Application of CFD in Indonesian Research: A review

Ambarita H1*, Siregar M R2, Kishinami K3, Daimaruaya M3, Kawai H3

1Sustainable Energy and Biomaterial Centre of Excellent, Faculty of Engineering, Universitas Sumatera Utara, Jl. Almamater Kampus USU Medan 20155, Indonesia
2Sarulla Operation Ltd., The Energy Building 51st floor, SCBD Lot 11A, Jl. Jend. Sudirman Jakarta 12190, Indonesia
3Mechanical Engineering Department, Muroran Institute of Technology, 27-1 Mizumoto-cho, Muroran, Hokkaido 050-8585, Japan

*Email: himsar@usu.ac.id

Abstract. Computational Fluid Dynamics (CFD) is a numerical method that solves fluid flow and related governing equations using a computational tool. The studies on CFD, its methodology and its application as a research tool, are increasing. In this study, application of CFD by Indonesian researcher is briefly reviewed. The main objective is to explore the characteristics of CFD applications in Indonesian researchers. Considering the size and reputation, this study uses Scopus publications indexed database. All of the documents in Scopus related to CFD which is affiliated by at least one of Indonesian researcher are collected to be reviewed. Research topics, CFD method, and simulation results are reviewed in brief. The results show that there are 260 documents found in literature indexed by Scopus. These documents divided into research articles 125 titles, conference paper 135 titles, book 1 title and review 1 title. In the research articles, only limited researchers focused on the development of CFD methodology. Almost all of the articles focus on using CFD in a particular application, as a research tool, such as aircraft application, wind power and heat exchanger. The topics of the 125 research articles can be divided into 12 specific applications and 1 miscellaneous application. The most popular application is Heating Ventilating and Air Conditioning and followed by Reactor, Transportation and Heat Exchanger applications. The most popular commercial CFD code used is ANSYS Fluent and only several researchers use CFX.

1. Introduction
Computational Fluid Dynamics, hereafter named as CFD, is a numerical technique that solves a set of fluid flow and related governing equations to simulate the fluid flow and other parameters. Before the CFD era, the fluid flow problems were solved using the pure theoretical method and experimental method. The history of CFD started in the early 1970's and the beginning of CFD was triggered by the availability of increasing computational tools and CFD methodology its self. The first applications of the CFD methods include simulation of transonic flows based on solution non-linear potential equation. At the beginning of 1980's, the CFD has been used to solve two-dimensional and three-dimensional Euler equations. Forced by rapidly increasing speed of computers and development of numerical techniques, in the mid of 1980's, the solution of the viscous flows governed by Navier-Stokes equations become feasible. In parallel, a variety of turbulence models developed with different degree of numerical complexity and accuracy [1].

CFD has now matured and has been widely used not only in aerodynamics application but also in many multidisciplinary subjects. In the present time, CFD has become an integral part of the engineering design and analysis environment of many industrials. This is because of its ability to predict the performance of a new designs or process before they are ever manufactured or implemented. It is now easily found in the literature that CFD plays an important role as one of the key tools in many applications such as Hydrodynamics, Powerplant, Turbomachinery, electronic cooling device, chemical process, Heating Ventilating & Air Conditioning (HVAC), Biomedical Engineering, etc. There is no
strict rule in dividing all of the CFD applications in literature. However, several researchers have been reported application of CFD in a particular area. Xia and Sun [2] published a review article on applications of CFD in the food industry. In the food industry, CFD is applied in several main processes such as drying, sterilization, refrigeration and mixing. It was shown that CFD has already proven to be a valuable tool to the food processing industry. However, there are still some problems that block the widespread use of CFD. Some of the materials in food processing differ in many ways from those is conventionally applied (air, water, and any fluids) and the specific knowledge in the food processing is extremely needed to be coupled with the CFD expert. Norton et al. [3] reviewed the applications of CFD in the modeling and design of ventilation systems in the agricultural industry. The results showed that advances in CFD technology have meant that field solutions can include the biological response of housed entities, and thereby enhance the realism of simulations. Bhutta et al. [4] reported their review of the CFD applications in various heat exchanger design. The results showed that CFD has been replacing the conventional methods used for the design, analysis, and optimization of heat exchangers. Easily accessible general purpose CFD commercial software such as FLUENT and CFX can fulfill the requirements of CFD analysis of a various type of heat exchangers. One of the most discussed issues in the heat exchanger analysis is the modeling of turbulent flow. In this issue, the turbulence model has been most widely used for heat exchanger design and optimization. The simulation results generally show a good agreement with experimental with discrepancy varies from 2% to 10%. Al-abidi et al. [5] reviewed the applications of CFD in the latent heat thermal energy storage. The results show that numerical solution in the analysis of Phase Change Material (PCM) is more accurate than the analytical solution and CFD software is used successively to simulate the application PCM as the latent heat thermal energy storage. Various CFD commercial software is used in PCM engineering includes ANSYS Fluent, Comsol Multiphysic, and Star-CCM. Among these the most widely used is ANSYS Fluent.

