The Netherlands Press
Journal of Artificial Intelligence in Fluid Dynamics

Article

An Overview on Applications of Computational Fluid Dynamics

Zaizul Ab Rahman

Research Centre for Theology and Philosophy
Faculty of Islamic Studies, Universiti Kebangsaan Malaysia,
Selangor, MALAYSIA.
E-mail: zaizul@ukm.edu.my

Abstract. CFD – Computational Fluid Dynamics was established as a method of selection for solving issues that involves one/more of following phenomena: fluids flow, transfer of heat, transfer of mass & chemical reaction. The current study covers the prominent aspects as well as actual CFD techniques application worked out over the last more than 5 decades in different sections related to industry with the purpose of how & where it might be implemented. As computer hardware and software resources improve, it is expected that CFD will provide even more chances for analysing industrial activity.

Keywords: CFD, Fluids flow, Transfer of Heat, Transfer of Mass.

Journal of Artificial Intelligence in Fluid Dynamics Volume 1 Issue 1
Received 30 May 2022
Accepted 27 June 2022
Published 10 September 2022
Online at https://www.jaifd.com/

1. Introduction

The field of CFD – Computational Fluid Dynamics draws upon a wide range of disciplines, including but not limited to flow technology, physics, computer applications, mathematics as well as mechanics. Specifically, it refers to a set of methods for predicting the behaviour of fluid systems by resolving the Navier-Stokes equations which ensures that mass, momentum, and energy are conserved. CFD, in its latest version as Computer-aided engineering (CAE) software, presents itself as a helpful instrument for exploring the design and performance domain of physical systems, as well as for diagnosing and troubleshooting system behaviour. When a large number of design iterations must be analysed or when physical testing is impractical due to considerations like size, expense, accessibility, or the existence of physical / environmental risks, CFD has the potential to serve as a useful supplementary or even replacement analytical approach. Flow technology, physics, computer applications, mathematics, and mechanics are just few of the fields that inform CFD – Computational Fluid Dynamics. To be more specific, it is a group of techniques for forecasting the behaviour of fluid systems by solving the Navier-Stokes equations in a way that preserves mass, momentum, and energy. Modern computational fluid dynamics (CFD) software in the form of Computer-aided engineering (CAE) applications shows itself as a useful tool for investigating the design and performance domain of physical systems and for detecting and fixing behavioural issues. CFD has the potential to be a valuable supplemental or even replacement analytical approach when a large number of design iterations must be analysed or when physical testing is impossible for reasons including size, cost, accessibility, or the presence of physical / environmental concerns. In situations where validation data for modelling approaches already exists or when operational data for validation can be easily accessed, CFD is particularly common. Major impetuses for the development of simulation techniques such CFD have been business and legal drives needing technological progress & efficiency improvements. The adoption of such simulation methods has also been pushed along by the improvement and expansion of academic knowledge, techniques, and computational resources. CFD is widely used in circumstances wherever validation data for modelling approaches already exists, or when operational data for validation can be easily acquired. Business and legal imperatives for technical advancement and efficiency improvements have been major impetuses for development of simulation techniques such as CFD. Increases in available computer capabilities, as well as advancements in the state of the art in terms of both theory and practise, have facilitated the widespread implementation of such simulation methodologies. Several scholars have made important contributions to CFD over the years. CFD – Computational Fluid Dynamics was established as a method of selection for solving issues that involves one/more of following phenomena: fluids flow, transfer of heat, transfer of mass & chemical reaction. The current study covers the prominent aspects as well as actual CFD techniques application worked out over the last more than 5 decades in different sections related to industry with the purpose of how & where it might be implemented. As computer hardware and software resources improve, it is expected that CFD will provide even more chances for analysing industrial activity. When two fluids of different temperatures are combined, temperature changes both in space and time, as discovered by Shivakumara et al. (2017) [1]. There is a phenomenon known as thermal stripping that can occur if this variation is large enough and is caused by high cycle thermal fatigue. As an alternative, computational fluid dynamics (CFD) is utilised since it cuts down on the need for costly and time-consuming trials during the design phase. By taking into account several different viscosity models, Laohasurayodhin et al. (2014) [2] determine the impact of pipe inclination on fluid flow via numerical simulation. Dump diffusers are employed in maritime gas turbines and cutting-edge aircraft engines, and Klein (1995) [3] performed an internal flow analysis of one using the k- turbulence model in ANSYS FLUENT. Researchers discovered that a larger diffuser angle had no effect. By using