lume 24, No. 13

25 October 2000

International Journal of

Energy Research

Editor **J. T. McMullan**

WILEY

IJERDN 24(13) 1123–1216 (2000)
ISSN 0363-907X

International Journal of

Energy Research

Aims and Scope:

The *International Journal of Energy Research* is dedicated to providing an interdisciplinary platform for the discussion of issues arising in energy research without the constraints imposed by aiming at a restricted audience.

The subject matter of the Journal is concerned with the development and exploitation of both traditional and new fuels and other sources of energy, and interdisciplinary subjects are encouraged. We solicit papers in, but are not limited to, the following areas: solar, nuclear, wavepower and wind energy, heat pumps, building, transport and district heating, energy conservation, synthetic fuels and industrial energy research and storage.

The Editors do not wish to restrict the areas of energy research suitable for publication, but would suggest some guidelines. Submitted material should report on original work, which may be in an area of scientific or technological development, or in the equally important area of feasibility. It is hoped that in all cases emphasis will be placed on some aspects of a particular field or group of energy possibilities, and that discussion of feasibility and possible strategy be aimed at a specific target rather than at a generalized concept of energy policy.

The Journal also carries reviews on important development areas and these may either be submitted in the normal way or invited by the editors. Contributions may be either normal 'full length' research papers or 'short communications'.

International Journal of
ENERGY RESEARCH (*Int. J. Energy Res.*)

CONTENTS

VOLUME 24, ISSUE No. 13 — 25 October 2000

Indexed or abstracted by 'ANTE: Abstracts in New Technologies & Engineering', 'API EnCompass (Ei)', 'Applied Mechanics Reviews', 'Bibliography & Index of Geology', 'Bulletin Signalétique du CNRS', 'Cambridge Scientific Abstracts', 'Chemical Abstracts Service', 'Current Contents®/Engineering, Computing & Technology (ISI)', 'Ei Page One', 'Ei COMPENDEX PLUS', 'Environment Abstracts (CIS)', 'Environmental Periodicals Bibliography', 'Fiz Energie', 'Fluidex (Elsevier)', 'Fuel & Energy Abstracts', 'Geobase (Elsevier)', 'Geographical Abstracts (Elsevier)', 'GeoRef', 'Geotimes', 'Groundwater & Soil Contamination Database', 'ICONDA (Raum & Bau)', 'INSPEC', 'International Development Abstracts (Elsevier)', 'International Petroleum Abstracts', 'ISI Alerting Services', 'London Energy News', 'Science Citation Index Expanded (also known as SciSearch®) (ISI)', 'Thermal Abstracts', 'VINITI (Russian Academy of Sciences)'

IJERDN 24(13) 1123–1216 (2000)
ISSN 0363-907X

INTERNATIONAL JOURNAL OF ENERGY RESEARCH
Int. J. Energy Res. 2000; **24**:1161–1169

Flexibly using results of CFD and simplified heat transfer model for pulverized coal-fired boilers

Minghou Xu[1,2,*], Xiuguang He[2], J. L. T. Azevedo[2] and M. G. Carvalho[2]

[1]*Huazhong University of Science and Technology, Wuhan 430074, People's Republic of China*
[2]*Instituto Superior Técnico, Technical University of Lisbon Av. Rovisco Pais, 1096 Lisboa Codex, Portugal*

SUMMARY

The efficient use of pulverized coal is crucial to the utility industries. The use of computational fluid dynamics (CFD)-based numerical models has an important role in the design of new boiler furnaces or in retrofitting situations. The results of CFD simulations can be used to better understand the complex processes occurring within the boiler furnace. The use of these results to support boiler operation and training of operators requires that the CFD models can be easily accessed and the results are easily analysed.

This paper discusses two ways to simulate the heat transfer process in boiler furnaces. The method directly applying CFD results is employed, in which the grid for solving the energy equation is the same as the flow grid in the CFD simulation while radiation heat transfer is solved in another relatively coarse grid. Comparison of the prediction results between CFD and Heat Transfer code (Simple model) is performed under boiler full load (100%) with one side wall fouling, as well as for different boiler loads (100, 98 and 95 per cent boiler full load, respectively). Finally, the flexible use of the results of CFD and the simple model for pulverized coal-fired boilers is presented. To facilitate the use of the system, a user-friendly interface was developed which enables the user to manipulate new calculations and to view results, namely performing 'what-if' analysis. Copyright © 2000 John Wiley & Sons, Ltd.