The above literature shows that the CFD which was originally developed for the aerodynamic purpose, it is now in present spread out to multidisciplinary applications. Today, CFD plays an important role in the design, analysis, and optimization of all related fluid engineering problems. There is significant growth in the CFD application in research and industry in the world includes in Indonesia. The objective of the present work is to review the applications of CFD published by Indonesian researchers that are reported in the literature. The results are expected to supply the necessary information in the development of research in Indonesia that related to CFD application.

2. Methods
The present work is divided into two sections. In the first section, the principle of computational fluid dynamics will be briefly presented including the governing equations and methodology of the solutions. In the second section, the growth of research paper related to CFD in the world and selected countries include Indonesia will be examined. In addition, the research articles, where at least one of Indonesian researcher as the author, are further reviewed. It is realized that there are many published research papers are found in the literature. In order to make clear and more focus discussion, only the research articles that are indexed in Scopus [6] will be used. The data were drawn from Scopus database that accessed on September 1, 2017.

3. Results and Discussions

3.1. Principles of CFD
The definition of CFD is the methodology that converts the governing equations into a set of linear equations and solves them iteratively. The fluid flow can be formulated using partial differential equations and named as governing equations. There are several different types of governing equations depend on the assumptions that have been made. The hierarchy of the governing equations is shown in
Figure 1. The real flow typically found in the real problem is turbulent flow regime. Here the energy is transferred from large scale motions to very smaller eddies until the scale become so small that the motion is dissipated by viscosity. In order to solve such problem numerically, in three-dimensional case, the computational domain needs to be divided into grids with a number with the order of \( \text{Re}^{9/4} \). Where \( \text{Re} \) is a Reynold number of the flow. For the case with big dimension, for instance, an aircraft with typical \( \text{Re} \) number is 30 million, that grid number is beyond the range of any current of the foreseeable computer. Thus, several assumptions have been made to simplify the real governing equations. The simplest to the most complex flow equations are shown in the figure. In the lowest hierarchy, the simplest flow is inviscid flow equation. In the inviscid calculations combined with boundary layer equation can result in accurate estimation of lift and drag coefficient. In the top of the hierarchy is Reynolds averaged Navier-Stokes (RANS) equations. This is a time-averaging of the Navier-Stokes equation and the continuity equation for incompressible flow.

Figure 1. The hierarchy of governing equation [7]

Introducing the time averaging mean value of velocity, as formulated in equation (1)  
\[
\overline{v}_i = \lim_{T \to \infty} \frac{1}{T} \int_{-T}^{T} v_i \, dt 
\]  
(1)

where \( \overline{v}_i \) is the mean value of velocity, the RANS governing equations of continuity and momentum equations can be written as follows.

\[
\frac{\partial \overline{v}_i}{\partial x_i} = 0 
\]  
(2)

\[
\rho \frac{\partial \overline{v}_i}{\partial t} + \rho v_i \frac{\partial \overline{v}_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \overline{\tau}_{ij} - \rho \overline{v}_i \overline{v}_j \right) 
\]  
(3)

In this equation, the additional term of \( \rho \overline{v}_i \overline{v}_j \) is known as Reynolds-stress tensor represents the transfer of momentum due to turbulent fluctuations.

\[
-\rho \overline{v}_i \overline{v}_j = -\rho \left( \overline{v}_i \overline{v}_j - \overline{v}_i \overline{v}_j \right) 
\]  
(4)

In the equation (3), the laminar viscous stresses \( \overline{\tau}_o \) can be calculated using the following equation:

\[
\overline{\tau}_o = 2 \mu \overline{S}_o = \mu \left( \frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i} \right) 
\]  
(5)

In order to close the above equations, there are several turbulence models proposed by researchers. The governing equations presented above are solved numerically by converting those equations into linear equation system in every grid. In general, the steps of CFD method are depicted in Figure 2. The fluid engineering problem needs to be modeled includes the dimensions, boundary, and other related parameters. After decided, the selected problem must be transferred into computational domain. Here, any available software can be used such as any CAD software, GAMBIT, ICEM or ANSYS Design.
Modeler. By using these software, the computational domain will be divided into grids and meshes. The mesh will be exported to the solver. There are several commercially CFD solvers available in the market. They are ANSYS Fluent, CFX, PHOENICS, and CFD2000. The self-developed code also can be used to solve the governing equations in every grid.

