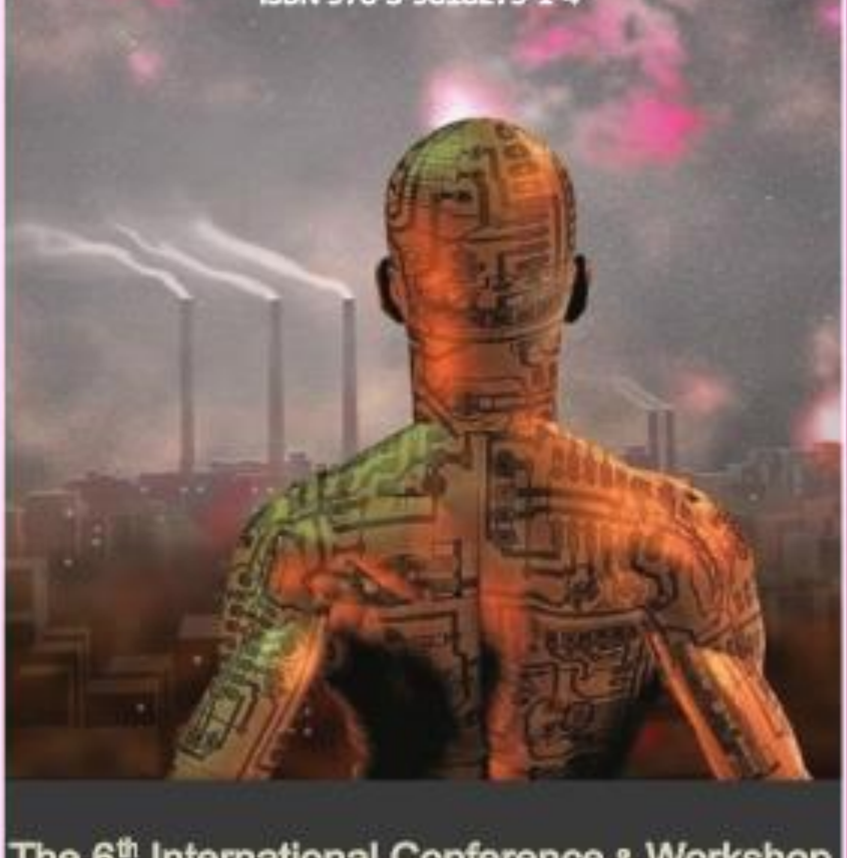
Proceeding

SCIENCE AND ENGINEERING FOR RELIABLE ENERGY

ISBN 978-3-9818275-1-4



The 6th International Conference & Workshop

REMOO-2016

Budva / MONTENEGRO 18-20 May

RESEARCH AND EDUCATION IN THERMAL AND POWER ENGINEERING WITH SUPPORT OF CFD TECHNOLOGY

Risto V. FILKOSKI[1)], Marija CHEKEROVSKA[2)], Florent BUNJAKU[3)]

[1)]University "Sts Cyril and Methodius", Faculty of Mechanical Engineering, Karpos II nn, 1000 Skopje, Macedonia, risto.filkoski@mf.edu.mk
[2)]University "Goce Delchev", Faculty of Mechanical Engineering, 6000 Shtip, Macedonia
[3)]University in Prishtina, Faculty of Education, Prishtina, Kosovo

Abstract

Advanced mathematical methods offer opportunities for an in-depth analysis, optimisation and examination of various options to increase the overall efficiency of the thermal energy facilities. Computational fluid dynamics (CFD) technique, as a powerful engineering tool, has been extensively used for modelling and investigation of operational behaviour of thermal energy systems. Advanced CFD techniques help researchers in performing research work efficiently and in interpretation of test results. The improved solver technologies and outstanding modelling possibilities also support teaching of numerical analysis and application fundamentals for a broad range of disciplines within the thermal, process and environmental engineering. In this work, the capabilities of CFD technique are demonstrated with practical examples in connection with certain specific issues: realistic representation of the object geometry, visualisation of the processes occurring in the analysed object, comparison between the different turbulence modelling approaches, the combustion process analysis at different levels depending on the specific needs, evaluation of the impact of the applied thermal radiation model on the results and analysis of solar collectors' efficiency in different operating conditions.

Keywords

Computational fluid dynamics (CFD), combustion, energy efficiency, fuel, fluidised bed.

1 Introduction

The importance of energy as an essential factor for economic growth, as well as for overall social development is very well known [1]. Advanced modelling techniques, such as computational fluid dynamics (CFD), are established as very useful tools for solving different energy and environmental problems, particularly suitable for prediction of aerodynamics and thermal processes (fluid flow, fuel burnout, heat transfer, formation and reduction of pollutants) in the thermal energy systems. In combination with experimental research and/or on-site measurements, they offer multiple benefits: cost and time reduction, possibility to reproduce the operating conditions, as well as detailed insight into the complex interacting physical phenomena and chemical reactions determining the combustion process [2,3]. The CFD significantly facilitate the investigation of the influence of various process and design parameters to the combustion efficiency and, consequently, to the overall plant efficiency and the emission of pollutants.

