Analysis of the capabilities of software products to simulate the behavior of dynamic fluid flows

Georgieva Nely, Delcheva Sivelina, Tsankov Petko
Trakia University, Faculty of Technics and Technologies of Yambol
38 “Graf Ignatiev” str., 4600 Yambol, Bulgaria

e-mail: nely.georgieva@trakia-uni.bg

Abstract. In practice there are many processes that have complicated and complex character. Most often, these are dynamic systems with a large number of variable parameters, which significantly complicates their study, analysis and prediction. For their study most commonly used simulations that allow you to gain knowledge of their systemic nature. In these systems include and the thermodynamic systems where, due to a change of the many influencing parameters: temperature, pressure, velocity, fluid flow rate, various physical processes take place. For their study is widely used computer simulations to help predict the behaviour of systems under various conditions. Simulations are closely related to direct geometric modelling to enable to interactive design research and rapid model creation, allowing engineers to experiment with design ideas and immediately see the result of their changes. As an object of the simulation is most often fluid, structural and thermal simulation applications, which are used both for research and for innovative practical solutions in manufacturing enterprises. Computer simulations allow to predict the behaviour of certain processes in a real environment through virtual testing of CAD models. The main stages contain a methodical sequence, which includes the construction of a mathematical model, its numerical solution and processing of the obtained results. The main equations that make up the mathematical models are the balance equations of mass, momentum and energy, which determine the distribution of velocities, pressures, densities, temperatures and other control parameters. For modelling of flows of liquids and gases for industrial tasks, turbulence, heat transfer and chemical reactions, etc. are taken into account. In the practice, a large number of software solutions to simulate the processes occurring in thermal and fluid flows offered by many manufacturing companies are used. The aim of the present study is to analyse the possibilities of the software products used in practice to simulate the behaviour of dynamic fluid flows under certain initial conditions. Based on the analysis and the formulated conclusions, the choice of a software module for specific thermal research and construction of a geometric model of the studied objects through which numerical (CFD) simulations will be performed using different models, research in different modes of work, as well as the validation of the created methodologies, will be substantiated.

Keywords: thermodynamic processes, thermodynamic systems, simulation of thermodynamic processes, simulation of fluid flows.

1. Introduction
The study of processes in thermodynamic systems is a compound process with a complex nature, as a large number of factors influences the behavior of fluids. For their study is used Computational fluid dynamics (CFD) which is the branch of Computer-aided engineering (CAE) that simulates fluid motion
and heat transfer using numerical approaches. It can be considered as a group of computational methodologies used to solve equations controlling fluid flow.

Historically, these methods were first developed to solve the linearized potential equations. Two-dimensional (2D) methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s [1].

With the further development of information technology and computer technology, there is a significant simplification of the work in studying the behavior of fluid flows, reducing the time for analyzing and simulating the processes occurring in thermal systems, increasing the quality and reliability of the results. To confirm this statement, an analysis of scientific publications with open access in this field, presented in the journals of the bibliographic database Scopus was done. It is one of the world's largest bases for scientific publications and includes over 5,000 publishers. It was found that the first available publications related to the computational dynamics of fluids and simulation of thermodynamic processes are from 1996 (Figure 1). Over the last decade, there has been significant growth and increased pace of research in these areas, which can be explained both by the rapid development of computer technology and the growing supply of computer applications for research in the field of CFD. The analysis of the trend shows that the growth rates of research in the field of CFD will be maintained in the coming years too.

![Figure 1. Dynamics of scientific publications with open access in the field of CFD, published in Scopus by years.](image)

The analysis by country shows China's undisputed leadership in the field of flow simulation research - one hundred publications for the period under review (Figure 2). Of the European countries leading in research are Italy, Portugal, Romania. For Bulgaria, the number of scientific publications for the period under review is three, which is significantly less than the number of publications from the leading countries. Figure 2 presents the results for countries with more than ten CFD publications.