![Figure 2 Typical Algorithm of CFD](image)

The method of converting the governing equations into a system of linear equations is known as spatial discretization method. There are several spatial discretization methods typically available in the commercial software. They are First and Second Order Upwind, Power Law, QUICK and Third-Order MUSCL. These discretization methods are ordered from the cheapest to the most expensive computational cost. The highest cost relates to high accuracy. As a note, the governing equations consist of several fields such as pressure and velocity fields also temperature field, turbulent kinetic energy field depends on the governing equations. These fields are coupled iteratively. There are several coupling schemes typically available in the commercial software such as SIMPLE scheme, SIMPLC scheme, and PISO scheme. The most commonly used is SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) proposed by Patankar and Spalding [8].

Even though at present the commercial CFD code has been matured and has been widely used in many industrial applications, however, there are several challenges in the applications CFD. In the present worldwide commercial and government codes are based on the algorithms developed in the 80s and 90s. These codes can handle complex geometry but are generally limited to the second order of accuracy. The turbulences are handled using modeling such as first and second order RANS and for the present highest performances computers, the turbulence can be solved using LES and DNS. The unsteady simulations are still very expensive and several questions on accuracy still remain. These are several challenges in the future development of CFD [7].

3.2. Growth of Research Paper on CFD

The database of Scopus has been used to explore research paper related on the CFD. The growth of research paper indexed by Scopus in the last 22 years, during 1996 to 2017 is shown in Figure 3. There are two research documents shown in the figure. The first is the document related to all CFD topics (shown by the red line and its number on the right). The second is the document related to CFD in a particular area of aerodynamics (shown by the blue line and its number on the left). As a note, the number of document in the year 2017 is lower than in the year 2016. This is because the year 2017 is
not yet finished. It will higher. The figure shows that the research paper increases with increasing time. The research paper related to CFD increases 10.2% in every year. This fact reveals that application of CFD in all research areas is increasing. In the specific application the aerodynamics, as the original application of CFD, the application of CFD is increasing with the averaged growth of 10.5%. The share of the research paper in the aerodynamic application is around 9.6%. This fact suggests that 9.6 of research papers out of 100 research paper related to CFD are related to aerodynamic applications.

![Figure 3 Worldwide document related to CFD indexed by Scopus](image1)

Figure 3 Worldwide document related to CFD indexed by Scopus

![Figure 4 Document related to CFD indexed in Scopus](image2)

Figure 4 Document related to CFD indexed in Scopus

Several countries also show the same trend with the worldwide trend. Figure 4 shows the research document related to CFD published by researchers with affiliation from Malaysia, Brazil, Thailand, and Indonesia. It can be seen the similar trend by these countries. In cumulative, the total document published by Malaysia, Brazil, Thailand and Indonesia researchers are 1419, 1641, 370, and 260 documents, respectively. Brazil is the most productive among these countries. During the last ten years (2006 to 2016), Brazilian researchers published 110 documents per year and followed by Malaysia, Thailand, and Indonesia with the averaged productivity of 110.8 documents, 29.5 documents, and 17.9
documents, respectively. It reveals that Indonesia shows the lowest productivity among these countries. However, there is a significant growth from Indonesian researchers. In the last ten years, Indonesia shows the average growth of document; it is 48.06% per year. The second is Malaysia with a growth of 33.75% and followed by Brazil and Thailand with yearly growth are 15.9% and 7.2%, respectively. This fact reveals that there is an increasing growth of application CFD in Indonesia in the last ten years.

3.3. Indonesia Research Articles related to CFD

For Indonesia case, the first document related on CFD published by Indonesian researcher was in June 1996 by Kindangen et al. [9]. Here, the CFD was used to investigate natural ventilation, and the results were compared with wind tunnel result. After this article, there are 260 documents indexed by Scopus produced by Indonesian researchers during period 1996 to 2017. The type of those documents consists of conference paper with 135 documents and followed by 125 research articles, one book, and one review. The distribution of those documents is presented in Figure 5(a). In this work, the 125 of research article are reviewed to explore the characteristics of CFD application published by Indonesian researcher.