KEY WORDS: pulverized coal combustion; heat transfer; CFD

1. INTRODUCTION

The efficient use of pulverised coal is crucial to the utility industry. To achieve a higher combustion efficiency, the major affecting factors such as the particle size distribution, gas and particle temperatures, local heat release, local oxygen concentration, kinetic parameters for coal devolatilization and char oxidation, and coal properties should be understood thoroughly.

In the past two decades, the use of CFD codes for modelling utility boilers is becoming a useful tool to predict the performance of boilers among the scientific and industrial communities

* Correspondence to: Minghou Xu, Instituto Superior Técnico, Technical University of Lisbon, Pavilhao de Maquinas, 2° andar, Av. Rovisco Pais, 1096 Lisboa Codex, Portugal.

Contract/grant sponsor: Commission of the European Community; contract/grant number: BRPR-CT96-0198; Contract/grant sponsor: FCT (Fundação para a Ciência e a Tecnologia) of the Ministry of Science and Technology of Portugal; Contract/grant number: PRAXIS XXI/BPD/16323/98 (No. 439.01).

Received 12 October 1999
Accepted 14 January 2000

(Xu *et al.*, 1998; Azevedo et al., 1994; Carvalho and Coelho, 1991; Boyd and Kent, 1986; Robinson, 1985; Gorner and Zinser 1986; Lockwood *et al.*, 1988 and Fiveland and Wessel, 1988). It helps engineers to optimise the operating conditions, reduce pollutants emission, investigate malfunctions in the equipment, evaluate different corrective measures and also improve the design of new boilers. Submodels for simulating the in-furnace processes such as mixing, radiative heat transfer and chemical kinetics have been developed. The results of CFD simulations can be used to better understand the complex processes occurring within the boiler furnace. The use of these results to support boiler operation and training of operators requires that the CFD models can be easily accessed and the results are easy to be analysed.

Heat transfer in the combustion chamber is one of the most important processes in the design and operation of combustion equipment. The evaluation of this process has received significant attention for many years. Traditionally, the empirical method based on indices and correlation is used in engineering calculations. Nowadays, the Zonal method, which is usually known as Hottel's zonal method (Hottel and Cohew, 1958; Hottel and Sarofin, 1967), discrete ordinates approximation (Khalil, 1982), the Monte-Carlo method (Hammersley and Handscomb, 1964; Howell, 1968), and the discrete transfer method (Lockwood and Shah, 1981; Docherty and Fairweather, 1988), are the most commonly used methods for calculating radiation heat transfer in practical engineering systems (Vishanta and Menguc, 1987).

This paper discusses two ways to simulate the heat transfer process in boiler furnaces. The method directly applying CFD results is employed in which the grid for solving the energy equation is the same as the flow grid in the CFD simulation while radiation heat transfer is solved in another relatively coarse grid. Comparison of the prediction results between CFD and Heat Transfer code (Simple model) is performed under boiler full load (100 per cent) with one side wall fouling as well as for different boiler loads (100, 98 and 95 per cent boiler full load, respectively). Finally, a flexible use of the results of CFD and the Simple model for pulverized coal-fired boilers is presented. To facilitate the use of the system, a user-friendly interface was developed which enables the user to manipulate new calculations and to view results, namely performing what-if analysis.

2. NUMERICAL MODELS

The numerical model is based on an Eulerian description for the continious phase and a stochastic Lagrangean description for the coal particles. The well-known k–ε eddy viscosity/diffusivity model is used to quantify turbulent mixing in the furnace. The representative coal particles are tracked in the combustion chamber using simulated instantaneous gas velocities. The energy balance to the coal particles is used to calculate the particle temperature along time and to describe coal evolution. Volatile release is computed by a first-order reaction rate model. Char combustion is modelled using a parallel process of surface kinetics and oxygen diffusion. The balance of radiative heat transfer is calculated using the discrete transfer method. The radiative properties of gas are computed using the wide band model. A detailed description of the models is presented in (Azevedo and Carvalho, 1993; Coimbra *et al.*, 1994; Yuan *et al.*, 1995 and Xu *et al.*, 1996).

