International Journal of Research and Engineering
ISSN: 2348-7860 (O) | 2348-7852 (P) | Vol. 5 No. 6 | June 2018 | PP. 430-433
Digital Object Identifier DOI® http://dx.doi.org/10.21276/ijre.2018.5.6.2

Computational Fluid Dynamics: Science of the Future

ORIGINAL ARTICLE

Author(s): [1]Senan Thabet, *[2]Thabit H. Thabit
Affiliation(s): [1]University of South Wales, Cardiff, UK
[2]Ninevah University, Mosul, Iraq
**Corresponding Author:* thabit.acc@gmail.com

Abstract: This paper will answer a list of questions regarding the computational fluid dynamics (CFD). It will give a brief discussion regarding the significance of CFD and will recount the pros and cons of applying CFD. The following assignment will also give an overview of the terms that come under the ambit of CFD like discretization, numerical grid, initial conditions, boundary conditions, sweep, convergence, and turbulence modeling. The researchers such as Guang Xu et al. (2017), Raase and Nordström (2015), and Frigg et al. (2009) concluded that CFD is the science of the future as it cares in all aspects of life in the present and the future, CFD science treats the fluids mainly the air and the water as good and bad, bad when the CFD tries to find a way through the air and the water to get the minimum resistant for cost-effective and less fuel burning for greener, healthier and better world in many applications such submarines, air crafts, automobiles, ships, trains, motorbikes and too many other applications.

Keywords: *Computational Fluid Dynamics, Physical Process, Simulation, Applications.*

I. INTRODUCTION

In a lot of scientific issues, from designing airplane engine to predicting the weather, there is a need to calculate the dynamics of liquids and gases. These complex simulations, known as computational fluid dynamics (CFD), amount to huge amounts of calculating time, if good accuracy needs to be achieved. There are examples where the simulation of a few turns of a propeller can take thousands of CPU hours on powerful workstations. In order to significantly reduce the simulation time for CFD, utilizing parallel processing will be of paramount importance.

II. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is a method that uses techniques from physics, applied mathematics and computer science to model, predict and visualize how fluids, that is, gas or liquid, flows [1]. A qualitative and quantitative prediction can be put together with the help of CFD which uses mathematical modeling tools, numerical computation and software tools to devise, understand, construct and therefore, predict the required scenarios.

2.1 THE APPLICATIONS OF CFD

Computational Fluid Dynamics can be used in different domains of science, which include [2]:

- Maximizing yield of equipment.
- Modeling ventilation and air flow for cooling purpose.
- Optimizing strategies for oil discovery and extraction.
- Forecasting weather and potential chances of disasters.
- Building efficient models in CFD simulation to ensure Energy and cost savings, CFD simulation is frequently used are Aerospace / Aeronautics, Automotive or Automobile, Building HVAC[(heating, ventilation, and air conditioning), Chemical / Petrochemicals, Energy / Power Generation, Manufacturing / Process Engineering, oil and gas industry, Product Design and Optimization, Oil and gas Industry, and Turbo Machinery, etc. In most of the instances, Computational Fluid Dynamics Modeling is used for deriving most accurate options for designing or the efficiency level of HVAC system.
- Applications of CFDs are not limited to the aforementioned domains and therefore, are extensively applied in various fields of study.

2.2 CFD AS A SIMULATION TOOL

CFDs are used as an alternative to actual experimentation since it provides a cost-effective means of observation and experimentation. Using simulations, CFDs allow scientists, engineers and practitioners to observe multiple scenarios based on desired measurements, and be able to reach more accurate conclusions. The reason behind this is that CFDs enable the simulations of experiments which in reality would be too expensive, difficult to conduct or impossible given the circumstances. Main reasons CFDs is a preferred tool by practitioners include [3]:

- Solving problems virtually that are otherwise impossible to conduct or tackle
- A cost effective method that doesn't lead to experimental costs
- Designing flows for actual implementable solutions rather than laboratory-sized models.

2.3 ADVANTAGES OF CFD

For complex modeling geometry, CFD provides quick results when associated with physical modeling techniques as they would take more time, space and costs [4].

CFD allows for the study of in depth queries in a more structured and visualized manner. This helps in creating new theoretical developments in several fields for advance developments [5]

CFD is also a cost effective measure as it coverts actual fluids into digital imagery for better, elongated and uncorrupted analysis. This is appreciated through the advancements in various field and their plummeting costs. CFD is also used for controlled simulations that are not created in real life situations. For e.g. a nuclear blast or a massive volcano explosion [5]

CFD gives the possibility to analyze different problems which are very difficult and dangerous in experimental way.