CFD, Nimadge and Chopade (2017) [4] looked into the flow of a steady, incompressible fluid via a T-junction. They set up an experiment to collect data on the flow of fluid at a T-junction of a pipe and then used those measurements to inform a CFD analysis carried out with the help of FLUENT and ANSYS. Computational fluid dynamics was used by Spooner et al. (2017) [5] to create computer simulations of manifold topologies with bifurcations/trifurcations that lead to head losses in a fluid system, and from those simulations extract quantitative information about the loss coefficients at the bifurcation's sharp edges. Computational Fluid Dynamics (CFD) was used by Gedik (2017) [6] to examine the flow of magnetorheological (MR) fluids in circular pipes subjected to a uniform magnetic field (CFD). Patel et al. (2005) [7] utilised CFD to examine the impact of pressure created on the wall of the pipe under various pipe networks, including Y-junctions, elbows, T-junctions, bends, contractions, expansions, valves, and many other components. Researchers Banjara et al. (2017) [8] a computational fluid dynamics study was conducted to examine the effect of mass flow rate & turbulent fluid flow velocity all along length of a pipe in a trifurcation at different Reynolds' numbers. Acharya (2016) [10] used CFD software COMSOL to solve Navier-Stokes incompressible fluid flow and conducted fluid flow experiments, measuring volumetric flow using an analogue flow metre to estimate the average velocity of fluid in a pipe. Mandal et al. (2014) [11] utilised CFD software, specifically ANSYS FLUENT, to evaluate the flow behaviour of a pair of immiscible fluids through such a horizontal pipeline, and they found that the total pressure of mixture decreases with the increasing in oil velocity in annular flow. Three-dimensional pressured fluid flow velocity profiles were studied by Ramos et al. (2014) [12], who also identified the most effective meshes for laminar and turbulent flow. They also found that the inclination for turbulence decreases friction factor. When analysing the flow through a pipe and calculating the losses in head caused by a change in the pipe's geometry, Kumar [14] used a CFD programme called ANSYS. Through CFD research, Sehgal et al. (2013) [15] determined that the resistance coefficient varies with changes in flow parameters such bend angle, pipe diameter, pipe length, and Reynold's number. Mandal (2013) [16] utilised CFD modelling to examine the boundaries of transitional flow patterns for moderately viscous oil-water two-phase flow over a horizontal pipeline. Resistance coefficient varies with the change in flow, as was discovered by Hirani and Kiran (2013) [17] using ANSYS CFX software to study the effect of turn/bend for a Y-shaped pipe. According to the results of a computational fluid dynamics (CFD) simulation study conducted by Sochi (2013) [18], branching junctions play crucial kinematic and dynamic functions in blood transport vasculature and industrial flow systems. T-junction and Y-junction flow fields were analysed by in and Li (2013) [19] using the shear stress transfer (SST) model in ANSYS/CFX. Different frequencies and phase variations were found to cause the velocity peak at the T-junction to change in a predictable manner. For fluid flow under different situations, Prabhakar (2012) [20] used the CFD simulation software ANSYS and developed a concept for establishing a link between rotational velocity, Reynolds Number, and drag coefficient. Computation of laminar and turbulent fluid flow via concentric annular pipes was presented by Motlagh et al. (2012) [21] using a residual-based variational multiscale modelling approach. Important research on numerical analysis & modelling of laminar flow in pipes were examined by Ismail and Adewoye (2012) [22]. The purpose of this paper is to provide a comprehensive overview of the Computational Fluid Dynamics (CFD) technique and its many practical applications across a wide range of industries. Fluid flow in heat transfer, branched-pipe systems, & bifurcated blood arteries all are discussed, as are the contributions made to our understanding of these phenomena. Sochi (2013) [18] conducted a CFD simulation investigation showing that branching junctions have important translational and rotational functions in blood transport vasculature as well as industrial flow systems. Using the shear stress transfer (SST) model in ANSYS/CFX, in and Li (2013) [19] analysed the flow fields at a T-junction and a Y-junction. Changing the frequency or phase of an oscillator causes the velocity peak just at T-junction to shift in a predictable way. Prabhakar (2012) [20] used the CFD simulation software ANSYS to develop a notion for establishing a relationship between rotational velocity, the Reynolds Number, and the drag coefficient for fluid flow in a variety of scenarios. Motlagh et al. (2012) [21] introduced a residual-based finite difference multiscale modelling technique for computing laminar and turbulent fluid flow through concentric annular pipes. Ismail and Adewoye (2012) [22] studied pivotal studies on numerical analysis and modelling of laminar flow in pipes. This paper's goal is to introduce readers to the Computational Fluid Dynamics (CFD) technique and discuss its many real-world applications in a variety of fields. Our progress in understanding fluid flow in heat transfer, branched-pipe systems, and bifurcated blood arteries is discussed. Also, computational fluid dynamics is used in estimation of convective heat transfer coefficients in nanofluids [23, 24, 25].

