

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/326264731>

A review on applications of computational fluid dynamics

Article · July 2018

CITATIONS

2

READS

5,487

3 authors:



[Ram Kumar Raman](#)

2 PUBLICATIONS 2 CITATIONS

[SEE PROFILE](#)



[Yogesh Dewang](#)

Maulana Azad National Institute of Technology, Bhopal

39 PUBLICATIONS 45 CITATIONS

[SEE PROFILE](#)



[Jitendra Raghuwanshi](#)

Lakshmi Narain College of Technology BHOPAL

8 PUBLICATIONS 6 CITATIONS

[SEE PROFILE](#)

Some of the authors of this publication are also working on these related projects:



Concepts in Engineering Design [View project](#)



Optimization tool box [View project](#)

A review on applications of computational fluid dynamics

Ram Kumar Raman¹, Yogesh Dewang² and Jitendra Raghuwanshi²

M.Tech Scholar, Department of Mechanical Engineering, LNCT Bhopal, India¹

Assistant Professor, Department of Mechanical Engineering, LNCT Bhopal, India²

Abstract

Computational fluid dynamics (CFD) is well established as a tool of choice for solving problems that involves one or more of the following phenomena: flow of fluids, heat transfer, mass transfer and chemical reaction. The present work reviews the salient features as well as practical application of CFD techniques carried out in the last more than 50 years in various sections related to industry with the objective of how and where it can be applied. It is concluded that CFD presents a number of opportunities in industrial work analysis, and the scope of this opportunity will further develop as both computational hardware and software resources becomes more advanced.

Keywords

CFD, Flow of fluids, Heat transfer, Mass transfer, Chemical reaction, Scope, Opportunity, Practical application.

1.Introduction

Computational Fluid Dynamics (CFD) is the combination of physics, flow technology, computer applications, mathematics and mechanics. It is a group of techniques aimed at solving the Navier-Stokes equations (or strictly, Reynolds-Averaged Navier-Stokes equations in most cases), thereby satisfying the conservation of mass, momentum and energy to predict the behavior of fluidic systems. In its modern guise as Computer-aided engineering (CAE) software, CFD presents itself as a useful tool for investigating domain space for physical system design and performance variables, and for diagnosing or troubleshooting system behavior. Typical scenarios where the application of CFD may complement or replace existing analytical techniques are when a high number of design variations are to be analyzed or where physical testing may be prohibited due to restricting factors, such as scale, cost, accessibility, or the presence of physical or environmental hazards. CFD is especially prevalent in cases, where modeling methodologies have been previously validated or operational data for validation is easily obtainable. Key factors influencing the uptake of simulation techniques such as CFD have included business and legislative drivers demanding technological development and efficiency improvements. The advancement and increased accessibility of knowledge, techniques and

computational resources from academia, software and hardware developers have also driven the use of such simulation techniques. In past years, a number of researchers contributed in the area of CFD. Shivakumara et al. (2017) [1] found that by mixing two fluids at different temperature, a spatial and time temperature fluctuation occurs. If this fluctuation is high, it may cause damages to the structure due to high cycle thermal fatigue and it is called as thermal stripping phenomena. Computational Fluid Dynamics (CFD) is also used alternatively, as it reduces the number of experiments required, cost and time required for the designing process. Laohasurayodhin et al. (2014) [2] find out the effect of angle of inclination on flow of fluid in a pipe by considering various viscous models through numerical simulation. Klein (1995) [3] carried out internal flow analysis of a dump diffuser used in marine gas turbines and modern aircraft engines by k- ω turbulence model in ANSYS FLUENT. They found that there will be no effect of diffuser when the diffuser angle is increased. Nimadge and Chopade (2017) [4] investigated the steady, incompressible fluid flow through a T-junction by CFD technique. They prepared experimental setup to obtain the reference data when fluid passes through T-junction of pipe and also utilized same data for CFD analysis through FLUENT and ANSYS are used for that purpose. Spooner et al. (2017) [5] developed manifold arrangements with bifurcations/trifurcations which lead to head losses in a fluid system and generated quantitative data of sharp-edged bifurcation loss coefficients obtained through Computational Fluid Dynamics (CFD). Gedik (2017) [6] undergone experimental and numerical study of Magnetorheological (MR) fluids flow in circular pipes under the influence of uniform magnetic field by Computational Fluid Dynamics (CFD). Patel et al. (2005) [7] investigated the effect of pressure developed on the pipe wall under different pipe-networks namely Y-junction, elbows, T-junctions, bends, contractions, expansions, valves and many other components through CFD technique. Banjara et al. (2017) [8] investigated the effect of mass flow rate and velocity of turbulent fluid flow along length of pipe in a trifurcation pipe branch under different Reynolds's number using CFD analysis. Acharya (2016) [10] conducted experiments for fluid flow and