A comprehensive review of the modelling approaches and techniques of combustion systems on solid fuels is given in [3]. Different standpoints, objectives, advantages and shortcomings of particular approaches to the modelling of two-phase flow, turbulence, combustion and thermal radiation are discussed in [4]. The permanent progress in the computers capability has enabled development and application of massive mathematical models of turbulent flows and thermal processes in the combustion systems and extensive use of CFD technique with satisfactory results [5-9]. Comprehensive 3-D models of industrial-scale combustors have been developed and successfully applied for years now [5-12]. Numerical codes describing the processes in combustion plants, based on a solution of differential conservation equations, have been a subject of numerous investigations [9-12]. An overview of different turbulence modelling approaches, including some widely known and applied (k-ε turbulence model, Spalart-Allmaras, LES, DNS etc.) and some specific (two-scale second-moment one-point turbulence closure, rescaled v^2-f model etc.) is given in [15]. Despite some weaknesses, the k-ε turbulence model, or some derivatives, like RNG k-ε model or k-ε-k_p two-phase turbulence model, are often used in combustion systems, mostly due to the simplicity. Very often, the standard k-ω model, based on Kolmogorov's work (1942), is employed as a reasonable compromise [16]. Gas phase conservation equations are mostly used as time-averaged, but some prefer the Favre-averaged equations instead [9]. Two-phase flow is usually described by Eulerian-Lagrangian approach and PSI-CELL method for taking into account the influences between phases. Usually the combustion sub-models separately treat the releasing of volatiles, char oxidation and gas phase reactions, like in [9,11,12,17]. Thermal radiation in the furnaces is modelled by means of various approaches, like discrete transfer method [17], the P-1 as a variation of the P-N model [18], the six-fluxes method [19], Monte Carlo method [2], or discrete ordinates (DO) method [9,10,11]. Generally, a comprehensive model of the furnace processes must balance sub-models sophistication with computational practicality.

2 Methodology

The CFD technique can be successfully employed to investigate and better understand the physical processes such as complex fluids flow, chemical reactions and heat transfer that occur within the designated energy objects. These processes are closely related to the interaction of phenomena associated with dissipation, diffusion, convection, boundary layers, turbulence, thermal radiation etc. Although the analytical methods are still widely practiced and experiments will continue to be significantly performed, the general trend is clearly toward greater reliance on the computational approach for industrial design, particularly when the flows are very complex. Whether the fluid flows are incompressible or compressible, most of their crucial aspects are nonlinear. As a consequence, mostly do not have any analytic solution, which appears as a motivation to seek numerical solutions for the partial differential equations.

Aside from the doubtless scientific-research possibilities, the CFD technique is a powerful educational tool to learn basic and advanced thermal-fluid science. There are several prime benefits of the CFD use in the educational process in the field of energy, thermal and environmental engineering. Recently, CFD is revolutionising the teaching of fluid mechanics and thermal science through visualisation of complex fluid flows. The experience with a more hands-on approach to better understand the concepts of fluid flow and heat transfer greatly deepens the students' understanding of the fluid-flow phenomena. In particular, the visualisation capability greatly enhances the students' intuition of the flow behaviour. Such an approach opens the door to new classes of problems that can be solved by engineering students, who are no longer limited by the narrow range of classical flow solutions.

When it comes to the problems of thermal engineering, the most sensitive issues are related to the processes of flow and turbulence, combustion, accompanying mechanisms of pollutants formation and radiation heat transfer. The choice of the turbulence model depends mostly on the flow physics, the accuracy level required, the established practice for a specific class of problems, the available time and computational resources for the simulation. There is not a single turbulence model that is universally accepted as being superior over the others for all classes of flow problems. A combustion plant on solid fuel in the most complex case, as an object for numerical modelling is characterised with a weakly-compressible particle-laden flow, chemical reactions of the released species, burnout of the char particles and heat transfer phenomena occurring in a turbulent flow. In order to properly include the turbulence in a comprehensive model of a combustion plant, one should make a choice between many different models [15].

Regarding the combustion, a possible approach is to consider species transport with chemical reactions including: gas phase reactions that may involve NO_x and other pollutants formation, volumetric and chemical kinetics of reacting flow, particle surface reactions in which the reaction occurs at the surface of a discrete phase particle, etc. There are different approaches to model gas phase reacting flows: generalised finite-rate model, non-premixed combustion model, premixed combustion model, partially premixed combustion model, composition PDF transport model etc. A simulation of pulverised coal combustion involves modelling of a continuous gas phase interaction with a discrete phase of coal particles. The particles, travelling through the gas, devolatilise and undergo char combustion, creating sources of fuel for gas phase reaction. In the works [4,21-23] species and chemical reactions are modelled using the mixture fraction / probability density function (MF/PDF) approach and the full equilibrium chemistry, where the turbulence-chemistry interaction is modelled using a specific function, i.e. double-delta PDF. The equilibrium chemistry model assumes that the chemistry is rapid, so that chemical equilibrium always exists at the molecular level. The poly-disperse coal particle size distribution is assumed to fit the Rosin-Rammler equation. An alternative approach would be to use the generalised finite rate formulation method, known as the Magnusen model [24]. The method is based on the solution of species transport equation for reactants and products and the chemical mechanism is explicitly defined.