The velocity, the temperature and the mixture fraction are prescribed at the inlet, whereas, the kinetic energy of the turbulence and its dissipation rate are estimated (see e.g. Carvalho, *et al.*, 1987). At the walls, the laws of the wall (Launder and Spalding, 1974) are employed. The platen superheaters are also included in the computation domain. At the exit, a zero gradient normal to the boundary is assumed for the dependent variables. The vertical velocity is then corrected to ensure mass conservation.

The governing equations are discretized over a staggered grid using the finite difference method and integrated over each control volume in the computational domain. Each of the equations is tridiagonal and can be solved using TDMA solvers. The equations are solved by performing iterations until the solution satisfies a preset accuracy and SIMPLER algorism (Patankar, 1980) of pressure correction is applied in the iteration.

3. STRATEGY OF CFD RESULT IMPLEMENTATION

The grid of flow field calculation in the CFD simulation is generally finer than that used for heat transfer calculation. The data to be transferred from CFD to heat transfer are flow field and heat release distribution. These data are used to solve the energy equation in the heat transfer process, and are not directly used in the radiation heat transfer calculation.

In the calculation of heat transfer process, there are two main parts. One is, to solve the energy equation and give the temperature profile in the furnace. Another is, radiation heat transfer calculation, and this is a much more computationally time-consuming process. For this reason, a relatively coarse grid is preferred in the radiation heat transfer calculation. If we use two different grids in heat transfer calculation, then we can construct a fine grid to solve energy equation. Specifically, this fine grid can be the same grid as used in the CFD simulation. This will enable the direct utilization of flow field data into the heat transfer modelling.

Another option is to solve the energy equation and radiation heat transfer using the same coarse grid. In this case, flow field data and heat release pattern generated from the CFD simulation has to be converted to the new grid structure. One reason to use this method is to keep the grid of the 3-D heat transfer simulation independent. Meanwhile, it is expected that solving energy equation and radiation heat transfer process in the same coarse grid will save CPU time. However, converting the flow field from a fine grid to a coarse one will cause problems in mass balance. Thus, the correction of mass balance for the coarse grid should be included.

3.1. Direct application of CFD results in Simple model

Two grid systems are employed in this strategy. The grid for solving the energy equation is the same as the flow grid in the CFD simulation. Radiation heat transfer is solved in another relatively coarse grid. Figure 1 gives a schematic diagram of the direct application of CFD results into 3-D heat transfer calculation.

To solve the energy equation at the same grid as the CFD flow grid has advantages. Firstly, flow field and heat release data from CFD can be used directly. Secondly, mass rebalance at each control volume is not necessary. However, the conversion of radiation source and new temperature profile between fine grid and coarse radiation grid is a disadvantage in this strategy.

3.2. Application of CFD results in simple model after converting to coarse grid

In comparison with the direct implementation of CFD results which uses two grid systems, we can use only one relatively coarse grid system to solve energy equation and simulate radiation heat transfer process. In this way, pre-treatment to convert the CFD data into such a single grid

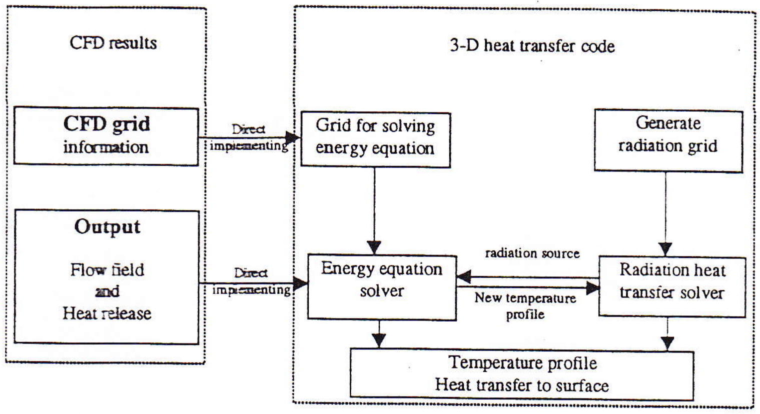


Figure 1. Direct application of CFD results into heat transfer simulation.