measured volumetric flow by analogue flow meter for obtaining the average velocity of fluid in a pipe and also employed CFD software COMSOL for solution of Navier-Stokes incompressible fluid flow. Mandal et al. (2014) [11] utilized CFD software namely ANSYS FLUENT for analyzing the flow characteristics of a pair of immiscible liquids (moderately viscous oil and water) through horizontal pipeline and found that in the annular flow, total pressure of the mixture decreases with increase in oil velocity due to the fact that pipe cross section is completely wetted with water. Ramos et al. (2014) [12] investigated the effect of mesh independence in a three-dimensional pressurized fluid flow for velocity profiles and found most efficient meshes for laminar and turbulent flow. Hosseini et al. (2014) [13] conducted CFD analysis of laminar single-phase flow of water in a hollow helical pipe at under various Reynolds numbers and found that with increasing Reynolds number and creation of centrifugal forces, a high velocity and pressure region occurs between two tubes, at the outer side of the hollow helical pipe walls. Besides, they observed that friction factor decreases as the tendency for turbulence increases. Kumar [14] utilized CFD software namely ANSYS for point by point study of flow through pipe and also for determination of losses in head due to change in geometry of pipe. Sehgal et al. (2013) [15] studied the effect of bend angle, pipe diameter, pipe length and Reynold's number on resistance coefficient through CFD analysis and found that resistance coefficient varies with change in flow parameters. Mandal (2013) [16] applied the technique of Computational fluid dynamics (CFD) simulation to investigate the transition boundaries of different flow patterns for moderately viscous oil-water two-phase flow through a horizontal pipeline. Hirani and Kiran (2013) [17] utilized ANSYS CFX software to investigate the effect of turn/bend for a Y-shape pipe and found that resistance coefficient vary with the change in flow.

Sochi (2013) [18] found through CFD simulation that branching junctions especially plays important kinematic and dynamic roles in blood transportation vasculature and industrial flow systems. in and Li (2013) [19] studied the flow fields of T-junction and Y-junction using shear stress transport (SST) model through ANSYS/CFX software. They found that the variation rule of velocity peak in T-junction with different frequencies and phase-differences. Prabhakar (2012) [20] utilized CFD simulation software ANSYS and is proposed an idea for

establishment of relationship among angular velocity, Reynolds Number and drag coefficient for fluid flow at varying conditions. Motlagh et al. (2012) [21] presented an application of residual-based variational multiscale modeling methodology for computation of laminar and turbulent fluid flow through concentric annular pipe flows. Ismail and Adewoye (2012) [22] investigated some important works done on numerical analysis and modeling of laminar flow in pipes. The objective of the present work is to review the various aspects of applications of Computational Fluid Dynamics (CFD) technique in various industrial sectors. In this paper the major emphasis is drawn on to the contributions made in the area of fluid flow through heat exchangers, branched-pipes and bifurcated blood vessels.

2. Overview of computational fluid dynamics

2.1 Computational Fluid Dynamics

Computational Fluid Dynamics is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. It is usually abbreviated as CFD and is defined as a branch of fluid mechanics that solves and analyzes fluid flow problems, using numerical methods and algorithms. In order to perform the calculations required to simulate the fluid surface interaction, defined by boundary conditions, computers need to be employed. High-speed supercomputers can applied be for finding better and fast solutions. The fundamental basis of almost each and every CFD problems is linked with the Navier-Stokes equations. These define almost any single-phase fluid flow. The Navier-Stokes equations can be simplified by removing certain parameters, which describe viscosity and leads to Euler equations. Further simplification can be done by removing the parameters that describe vorticity and hence yields the full potential equations. After the simplifications are done, these equations can be then linearized so as to obtain the linearized potential equations.