When it comes to the heat transfer modelling, the thermal conditions in the considered domain should be further analysed. Specifically, in the combustion plants, the selection of a thermal radiation model has a central role, due to the temperature level. The selection of the most appropriate thermal radiation model in certain conditions depends on various factors, such as the optical thickness, the possibility for inclusion the scattering and emissivity effects, the way the model is treating the effects of the presence of discrete phase and the model behaviour in the case of medium with localised heat sources. The P-1 model, as simplified P-N differential approximation, has certain specific advantages over other models in treating the radiate energy transfer in a grey absorbing and emitting medium with presence of particulates. It is relatively simple, treats the radiative transfer equation as an easy-to-solve diffusion equation and it can be easily applied to complicated geometries. The DO and the P-1 models are utilised in [18, 25-28], since they effectively comprise the influence of the discrete phase presence in the boiler furnace, unlike some other frequently used models in thermal engineering applications, like the Discrete Transfer Radiation Model (DTRM) [17]. The optical thickness of the radiating medium aL, where a is absorption coefficient and L is characteristic path length, is one of the indicators of which model to use in the analysed problem [4, 23]. If the optical thickness is large, aL>>1 or aL>1, the P-1 model should typically be used. The DTRM and the DO model work across the range of optical thicknesses, but are substantially more resources demanding. The P-1 and DO models account for scattering and emissivity, while the DTRM neglects it. Also, only the P-1 and DO models account for exchange of thermal radiation between the gas and particulates. In problems with localised heat sources, such as solid fuel particles, the DO model is probably the best suited for computing radiation [4].

3 Presentation of Some Case Studies

The CFD technique offers an opportunity for very good visualisation of the designated object geometry. Also, it has a power to capture the complex flow characteristics, as well as an in-depth insight of the processes and phenomena taking place in the analysed system and good presentation of the obtained results. Some typical examples are presented in Figures 1 to 4 [29, 21, 22]. An expressive visualisation of the object geometry is given in Fig. 1, showing the numerical domain that consists of a circulating fluidised bed (CFB) combustor and two cyclones [29]. Another example is a computational domain of a furnace and a cyclone of a hot-water boiler at Chalmers University, Goteborg, Fig. 2 [21]. An interesting case where the CFD technique can provide excellent visual illustration of the two-phase flow is given in Figures 3 and 4 [21, 22]. In this case, a subject of investigation is a middle scale utility boiler on pulverised coal with tangential disposition of burners.

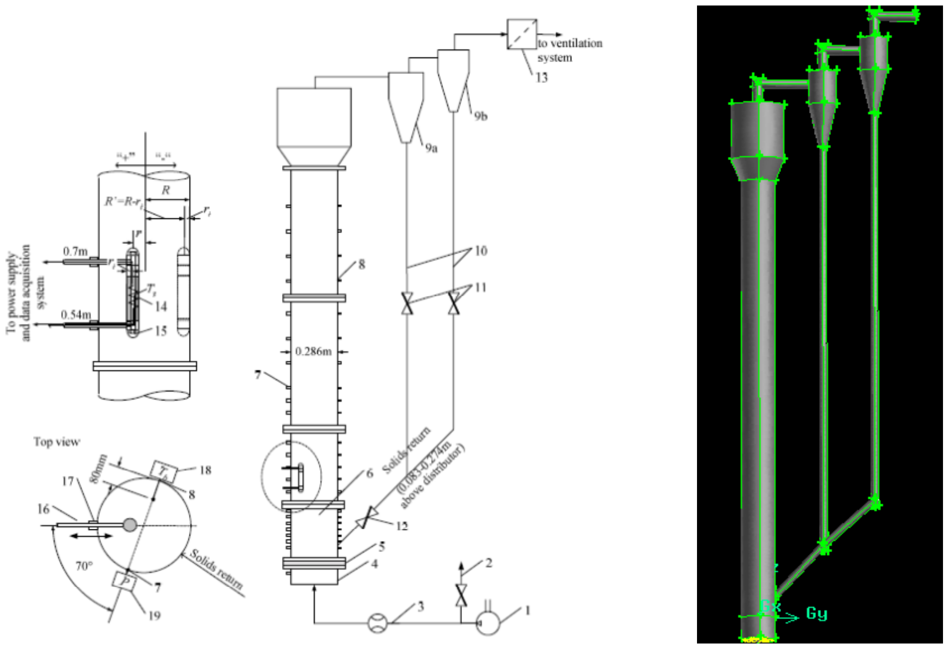


Fig. 1. Circulating fluidised bed system and a numerical domain for CFD simulation [29]

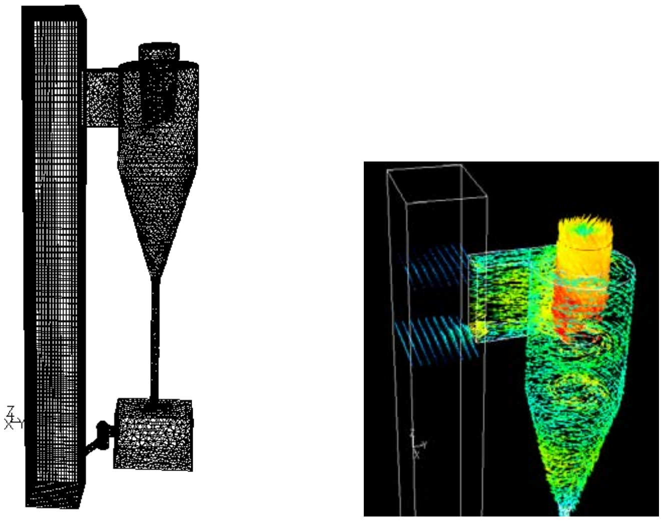


Fig. 2. Meshed computational domain of a hot water boiler and velocity vectors in the cyclone [21]