![Figure 5 Number of document and article distribution](image)

The review shows that CFD research papers can be divided into two modes. The first mode is the research related to the CFD method and the second mode is the application of CFD to solve the engineering problem. In the first mode, typically a new method (algorithm, scheme, discretization technique, etc.) is proposed and compared with the previous one. There are 8 of research articles deal with the first mode. Febrianto and Zuhal [10] proposed vortex in cell [VIC] method to predict flutter phenomenon of 2D bridge deck model. The performance of the proposed method to simulate moving body is tested by simulating fluid-structure interaction and flutter phenomena of long-span bridge. The result of the proposed mode showed good agreement with another numerical method and experimental results. Tsai et al. [11] presented a coupled wave-vegetation simulation for the moving object of the coastal vegetation on tsunami wave height damping. Solitary wave propagation on a group of emergent cylinders was employed to idealize the problem. To model the turbulence, general RANS equation combined with renormalization group turbulent closure. The general moving object (GMO) model is employed to simulate the coupled motion of vegetation with wave dynamically. Djojodihardjo and Safari [12] reported a development the foundation for the computational scheme for the calculation of the influence of the acoustic disturbance to the aeroelastic stability of the structure. Wahyudi et al. [13] proposed a new method to study the gas-solids flow and heat transfer in fluidized beds with an immersed tube. The new method is a fully three-dimensional model combined CFD and discrete element method. At first, a critical bed thickness is determined at which the bed can be regarded as fully 3D. The result shows that the model can successfully reproduce the typical relationship between pressure drop and gas
velocity, and flow and heat transfer characteristics such as the four distinct stages of bubble transit through the tube and peak of heat transfer coefficient between the tube and the bed for certain gas velocity. This phenomenon was well predicted by the proposed method which is not well-predicted by previous 2D CFD-DEM and 2D CFD-3D DEM models. van Groesen and Andonowati [14] proposed a new approach to solve fully nonlinear treatment of the dynamic wave-ship interaction for potential flows. Hamilton description of the inviscid free surface wave was extended in a straightforward consistent way to include ship-wave interaction. By describing the fluid flow motion in horizontal variables only, computational complexity was reduced. The internal fluid motion is modeled in a consistent approximative way. The Lagrangian variational principle is used to develop the equations for the wave-ship interaction. The results show how to obtain consistently the extension of known Hamiltonian formulation for free surface waves to a Hamiltonian formulation of the fully nonlinear wave-ship interaction while retaining the dimension reduction of dealing only with the fluid potential at the free surface and wetted ship hull. The main observation of their work is that the Hamiltonian formulation avoids the calculation of the dynamics part of the pressure which becomes part of a canonical momentum variable that is updated by the hydrostatic forces.

Gun Gun et al. [15] investigated performance of several turbulence models by comparing the numerical results and experimental one. Three models were tested, they are k-epsilon model, renormalization group (RNG) k-epsilon and Reynolds stress model (RSM). The results revealed that those three turbulence models provide good results to predict the distribution of speed and pressure of the fluid flow in the wind tunnel. In the case of predicting the turbulent kinetic energy, turbulent dissipation rate and turbulent effective viscosity, the k-epsilon model showed a good agreement in comparison with RSM model. Nugroho et al [16] proposed an analytical solution to three-dimensional incompressible Navier-Stokes equation with the continuity equation. For simplicity, the spatial and temporal coordinates are transformed into a single coordinate $\xi$. The form of solution $V = \nabla \Phi + \nabla \times \Phi$ was proposed. Here $\Phi$ is a potential function that is defined as $\Phi = P(x, \xi)R(\xi)$. As a result, more general solutions are also obtained based on the particular solution of $P$ and the explicit analytical solutions are found to be mathematically similar for the cases of zero and constant pressure gradient. The conclusion was the selection of variables for the potential function can be interchanged from the beginning, resulting in similar explicit solutions.

In the second mode of the research, the CFD code, typically the commercial one, is used as a tool to provide analysis, design, and optimization of an engineering problem or process. As expected, this mode is the most used mode by Indonesian researchers. Figure 5(b) shows the application distribution of research articles related to CFD. It consists of Aerodynamics (9%), Biomedical (4%), Combustion (7%), Drying (2%), Heat Exchanger (9%), Heating Ventilating and Air Conditioning, hereafter as HVAC (15%), Hydropower (4%), Marine (6%), Powerplant (6%), Reactor and Separation Process (14%), Transportation (10%), Wind Power (6%), and Miscellaneous (8%). This distribution shows that for Indonesian researchers the CFD most used in HVAC application with 19 research articles and followed by Reactor and Transportation applications with 18 and 13 research articles, respectively. The typical research article for each application is reviewed below.