2.2 Numerical calculation method

From last few decades, there are varieties of numerical solutions had emerged. The main difference lies in the regional discrete approach, the equation of discrete and algebraic equations for solution. The numerical methods used in CFD numerical solution are: finite difference method, finite volume method and finite element method. The most widely solver method for CFD is the finite volume method.

2.3 Finite Difference Method

Finite difference method (FDM) is the earliest method of computer numerical simulation. The method divides the solution region into a separation grid, and uses a finite number of grid nodes instead of a continuous solution area. The finite difference method is used to discretize the derivative of the control equation with the difference of the function value on the grid node to establish the algebraic equations with unknown values on the grid nodes. The method is an approximate numerical solution that directly turns the differential problem into algebraic problem.

2.4 Finite volume method

Finite Volume Method (FVM), also known as the control volume method, is a commonly used method for spatial discretization. It is mainly from the equations of conservation type, and make integral in its control volume, and further solve the conservation equation in integral form. These all control differential equations have a common form, this form of consistency is the basis of a common solution.

2.5 Finite element method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. It is based on the variation principle and the weighted margin method. The basic solution idea is to divide the calculation area into a finite number of non-overlapping units. In each unit, select some of the appropriate nodes as the interpolation point for the function. Then, the variables in the differential equation are rewritten as a linear expression (shape function) consisting of the node value of each variable or its derivative and the selected interpolation function. Finally, the differential equation is discretized and solved by means of the variation principle or the weighted margin method. Different finite element methods are used to form different weight functions and interpolation functions.

2.6 Softwares of CFD

- ADINA
- ANSYS
- AUSM
- Avizo (software)
- FLACS
- OpenFOAM
- TELEMAT

3. Practical application of computational fluid dynamics in

3.1 Biomedical Engineering

Computational fluid dynamics is widely used to solve complex problems in biomedical field. CFD is becoming a key component in developing update designs and optimizations through computational simulations, resulting in lower operating costs with enhanced efficiency. Many simulations and clinic result have been used to study the analyses in biomedical applications, particularly in blood flow and nasal airflow. The study of blood flow analysis includes the circulations of blood of ventricle functions, coronary artery and heart valves. Meanwhile, the nasal airflow analysis consist of the basic airflow in humane nose, drug delivery improvement and virtual surgery. Various applications of CFD in biomedical engineering are listed below.

- Heart pumping
- Blood flows through arteries and vein
- Air flow in lungs
- Joint lubrication
- Cell-fluid interface
- Tendon-sheath
- Gas exchanger
- Artificial organ design
- Vocal tract analysis
- Perfusion in tissues
- Life support system
- Nose and sinus flows
- Spinal fluid flow
- Cardiac valve design
- Microbe locomotion

3.2 Mechanical engineering

- **Heat exchanger:** Computational fluid dynamics has been employed for the following areas of study in various types of heat exchangers: fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase.
- **Fluid flow throw pipe:** Computational fluid dynamics is used to determine velocity and pressure of flow of fluid in a pipe and compare it with the result obtained in the laboratory i.e. fluid flow module. It is also used to observe the heat transfer during the flow of fluids in a pipe and compare it with the results obtained in the laboratory i.e. heat transfer module.
- **Combustion in IC engines:** Computational fluid dynamics is used in the development of model of a spark ignition (SI) engine and the application of the engine model into an undergraduate internal combustion (IC) engine. The development of

model helps engineering students better understand the combined effects of chemical reactions, species transport, flow patterns and temperature distributions in the SI engines.

- **Aerodynamics of aircraft:** Lift and drag
- **Automotive:** External flow over the body of a vehicle or internal flow through the engine, combustion, Engine cooling
- **Turbo machinery:** Turbines, pumps, compressors
- Flow and heat transfer in thermal power plants and nuclear power reactors
- HVAC
- Manufacturing–Casting simulation, injection molding of plastics
- Marine engineering: loads on off-shore structures
- Hydrodynamics of ships, submarines, torpedo etc.

3.3 Missile engineering

- Due to the increased complexity of contemporary buildings and a Characterising missile airframe aerodynamic loads (force and moments) and the aerothermal environment is acritical aspect in missile engineering. The aerodynamic information is used to assess stability and control, mission performance and maneuverability.
- CFD is employed to supplement and extend test results of experimental approach.
- In addition, the flow visualization provided by CFD can be invaluable in all phase of wind tunnel testing. Surface pressure distributions, wake line filaments and other visualization aids provide opportunities to “observe” the flow and its interaction with various airframe components.
- CFD bridge added intelligence to critical design decisions during the wind tunnel testing phase. For example, inviscid computations significantly assist in the scale-model design.