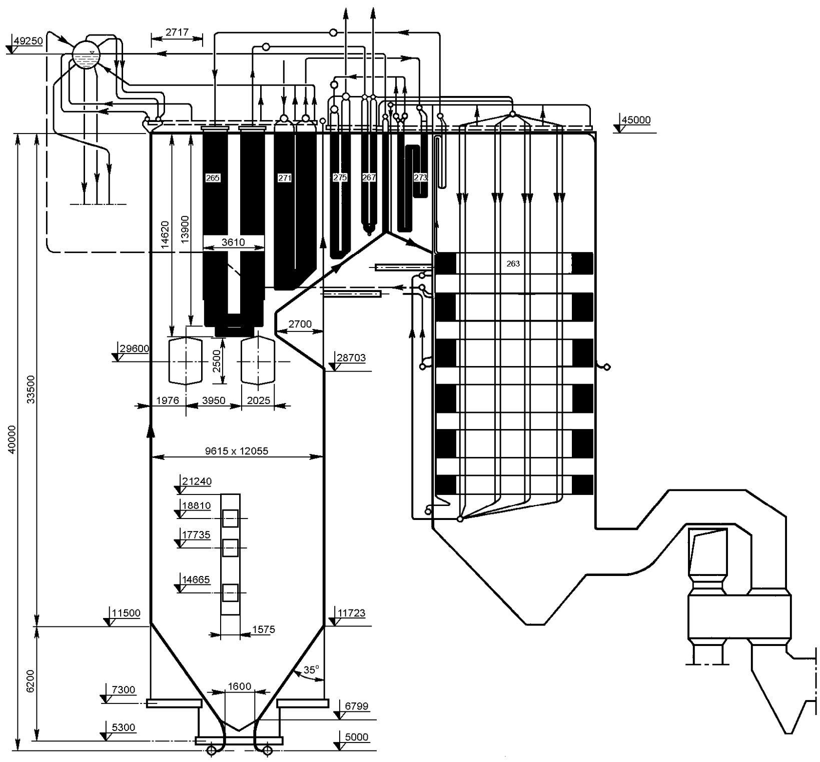


Fig. 3. Schematic presentation of the boiler OB-380 [21, 22]

Often, the objective of the CFD study is to optimise the air and fuel introduction in combustion plants, as very important issue for improvement of their energy and environmental performances. By using proper CFD tool, it is possible to analyse the effect of air redistribution on the flow field, temperature profiles and combustion efficiency. In the case of a utility boiler on pulverised coal, depicted in Figure 5, operating in 225 MWe power plant unit and fired with low quality lignite [4], the CFD modelling technique was used for investigation of the aerodynamic behaviour of the gas-solids mixture, combustion efficiency, temperature profiles and gaseous combustion products concentrations.

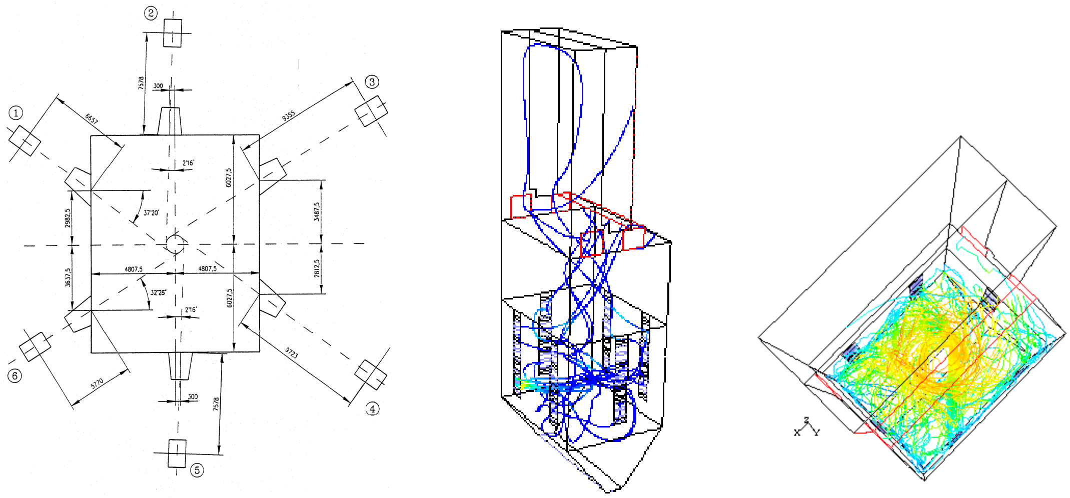
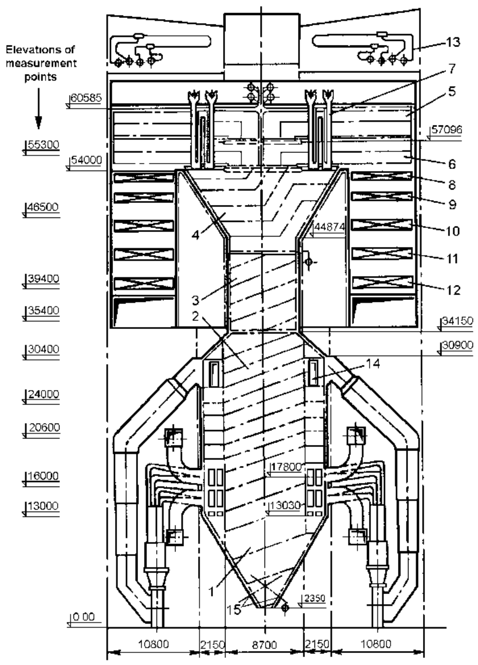
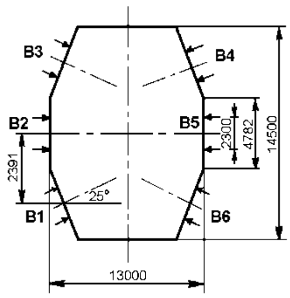


Fig. 4. The boiler OB-380 burners disposition and particles trajectories [21, 22]





