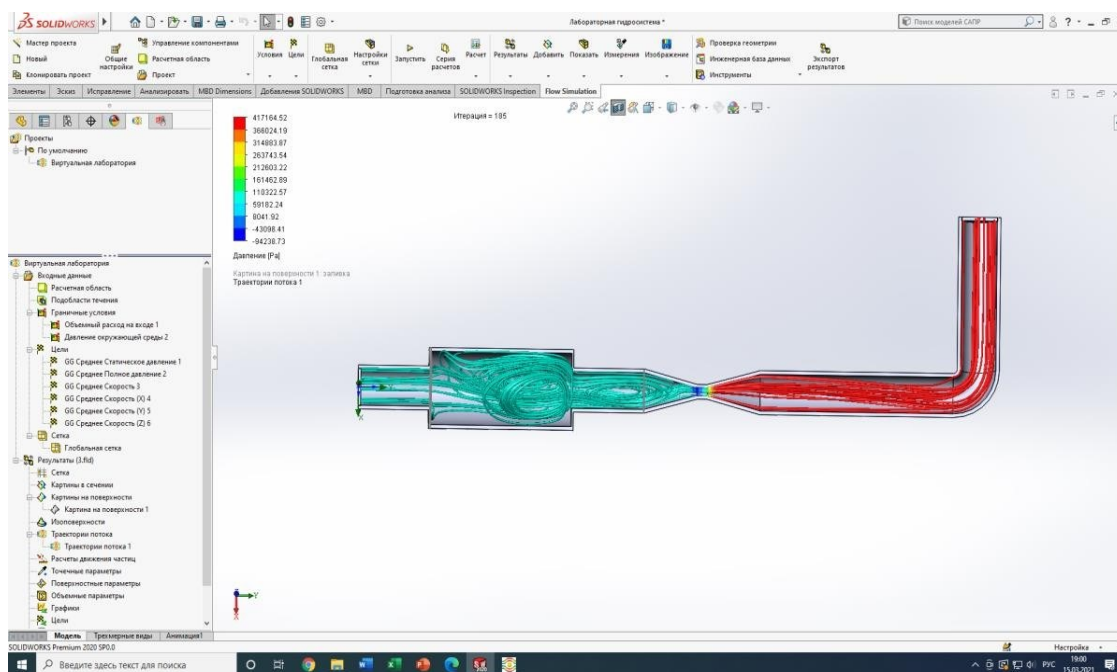


Figure 2. Water flow pressure field

Based on the performed calculation, it is possible to plot pressure and velocity graphs at any selected point of the structure, to visualize the movement of the water flow. It seems appropriate to conduct a simulation of the hydraulic regime of fluid flow when changing the geometric parameters of the structure, varying the characteristics of the volume flow at the inlet. When changing the type of task from internal to external, it is possible to simulate the flow of water from nozzles of various geometric configurations.

Thus, we get a flexible and convenient virtual laboratory workshop. Currently, this work is actively used in the educational process both when studying at the university and in the process of learning with the use of distance learning technologies.