### 3.3.1 Aerodynamics applications

In the aerodynamics application, there are 11 research articles resulted by Indonesian researcher indexed in Scopus. Bangga and Sasonoko [17] provided CFD predictions of a pitching NACA 0012 air foil at a reduced frequency of 0.1 and at small Reynolds number vane of 135,000. The turbulence flow was modeled using adjusted the k-epsilon URNAS model. Here the dumping factor was introduced as a function of wall distance in the buffer zone region. Parametric studies on the involving variables were conducted. Their study success to demonstrate the fluid flow analysis using the CFD ANSYS Fluent. Effendy et al. [18] investigated the cooling performance of blade trailing edge cutback of the gas turbine that typically used in the aircraft. Effects of the lip thickness to slot height ($t/H$) ratio was explored. Their study was aimed to enrich the information on the dynamic flow interaction process between the internal coolant fluid and external warm mainstream fluid in comprehensive manner. Figure 6 shows
the isometric view of computational domain (a), schematic view of trailing edge cutback, and (c) turbulent flow structure at several t/H ratios. The CFD with detached-eddy simulation (DES) combined with SST k-omega turbulence model can be used to simulate the dynamic flow interaction process of trailing edge cutback of a gas turbine. The observed vortex shedding and its characteristics in the near wake region (as shown in Figure 6c) are found to play an important role in determining the dynamics process of the cold and warm airflow mixing, which in turn have significant influences on the prediction accuracy of the near-wall heat transfer performance. Those papers are only two out of 11 articles Scopus indexed published Indonesian researchers.

Biomedical engineering

Biomedical engineering also uses CFD as a tool to provide analyses. There are five research articles have been found in literature indexed in Scopus. Three of those works are briefly presented here. Tenekecioglu et al. [19] employed CFD ANSYS Fluent to simulate blood flow in a porcine coronary artery model. The objective was to evaluate the impact of the scaffold design on the local hemodynamic microenvironment. Figure 7 shows the CFD results of the analysis. The conclusion of the study is that

Figure 6 the isometric view of computational domain (a), schematic view of trailing edge cutback, and (c) turbulent flow structure at several t/H ratios [18]
the CFD assessment can be used to guide improvements in the scaffold design for a more "hemo-compatible" geometry. Sanka et al. [20] employed CFD to simulate the water flow over the surface of the crab carapace and the interactions of water with the micro-protuberant setae of the carapace surface. The objective of the modeling was to determine how the micro-protuberant setae of the crab carapace may instigate micro-organism nucleation by altering fluid motion. In their work Fiji-Image and Rhino 3D (CAD modeling software) were used to develop the computational domain and Comsol Multi-physics was used to execute the numerical analyses. Figure 8 shows the velocity field and streamline over Tiarinia cornigera carapace. In total, there are five research articles related to the application of CFD in biomedical engineering. In this work only these two papers are briefly reviewed.

3.3.3 Combustion
The combustion process is also a field in engineering that typically employs CFD to simulate the fluid flow during the combustion process. Even though combustion process is a perfect application of CFD, however only 7% of the research articles are related to this application published by Indonesian researcher. This might be because the CFD application in combustion needs more numerical effort. Also, the combustion process is integrated with many familiar CFD code, such as ANSYS Fluent, only in a few years later. One of these paper is the study by Ambarita et al. [21] which employed ANSYS Fluent commercial code to simulate combustion process in a small compression ignition engine run on duel-fuel (diesel-fuel) mode. In the case of thermal efficiency, the CFD and experimental results showed a good agreement.
3.3.4 Drying

In the drying process, CFD is typically used to simulate simultaneous heat and mass transfer in the fluid flow of drying medium. For instance, Simanjuntak et al. [22] explored the drying process of coal in swirl fluidized bed coal drying. Here, CFD commercial code was employed to investigate the effect of different angles of guide van in a transient three-dimensional computational domain. The results showed that CFD commercial code could be used to simulate the drying process.

3.3.5 Heat Exchanger

Recently, heat exchanger industry and researchers employ CFD commercial code to provide analysis, design, and optimization to the heat exchanger. The literature review shows that 9% of the research papers deal with heat exchangers. Afrianto et al. [23] used CFD to investigate LNG flow and heat transfer characteristic in a heat exchanger. In their work, the commercial code Fluent was used to simulate the LNG flow and heat transfer characteristics in a shell and tube heat exchanger. The shear stress transport (SST), the k-omega turbulence model, was adopted. The CFD code successfully depicts the flow characteristics inside the heat exchanger as shown in Figure 9.

Figure 9 Contour of fluid distributions: (a) shell side wall temperature, (b) tube side LNG temperature, and (c) tube side density distribution [23]

Santosa et al. [24] employed CFD code Fluent to investigate local refrigerant and air heat transfer coefficient in plain fin-and-tube gas cooler coils, a type of heat exchanger. The objective is to fill the absence of investigation on the analysis of heat transfer coefficients of CO2 gas coolers in literature. The CFD result is shown in Figure 10. The k-epsilon model was used to model the turbulence model. The results reveal that CFD modeling was found to be able to adequately represent the heat transfer characteristics of the gas cooler and be an effective simulation tool for the determination of local air and refrigerant-side heat transfer coefficient in the coil.
3.3.6 Heating Ventilating and Air Conditioning
This application is the most popular for CFD applications by Indonesia researchers. About 15% of the research articles dealt with HVAC. In this application, CFD is employed to simulate temperature and velocity distributions. Using these fields, the ergonomic parameters can be estimated. The applications are not aimed only for the human, but also for livestock such a study reported by Yani et al. [25]. In the paper, CFD is used to design the housing of the broiler. Solid Work Flow simulation as a CFD code was used.

