ANALYSIS ON APPLICATIONS ON COMPUTATIONAL FLUID DYNAMICS

C. Haritha,S. Aruna
Department of Mathematics & Humanities, Mahatma Gandhi Institute of Technology, Hyderabad
Corresponding Author: aruna_siripurapu@yahoo.com

ABSTRACT
The majority of major fire accidents that occur in industrial plants involve hydrocarbon fuels. In recent decades, computer fluid dynamics (CFD) modelling in the field of fire risk analysis has been used widely, given the need for a thorough understanding of phenomena associated mit hydrocarbon fire. The present work reviews the key features and the practical application of CFD techniques in various industries in the last more than 50 years with the aim of how and where they can be used.It has been concluded that there are numerous possibilities CFD offres, and as both computer hardware and software resources become more advanced the extent of this opportunity will further develop.

1. Introduction

The combination of physics, flow techniques, computer applications, maths, and mechanics is Computational Fluid Dimensioning (CFD). It is a set of techniques designed to solve the equations of NavierStokes (or strictly speaking, in most cases Reynolds-Averaged Navier-Stokes), thus satisfying the conservation of mass, dynamism and energy for the prediction of the behaviour of liquid systems. CFD is a modern tool for the purpose of researching domain space for physical systems design and performance variables and for the diagnosis or resolvement of system behaviour. CFD is computer-aided engineering software (CAE).

Typical scenario for complementing or substituting the application of CFD existing analytical techniques involves the analysis of a large number of design variations or forbids physical tests because of limiting factors such as scale, cost, accessibility, physical or environmental hazards. In case of previously validated modelling methods or operational data for validation easy to obtain, CFD is particularly prevalent. The business and legislative drivers demanding technical development and improved efficiency were key factors influencing the use of simulation techniques such as CFD.