In the practice, a large number of software solutions to simulate the processes occurring in thermal and fluid flows offered by many manufacturing companies are used. A study of the suggesting computer
applications on the market for simulations of thermodynamic processes found that almost all manufacturers of software for Computer-Aided Design (CAD) offer modules (standalone or embedded in a platform) for CFD [2, 3, 4, 5, 6], which have different options - for a more general application and specialized in certain areas. Manufacturers such as Dassault Systèmes [7], for example, offer access to a fluid simulation module built into the 3DEXPERIENCE platform, while Autodesk offers separate software - Autodesk CFD, which is compatible with other software products offered by the company, and the company will soon release an update, which will allow direct export from Fusion 360 [8]. There are also a large number of free programs on the market in this area.

The use of appropriate software for simulation, study and prediction of thermodynamic processes is a key point in research that will ensure the quality and reliability of the results. Therefore, the choice of a computer application CFD is important and is the first step in designing a scientific experiment.

The aim of the present study is to analyze the capabilities of the most commonly used in practice software products for simulating the behavior of dynamic fluid flows under certain given initial conditions.

2. Exposition

The market of simulation and analysis software is growing on the back of rising technological advancements, increasing demand for innovation and superior quality products from different regions across the globe, high expenditure on aerospace and defence across the globe and boost in need for application-specific simulation software. The software products available on the market for simulating the behavior of dynamic fluid flows are many and varied, which makes it difficult to choose the most appropriate software product for conducting research under specific initial conditions. The worldwide Computer-Aided Engineering (CAE) market for 2017 is expected to reach $ 3.8 billion [9]. CFD applications are the second most widely used CAE applications and have seen significant growth worldwide [10]. According to MARKET REPORTS for Computational Fluid Dynamics market, published on 31.08.2020 [11] the largest market share in the world is held by several key players - Mentor Graphics, Altair Engineering, Comsol, Numerica International, Ansys, Autodesk, EXA,
Convergent science (Figure 3). The largest consumers in the field are the automotive industry, the aerospace, and defence industry, the electrical and electronics industry, the industrial machinery industry, the remote control hobby car industry, and others.

![Figure 3. Share of Computational Fluid Dynamics market by players,](https://www.amplemarketreports.com)

A study in the Google Scholar and CrossRef databases, published in [12], identified the areas of application of CFD applying in the field of Energy Engineering Research, as well as the most commonly used applications for CFD analysis in accordance with their purpose. The obtained results differ from the presented information on the market share of key players. This imposes the conclusion that CFD applications used primarily in research differ from those used in practice. The analysis indicated that Phoenics was the very first commercially available CFD code (released in 1981), but currently Ansys is the most widely used CFD software nowadays (over 40% market share), with both major codes CFX (acquired in 2003) and Fluent (acquired in 2006). Among the open source CFD software, OpenFOAM from ESI Group is the most widely used.

An analysis of the CFD applications used in the scientific articles in the field of Energy Engineering Research, published in the Scopus database, presented in Fig.1 [13, 14, 15, 16, 17, 18, 19, 20]. The obtained results are presented in Figure 4.

![Figure 4. Analysis of the frequency of use of CFD software products in research published in Scopus.](https://example.com)
A study of the capabilities of simulation software products found that the most commonly used fluid flow simulation modules are Ansys Fluent, OpenFoam and Comsol Multiphysics.

Ansys is a fluid modelling tool well known among those working in this field. Ansys Fluent is a software module from the Ansys fluid simulation software product used to predict fluid flow, heat and mass transfer, and other phenomena [21, 22, 23, 24, 25]. To improve performance, Ansys offers several packages that include Fluent in addition to other support software. The “CFD Premium Bundle” includes Fluent, Workbench, SpaceClaim, Ensight (a post-processing software package), CFX (another CFD solver) as well as CFD-Post (a post-processing tool). Fluent can be used in both Windows and Linux-based versions, it should be noted that SpaceClaim is currently only available for Windows operating systems. Ansys uses import from another CAD package to initially create geometry, such as SolidWorks or AutoDesk Inventor. This geometry is then introduced into SpaceClaim to separate and mark parts and boundaries, as well as to create all the necessary areas to improve the network (“bodies of influence”). The SpaceClaim geometry file is then imported into Fluent for pre-processing, linking and starting the simulation [26].