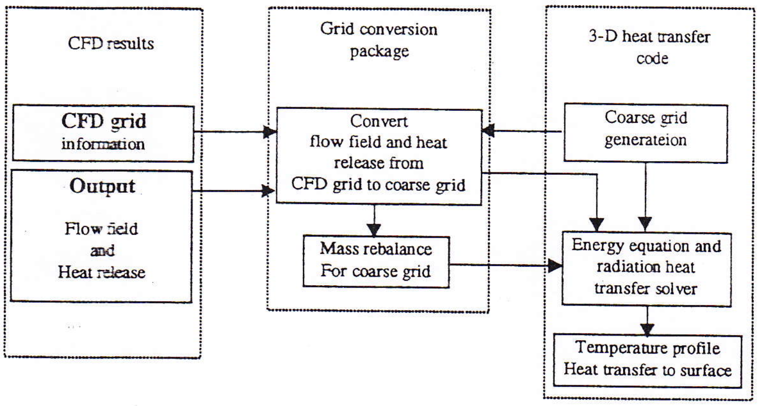


Figure 2. Application of CFD results after converting to coarse grid.

system is the first step. As the flow field conversion from the fine grid to the coarse one causes unsatisfactory mass balance for each control volume in the coarse grid system, additional work is necessary to deal with the mass rebalance and makes this method complicated. Figure 2 shows the strategies for this implementation process.

3.3. *Validation and comparison*

Only validation of direct implementation of CFD results is presented in this study. The CFD results represent the prediction data of the boiler performance under specific operation conditions. The implementation of CFD results into heat transfer calculation is based on the assumption that the flow field and heat release pattern will not change much when the radiation properties of boiler walls change in a certain range.

Comparison of the prediction results between CFD and Heat Transfer code (or Simple model) was performed under 100 per cent boiler load with one side wall fouling, and for different boiler loads (100, 98 and 95 per cent boiler full load, respectively). While CFD model predicts the full performance of the boiler such as heat transfer to walls in a relatively long computation time, the Simple model predicts only the heat transfer to walls and temperature distribution in about 3 min, keeping the flow field and heat release pattern the same as those of CFD model.

Figures 3–5 present the comparison of the CFD predictions and the Simple model predictions for fouling on side boiler wall under 100 per cent boiler full load. The flow field and heat release data used in Simple model were taken from CFD under clean wall (no fouling on walls) conditions.

Figures 6–8 present the comparison of the CFD predictions and the Simple model predictions for different boiler loads. The flow field and heat release data used in the Simple model were taken from CFD simulation under the same boiler load conditions.

From these two kinds of comparison, we can see that the Simple model has the same sensitivity to the change of wall property and boiler load as the CFD code does. However, the total heat flux to walls and furnace exit temperature are under-predicted by the Simple model. The difference of the predictions between these two codes is under 1.2 per cent and hence negligible. This means

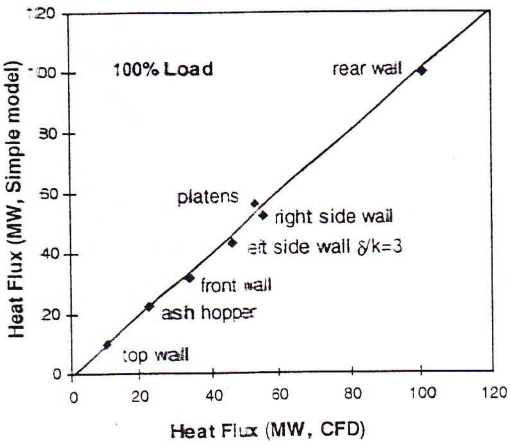


Figure 3. Comparison of heat flux to walls between CFD and Simple model prediction for $\delta/k = 3$ (fouling on side wall).

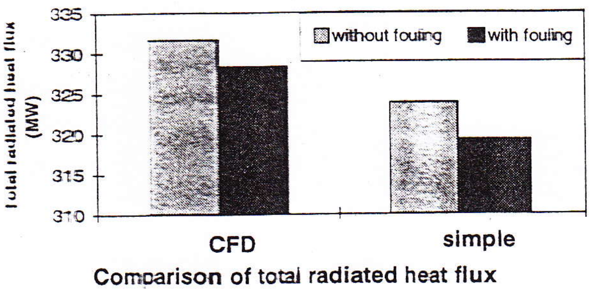


Figure 4. Comparison of total heat flux to walls between CFD and Simple model prediction for $\delta/k = 3$ (fouling on side wall).