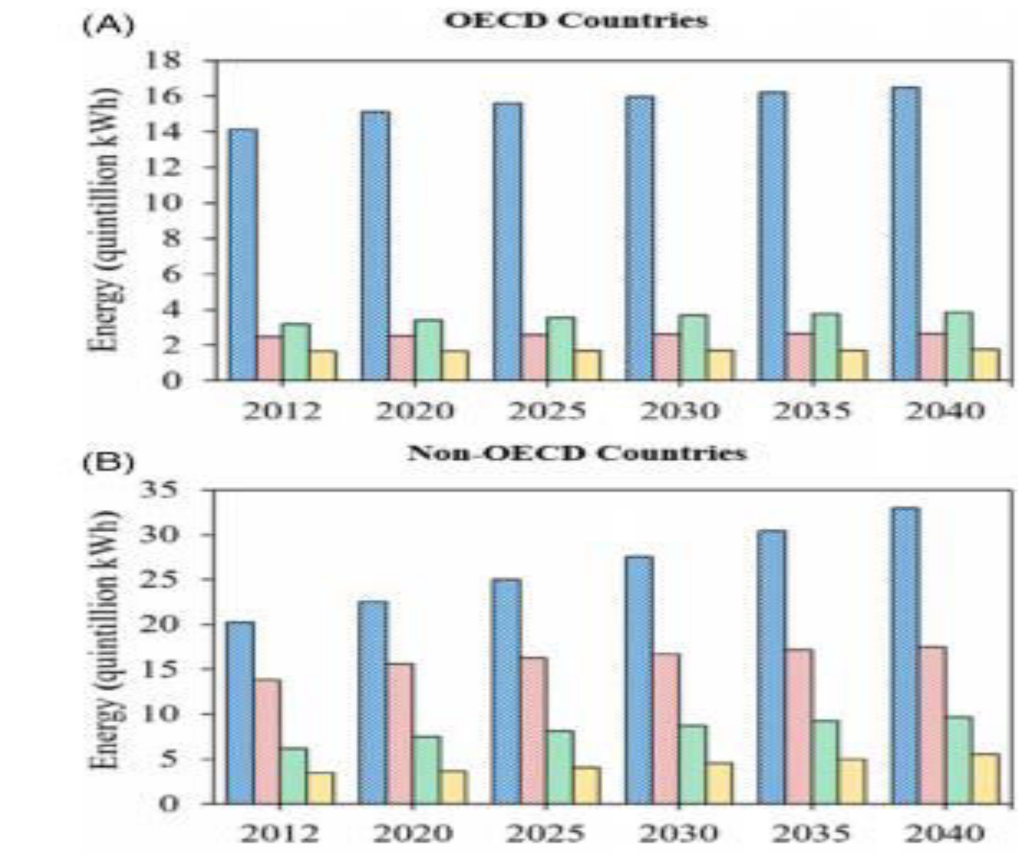

FIGURE 1 Worldwide energy consumption in the industrial sector for: A, OECD countries; and B, non-OECD countries.

In 2016, the United States Energy Information Authority (EIA) reported that it consumed approximately 54 percent of the energy supply in the world for multiple purposes. [1] Depending on their economic activities and technological development, the diversity and amount of fuel consumed vary significantly between countries. Those non-OECD countries have a maximum consumption growth averaging 1.5 percent per annum between 2012 and 2040, while those belonging to the OECD have a 0.5 percent annual consumption growth (Figure 1).In the worldwide industrial sector, the average energy consumption is estimated to increase by 1.2 percent per year to 2040. by 2040.

Among the different sources of energy available, hydrocarbons are the most widely used and will continue to be found in the near future. One of the main advantages of these fuels is the high heat values that cause huge amounts of energy to be released. Also hydrocarbon earth reservoirs are easily identified and thus easily extracted and transported to industrial sites.Consequently, the process of generating energy from hydrocarbon energy is cheaper and more economical than other non-conventional forms of energy, for example renewable energy. The quantity of hydrocarbon fuel on Earth is however limited, and it is recognised that soot- and smos-free solid particles can significantly harm the environment and human health.

2. Literature Review

Shivakumara et al. (2017)[1] found that spatial and time temperature fluctuations occur by mixing two different fluids at different temperatures. If this fluctuation is high, this can cause structural damage due to a high cycle of thermal tiredness and is called a thermal strip phenomenon. Alternatively, the Computational Fluid Dynamics (CFD) is used because it reduces the number of experiments required for the design process, the cost and time required.

The effect of the inclination angle on flow of fluid in a pipe is detected by numerical simulation in Laohasurayodhin, etc. (2014)[2].

The dump diffuser used in marine gas turbines and modern aircraft motors was analysed internally by Klein (1995)[3] by means of k- till turbulence model in ANSYS FLUENT. They found that when the diffuser angle is increased, there is no diffuser effect.

The steep, incompressible fluid flow by a CFD-technique T-junction was investigated by Nimadge and Chopade (2017)[4]. You prepared experimental setup for the acquisition of reference information on the passage of fluid via T-pipe junction and used the same data for CFD analysis via FLUENT and ANSYS.

Spooner et al (2017)[5] have developed a multitude of bifurcation/trifurcation arrangements that have resulted in head losses of a fluid system and generated quantitative data of a sharply-edged fluid dynamics loss coefficients (CFD).

Gedik (2017) [6] is subjected to an experimental and numerical study of magnetic theological (MR) fluids in circulatory pipes under the effects of computer fluid dynamics on a uniform magnetic field (CFD).The pressure effects on pipe wall under the various piping systems Y-joints, coleboxes, T-joints, bends, contractions, valves and many other components through the CFD technique were investigated by Patel et al. (2005)[7]. Banjara et al. (2017)[8] have investigated the effect on a trifurcation pipe branch, using the CFD analysis, of mass flow rate and speeds of turbulent fluid flow throughout the pipe length under various reynolds.