Three-dimensional pressured fluid flow velocity profiles were studied by Ramos et al. (2014) [12], who also identified the most effective meshes for laminar and turbulent flow. They also found that the inclination for turbulence decreases friction factor. When analysing the flow through a pipe and calculating the losses in head caused by a change in the pipe's geometry, Kumar [14] used a CFD programme called ANSYS. Through CFD research, Sehgal et al. (2013) [15] determined that the resistance coefficient varies with changes in flow parameters such bend angle, pipe diameter, pipe length, and Reynold's number. Mandal (2013) [16] utilised CFD modelling to examine the boundaries of transitional flow patterns for moderately viscous oil-water two-phase flow over a horizontal pipeline. Major impetuses for the development of simulation techniques such CFD have been business and legal drives needing technological progress & efficiency improvements. The adoption of such simulation methods has also been pushed along by the improvement and expansion of academic knowledge, techniques, and computational resources. Several scholars have made important contributions to CFD over the years. The purpose of this paper is to provide a comprehensive overview of the Computational Fluid Dynamics (CFD) technique and its many practical applications across a wide range of industries. Fluid flow in heat transfer, branched-pipe systems, & bifurcated blood arteries all are discussed, as are the contributions made to our understanding of these phenomena.

2. Overview of Computational Fluid Dynamics

2.1. Computational Fluid Dynamics

Computational fluid dynamics is the use of computer simulation to study systems with fluid flow, heat transfer, and associated phenomena like chemical reactions. Commonly abbreviated as "CFD," this subfield of fluid mechanics focuses on the use of numerical methods and algorithms to the study of fluid flow issues. It is necessary to use computers to do the computations necessary to model the fluid surface contact, determined by boundary conditions. Supercomputers with massive processing power can be used to quickly and accurately solve problems. The Navier–

Stokes equations form the backbone upon which nearly all CFD problems are built. One-phase fluid flows are characterised by these characteristics. Eliminating the parameters that describe viscosity in the Navier–Stokes equations yields the simpler Euler equations. Parameters describing vorticity can be removed to simplify the equations for full potential further. In order to produce the linearized potential equations, these equations must first be simplified.
It is necessary to use computers to do the computations necessary to model the fluid surface contact, determined by boundary conditions. Supercomputers with massive processing power can be used to quickly and accurately solve problems. The Navier–Stokes equations form the backbone upon which nearly all CFD problems are built.

To investigate systems including fluid movement, heat transfer, as well as associated phenomena like chemical reactions, researchers have turned to a field called computational fluid dynamics. In this branch of fluid mechanics, known as computational fluid dynamics (CFD), numerical approaches and methods are used to examine fluid flow concerns. The computations required to model the fluid surface contact, governed by boundary conditions, can only be done on computers. Problems can be solved swiftly and precisely with the help of supercomputers due to their enormous processing capability. The foundation for practically all CFD problems can be found in the Navier–Stokes equations. These traits define fluid flows that only involve a single phase. The Navier–Stokes equations can be reduced to the simpler Euler equations by omitting the parameters that describe viscosity. It is possible to further simplify the equations for maximum potential by removing the vorticity parameters. The potential equations need to be simplified before they can be linearized. Because of the complexity of computing the fluid surface contact based on boundary conditions, numerical modelling has become indispensable. Problems can be swiftly and precisely solved with the help of supercomputers due to their enormous processing capability. Navier–Stokes equations are the foundation of practically all CFD issues.