1 – Furnace hopper; 2 – Lower radiation part; 3, 4 - Middle radiation part; 5 – Upper radiation part I; 6 - Tube walls in the upper radiation part II; 7 – Superheater; 8, 9 – Convective superheaters; 10 – Transition zone; 11, 12 – Economisers; 13 – Heat exchanger "steam-steam"; 14 – Recirculation openings; 15 – Lower air inlet nozzles

Fig. 5. A schematic representation of the utility boiler Pp-670-140 GOST 3619-76 (P-65) - layout and horizontal intersection showing the burners' disposition [4]

The work was focused on an analysis of the changes of temperature field and combustion efficiency with a design change by adding a number of lower air inlet nozzles at the bottom of the furnace (Figure 5), so that a part of the total air quantity is introduced through them. The simulations have been performed on a basis of commercial CFD code, adapted for pulverised coal industrial and utility scale boiler furnaces. The geometry outline of the domain, representing the boiler furnace and a part of the convective tract, as well as the numerical meshes, are presented in Fig. 6.

As a consequence of the lower air introduction, the local flow field is changed (Fig. 7), additional swirls are created and part of the coarse combustible particles is entrained by the lower air flow to the upper furnace regions, enabling them to undergo complete combustion [4]. Examples of traces of groups of particles released from the burners group assigned as B1 (Fig. 5) in the cases with (a) and without (b) lower air introduction are presented in Fig. 8, showing obvious reduction of the number of particles that fall through the hopper hole [4].

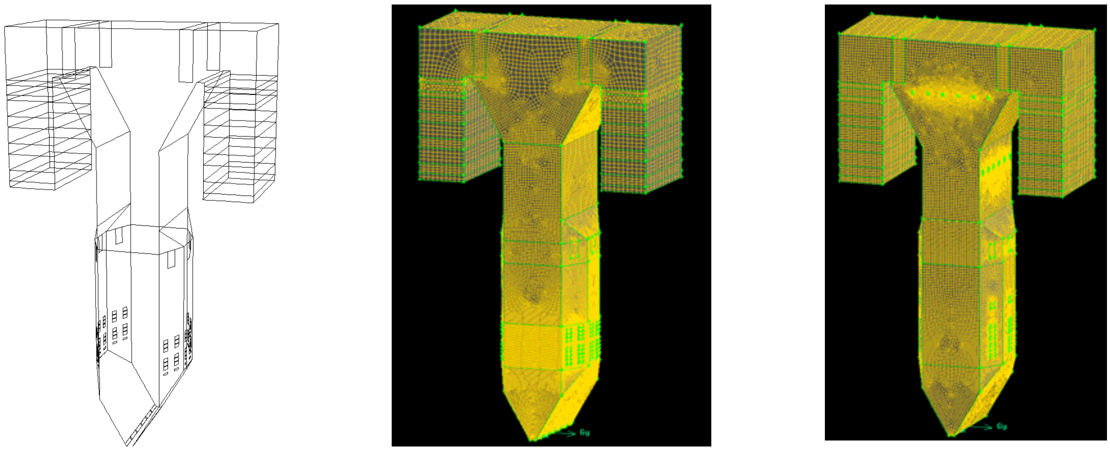


Fig. 6. Geometry outline and the numerical mesh (before and after the boiler modernisation)

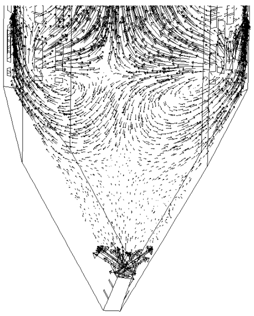


Fig. 7. Velocity vectors in the central vertical cross-section [4]

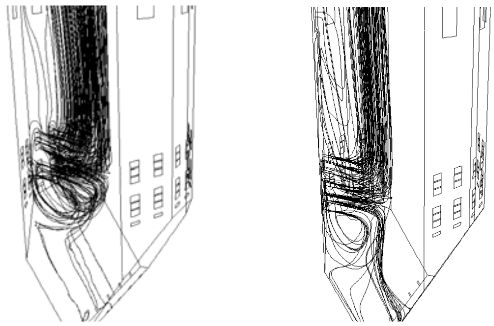
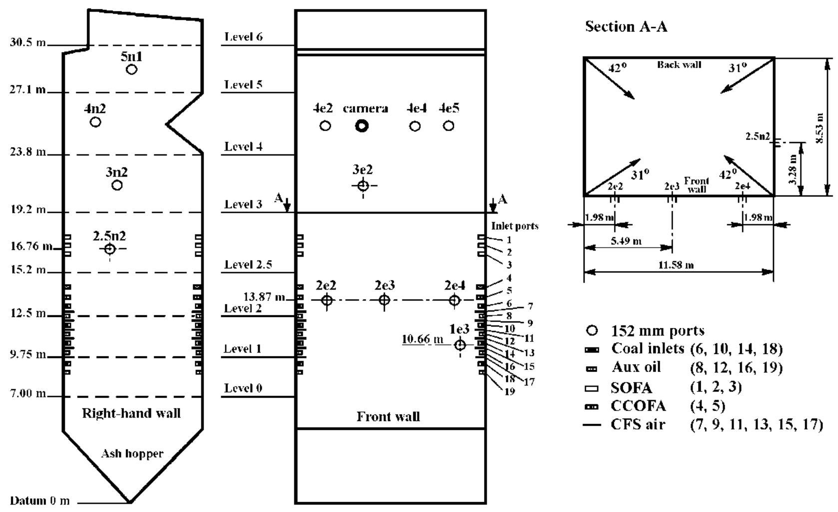


Fig. 8. Traces of groups of particles released from the group of burners B1: a) case with lower air introduction; b) case without lower air introduction [4]

Another interesting case of a combustion system on pulverised coal and multiple fuel and air inlet ports is presented in Fig. 9 [23]. The numerical mesh consists of 688,886 volume cells, and it is created taking into consideration the zones where large gradients of variables can be expected, such as the near-burner-regions (Fig. 10).