3.3.7 Hydropower
There are five research articles related to the application of CFD to hydropower found in literature indexed in Scopus affiliated by Indonesian researcher. Budiman et al. [26] employed CFD ANSYS Fluent to calculate the load for various opening angles of the guide vane. To model the turbulence mode, the k-epsilon standard was used. The CFD results of the study are shown in Figure 11. Arini et al. [27] employed CFD to simulate fluid flow around vertical axis tidal turbine to explore the operating performance such as torsion and output power. The profile of the turbine is NACA0015, and the turbulence flow was modeled using RANS.

3.3.8 Marine
Marine also employs CFD to simulate fluid flow. There are eight research papers have been found in the literature. Tsai et al. [28] used open-source CFD IHFOAM to investigate wave-damping
performance that considering vegetation motion coupled with tsunami wave. The fractional area/volumes obstacle representation technique is used to simulate rigid body motion dynamically coupled with fluid flow. The free water surface tracking in the model is accomplished by using the volume of fluid (VOF) method. And the turbulence flow is modeled using k-epsilon model. The simulated results show that the damping of wave height and turbulent kinetic energy by the moving cylinders. Also, the wave decay by the coastal vegetation may be overestimated if the vegetation was represented as a stationary state. Wulp et al. [29] reported a study on numerical simulations of river discharges, nutrient flux and nutrient dispersal in Jakarta Bay, Indonesia. The Delf3D flow model was used to simulate flow characteristics by solving the shallow water equations for flow. The method also incorporates nutrient loads such as total nitrogen and total phosphorus. The nutrition distributions during flood flow and during ebb flow are shown in Figure 12.

![Figure 12 Nutrition distribution, total nitrogen gradient during (a) flood flow and (b) ebb flow](image)

3.3.9 Powerplant

Several research articles related to application of CFD in powerplant have been found in Scopus indexed literature. Typically, the CFD codes are used to explore the characteristic of fluid flow during energy conversion in the powerplant. Daryus et al [30] reported a study on a micro gas turbine bioenergy prototype X-3 which is known as a prototype turbine designed to be able to operate on any kind of fuel, include renewable and it is aimed as a powerplant for green building application. In order to explore the flow phenomena in the combustor of the turbine. In this study two type of turbulence models were tested, STD k-epsilon and RNG k-epsilon modes.

3.3.10 Reactor

In this in this application, the reactor includes the apparatus that maintain and control a nuclear reaction and also the container to hold chemicals undergoing a chemical reaction. There are 18 research articles found in the literature. Wahyudi et al. [31] used CFD to analyze fluid flow and heat transfer mechanism in low temperature packed and fluidized beds. In this case, there are two flows will be analyzed, the fluid flow and the solid flow. In order to combine these two flows, the CFD-DEM model is employed. The typical results of flow fields and solid flow are presented in Figure 13. It was demonstrated that both the gas and solids flow are 3D in nature. The unique feature of 3D orientations of a gas velocity vector field around the bubble plays an important role in generating in a consistency of the size and upward movement of the bubble. Muharam et al. [32] used CFD to model hydrotreating reactor to produce renewable diesel from non-edible vegetable oils. The reactor is packaged with spherical catalyst particles. The model was two-dimensional axisymmetric. Simanungkalit and Rinaldi [33] reported a study on CFD analysis of the fast pyrolysis for empty fruit bunches in Fluidized-Bed Reactor. In order to model the two-phase flow, the Eulerian approach was used. The approach considers that discrete secondary phase (solid) and continuum primary phase (fluid) are similar and illustrated for the primary
phase. ANSYS Fluent was used to perform the analysis. The contours of solid phase distribution, reactor temperature, and bio-oil vapor concentration were presented. Ramadhan et al. [34] employed CFD analysis to perform a preliminary study for design core of nuclear research reactor of Triga Bandung using fuel element plate MTR. Here porous and non-porous models of the fuel element plate were considered. The model was created using CAD software based on a three-dimensional picture of the fuel plates. However, the governing equations and type of CFD code are not declared in the report. The CFD results of the velocity field and temperature distributions were presented. The above works are only a few samples of the research papers affiliated by Indonesian researchers.

