UDC 621.22:62

DESIGN AND RESEARCH OF EQUIPMENT CHARACTERISTICS USING COMPUTATIONAL FLUID DYNAMICS TOOLS

***O.O. Zhuha*[1], *D.V. Davydov*[1]**, *E.V. Manoilo*[2]

[1] Graduate Student of the Department of ChE&IE NTU "KhPI", Kharkiv, Ukraine

[2] associate professor of the Department of ChE&IE, Candidate of Technical Sciences, NTU "KhPI", Kharkiv, Ukraine

Ievgeniia.Manoilo@khpi.edu.ua

Computational fluid dynamics (CFD) can provide insights into gas flow that cannot be obtained analytically or experimentally and improve performance by reducing losses. In addition, expensive prototyping can be done after establishing convincing characteristics using CFD models. The results of such studies will be useful for visualizing the flow pattern and identifying any deficiencies in it. With the advent of high-speed computing, the application of CFD is becoming widespread in engineering design and analysis.

Calculation using computational fluid dynamics (CFD) methods is an effective way to model and analyze work processes. To perform CFD calculations, it is necessary to construct a three-dimensional geometry of the working element and set the parameters of the working environment. Next, boundary conditions are set for the model, and a suitable turbulence model is selected for modeling the flow in the system. This can be a k-ε, k-ω, SST, or other model, depending on the flow characteristics. Mesh resolution, time step, and other numerical parameters are set to ensure accurate and efficient calculation. After that, CFD calculations are performed with analysis of the results obtained, including the distribution of pressure, velocity, temperature, and other parameters along the flow in the working element and beyond its boundaries.

Based on the analysis of the numerical results obtained, changes are made to the geometry of the structure under investigation to further improve the efficiency and performance of the installation. These steps can be repeated several times in the process of optimizing the geometry of the flow path. It is important to have experience and knowledge in using CFD software such as ANSYS Fluent, OpenFOAM, and others to successfully perform such calculations.

Simulation-based design uses advanced virtual technologies to manage the design and development process by obtaining a product or compiling qualitative responses and quantitative indicators prior to creating a physical prototype. This approach combines traditional CAE modeling tools, artificial intelligence predictions, system modeling, and process simulation in the development of concepts and their subsequent analysis.

Engineers can take full advantage of simulation technology to validate their design concepts early in the product development process, well before the detailed design stage. This reduces the risk of costly design changes and tooling rework later on, making it a powerful tool for engineers seeking to optimize their product development workflows. High level of detailization ensures effective development of the product and the manufacturing process.

The use of simulation-based design allows for early verification of design concepts, justification of design decisions, increased efficiency in the use of available resources, optimization of production methods, and reduction of production costs. Using parametric modeling and simulation tools, engineers can quickly and efficiently explore a range of proposed options and, based on an iterative process, identify cost-effective design solutions without large-scale physical prototyping. Thus, simulation-based design provides engineers with opportunities to optimize designs and improve their performance when designing modern technological equipment.