An extensive testing programme is performed on a solar collector experimental set-up, installed on a location in Shtip (Republic of Macedonia), latitude 41^{o} 45' and longitude 22^{o} 12', in order to investigate the effect of the sun tracking system implementation on the collector efficiency, Fig. 11 [30]. The set-up consists of two flat plate solar collectors, one with a fixed surface tilted at 30^{o} towards the South, and the other one equipped with dual-axis rotation system. The study includes development of a 3-D mathematical model of the collectors system and a numerical simulation programme, based on the computational fluid dynamics (CFD) approach, with the main aim is to provide information on conduction, convection and radiation heat transfer, so as to simulate the heat transfer performances and the energy capture capabilities of the fixed and moving collectors in various operating modes.



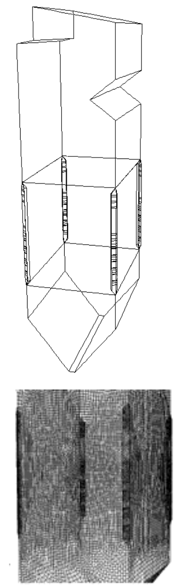


Fig. 9. Schematic representation of the boiler furnace with disposition of air and fuel inlets, measurement ports and direction of burners [23]

Fig. 10. Furnace geometry and the numerical mesh in the burners' region [23]

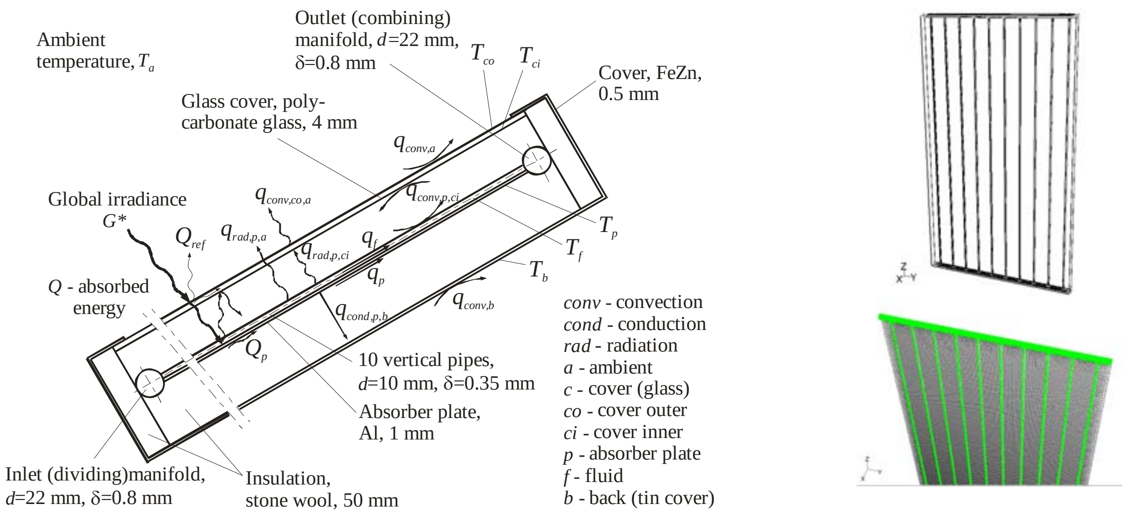


Fig. 11. Illustration of the energy balance of the collector, the outline of the numerical domain and the mesh [30]

4 Results

Some typical results of the provided numerical simulations of the objects presented in the previous section are given in the following figures. The modelling results in all the case studies [4, 21-23] were compared with a test matrix of measurements at different boiler operating conditions. In the case presented with Figs. 5-7, the comparison between the numerically obtained and measured temperature profiles, as well as energy loss due to inefficient combustion shows satisfactory compliance. Simulation results of typical temperature distribution in the central vertical intersection of the computational domain at 97 % boiler load, as well as in several horizontal intersections are presented in Fig. 12 [4]. The plots highlight the flame shape and furnace high temperature regions outside the near-burner-flame boundaries. The calculated temperature profiles along the furnace height at four operating modes and average measurement values at different elevations are displayed in Fig. 13 [4]. Despite the appearance of certain discrepancies, the general trend-line is well predicted with the calculations.

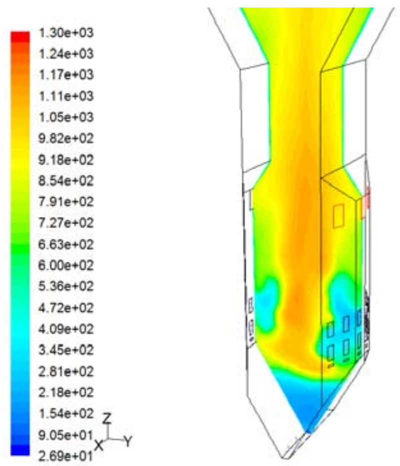


Fig. 12. Temperature contours (^oC) [4]