3.4 Architecture

- Growing interest in improving building performance in terms of the environmental impact CFD technique is used to study internal as well as external airflow.
- Earlier wind tunnel testing (WTT) was the only method followed for external airflow analysis but nowadays WTT is replaced by CFD because with the help of WTT values at few points where obtained but in case of CFD values at various points can be obtained.

3.5 Food Industry

Sterilization:

- It is known that consumer demands for food products focus on safety, product quality and cost. Therefore, it is of great necessity quality and assure safety of the food supply. Sterilization is an important technique for food storage and preservation. CFD can be used to study both temperature distribution and flow pattern of food in the process of sterilization so as to optimize the quality of food products.

Refrigeration:

- The consumption of frozen foods has increased continually in the past years because frozen foods have demonstrated good food quality and safety record.
- Refrigeration can slow down bacterial growth and preserve food.
- Therefore, researchers have recently applied CFD in the modelling of heat and mass transfer in foods during refrigeration (chilling and freezing).

3.6 Other Applications

- **Automobile industry:** CFD has emerged as a powerful tool for predicting flow and thermal distributions in vehicle systems. It is being accepted more and more in day-to-day design of automotive components. It can significantly reduce the need for prototype test, as well as cost and design cycle time. Once a model has been created, it is usually very easy to determine the effects of geometry changes, flow parameter changes and material property changes. With continued developments in computer hardware and numerical techniques, CFD can be accepted to be accepted like Finite Element Analysis (FEA) in the solid mechanics word.
- **Turbomachinery design:** CFD has been increasingly used to successfully design turbomachinery as well as carry out analysis and validation requirements for existing design.
- **Chute:** Chute (gravity), a channel down which falling materials of fluids are guided. Air velocity distributions and air mass flow rates can be predicted through CFD software.
- Ocean current
- Defense
- Health care
- Water waste treatment
- Loads on offshore structure in marine engineering
- Weather prediction in meteorology
- Flow inside rotating passages and diffusers in turbo-machinery

- External and internal environment of buildings like wind loading and heating or Ventilation system
- Mixing and separation or polymer moldings in chemical process engineering
- Distribution of pollutants and effluent in environmental engineering
- Chemical engineering

4. Benefits and limitations of CFD

Benefits

Insight- if there is a device or system design which is difficult to analyze or test through experimentation, CFD analysis enables us to virtually sneak inside the design and see how it performs. CFD gives a deep perception into the designs. There are many occurrences that we can witness through CFD which wouldn't be visible through any other means.

Foresight- under a given set of circumstances, we can envisage through the CFD software what will happen. In a short time we can predict how the design will perform and test many variants until we arrive at an ideal result. Efficiency- the foresight we gain helps us to design better to achieve good results. CFD is a device for compressing the design and development cycle allowing for rapid prototyping.

Limitations

Even if there are many advantages of CFD, there are few shortcomings of it as follows:

CFD solutions rely upon physical models of real world processes.

- Solving equations on a computer invariably introduces numerical errors.
- Truncation errors due to approximation in the numerical models.
- Round-off errors due to finite word size available on the computer.
- The accuracy of the CFD solution depends heavily upon the initial or boundary conditions provided to numerical model.

5. Result and discussion

Computational fluid dynamics presents different types of results. In this paper result attention is focused mainly on diffuser angle. The diffuser angle is changed and other geometries are kept constant. Different diffuser angles taken are 5°, 10°, 16°, 20° and 22°. The effect of diffuser that is decrease in velocity and increase in static pressure is taken into account and accordingly diffuser angles are picked. As the diffuser angles is below 5° and more than 22° the effect of diffuser will be absent. Pressure contours, velocity contours, turbulent kinetic energy

contours, streamlines and velocity vectors are represented on the different planes created.