2.2. Numerical calculation method

Numerous numerical solutions have arisen during the past few decades. The key distinction is the use of discrete and algebraic equations in the regional discrete approach to problem solving. CFD numerical solution employs the finite element approach, the finite volume method, and the finite difference method. The finite volume approach is the most popular CFD solver.

2.3. Finite Difference Method

The first technique used for computer numerical simulation was the finite difference method (FDM). The solution area is separated into a grid using this technique, so that the approach can work with discrete nodes rather than one continuous one. By making the use of finite difference method, the derivative of control equation is discretized with difference of function value just on grid node, leading to algebraic equations with unknown parameters on the grid nodes. The procedure provides a rough numerical solution that transforms the differential issue into an algebraic one.

2.4. Method of Finite Volume

Among the many techniques for spatial discretization is the method of finite volume (FVM), sometimes also known as the method of control volume. It derives mostly from conservation-type equations, and incorporates an integral in its governing volume; it then uses an integral to solve the conservation equation. Each of these differential equations with all controls follows a standard form, and this uniformity provides the foundation for a universal solution.

2.5. Finite element method

Structure analysis of solids makes use of the finite element technique (FEM), which can also be used to fluids. It uses the weighted margin method and the variation concept. Separating the calculation area into a defined set of non-overlapping units is the foundational notion behind the proposed approach. Pick a few convenient nodes within each unit to use as the function's interpolation points. Then, using the node value or derivative of each variable and the selected interpolation function, the outputs of the variables with in differential equation are turned it into linear expression (shape function). Final steps involve discretizing the differential equation and then solving it with the variation principle or the weighted margin approach. Weight functions and interpolation functions are formulated using various finite element techniques.

3. Practical application of Computational Fluid Dynamics

3.1. Biomedical Engineering

The discipline of medicine makes extensive use of computational fluid dynamics to address intractable issues. Computer fluid dynamics (CFD) is increasingly being used as a tool in the development of updated designs and optimizations by means of computational simulations, leading to lower operating costs and improved efficiency. The analyses in biological applications, especially blood flow and nasal airflow, have been the subject of much simulation and clinical testing. Ventricle function, coronary artery function, and heart valve function are all part of the blood circulations that are analysed in the research of blood flow analysis. Nasal airflow analysis, meanwhile, includes things like the natural flow of air in a human nose, better medicine delivery, and computer-generated operations. Here are some examples of how CFD is used in biomedical engineering.

Medicine relies heavily on computational fluid dynamics to solve intractable problems. In order to reduce operating costs and maximise efficiency, more and more industries are turning to computational simulations using computer fluid dynamics (CFD). Significant time and effort have been spent simulating and clinically evaluating the analyses for use in biological applications, particularly blood flow and nasal airflow. The examination of blood flow includes the study of ventricular function, coronary artery function, as well as cardiac valve function. The study of nasal airflow, meanwhile, encompasses a wide range of topics, from the way air flows in a person's nose to the efficiency with which drugs are administered to entirely synthetic procedures. Below are a few applications of CFD in the field of biomedical engineering:

- Pumping of Heart
- Flowing of blood through veins and arteries
- Flow of air in lungs
- Combined lubrication
- Interface of cell fluid
- Tendon-sheath
- Gas exchanger
- Design of artificial organ
- Analysis of vocal tract
- Tissues perfusion
- Life support system
- Flows of nose & sinus
- Flow of spinal fluid
- Design of cardiac valve
- Locomotion of microbe