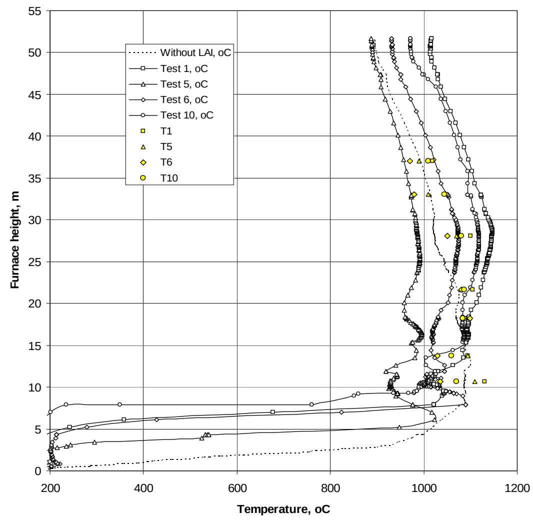


Fig. 13. Temperature along the central axis: CFD-Test 1 – simulations, test 1; CFD-Test 5 – simulations, test 5;CFD-Test 6 – simulations, test 6; Test 1, 5, 6, 10 – measurements [4]

A comparison between the energy losses due to fuel loss through the furnace bottom in the considered cases, elaborated in [4], shows good match between the measurements and CFD results. The described methodology gives a possibility to conduct an optimisation of the ratio between the lower air and total air flow rates, as well as optimisation of the lower air blow velocity with regards to the reduction of energy loss due to incomplete combustion.

As for the case of the utility boiler on pulverised coal depicted in Figures 9 and 10, and described in more detail in [23], typical horizontal temperature profiles obtained with CFD modelling, compared to measurement data, are presented with diagrams in Figure 14 [23].

The feasibility of the proposed method for the solar collector efficiency [30] was confirmed by experimental verification, showing significant increase of the daily energy capture by the moving collector, compared to the immobile collector unit. The comparative analysis, presented here for just one set of operating conditions in Fig. 15, demonstrates a good agreement between the experimental and numerically predicted results at different running conditions, which is a proof that the presented CFD modelling approach can be used for further investigations of different solar collectors configurations and flow schemes.

The obtained results regarding the flow field, particles trajectories, temperature profiles, combustion efficiency and other important parameters in the described and other cases [4, 21-23] are in the expected limits and the comparison with the measurements shows quite satisfactory agreement.

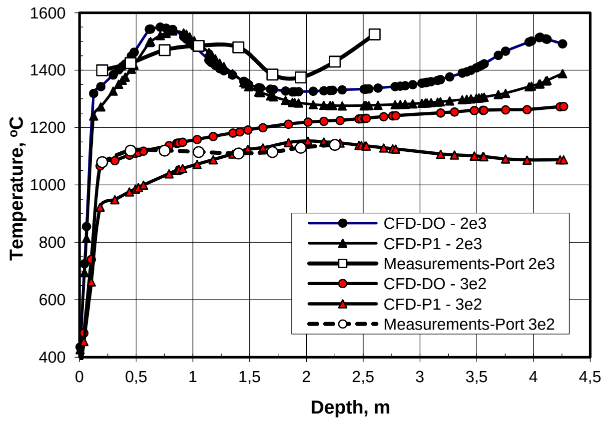


Fig. 14. Temperature profiles on the east (right-hand) wall for three levels of SOFA: port 2e3 and port 3e2 [26]

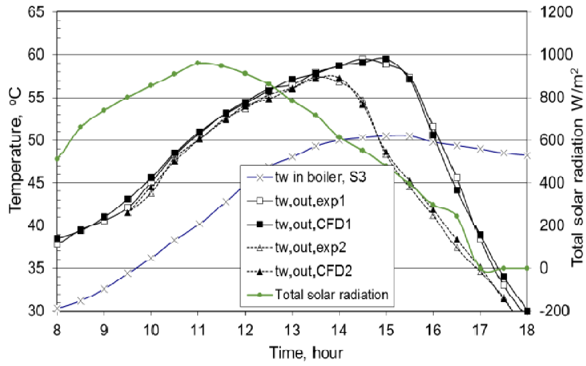


Fig. 15. Solar radiation and working fluid temperature change, experimental and CFD results, 20.03.2010: 1 - moving collector, 2 - static collector [30]

5 Conclusion

The described case studies and the overall experience with CFD modelling technique show that it can be very successfully applied for educational purposes, in order to gain a thorough understanding of the flow physics and the fundamentals of the numerical techniques and models. In the same time, it is a powerful design tool for practical energy systems, as well as a strong research tool for solving complex flow and thermal problems. Properly tuned CFD models, on a basis of the comparisons with available site records and tests, can produce very realistic insight into the different processes and phenomena, thus enabling better understanding, in-depth analysis and optimisation of the design and operating variables.

References

[1] Kandpal, T.C., Garg, H.P., Energy education, Applied Energy 64, (1999), p.71-78

[2] Schnell, U., Numerical modelling of solid fuel combustion processes using advanced CFD-based simulation tools, Prog. in Comp. Fluid Dynamics, 1, (2000), 4, pp. 208-218

[3] Diez L.I., Cortes C., Campo A., Modelling of pulverised coal boilers: review and validation of on-line simulation techniques, Applied Thermal Engineering, 25, (2005), pp. 1516-1533

[4] Filkoski R. V., Joleska B. L., Petrovski I. J., Assessment of the Impact of Under-Fire Air Introduction on the Pulverised Coal Combustion Efficiency, Chem. Eng. Trans., 34, (2013), 25-30

[5] Ustimenko, B.P., Dzhakupov, K.B., Kroly, V.O., Numerical modelling of aerodynamics and combustion in combustiors and technology plants, Science Publ., Alma Ata, 1986 (In Russian)