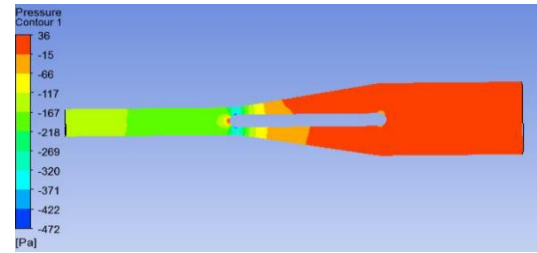


Figure 1 Pressure contour along plane at 10° diffuser angle

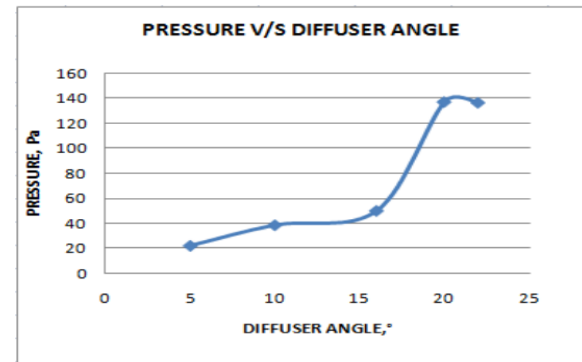


Figure 2 Pressure versus diffuser angle

Figure 2 shows the pressure versus diffuser angle plot. Pressures obtained during analysis for different diffuser angles are plotted. Pressure gradually increases for every diffuser angle change. From 5° diffuser angle to 22° diffuser angle pressure increases. This gradual increase can be seen in the graph plotted. Due to the flow recirculation and boundary layer separation the pressure gradient increases while the velocity of the fluid decreases. Boundary layer separation takes place when the flow near the wall or edge recirculates or reverses in the diffuser.

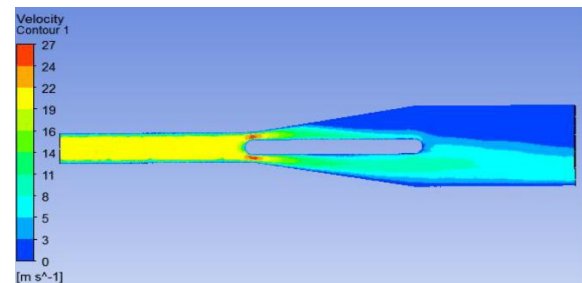


Figure 3 Velocity contour along plane at 10° diffuser angle

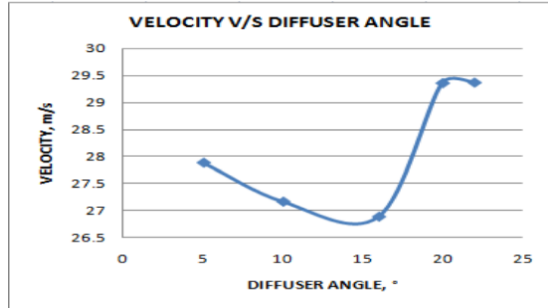


Figure 4 Velocity versus diffuser angle

Figure 4 shows the velocity versus diffuser angle plot. Velocities obtained during analysis for different diffuser angles are plotted. From diffuser angle 5° to 16° the velocities goes on decreasing. From 16° to 22° the velocity keeps on increasing. From this we can tell that diffuser angle more than 16° no more behaves like a diffuser. Effect of diffuser is no more after 16°. Due to the boundary separation the velocity gradient decreases or sometimes goes to negative velocity gradient. Boundary layer separation depends on the distribution of undesirable velocity gradient along the surface. Due to flow recirculation the velocity in the diffuser reduces.