3.2. Mechanical Engineering

- **Heat exchanger:** Maldistribution of fluid flow, fouling, loss due to pressure, and thermal analysis during optimization and design of heat exchanger are all examples of applications of computational fluid dynamics.
- **Fluid flow throw pipe:** The fluid flow module in the lab is compared to the results produced using CFD to find the velocity & pressure of the flow of fluid through the pipe. Transfer of heat while fluid flows through pipe can be monitored and compared to laboratory data using this module.
- **Combustion in IC engines:** Both the creation of a design of a spark ignition (SI) engine and its implementation in an internal combustion (IC) engine used in an undergraduate course require the use of CFD. Modelling the interplay of chemical processes, species transport, flow patterns, & temperature distributions in spark ignition engines has helped engineering students gain a deeper understanding of the system.
- **Aerodynamics of aircraft:** Lift and drag
- **Automotive:** Ventilation, whether it is external over the vehicle's body or internal over the engine, combustion, and engine cooling.
- **Machinery of turbo:** Turbines, pumps, compressors
- Flow & heat exchange in thermal power plants & nuclear power reactors
- HVAC
- Manufacturing–Casting simulation, injection moulding of plastics
- **Marine engineering:** Off-shore structures' loads
- Ships hydrodynamics, submarines, torpedo etc.

3.3. Missile Engineering

- Because of the complexities of contemporary buildings as well as a Characterizes missile airframe aerodynamic load as well as the environment of aerothermic is acritical issue in missile engineering. The information of aerodyamics is utilized to measure balance and stability, mission performance & maneuverability.
- Computational Fluid Dynamics is employed to supplement & extend test results of experimental method.
- In addition, the visualization flow given by Computational Fluid Dynamics could be essential in all step of wind tunnel testing. Surface pressure distributions, wake line filaments as well as other visualization aids enable possibilities to "observe" the flow as well as its interaction with various aircraft components.
- Computational Fluid Dynamics bridge added intelligence for decisions of critical decisions during the testing phase of wind tunnel. For instance, the construction of a scale model benefits greatly from the use of inviscid computations.
- The use of Computational Fluid Dynamics (CFD) provides a bridge of additional intelligence for making crucial decisions during wind tunnel testing. For instance, inviscid computations are quite helpful for building a scale model.

3.4. Architecture

- Getting interested in enhancing performance of buildings in terms of the environmental impact CFD technology is used to examine internal as well as exterior airflow.
- In the past, exterior airflow analysis was solely done with wind tunnel testing (WTT), but presently, CFD has mostly supplanted WTT because it can produce data at a wider range of locations than WTT can.
- In order to improve the environmental performance of buildings, CFD technology is being utilised to analyse the airflow from within and outside the structures.

3.5. Food Industry

Sterilization:

- It's common knowledge that when it comes to food, customers care most about three things: security, quality, and price. As a result, ensuring the high quality and security of the food supply is crucial. A crucial method for long-term food storage and preservation is sterilisation. Examining the flow pattern and temperature distribution of food during sterilisation with CFD can help improve the quality of food items.
- It is generally accepted that safety, quality, and affordability are the top three priorities for consumers when it comes to food purchases. Therefore, it is essential to keep the food supply safe and of good quality. Sterilization is an important technique for long-term food storage and preservation. The quality of food can be enhanced by using CFD to analyse the flow pattern and temperature distribution of food during sterilisation.

3.6. Refrigeration:

- Frozen meals have consistently shown an excellent quality and food safety record, which has led to a steady increase in their popularity among consumers.
- To prevent spoilage and slow bacterial growth, refrigeration is a must.
- Because of this, CFD has recently been used for the modelling of heat as well as mass transfer in food during refrigeration.

4. Benefits and Limitations of CFD

Insight: CFD analysis allows us to practically peek inside a design to understand how it operates, which is very useful for devices and systems that are difficult to evaluate or test through experimentation. In-depth understanding of the designs is provided through CFD. The use of CFD allows us to see a wide variety of phenomena that would otherwise be hidden from view. With the help of CFD software, we can predict what will occur in a certain situation. We can foresee how the design will function and test numerous variants rapidly to get the best one. Effectiveness - the insight we receive enables us to make more well-thought-out designs, improving the quality of our output. Rapid prototyping is made possible by CFD since it shortens the design & analysis cycle. Limitations However, despite CFD's many benefits, it does have a few drawbacks. CFD answers are based on physical models of actual processes.