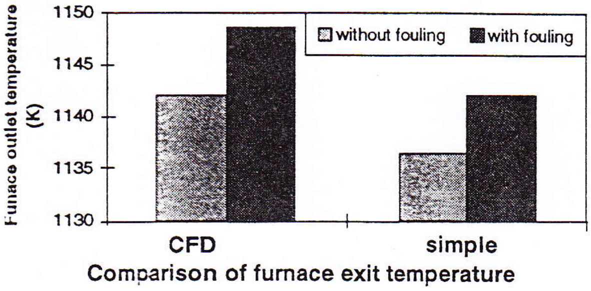


Figure 5. Comparison of furnace exit temperature between CFD and Simple model prediction for $\delta k = 3$ (fouling on side wall).

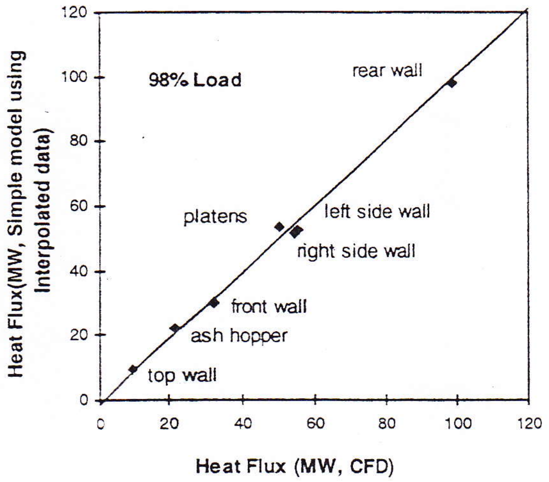


Figure 6. Comparison of heat flux to walls between CFD and Simple model prediction for different boiler loads.

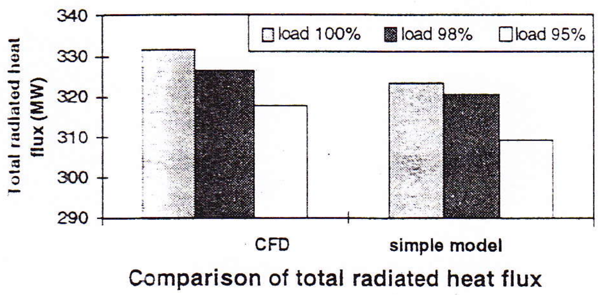


Figure 7. Comparison of total heat flux to walls between CFD and Simple model prediction for different boiler loads.

that, the heat transfer to walls and temperature distribution can be precisely simulated in a very short time in case that the CFD result database is big enough. Hence, the Simple model results can be used in time to adequately guide the boiler operation.

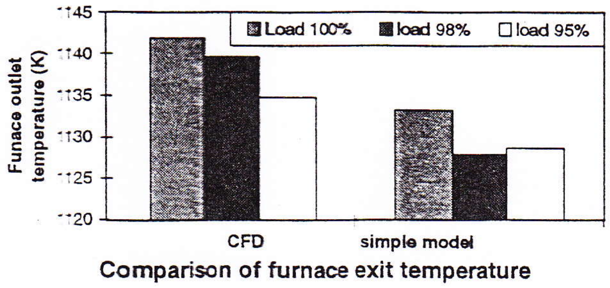


Figure 8. Comparison of furnace exit temperature between CFD and Simple model prediction for different boiler loads.

4. FLEXIBLY USING RESULTS OF CFD AND THE SIMPLE MODEL SIMULATION

In the past two decades, the use of CFD codes for modelling utility boilers has become a useful tool to predict the performance of boilers among the scientific and industrial communities. It is well known that the results of CFD simulations can be used to better understand the complex processes occurring within the boiler furnace. However, the use of these results to support boiler operation and training of operators requires that the CFD models can be easily accessed and the results are easy to be analysed. There are several ways to use the CFD results and these will be discussed below.