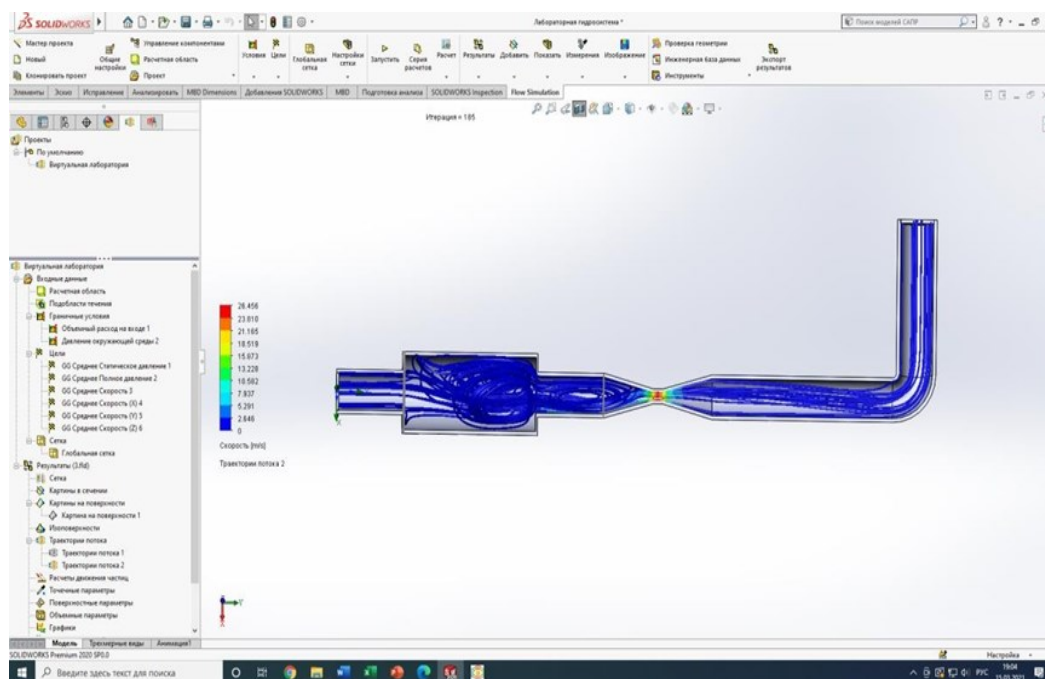


Figure 3. Water flow velocity field

*Conclusion.* In the process of virtual modeling, the results of solving computational fluid dynamics problems can serve as boundary conditions for solving problems in the field of strength calculations of structures, including main pipelines. The use of numerical hydrodynamics models can significantly improve the adequacy of virtual modeling of technical systems.

The ability to solve such problems in a single CAD/CAE environment of SolidWorks allows you to carry out design optimization cost-free field tests and experiments. The considered example of creating a virtual laboratory allows us to conclude that it is possible to create a «digital double» of a real laboratory with the possibility of conducting laboratory work on the hydraulic electronic educational environment of the university. Of considerable interest is the possibility of implementing interdisciplinary projects in the SolidWorks tool environment in solving complex engineering problems, the solution of which requires modeling of hydro-gas dynamic flows in the systems being designed.

The use of the SolidWorks Flow Simulation tool environment in the practice of training students and masters in technical areas of training allows them to form professional competencies for solving problems in the field of engineering systems design when they are exposed to a complex of physical factors.

#### References

1. Alyamovskij, A.A. SOLIDWORKS Simulation iFloEFD. Praktika, metodologiya, ideologiya / A.A. Alyamovskij. – Moskva: DMK Press, 2018. – 658 s. – ISBN 978-5-97060-646-9. – Tekst: elektronnyj // Lan': elektronno-bibliotekhnayasistema. – URL: <https://e.lanbook.com/book/131715> (data obrashcheniya: 19.10.2022).
2. Sobachkin A., Dumnov G., Numerical Basis of CAD-Embedded CFD. NAFEMS WorldCongr. 2013, 1-20.
3. Antipova M.S., Dyad'kin A.A., Zapryagaev V.I., Krylov A.N. Komp'yuternoe modelirovanie istecheniya holodnoj sverhzhukovoj strui iz konicheskogo sopla s ispol'zovaniem programmogo paketa FloEFD. // Kosmicheskaya tekhnika i tekhnologii. – 2016. – № 1(12). – S. 5-11.