With a strong history in CFD, cutting edge and continuous upgrades to the code base and capabilities, a reputation for providing accurate and validated results for numerous flow cases/types, a stout user base and community, and dedicated customer support with a vast array of online resources and webinars, overall Fluent is a strong multi-physics software platform. Recently added capabilities have greatly increased its ease of its. As advantages of Ansys the following characteristics can be noted: powerful, efficient and validated numerical methods, full suite of physics and multiphysics capabilities, and as disadvantages: requirement for standalone software for pre-processing (SpaceClaim) and superior post-processing (Ensight), cost.

Comsol Multiphysics is a simulation software for modelling projects, devices and processes in the field of engineering, manufacturing and research [27]. The Fluid Flow & Heat Transfer Modules module provides opportunities for workflow research, pre-processing and post-processing to ensure maximum productivity and accuracy.

Visual-CFD is a complete user interface for computational fluid dynamics (CFD), created for OpenFOAM [28, 29, 30] and provided in the Visual-Environment of ESI group. OpenFOAM has all the advantages of open source programs as broad user base, tutorials and example problems online, and the ability to customize the code base to your liking, increased acceptance in academia and industry and most of all that it is free. The inconvenience is that an additional program is needed to visualize the results and a need to study and installing the additional post-processing software package as well as a limited possibility when working under Windows.

The considered three modules for fluid flow simulation have the ability to work with both operating systems - MS Windows and Linux.

To compare the capabilities of the three of the most used software products according to the results of the research, a study was conducted based on the information provided by the manufacturers. The results are presented in Table 1.

The sequence of working with these tools is analogous and includes the following steps:

- Defining the problem to be solved;
- determination of an appropriate simulation method;
- mathematical analysis of the chosen simulation method;
- creation of a theoretical model of the studied model;
- validation of the simulation.

From the analysis, it can be argued that in terms of application areas, the methodology of creating simulations and the sequence of work, the tools in question have equal capabilities. The main advantage of Ansys Fluent is the presence of a built-in module for CAD design. The integration of such a module in computer calculations of fluid studies in the working environment of the designer gives extremely great advantages:
Table 1. Comparative analysis of the capabilities of the most commonly used CFD applications for Energy Engineering Research

| Comparison indicators                                      | CFD applications for Energy Engineering Research |
|------------------------------------------------------------|-------------------------------------------------|
|                                                            | Ansys Fluent | Fluid Flow & Heat Transfer Modules | Visual-CFD |
| Built-in CAD design module                                 | +            | -                                  | -          |
| Areas of application:                                      |              |                                    |            |
| • Non-compressible and compressible flows                  | +            | +                                  | +          |
| • Laminar and turbulent flows                              | +            | +                                  | +          |
| • Single-phase and multi-phase flows                       | +            | +                                  | +          |
| • Heat and mass transfer, transient hydraulic processes    | +            | +                                  | +          |
| • Acoustics in pipe and duct networks                      | +            | +                                  | +          |
| Methods and equations used for simulations                 |              |                                    |            |
| • Finite volume method                                     | +            | +                                  | +          |
| • Reynolds-Averaged Navier–Stokes models (RANS) as k-ε and k-ω for turbulent flow | + | + | + |
| • Modeling of two-phase flows by the Lagrange method       | +            | +                                  | +          |
| • Prandtl equation for laminar, boundary, self-similar flow| +            | +                                  | +          |
| • Impulse equation of Karman                               | +            | +                                  | +          |

- Full associativity between the three-dimensional model and the set mesh. Changing the geometric shape of the model leads to updating the networking in compliance with the specified boundary conditions.
- Ability to conduct experiments by performing variations of a parameter or group of parameters from the geometric model in order to improve the values of the optimized quantities.
- Ability to compare the optimized values of several alternative geometric configurations of the product under identical boundary conditions and crosslinking characteristics.
- The results of the fluid analysis can be taken into account in a timely manner in the three-dimensional model by changing the geometric shape in critical areas.