For systems and devices that are challenging to analyse or verify experimentally, CFD analysis provides a "look inside" to understand how they operate. The designs are better understood thanks to CFD's detailed analysis. Many phenomena that would otherwise be invisible are now visible thanks to CFD. The future of a given condition can be foreseen with the help of CFD software. We can anticipate the design's performance and fast test multiple iterations to settle on the optimal one. Efficiency - the new information helps us develop better plans, which ultimately leads to higher-quality work. CFD shortens the design & analysis cycle, which allows for rapid prototyping. Limitations To be sure, CFD has numerous uses, but it also has some downsides. The mechanisms that inform CFD's conclusions are physically grounded.

- Numerical errors are inevitable when using a computer to solve an equation.
- Approximation flaws in numerical models that cause truncation.
- Inaccuracies resulting from rounding down or up because of the limited size of computer words.
- CFD solution correctness is highly sensitive to the beginning and boundary conditions used in the numerical model.
- Rounding errors caused by the constraints of the length of computer words.

5. Results and Discussion

Many forms of information can be gleaned from CFD simulations. The results of this research pay special attention to the diffuser angle. This just modifies the diffuser's angle; all other dimensions remain unchanged. The angles of the diffusers used range from 5 degrees to 22 degrees. Diffuser angles are selected with consideration given to the diffuser's effect, which is a reduction in velocity and an increase in static pressure. Below 5 degrees and above 22 degrees, the diffuser's impact disappears. Different planes are made to show off things like pressure contours, velocity contours, streamlines, and velocity vectors, as well as turbulent kinetic energy contours.

It's possible to learn a lot from CFD simulations. This study's findings highlight the importance of the diffuser angle. Any other dimensions of the diffuser are unaffected by this change in angle. Diffusers with inclinations between 5 and 22 degrees are commonly employed. The diffuser's impact, which is to slow the air down and raise the static pressure, plays a role in the decision of the optimal diffuser angle. The diffuser loses effectiveness between 5 and 22 degrees. Pressure contours, velocity contours, streamlines, velocity vectors, and turbulent kinetic energy contours are all displayed on specialised planes.

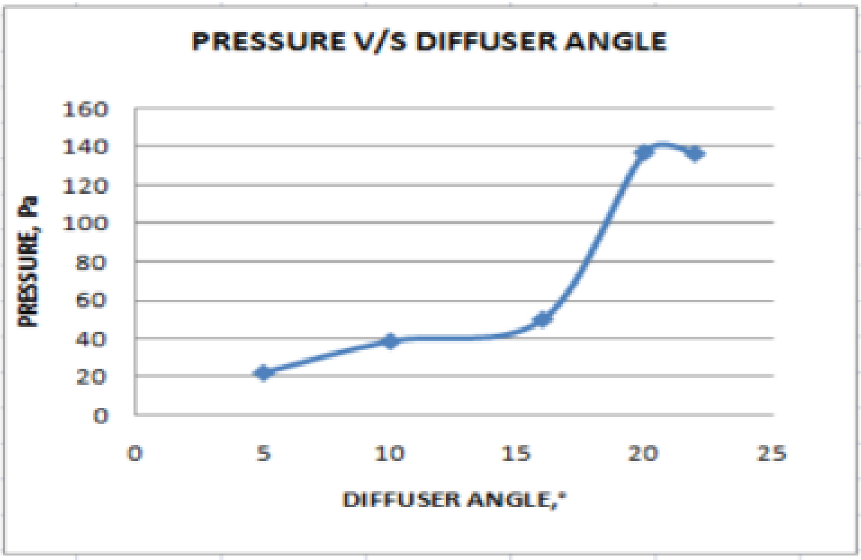

Fig.1: Pressure versus diffuser angle

Pressure versus diffuser angle is depicted in Fig. 1. Analytical results for various diffuser angles are presented beside the corresponding pressures. With each new angle the diffuser is set at, the pressure steadily rises. Pressure rises from a diffuser angle of 5 degrees to that of 22 degrees. A graph showing this steady rise is presented. The pressure gradient and flow velocity are both increased by flow recirculation & boundary layer separation. Boundary layer separation happens when the flow around the diffuser's walls and edges is redirected.