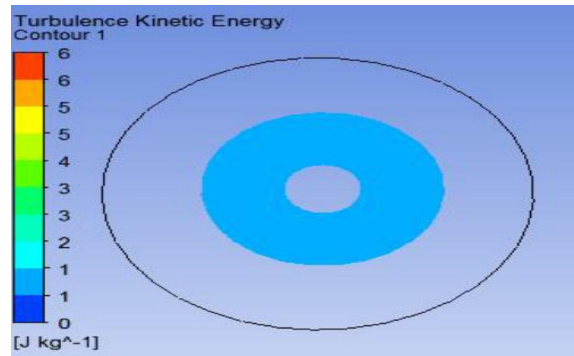


Figure 5 Turbulence kinetic energy along plane at 10° diffuser angle

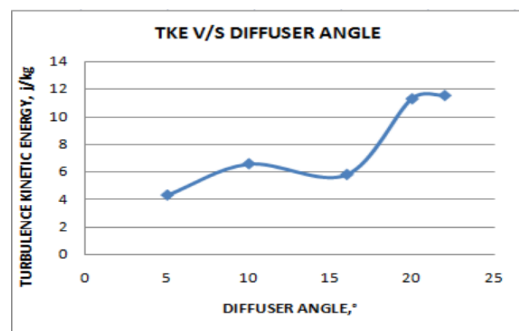


Figure 6 Turbulence kinetic energy versus diffuser angle

Figure 6 shows turbulence kinetic energy versus diffuser angle plot. Turbulence kinetic energy obtained for different diffuser angles are presented on the graph. Turbulence kinetic energy increases as the diffuser angle increases. From 10° diffuser angle to 16° diffuser angle the turbulence kinetic energy decreases slightly and for diffuser angle more than 16° turbulence kinetic energy increases. It is associated with eddies as the flow in the diffuser is turbulent. As eddies produced near the walls and edges increases the turbulence kinetic energy also increases.

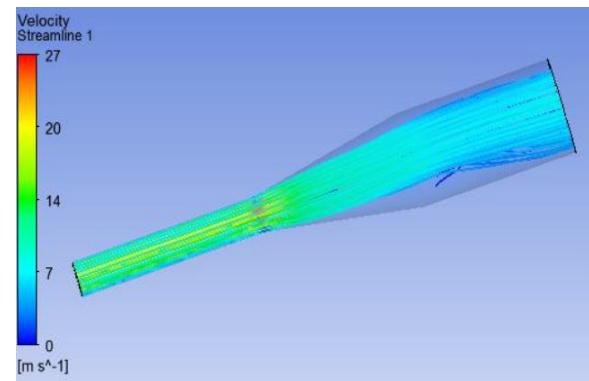


Figure 7 Streamlines along plane at 10° diffuser angle

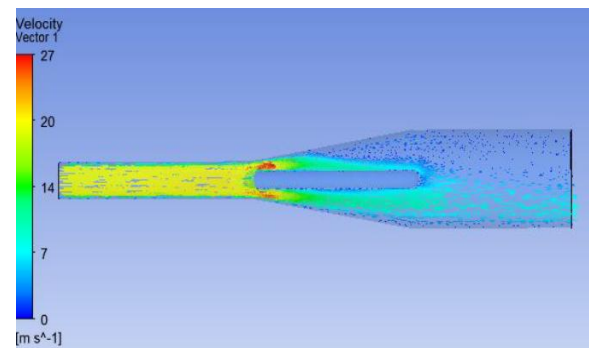


Figure 8 Velocity vectors along plane at 10° diffuser angle

From the above figures and analysis for the 10° diffuser angle the maximum velocity is 27.17 m/s, pressure is 38.75 Pa and turbulent kinetic energy is 6.55 J/kg. Similarly analysis has been carried out for different diffuser angles. Streamlines and vectors are also represented for all the diffuser angles. Due to the flow recirculation and boundary layer separation the velocity gradient of the fluid decreases and pressure gradient increases in the diffuser.

6. Conclusion and future work

Computational Fluid Dynamics (CFD) is already gaining importance in the industry. Some of the companies reaping benefits of the technology are 3M; Air products; Argonne National Lab; Bechtel; BP Amoco; Chemineer; Cray; Dow Chemical; Dow Corning; DuPont; Eastman Chemical etc. In the body of work presented in this paper, CFD is studied by considering various aspects of problems related to industries as well as other areas. A brief and concise review of the contributions made by the previous researchers in the area of CFD is also presented. Some researchers show that the increasing developments of computational fluid dynamics in recent years have opened the possibilities of low cost. There has been considerable growth in the development and application of CFD recently in the area of drying, sterilization, mixing and refrigeration. However, the simulation results should be validated by experiments because CFD use many approximate models as well as a few assumptions. Although there are still some obstacles such as inability in accurate simulation of large 3-D problems on an affordable computer, in particular, in large-scale sophisticated plants, the trend of widespread application of CFD in the industrial processing will continue in the 21st century.