![Image](image.png)

Figure 13 (a) Fluid flow distribution and solid flow distribution in fluidized bed [31]

### 3.3.11 Transportation

Transportation industry and research is a traditional customer of CFD commercial code. This trend is also shown in the research articles affiliated by Indonesian researchers. There are 13 research articles found in literature, some of those reviewed here. Even though road transportation is the most used transformation, however, in Indonesia case, water transportation has the biggest share in comparison with road transportation. Taha et al. [35] used CFD code ANSYS Fluent to simulate fluid flow analysis to investigate aerodynamics characteristics of "Merdeka 2" a prototype of a solar vehicle that participated in the world solar challenge. The design concept of the car was based on box fish which was also claimed the concept design of Mercedes Benz minivan. Reynold-averaged Navier-Stokes with the k-epsilon model was used to model the turbulent flow. The CFD results and experimental data do not agree well. Syamsuar et al. [36] reported a study on using CFD to investigate outer wing of flying boat remote control model. The objective was to explore the aerodynamics and hydrodynamics characteristics of the model during hydro planning. Samuel et al. [37] used CFD to investigate the resistance components of converting a traditional monohull fishing vessel into Cataraman form. In the analysis, the real Cataramans size was modeled into 1:10 model.
3.3.12 Wind power
In this section, CFD is typically used to explore the fluid flow characteristics and comparison with the experimental results. About 6% of the research articles deal with wind power. Sanusi et al. [38] employed CFD code ANSYS Fluent to simulate the fluid flow of three blades models of Savonius Wind Turbines. The objective was to improve the performance of the wind turbine by combining a conventional blade with an elliptical blade into a combined blade rotor. The type of governing equations was not explored. The velocity and pressure contours of conventional and proposed design resulted by CFD code were plotted, and examinations were made based on these CFD results. Bangga et al. [39] employed CFD code to investigate a single bladed vertical axis wind turbine under a dynamic stall. The blade was developed based on NACA 0012 profile and was operated under stalled conditions at tip speed ratio of 2. The analysis was performed using ANSYS Fluent, and the turbulence was modeled using SST turbulence model. Soebiyan et al. [40] explored the potency of wind energy in high-rise buildings in humid tropical climate using CFD as a tool. The wind speed data were used to perform the CFD analysis. It was concluded that in the case of Bina Nusantara University wind energy could replace 3.27% of main building energy consumption and this is comparable to 127,650 kgCO2 emission reduction.

3.3.13 Miscellaneous
Several research articles related to miscellaneous such as sport, gun, bearing, etc. are also found in the literature. One of the specific studies was published by Mubin et al. [41]. In the study, CFD code is used to explore the drag and lift coefficient of sepak takraw ball at different face orientations. The sepak takraw ball was claimed to be unique since it is not enclosed like other balls such as golf, tennis and soccer balls. In the study, CFD code of CFX was employed to perform the analysis. The turbulent flow was modelled using RANS turbulence model. The typical results of the CFD software are shown in Figure 14. It was shown that CFD is a powerful tool for determining the drag and lift of a sepak takraw ball.
Figure 14 The actual and model of the ball (a) and flow visualization (b)

4. Conclusions
In this work, the document related to CFD indexed in Scopus affiliated by Indonesian researcher has been briefly reviewed. The following conclusions are drawn in this study.

- To the present, there are 260 documents related to CFD affiliated by Indonesian researchers found in literature indexed by Scopus. These documents divided into research article 125 titles, conference paper 135 titles, book one title and review one title.
- In comparison to neighbor countries such as Malaysia and Thailand also with Brazil, a country with a population closer to Indonesia, the document productivity is relatively lower. Indonesia is comparable with Thailand. However, Indonesia shows a significant growth, especially in the last five years.
- In the research articles, only limited researchers focused on the method development. The research articles, almost all, focus on using the specific application or using CFD as a research tool in the specific application such as aircraft and heat exchanger.
- The topics of the 125 research articles can be divided into 12 specific applications and one miscellaneous application. The most popular applications are HVAC followed by Reactor, Transportation and Heat Exchanger applications.
- The most popular commercial CFD code used by Indonesia researcher is ANSYS Fluent, and only several researchers use CFX.