4. Kryukov YU.A. Razrabotka metodov matematicheskogo modelirovaniya laminarnykh techeniy vyzkoj neszhimaemoy zhidkosti v sloe s mezhfaznoy granicej. Dissertaciya na soiskanie uchenoj stepeni kandidata tekhnicheskikh nauk. Samara. – 2018.
5. Ahmetov YU.M., Kalimullin R.R., Konstantinov S.YU., Hakimov R.F., Celishchev D.V. Issledovanie gidrodinamicheskikh i termodinamicheskikh processov vyslokonapornogo mnogofaznogo techeniya zhidkosti. // Vestnik UGATU. – 2012.- Tom.16, №2(47). – S.163-168.
6. Val'ger S.A., Fyodorov A.V., Fyodorova N.N. Modelirovanie ne szhimaemykh turbulentnykh techeniy v okrestnosti plohoobtekaemykh tel s ispol'zovaniem PK ANSYS Fluent. //Vychislitel'nye tekhnologii. - 2013.-Tom.18, №5.
7. Platonov D.V., Minakov A.V., Dekterev A.A., Sentyabov A.V. CHislennoe modelirovanie prostranstvennykh techeniy s zakrutkoj potoka. Komp'yuternye issledovaniya i modelirovanie. 2013 Tom. 5 №4. – S. 635-648.
8. Regotov G.A. CHislennoe modelirovanie processa psevdoozhizheniya v ANSYS Fluent // Universum: tekhnicheskie nauki: elektron. nauchn. zhurn. 2019. № 7(64).URL:[https://7universum.com/ru/tech/archive/item/7644\(dataobrashcheniya:18.07.2022\)](https://7universum.com/ru/tech/archive/item/7644(dataobrashcheniya:18.07.2022)).
9. Ryzhkov V.V., Zverev A.V., Rodionov I.A. CHislennoe modelirovanie processov teplomasso perenosa v mikroflyuidnom teplovom datchike potoka//VestnikMGTUim.N.E.Baumana. Ser.Priborostroenie. – 2017. – № 5. – S. 70-84.
10. SHustrova M.L., Aminev I.M., Bajtimirov A.D. Credstva chislenного modelirovaniya gidrodinamicheskikh parametrov processov. Elektronnyj resurs. URL: [https://cyberleninka.ru/article/n/sredstva-chislenного-modelirovaniya-gidrodinamicheskikh-parametrov-protsessov/viewer\(dataobrashcheniya:18.07.2022\)](https://cyberleninka.ru/article/n/sredstva-chislenного-modelirovaniya-gidrodinamicheskikh-parametrov-protsessov/viewer(dataobrashcheniya:18.07.2022)).
11. Smirnov E.M., Zajcev D.K. Metod konechnykh ob'emov v prilozhenii k zadacham gidrogazodinamiki i teploobmena v oblastiakh slozhnoy geometrii.Nauchno-tekhnicheskie vedomosti. 2004. – №2. Elektronnyj resurs. URL: [https://aero.spbstu.ru/publ/smironov3.pdf\(dataobrashcheniya:18.07.2022\)](https://aero.spbstu.ru/publ/smironov3.pdf(dataobrashcheniya:18.07.2022)).
12. Sinitsyna A., Borisov B., Gavryushina O. Study of the influence of the design parameters of the wet end of the gear pupm with external gearing, using the software package MATLAB 2018a. IOP Conference Series: Materials Science and Engineering, Tom 492, Issue 1, doi:10.1088/1757-899X/492/1/012007
13. Cerpinska, M., Irbe, M., Pupurs, A., Burbeckis, K. Modeling of water flow for waterjet propulsion system. Engineering for Rural Development. – 2021. – Tom 20. – P. 21-25.
14. Ferroudji, F., Ouattas, T., Khélifi, C. Design, modeling and finite element static analysis of a new two axis solar tracker using SolidWorks/COSMOSWorks. Applied Mechanics and Materials, 2013, Tom 446-447, p. 738-743.
15. Li, W.-M., Xie, L.-Y. On the SolidWorks-based virtual design integrated hydraulic manifold blocks. Dongbei Daxue Xuebao/Journal of Northeastern University, 2006. – Tom 27. Issue 6. – P. 681-684.