3. CFD Modelling

Turbulence models

The mixing phenomenon of fluid is described in three main, well known approaches: the Reynols average Navier-Stokes(RANS), the direct digital simulation (DNS), and large eddy simulations (LES). The latter were mainly designed to determine the turbulent viscosity of the flow µt, which is a key parameter for the CFD fire simulation turbulence phenomenon. The RANS approach only solves the Navier-Stokes equations for the medium flow variables and only calculates the big movements. RANS has been used extensively in industrial CFD applications in recent years, due to its simplicity and low computer requirements.One of the most common means of closing RANS equations is by k− − µtwo-equation eddy-viscosity model.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

where ρ represents the fuel density, k the turbulent kinetic energy, ε the dissipation rate, and Cμ a Constant model with frequently assumed value 0.09. However, the limits of the RANS approach were already recognised for simulating the highly oscillating flood-induced flow
The DNS approach on the other hand tries to solve the equations with their entire complexity and resolves edges across the entire range of time and space scals.
DNS requires high computing resources, and nowadays only a few research problems can be applied.LES is an intermediate approach between DNS and RANS when the small eddies of the flow are being filtered out. It also provides detailed information in a reasonable computational time about instantaneous airflow and turbulence. Smagorinsky is one of the most frequently used LES closure methods.

$$\mu_t = \rho(C_s\Delta)^2|S|$$

If Cs is a 0.2 model constant, S is tensor stress rate, and iv is filter size of the LES. iv (ie, cubic root of the cell volume). The model generally overestimates the turbulent viscosity within the neighbouring wall areas, making it inappropriate for certain transitional streams. There are more complex LES closing methods, for example the one developed by Deardoff, to correct the defects resulting from the Smagorinsky model[2]

$$\mu_t = \rho C_d \Delta \sqrt{k_{sgs}}$$

Where the cd is the 0,1-value model and ksgs is represented in a sub-grid-scaling stress by turbulent kinetic energy, which is a major process of small scale.

Combustion models

Through combustion models such as eddy breaking up (EBU), the nozzle dissipation system (EDC) as well as the laminar flamelet method, the products released due to the reaction to the fire can be measured (LFM). The EBU, the earliest model of combustion for turbulent flames, was based on an unlimitedly fast stoichiometric chemical reaction in just one step. It has certain limitations despite the wide use of the EBU model in different applications. It cannot represent many gas species; it can make no difference if the mixture is lean/rich.

Conclusions

Many computer simulations involving hydrocarbon fires have been performed during recent decades in open environments, as demonstrated by this comprehensive literary review. In computational simulations, large and medium fires were preferably modelled. FDS was the CFD code most used to fire jets and fire pools. The main aim of most simulations was to validate a CFD tool of interest by comparing the forecasts made with experimental results. There was generally a reasonable agreement but, instead of using quantitative comparison criteria, most of the simulations were qualitatively validated.

References

[1] U.S. Energy Information Administration (EIA), International Energy Outlook 2016, 1st ed., US Department of Energy, Washington, DC 2016, p. 19.
[2] F. I. Khan, S. A. Abbasi, J. Loss Prev. Process Ind. 1999, 12, 361.
[3] E. Palazzi, B. Fabiano, Process Saf. Environ. Prot. 2012, 90, 121.
[4] M. Masum, A. Rahman, S. Ahmed, F. Khan, Reliab.Eng. Syst. Saf.2015, 143, 19.
[5] J. Casal, Evaluation of the Effects and Consequences of Major Accidents in Industrial Plants, 2nd ed., Elsevier, Amsterdam, The Netherlands 2017.
[6] P. J. Rew, H. Spencer, T. Maddison, Hazards XIV: Cost Effective Safety: A Three-day Symposium Organised by the Institution of Chemical Engineers (North Western Branch) and Held at UMIST, Manchester UK 1998, 1st ed., Institution of Chemical Engineers, Rugby, UK 1998.
[7] A. Palacios, A. Muñoz, J. Casal, AIChE J. 2008, 55, 256.
[8] K. S. Mudan, Prog. Energy Combust.Sci. 1984, 10, 59.
[9] A. Hamins, S. J. Fischer, T. Kashiwagi, Combust. Sci. Technol. 1994, 97, 37.
[10] T. Steinhaus, S. Welch, R. O. Carvel, J. L. Torero, The rm. Sci. 2007, 11, 101.