References
[1] Blazek J 2001 Computational Fluid Dynamics: Principles and Applications (Amsterdam: Elsevier)
[2] Xia B and Sun DW 2002 Computers and Electronic in Agriculture 34, 5-24
[3] Norton T, Sun DW, Grant J, Fallon R and Dodd V 2007 Bioresource Technology 98, 2386-2414
[4] Bhutta M M A, Hayat N, Bashir M H, Khan A R, Ahmad K N and Sarfaraz K 2012 Applied Thermal Engineering 32, 1-12
[5] Al-abidi A A, Mat S B, Sopian K, Sulaiman M Y and Mohammed A Th 2013 Renewable and Sustainable Energy Reviews 20, 353-363
[6] www.scopus.com [Accessed September 1, 2017]
[7] https://pdfs.semanticscholar.org/de25/08faa97d1144ee1185d9a322c821803e4e96.pdf
[8] Patankar S V and Spalding D V 1972 International Journal of Heat and Mass Transfer 15, 1787-1806
[9] Kindangen J, Krauss G and Kindangen J 1996 Architectural Science Review 39, 113-120
[10] Febrianto E V and Zuhal L R 2017 ARPN Jurnal of Engineering and Applied Science 12(10), 3040-3045
[11] Tsai C P, Chen Y -C, Sihombing T O and Lin C 2017 Natural Hazards and Earth System Sciences 17(5), 693-702
[12] Djojodihardjo H and Safari I 2013 CMES-Computer Modeling in Engineering and Sciences 91(3), 205-234
[13] Wahyudi H, Chu K, and Yu A 2016 International Journal of Heat Transfer 97, 521-537
[14] van Groesen E and Andonowati 2017 Applied Mathematical Modeling 42, 133-144
[15] Gun Gun R G, Siswantara A I, Budiarso, Daryus A, and Pujowidodo H 2016 International Journal of Technology 7(8), 1362-1371
[16] Nugroho G, Ali A M S, and Abdul Karim Z A 2009 Applied Mathematics Letters 22(11) 1639-1644
[17] Bangga G and Sasongko H 2017 Journal of Applied Fluid Mechanics 10(1), 1-10
[18] Effendy M, Yao Y F, Yao J and Marchant D R 2016 Applied Thermal Engineering 99, 434-445
[19] Tenekecioglu E, Torii R, Bourantas C, Abdelghani M, Cavalcante R, Sotomi Y, Crack T, Su S, Santos T, Onuma Y and Serruys P W 2017 *International Journal of Cardiology* **227**, 467-473

[20] Sanka I, Suyono E A, Rivero-Muller A and Alam P 2016 *Acta Biomaterialia* **41**, 52-59

[21] Ambarita H, Widodo T I and Nasution D M 2017 *Journal of Physics: Conference Series* **801**(1), 012095

[22] Simanjuntak M E, Prabowo, Ichsan D and Widodo W A 2016 *JP Journal of Heat and Mass Transfer* **13**(4), 497-51

[23] Afrianto H, Tanshen Md. R, Munkhbayar B, Suryo U T, Chung H and Jeong H 2014 *International Journal of Heat and Mass Transfer* **68**, 110-118

[24] Santosa I M C, Gowreesunker B L, Tassou S A, Tsamos K M and Ge Y 2017 *International Journal of Heat and Mass Transfer* **107**, 168-180

[25] Yani A, Suhardiyanto H, Erizal and Purwanto B P 2014 *Media Peternakan* **37**(1), 17-23

[26] Budiman B A, Suharto D, Djodikusumo I, Aziz M and Juangsa F B 2016 *Advances in Mechanical Engineering* **8**(7), 1-8

[27] Arini N R, Arywibowo T H and Nugroho S 2014 *International Journal of Applied Engineering Research* **9**(22), 17793-17800

[28] Tsai C P, Chen Y C, Sihombing T O and Lin C 2017 *Natural Hazards and Earth System Sciences* **17**, 693-702

[29] van der Wulp, Damar A, Ladwig N and Hesse K J 2016 *Marine Pollution Bulletin* **110**, 675-685

[30] Wahyudi H, Chu K and Yu A 2016 *International Journal of Heat and Mass Transfer* **97**, 521-537

[31] Munarman Y, Nugraha O A, Leonard D 2017 *Chemical Engineering Transactions* **56**, 1561-1566

[32] Simanjungkalit S P and Rinaldi N 2013 *Journal of Engineering and Applied Science* **8**(9-12), 290-294

[33] Ramadhlan A I, Suwonon A, Umar E and Tandian N P 2017 *Engineering Journal* **21**(3), 173-181

[34] Taha Z, Passarella, Sugiyono, Abd Rahim N, Md Sah J and Ahmad-Yazid N 2011 *Advanced Science Letter* **4**, 2807-2811

[35] Syamsuar S, Djatmiko E B, Mujahid A S, Erwandi and Subchan 2015 *Jurnal Teknologi (Science & Engineering)* **76**(1), 221-228

[36] Samuel, Iqbal M and Utama K A P 2015 *International Journal of Technology* **3**, 432-441

[37] Sanusi A, Soepparaman S, Wahyudi S and Yulianti L 2017 *International Journal of Fluid Machinery and Systems* **10**(1), 54-62

[38] Bangga G, Hotomo G, Wiranegara R and Sasonko H 2017 *Journal of Mechanical Science and Technology* **31**(1), 261-267

[39] Soebiyan V, Saragih J F B and Tedja M 2017 *Chemical Engineering Transaction* **56**, 241-246