The integrated approach to computer-aided Fluid analysis has great advantages to accelerate the development cycle. The main characteristics of the studied system, the most appropriate geometric configuration, for example in order to reduce losses along the route along which passes the fluid, pressure drop, or improve heat transfer can be analyzed and pre-simulated by the constructor.

Another important advantage that Ansys Fluent has is that it is offered with a free academic license for one year, which explains its wide use in research.

3. Conclusion
From the study on the frequency of use of CFD simulation tools in the field of Energy Engineering Research in scientific publications in the bibliographic database Scopus and the available information about these instruments, presented by their manufacturers, the following conclusions can be drawn:
- The key players on the market of the CFD applications are highly focused on innovation in CFD design to improve efficiency and working with them. The best long-term growth opportunities in this sector can be reached by ensuring ongoing process improvements and financial flexibility to invest in optimal strategies.
The study conducted in the Scopus bibliographic database regarding the frequency of scientific publications in the field of CFD analysis showed that over the last decade there has been significant growth and increased pace of research in this area, which can be explained by the rapid development of computer technology, as well as the growing supply of computer applications for CFD research.

The study of the market share of the most commonly used CFD instruments and their use in research was found that there were some differences in the choice of preferred software for simulation of processes in heating systems in the practice and in the scientific literature. While in production conditions the most commonly used application is Mentor Graphic, in scientific publications it is Ansys Fluent.

From the analysis of the software used for Energy Engineering Research in the scientific publications in the bibliographic database Scopus, it was found that most often researchers use Ansys Fluent, OpenFoam and Comsol Multiphysics.

In the study of the capabilities of the three software on the information offered by their manufacturers, it can be argued that in terms of application areas, simulation methodology, and sequence of work, the considered tools have equal capabilities.

The main advantage of Ansys Fluent over the other two stimulation modules is the presence of a built-in platform for creating a geometric model, which leads to improved tool operation, creating associativity between the three-dimensional model and the set mesh, accelerating the cycle development, and using parametric simulation. In addition, Ansys Fluent is available with an academic license for one year, which explains its widespread use in research.

The two simulation modules Fluid Flow & Heat Transfer Modules of Comsol Multiphysics and Visual-CFD of OpenFOAM do not have a built-in platform for creating a geometric model. They use an importing of geometric patterns from various CAD design software.

Acknowledgments

The studies presented in the report are supported under scientific project 4 FTT/2020 „Investigation of thermal processes with ANSYS Fluent”.

References

[1] Milne-Thomson, L.M. (1973). Theoretical Aerodynamics. Physics of Fluids A. 5. Dover Publications. p. 1023. ISBN 978-0-486-61980-4
[2] Vasiliev V.A., Kalmykova M.A. Analysis and selection of software products for solving engineering problems of instrumentation, Electronic scientific and practical journal "Modern Engineering and Technology", 8/6/2020, ISSN 2225-644X, Russian
[3] Cradle CFD - Smart Multiphysics Focused Computational Fluid Dynamics, https://www.mscsoftware.com/product/cradle-cfd
[4] FloEFD for Solid Edge – thermos-fluid simulations, https://www.solidedge.bg/termo-fluidni-simulatzi-i-analiz
[5] https://www.capterra.com/simulation-software/
[6] Top Computational Fluid Dynamics (CFD) Software, https://www3.technologyevaluation.com
[7] https://www.marketsandmarkets.com/Market-Reports/simulation-software-market-263646018.html
[8] https://www.autodesk.com/solutions/simulation/cfd-fluid-flow
[9] https://www.statista.com/statistics/732384/worldwide-computer-aided-engineering-market-revenues/
[10] https://www.marketintellica.com/report/M49450-global-cae-market-report-2019
[11] https://primefeed.in/market-reports/4824591/computational-fluid-dynamics-market-demonstrates-a-spectacular-growth-by-2026-exa-convergent-science-autodesk-numeca-international-mentor-graphics-ansys/
[12] Alfredo Iranzo, CFD Applications in Energy Engineering Research and Simulation: An
Introduction to Published Reviews, Processes 2019, 7, 883; doi:10.3390/pr7120883, www.mdpi.com/journal/processes