References

- [1] Shivakumara N .V., kumar Sanath K.H., Kumara swamy K.L.. CFD analysis of t pipe junction in nuclear reactor cooling circuit. *International Journal of Innovative Research in Science, Engineering and Technology*. 2017.
- [2] Laohasurayodhin R, Diloksumpan P, Sakiyalak P, Naiyanetr P. Computational fluid dynamics analysis and validation of blood flow in Coronary Artery Bypass Graft using specific models. In *Biomedical Engineering International Conference (BMEiCON)*, 2014 (pp. 1-4). IEEE.
- [3] Klein A. Characteristics of combustor diffusers. *Progress in Aerospace Sciences*. 1995; 31(3):171-271.
- [4] Nimadge MG, Chopade MS. CFD analysis of flow through T-junction of pipe. *International Research Journal of Engineering and Technology (IRJET)*. 2017; 4:2395-0056.
- [5] Sukhapure K, Burns A, Mahmud T, Spooner J. Computational fluid dynamics modelling and validation of head losses in pipe bifurcations.
- [6] Gedik E. Experimental and numerical investigation on laminar pipe flow of magneto-rheological fluids under applied external magnetic field. *Journal of Applied Fluid Mechanics*. 2017; 10(3).
- [7] Patel T, Singh SN, Seshadri V. Characteristics of Y-shaped rectangular diffusing duct at different inflow conditions. *Journal of aircraft*. 2005; 42(1):113-20.
- [8] Zhang Y, Bazilevs Y, Goswami S, Bajaj CL, Hughes TJ. Patient-specific vascular NURBS modeling for isogeometric analysis of blood flow. *Computer methods in applied mechanics and engineering*. 2007; 196(29-30):2943-59.
- [9] Gujarathi YS. A comprehensive study on numerical and computational aspects of turbulence modelling.
- [10] Acharya S. Analysis and FEM Simulation of Flow of Fluids in Pipes: Fluid Flow COMSOL Analysis.
- [11] Desamala AB, Dasamahapatra AK, Mandal TK. Oil-water two-phase flow characteristics in horizontal pipeline—a comprehensive CFD study. *International journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering, World Academy of Science, Engineering and Technology*. 2014; 8:360-4.
- [12] Martins NM, Carriço NJ, Covas DI, Ramos HM. Velocity-distribution in pressurized pipe flow using cfd: mesh independence analysis. In *Third IAHR Europe Congress, Porto, Portugal, Apr 2014* (pp. 14-6).
- [13] Ahmadloo E, Sobhanifar N, Hosseini FS. Computational Fluid Dynamics Study on Water Flow in a Hollow Helical Pipe. *Open Journal of Fluid Dynamics*. 2014; 4(2):133.
- [14] Kumar VI. Simulation and flow analysis through different pipe geometry (Doctoral dissertation).
- [15] Singh B, Singh H, Sebgal SS. CFD analysis of fluid flow parameters within a Y-shaped branched pipe. *International Journal of Latest Trends in Engineering and Technology (IJLTET)*. 2013; 2(2):313-7.
- [16] Desamala AB, Dasari A, Vijayan V, Goshika BK, Dasmahapatra AK, Mandal TK. CFD simulation and validation of flow pattern transition boundaries during moderately viscous oil-water two-phase flow through horizontal pipeline. In *proceedings of world academy of science, engineering and technology 2013* (p. 1150). World Academy of Science, Engineering and Technology (WASET).
- [17] Hirani AA, Kiran CU. CFD simulation and analysis of fluid flow parameters within a Y-Shaped Branched Pipe. *IOSR Journal of Mechanical and Civil Engineering*, 10 (1). 2013:31-4.
- [18] Sochi T. Fluid flow at branching junctions. *International Journal of Fluid Mechanics Research*. 2015; 42(1).
- [19] Li X, Wang S. Flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic pipe system. *Chinese Journal of Aeronautics*. 2013; 26(4):1080-92.
- [20] Prabhakar R. CFD analysis of newtonian fluid flow phenomena over a rotating cylinder (Doctoral dissertation).
- [21] Motlagh YG, Ahn HT, Hughes TJ, Calo VM. Simulation of laminar and turbulent concentric pipe flows with the isogeometric variational multiscale method. *Computers & Fluids*. 2013; 71:146-55.
- [22] Ismail OS, Adewoye GT. Analyses and modeling of laminar flow in pipes using numerical approach. *Journal of Software Engineering and Applications*. 2012.