4.1. Direct application of CFD results

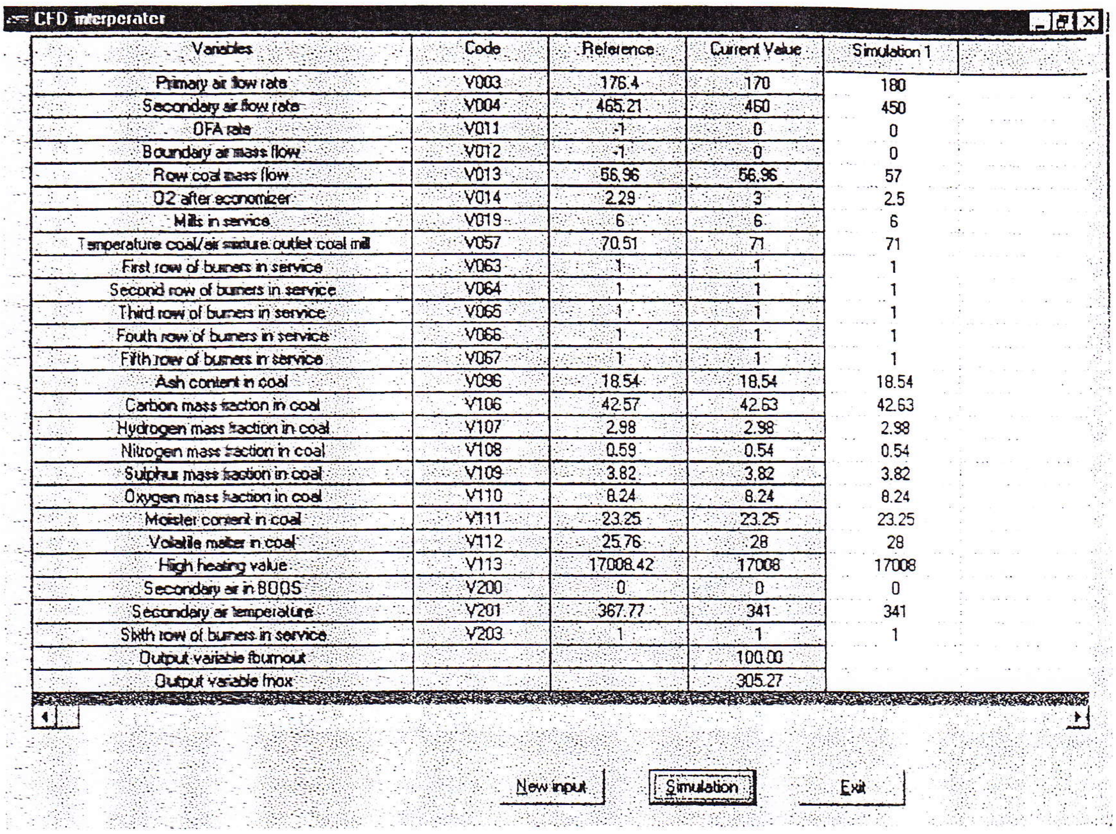
Direct application of CFD results is a useful way to predict the performance of boilers among the scientific and industrial communities. The results of CFD simulations can be used to better understand the complex processes occurring within the boiler furnace. However, such a kind of direct use cannot be easily accessed since the calculation takes a long time, say, several hours or even longer. This means that it can be only used in boiler design but not in supporting boiler operation and training of operators.

4.2. Application of CFD results by interpolation or extrapolation

The application of CFD results by interpolation or extrapolation is a way to use the results of existing CFD cases in the current situation. Comparison of the input data of the current case with those of the nearest existing case or two nearest cases should be conducted. Then, the results of the current case can be the same as the nearest existing case or interpolated/extrapolated by two nearest cases. Such a kind of application of CFD results is efficient in case that the database of the existing cases is large enough so that the current case can be easily matched.