2.4 DISADVANTAGES OF CFD

Even though there are various advantages of using CFD, there are cons to the application as well. For instance, it can differ to different physical model results and can lead to an impasse as to which should be taken as the more authentic result [4].
Results are also 20-40 % less comparable to a physical model result which indicates the importance of real time labor works which is more authentic in accurate data [4].

CFD has a bright future ahead as its micro use such as interpretations of internal climate of greenhouses can be applied on a macro level scale.

III. MODELING OF A PHYSICAL PROCESS

Modeling physical process is an activity to analyze, evaluate, simulate and define the common knowledge in a referencing manner that can be easily understood by the people. Its main aim is to provide crystal clear information about the object through simulations and convey the conceptual understanding regarding physical model [6].

Modeling a physical object varies with the situation and requires the relevant aspects from conceptual method to operationalize method by visualizing the subject. It is an essential part of different scientific fields which paves a way to create innovation of new objects such as modeling of electromagnetic system induction heating system and etc.

3.1 THE IMPORTANCE OF MODELLING PHYSICAL PROCESS

Modeling physical process hopes that the material which is presented will be useful for designer dealings and analyze different design of devices [13]. All the models are built on simulacra that is consideration of reality and it is much useful than empirical objects [7].

Modeling of physical process seeks attention to represent the specific topic into logical and objective way. It covers the aspect of various fields such as science education, philosophy of science and knowledge visualization. There is huge collection of methods and techniques bring the optimum result of any specified scientific modeling [8].

3.2 THE IMPORTANCE OF CHOOSING PROPER DIMENSIONALITY AND CO-ORDINATE SYSTEM

The main purpose of using dimensions and co-ordinate system is to brief a complete description of physical model. This system helps to determine the placement and position of object as well specify the facts and figures according to its numerical values. Dimensions show the size and features as well specific detail which shows its manufacturing quality. These are the most essential and quantitative description to create an object into the substantial position and generate visualization which can be generally accepted and easy to interpret [9].

The coordinates are real numbers mostly in general mathematics but as it gets into abstract system it may be complex number such as in Space. There is different coordinates system but it is only liable with respect to the conditions [10].

Following are some of the terms which are used in CFD.

- Discretization:

In CFD discretization is basically a discrete solution of the equation. Execution of differential equation through a set of algebraic nodes, instead of continuous solution in CFD, discretization generates useful solutions. Discretization process involves approximating derivatives, which produce the result close to the actual result. The Formula used to perform the computation is

$$\partial 2U / \partial x2 = \partial / \partial x (\partial U / \partial x)$$
$$\approx ((UE - UP)/\Delta x - (UP - UW) / \Delta x) / \Delta x$$

Using this formula result is a set of algebraic equation so U-velocity can be write in the form of matrix. So the result can be calculate much easier and get the approximate result [11].

- Numerical Grid:

A geometrical shaped box which has a physical domain is called grid. To identify the discrete figures where physics laws can be applied. In CFD Numerical grid is the first process that involved numerical computations of physics equations. The result of equations based on the numerical grid [11]. A numerical grid which has constructed good with less complexity gives the approximate solutions. There are many methods to construct the numerical grid few are listed below [10]:

A. Algebraic Methods
B. Differential equations Methods
C. Variation Methods
D. Unstructured grid Methods

These methods are used in CFD to construct or generate a numerical grid using the numerical values and equations

- Initial Conditions:

In CFD initial conditions are the velocity, time, pressure and any field values at that time t = 0. It allows user to use the result of previous projects in the current project and helps to perform the calculations faster and produce approximate solutions. There is also a boundary condition in which the main factor is space but in initial conditions the main factor is time [12]. The formula for initial conditions with respect to time is written below:

$$Vt + AV^{-} x + BV^{-} y + CV^{-} z = F^{-} x + G^{-} y + H$$

Some other quantities may be assigned as initial condition: Temperature, Scalar, and Humidity.

- Boundary Conditions:

It deals with the way fluid ingress and emits in a simulation model. This process is also defined as the alterations that occurred between the model and its surrounding. In other way we can also say that boundary condition connects the model with its surrounding. This condition could be pressure on model, heat fluidity, velocity of model and volumetric flow rates. Boundary condition alters depending on their environment respectively [11].

Boundary conditions connect the simulation model with its surroundings, without them, the simulation is not defining, and in most cases cannot proceed. Most boundary conditions can be defined as either steady state or transient. Steady state boundary conditions persist throughout the simulation. Transient boundary conditions vary with time, and are often used to simulate an event or a cyclical phenomenon.