[6] Fiveland, A.W., Wessel, A.R., Numerical Model for Predicting Performance of 3-D Pulverized-Fuel Fired Furnaces. Jour. of Eng. for Gas Turbines and Power, 110, (1988), 11, pp.117-126

[7] Eaton, A.M., Smoot, L.D., Hill, S.C., Eatough, C.N., Components, Formulations, Solutions, Evaluation, and Application of Comprehensive Combustion Models. Progress in Energy and Combustion Science, Vol. 25, (1999), 4, pp.387–436

[8] Smoot L.D., A Decade of Combustion Research. Progress in Energy and Combustion Science, 23, (1997), 3, pp.203-232

[9] Hill, S.C, Smoot, L.D., A Comprehensive Three-Dimensional Model for Simulation of Combustion Systems: PCGC-3. Energy & Fuels, 7, (1993), 6, pp.874-883

[10] Zhou L.X., Li L., Li R. X., Zhang J., Simulation of 3-D Gas-Particle Flows and Coal Combustion in a Tangentially Fired Furnace Using a Two-Fluid-Trajectory Model, Powder Technology, 125, (2002), 2, pp. 226-233

[11] Yin C., Caillat S., Harion J.L., Baudoin B., Perez E., Investigation of the Flow, Combustion, Heat-Transfer and Emissions From a 609 MW Utility Tangentially Fired Pulverized Coal Boiler. Fuel, 81, (2002), 8, pp.997-1006

[12] He B., Chen M., Yu Q., Liu S., Fan L., Sun S., Xu J., Pan W. P., Numerical Study of the Optimum Counter-Flow Mode of Air Jets in a Large Utility Furnace, Computers & Fluids, 33, (2004), 9, pp.1201-1223

[13] Pallares J. et al., Integration of CFD codes and advanced combustion models for quantitative burnout determination, Fuel (2007), doi:10.1016/j.fuel.2007.01.036

[14] Knaus, H., Schnell, U., Hein, K.R.G., On the modelling of coal combustion in a 550 MWe coal-fired utility boiler, Progress in Comp. Fluid Dynamics, Vol.1, (2001, 4, pp.194-207

[15] Rodi W., Fueyo N. (Ed.), Engineering Turbulence Modelling and Experiments 5, Proc. of the 5th Int. Symp. On Engineering Turbulence Modelling and Measurements, Mallorca, Elsevier, 2002

[16] Wilcox, D.C., Turbulence Modeling for CFD, DCW Ind., Inc., La Canada, California, 1998

[17] Boyd, R.K., Kent, J.H., Three-Dimensional Furnace Computer Modelling. Proc. of the 21st Symp. (Int.) on Combustion, The Combustion Institute, Pittsburgh, 1986, pp. 265-274

[18] Ratzel, III, A. C., Howell, J. R., Two-Dimensional Radiation in Absorbing-Emitting Media Using the P-N Approximation, Journal of Heat Transfer, Trans. of the ASME, 105, (1983), pp.333-340

[19] Bermudez de Castro A., Ferin J.L., Modelling and Numerical Solution of a Pulverized Coal Furnace, Proc. of the 4th Int. Conf. on Techn. and Combustion for Clean Environment, Lisbon, Portugal, 1997, pp.1-9

[20] Fan, J., Qian L., Ma, Y., Sun, P., Cen, K., Computational Modelling of Pulverized Coal Combustion Processes in Tangentially Fired Furnaces. Chem. Eng. Jour., 81, (2001), 1, pp. 261-269

[21] Filkoski, R.V., Modelling of Thermal Processes and Optimisation of Energetic-Environmental Characteristics of Modern Boiler Plants, Ph. D. Thesis, University "Sts Cyril and Methodius", Skopje, R. Macedonia, 2004

[22] Filkoski, R.V., Petrovski, I.J., Karas, P., Optimisation of pulverised coal combustion by means of CFD/CTA modelling, Thermal Science (An Int. Journal), 10, (2006), 3, pp.161-179

[23] Filkoski R. V., Pulverised-Coal Combustion with Staged Air Introduction: CFD Analysis with Different Thermal Radiation Models, The Open Thermodynamics Journal, 4, (2010), 4, 2-12

[24] Magnussen, B.F., Hjertager, B.H., On Mathematical Models of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, 16th Symp. on Combustion, Cambridge, 1976

[25] Fiveland W.A., Three-Dimensional Radiative Heat-Transfer Solutions by the Discrete-Ordinates Method, J. Thermophysics, 2, (1988), 4, pp. 309-316

[26] Siegel R., Howel J. R., Thermal Radiation Heat Transfer, Hemisphere Publ. Corp., Washington D.C., 1992

[27] Guo Z., Kumar S., Three-Dimensional Discrete Ordinates Method in Transient Radiative Transfer, Journal of Thermophysics and Heat Transfer, 16, (2002), 3, pp. 289-296

[28] Dorri-Nowkoorani F., Dougherty R. I., Modified P_N for Correlation Transfer in One-Dimensional Scattering and Absorbing Media, Journal of Thermophysics and Heat Transfer, 16, (2002), 4, pp. 529-536

[29] Stefanova A., Bi X., Lim J.C., Grace J.R.: Heat Transfer from a Vertical Tube in a Turbulent Fluidized Bed, 5th Symp. of SE European Countries TPST-2005, Sunny Beach, Bulgaria, 2005

[30] Chekerovska M., Filkoski R. V., Efficiency of solar-tracking liquid flat-plate solar energy collector, Thermal Science (An International Journal), 19, (2015), 5, pp. 1673-1684