#### Список литературы

1. Алямовский, А.А. SOLIDWORKS Simulation и FloEFD. Практика, методология, идеология / А. А. Алямовский. – Москва: ДМК Пресс, 2018. – ISBN 978-5- 97060-646-9. – Текст: электронный // Лань: электронно-библиотечная система. – URL: <https://e.lanbook.com/book/131715> (дата обращения: 18.07.2022).
2. Sobachkin A., Dumnov G., Numerical Basis of CAD-Embedded CFD. NAFEMS WorldCongr. – 2013, 1-20.
3. Антипова М.С., Дядькин А.А., Запрягаев В.И., Крылов А.Н. Компьютерное моделирование истечения холодной сверхзвуковой струи из конического сопла с использованием программного пакета FloEFD. // Космическая техника и технологии. – 2016. – № 1(12). – С. 5-11 (дата обращения 18.07.2022).
4. Крюков, Ю.А. Разработка методов математического моделирования ламинарных течений вязкой несжимаемой жидкости в слое с межфазной границей. Диссертация на соискание ученой степени кандидата технических наук. Самара. – 2018.
5. Ахметов Ю.М., Калимуллин Р.Р., Константинов С.Ю., Хакимов Р.Ф., Целищев Д.В. Исследование гидродинамических и термодинамических процессов высоконапорного многофазного течения жидкости. // Вестник УГАТУ. – 2012. – Т. 16. – № 2(47). – С.163-168.
6. Вальгер С.А., Фёдоров А.В., Фёдорова Н.Н. Моделирование несжимаемых турбулентных течений в окрестности плохообтекаемых тел с использованием ПК ANSYS Fluent. // Вычислительные технологии. – 2013. – Том 18. – № 5.

7. Платонов Д.В., Минаков А.В., Дектерев А.А., Сентябов А.В. Численное моделирование пространственных течений с закруткой потока // Компьютерные исследования и моделирование. – 2013. – Т. 5. – № 4. – С. 635-648.
8. Реготов Г.А. Численное моделирование процесса псевдооживления в ANSYS Fluent // Universum: технические науки: электрон. научн. журн. 2019. № 7 (64). URL: <https://7universum.com/ru/tech/archive/item/7644> (дата обращения: 18.07.2022).
9. Рыжков В.В., Зверев А.В., Родионов И.А. Численное моделирование процессов тепломассопереноса в микрофлюидном тепловом датчике потока // Вестник МГТУ им. Н.Э. Баумана. Сер. Приборостроение. 2017. № 5. – С. 70-84. DOI: 10.18698/0236-3933-2017-5-70-84.
10. Шустрова, М.Л., Аминев И.М., Байтимиров А.Д. Средства численного моделирования гидродинамических параметров процессов. Электронный ресурс. URL: <https://cyberleninka.ru/article/n/sredstva-chislennogo-modelirovaniya-gidrodinamicheskikh-parametrov-protseessov/viewer> (дата обращения: 18.07.2022).
11. Смирнов Е.М., Зайцев Д.К., Метод конечных объемов в приложении к задачам гидрогазодинамики и теплообмена в областях сложной геометрии. Научно-технические ведомости. 2004. №2. Электронный ресурс. URL: <https://aero.spbstu.ru/publ/smirnov3.pdf> (дата обращения: 18.07.2022).
12. Sinitsyna A., Borisov B., Gavryushina O. Study of the influence of the design parameters of the wet end of the gear pump with external gearing, using the software package MATLAB 2018a. IOP Conference Series: Materials Science and Engineering, Tom 492, Issue 1, doi:10.1088/1757-899X/492/1/012007
13. Cerpinska, M., Irbe, M., Pupurs, A., Burbeckis, K. Modeling of water flow for waterjet propulsion system. Engineering for Rural Development, 2021, Tom 20, p.21-25
14. Ferroudji, F., Ouattas, T., Khélifi, C. Design, modeling and finite element static analysis of a new two axis solar tracker using SolidWorks/COSMOSWorks. Applied Mechanics and Materials, 2013. – Tom 446-447. – P. 738-743.
15. Li, W.-M., Xie, L.-Y. On the SolidWorks-based virtual design integrated hydraulic manifold blocks. Dongbei Daxue Xuebao/Journal of Northeastern University, 2006, Tom 27, Issue 6, p. 681-684.