Figure 1 shows the relationship between pressure and diffuser angle. Calculated pressures and diffuser angles are shown. The pressure increases proportionally with the diffuser's angle of rotation. There is an increase in pressure from a diffuser angle of 5 degrees to an angle of 22 degrees. A graph depicting this increase over time is provided. Flow recirculation and boundary layer separation both enhance the pressure gradient and flow velocity. The boundary layer is separated when the flow is steered away from the walls and edges of the diffuser.

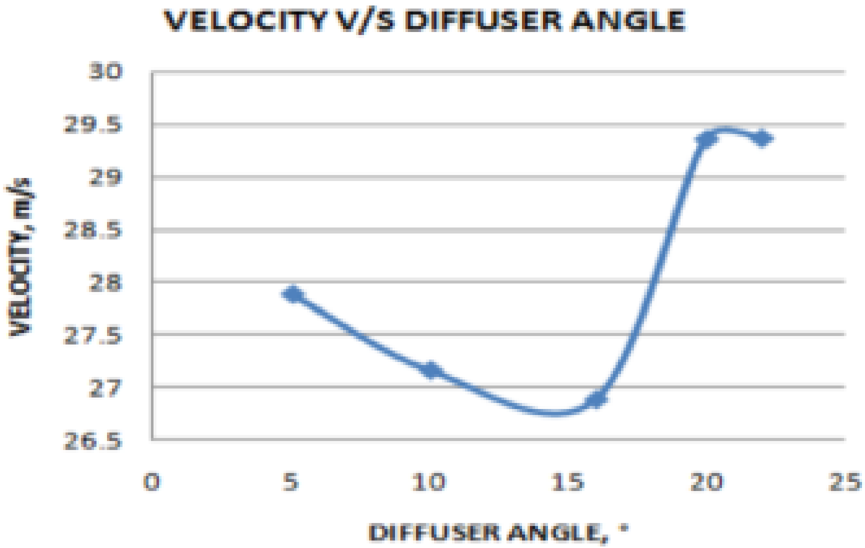

Fig.2: Velocity versus diffuser angle

The velocity versus diffuser angle curve is presented in Figure 2. The results of the analysis for various diffuser angles are presented together with the corresponding velocities. The speeds gradually drop from a diffuser angle of 5 degrees to 16 degrees. Velocities keep rising from 16 degrees to 22 degrees. This suggests that at an angle of greater than 16 degrees, the diffuser's effects are no longer diffuse. After 16 degrees, the diffuser is no longer effective. When two objects are separated by a barrier, the velocity gradient between them diminishes, and may even become negative. Separation of the boundary layer is influenced by the pattern of the unfavourable velocity gradient along the surface. The velocity in the diffuser is slowed because of flow recirculation.

Figure 2 depicts a curve of velocity versus diffuser angle. The study is provided alongside the velocities that occur from using different diffuser angles. From a diffuser angle of 5 degrees - 16 degrees, the speeds decrease significantly. Velocities increase from 16 to 22 degrees. This indicates that the diffuser's effects become concentrated at an angle greater than 16 degrees. When the temperature rises above 16 degrees, the diffuser stops working. Barriers reduce, and in some cases eliminate entirely, the velocity gradient between two objects. The structure of the adverse velocity gradient throughout the surface affects the splitting of the boundary layer. Because of the recirculation of the fluid, the speed of the gas passing through the diffuser is reduced.