- Sweep:

Mesh is actually a spongy web like structure. Sweep meshing or sweep is an important component of CDF process. In CFD process we mesh our model to redefine the structure of our model. The sweep meshing starts with specific point or segment in our model where we want to apply sweeping. Sweeping includes reducing or increasing mesh cells in the layers of model. After manual or automatic settings of system, the sweeping process starts here the user selects the particular part where the cross-section of model aids him/her whether any alterations in model is needed or not [10].

- Convergence:

CFD modeling is an iterative process and reoccurring processes doesn’t possesses exact solutions. The reason is the mathematical equation involves in CFD are not linear equation hence the variable could have multiple values as a result convergence occurs in turbulence modeling. The solution to convergence is to define an accuracy level that could be closed to what we require as our results. Now, it solely depends upon what criterion is required to minimize convergence [12].

- Turbulence Modeling:

Turbulence modeling uses regressing approach that means that the new value that we achieved is the mean flow of previous values and the accuracy attained after each iteration. There are some common methods for turbulence modeling that are zero equation model, one equation model, two equations model and seven equations model. These are all time averaged methods. In order to increase effectiveness of turbulence model we need it to be accurate, simple and economical [12].

IV. CONCLUSION

Computational Fluid Dynamics (CFD) is the application of different fields of science and mathematics to study fluid, liquid or gas, that are particularly in motion through computational means. These studies and analysis consist of vast mathematical equations that are used to comprehend how a system is performing and how optimal results can be obtained. So as mentioned previously there are many disadvantages of applying CFD but still the advantages are considered to be more operational functional in many fields application.

CFD is the science of the future as it cares in all aspects of life in the present and the future, CFD science treats the fluids mainly the air and the water as good and bad, bad when the CFD tries to find a way through the air and the water to get the minimum resistant for cost effective and less fuel burning for greener, healthier and better world in many applications such submarines, air crafts, automobiles, ships, trains, motorbikes and too many other applications, in the same time the CFD deals with the fluids as good environment when it tries to get the most out of it, for instance generate electricity by using the wind mill and the tidal stream turbine, taking every single effort to get most out of the wind and the tidal steam to generate clean electricity for greener world.

V. DECLARATION

All authors have disclosed no conflicts of interest and the project was self-funded

REFERENCE

[1] Guang Xu, Kray D. Luxbacher, SaadRagab, JialinXu, and Xuhan Ding, 2017, Computational fluid dynamics applied to mining engineering: a review, International Journal of Mining, Reclamation and Environment, Vol. 31, Iss. 4, pp. 251-275.

[2] Scott, Gordon, and Richardson, Philip, 1997, The application of computational fluid dynamics in the food industry, Trends in Food Science & Technology, Vol. 8, Iss. 4, pp. 119-124.

[3] Gandhi, Priya, Brager, Gail, and Dutton, Spencer, 2014, Mixed Mode Simulation Tools, Internal Report, University of California.

[4] Linfield, K. W., and Mudry, R. G., 2008, Pros and cons of CFD and physical flow modeling.

[5] Tu, J., Yeoh, G., and Liu, C., 2012, Computational Fluid Dynamics – A practical Approach, 2nd Edition, Elsevier, UK.

[6] El Hagrasy, A.S., Hennenkamp, J.R., Burke, M.D., Cartwright, J.J., and Litster, J.D., 2013, Twin screw wet granulation: Influence of formulation parameters on granule properties and growth behavior, Powder Technol .

[7] Draper, Norman R., and Box, George E.P., 1987, Empirical model-building and response surfaces, John Wiley and sons.

[8] Frigg, R., Hartmann S., and Imbert C., 2009, Special Issue Models and Simulation, Synthese, Springer Netherlands, Vol. 169, Iss. 3.

[9] Tsuboi, N., Daimon, Y., Hayashi, A., 2008, Three-dimensional numerical simulation of detonations in coaxial tubes, Shock Waves, 18: 379-392.

[10] Nomizu, Katsumi, 2001, Selected Papers on Classical Analysis, American Mathematical Society Translations: Series 2, USA.

[11] Ferziger, Joel H., and Peric, Milovan, 2002, Computational Methods for Fluid Dynamics, 3rd Edition, Springer, Germany.

[12] Raase, Sebastian, and Nordström, Tomas, 2015, On the Use of a Many-core Processor for Computational Fluid Dynamics Simulations, Procedia Computer Science, Vol. 51

[13] Thabit, Thabit H., and Younus, Saif Q., 2018, Risk Assessment and Management in Construction Industries, International Journal of Research and Engineering, Vol. 5, No. 2, pp. 315-320.