[13] Arkian A.H., Najafi G., Gorjian S., Loni R., Bellos E., Yusaf, T., Performance assessment of a solar dryer system using small parabolic dish and alumina/oil nanofluid: Simulation and experimental study, Volume 12, Issue 24, 12 December 2019, Article number 4747

[14] Gaurav Krishnayatra, Sulekh Tokas, Rajesh Kumar, Mohammad Zunaid4, 3 Dimensional CFD analysis of Laminar flow Natural Convection of Hollow Cylinder with Annular Fins, Proceedings of the World Congress on Mechanical, Chemical, and Material Engineering 2019, Article number 181

[15] Iturbide Jiménez, F., Mendoza Jasso, A.J., Antonio García, A., Santiago Alvarado, A., Design and construction of an apparatus to visualize incompressible fluid flow in several regimes, Revista Mexicana de Fisica EVolume 64, Issue 2, July 2018, Pages 133-138

[16] Kalash, A.R., Shijer, S.S., Habeeb, L.J., Thermal Performance Improvement of Double Pass Solar Air Heater, Journal of Mechanical Engineering Research and Developments, Volume 43, Issue 5, 2020, Pages 355-372

[17] Kocicioglu, I.Isak & Nasiri Khalaji, Mansour & Uğurlu, Ahmet & Doğan, Nihat. (2013). Ansys-fluent investigation of heat and flow in cross-flow heat exchanger (HRV) with angle square pin-fins.

[18] Rao, P.D., Nageswara Rao, B., CFD simulations and validation through test data of a double pipe counter flow heat exchanger, International Journal of Mechanical Engineering and Technology Volume 8, Issue 5, May 2017, Pages 818-831

[19] Ridha, H., Oleiwi Ch., Numerical investigation for liquid - Solid inclined fluidized bed, International Journal of Heat and Technology, Volume 38, Issue 1, March 2020, Pages 137-144, ISSN: 03928764

[20] Valaparla, R.K., Balasubramanian, K., Kiran Kumar, K., Numerical Investigation of Heat Transfer and Fluid Flow Characteristics in Circular Wavy Microchannels with Sidewall Rib, Volume 15, Issue 2, 1 June 2020, Article number 20190052, ISSN:2194615

[21] Doroudi, Shahed & Bussmann, Markus & Tandra, Danny & Tran, Honghi. (2014). ANSYS Fluent for Modeling Sootblower Jets.

[22] Zou, Ying & Zhao, Xingwang & Chen, Qingyan. (2017). Comparison of STAR-CCM+ and ANSYS Fluent for simulating indoor airflows. Building Simulation. 11. 10.1007/s12273-017-0378-8.

[23] Wijayanta, A.T.; Kristiawan, B.; Pranowo; Premono, A.; Aziz, M. Computational Fluid Dynamics Analysis of an Enhanced Tube with Backward Louvered Strip Insert. Energies 2019, 12, 3370.

[24] Computational Fluid Dynamics (CFD) Simulation, https://www.ansys.com/products/fluids

[25] https://www.ansys.com/

[26] https://www.resolvedanalytics.com/theflux/comparing-cfd-software-part-4-comprehensive-cfd-software-packages

[27] https://www.comsol.com/

[28] CFD Direct - The Architects of OpenFOAM, https://cfd.direct/

[29] Welahettige, Prasanna & Vaagsaether, K.. (2018). Comparison of OpenFOAM and ANSYS Fluent. 1005-1012. 10.3384/ecp171421005

[30] https://www.openfoam.com/