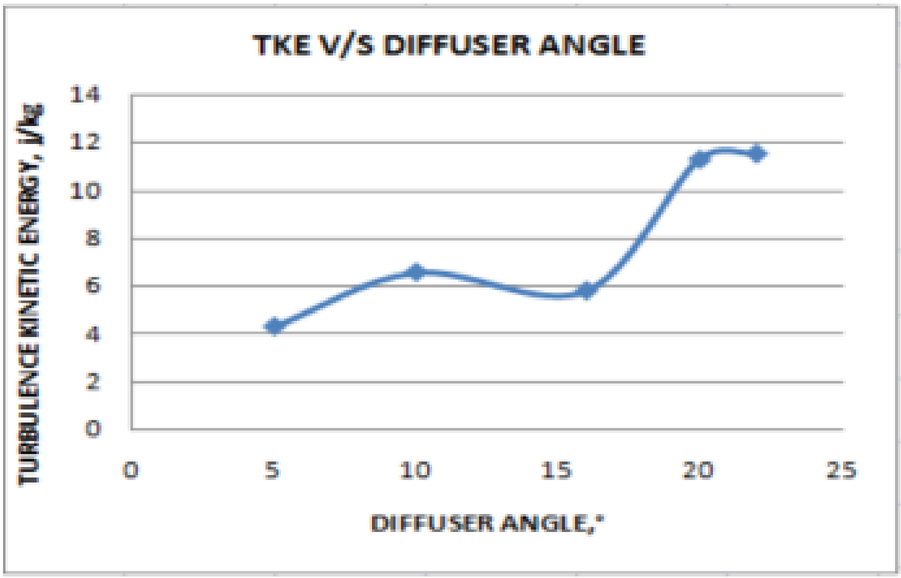

Fig.3: Turbulence kinetic energy versus diffuser angle

This is demonstrated by a plot of the kinetic energy of the turbulence vs the diffuser angle in Figure 6. A plot of the kinetic energy of turbulence is shown for a variety of diffuser angles. As the diffuser angle grows, so does the kinetic energy of the turbulence. The kinetic energy of turbulence reduces marginally between 10 and 16 degrees of diffuser angle, while it grows for diffuser angles greater than 16 degrees. Since the flow in a diffuser tends to be turbulent, eddies are often found there. The kinetic energy of the turbulence grows as the number of eddies formed close to the walls and edges does.

Figure 6 is a plot of the turbulence kinetic energy versus the diffuser angle, which demonstrates this. For different diffuser angles, a graphic of the turbulence's kinetic energy is provided. In other words, the turbulence's kinetic energy increases with increasing diffuser angle. Between 10 and 16 degrees, the kinetic energy of turbulence decreases slightly, but above 16 degrees, it increases. There are many eddies in a diffuser because the flow is so turbulent. As more eddies form in close proximity to the walls and edges, the kinetic energy of turbulence increases. Section headings should be flat left and set in Garamond, size 12. To maintain consistency, both the main sections and their sub-sections should be numbered and formatted with a flush left. Roman numerals are used for the sections.

6. Conclusion

The importance of Computational Fluid Dynamics (CFD) is growing rapidly. Companies such as 3M, Air Products, Argonne National Laboratory, Bechtel, BP Amoco, Chemineer, Cray, Dow Chemical, Dow Corning, DuPont, Eastman Chemical, and many more are benefiting from this technology. This paper presents a collection of work that investigates CFD by focusing on challenges from a wide range of industries and other contexts. In addition, we provide a brief summary of the work done by earlier researchers in the field of CFD. Recent advances in computational fluid dynamics, according to some experts, have expanded access to low-cost options. The fields of drying, sterilising, mixing, and refrigeration have all seen significant growth in the development & application of CFD in recent years. Because CFD relies on a large number of approximations and assumptions, the simulation results should be checked in the real world. Widespread use of CFD in industrial processes will continue in the 21st century despite challenges such as the difficulties to accurately simulate huge 3-D issues on a cost-effective computer, especially in large-scale sophisticated plants

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

[23] Seshu Kumar Vandrangi, Sampath Emani, KV Sharma, Gurunadh V. Computational analysis to determine the heat transfer coefficients for SiO2/60egw and SiO2/40egw based nano-fluids. Annales de Chimie-Science des Matériau. 2018 Jan 1;42(1):103-14.

[24] Seshu Kumar Vandrangi, Suhaimi bin Hassan, Sharma KV, Reddy P. Comparison of nanofluid heat transfer properties with theory using generalized property relations for EG-water mixture. InMATEC Web of Conferences 2017 (Vol. 131, p. 03004). EDP Sciences.

[25] Seshu Kumar Vandrangi, Sampath Emani, Hassan S, Sharma KV. Fluid dynamic simulations of EG-W (ethylene glycol–water) mixtures to predict nanofluid heat transfer coefficients. Environmental Technology & Innovation. 2020 Nov 1;20:101113.