4.3. Flexibly using results of CFD and the Simple model simulation

Obviously, the database of CFD case results cannot be as large as required since the storage of the computer system is limited. This means that the method to apply the CFD results by interpolation/extrapolation is impossible. However, it is quite easy to set only several main parameters such as boiler load, primary and secondary air flows, rows of burners and mills in service, etc., in



Variables	Code	Reference	Current Value	Simulation 1
Primary air flow rate	V003	176.4	170	180
Secondary air flow rate	V004	465.21	460	450
OFA rate	V011	-1	0	0
Boundary air mass flow	V012	-1	0	0
Row coal mass flow	V013	56.96	56.96	57
O2 after economizer	V014	2.29	3	2.5
Mills in service	V019	6	6	6
Temperature coal/air mixture outlet coal mill	V057	70.51	71	71
First row of burners in service	V063	1	1	1
Second row of burners in service	V064	1	1	1
Third row of burners in service	V065	1	1	1
Fouth row of burners in service	V066	1	1	1
Fifth row of burners in service	V067	1	1	1
Ash content in coal	V096	18.54	18.54	18.54
Carbon mass fraction in coal	V106	42.57	42.63	42.63
Hydrogen mass fraction in coal	V107	2.98	2.98	2.98
Nitrogen mass fraction in coal	V108	0.59	0.54	0.54
Sulphur mass fraction in coal	V109	3.82	3.82	3.82
Oxygen mass fraction in coal	V110	8.24	8.24	8.24
Moister content in coal	V111	23.25	23.25	23.25
Volatile matter in coal	V112	25.76	28	28
High heating value	V113	17008.42	17008	17008
Secondary air in BOOS	V200	0	0	0
Secondary air temperature	V201	367.77	341	341
Sixth row of burners in service	V203	1	1	1
Output variable fburnout			100.00	
Output variable fnox			305.27	

Figure 9. Interface to integrate Simple model and CFD case interpolation/extrapolation.

the matching process so that the necessary existing cases in the CFD database will be limited. In this circumstance, the Simple model should be employed to consider other parameters not included in the previous matching process. This integration is exactly an effective way to flexibly use results of CFD and the Simple model simulation. An interface to show this way is illustrated in Figure 9.

5. CONCLUSIONS

Two ways to simulate the heat transfer process in boiler furnaces were discussed in this paper. The better method directly applying CFD results was employed in which the grid for solving the energy equation is the same as the flow grid in the CFD simulation while, radiation heat transfer is solved in another relatively coarse grid. Comparison of the prediction results between CFD and the Simple model performed under boiler full load (100 per cent) with one side wall fouling as well as for different boiler loads (100, 98 and 95 per cent boiler full load, respectively) showed that the Simple model has the same sensitivity to the change of wall property and boiler load as the CFD code does. However, the total heat flux to walls and furnace exit temperature are a little bit under-predicted by the Simple model and hence negligible. A flexible use of results of CFD and the

Simple model for pulverized coal-fired boilers was also presented. To facilitate the use of the system, a user-friendly interface was developed which enables the user to manipulate new calculations and to view results.

ACKNOWLEDGEMENTS

This work was sponsored by the Commission of the European Community through subprogram ACORDE of the BRITE/EURAM Program under Contract No. BRPR-CT96-0198. The authors are indebted to EDP (Portugal) for the contribution to the experimental data. Partial support from PRAXIS XXI/BPD/16323/98(No. 439.01) of FCT (Fundação para a Ciência e a Tecnologia) of the Ministry of Science and Technology of Portugal to Dr Minghou Xu is gratefully appreciated.

REFERENCES

Azevedo JLT, Carvalho MG. 1993. *Second International Conference on Combustion Technologies for a Clean Environment*, Lisbon, Portugal.

Azevedo JLT, Coelho LMR, Carvalho MG. 1994. *Combustion Related Organizations-Common and Unified Symposium*, Salsomaggiore Terme, Italy: 2–17.

Boyd RK, Kent JH. 1986. *Twenty-first Symposium (International) on Combustion*. The Combustion Institute, Pittsburgh 265–274.

Carvalho MG, Coelho P. 1991. *Engineering in Computers* **7**:227–324.

Carvalho MG, Durão DFG, Pereira JCF. 1987. *Engineering in Computers—International Journal of Computer Aided Engineering Software* **4**:23–34.

Coimbra CFM, Azevedo JLT, Carvalho MG. 1994. *Fuel* **73**:1128–1134.

Docherty P, Fairweather M. 1988. *Combustion and Flame* **71**:79–87.

Fiveland WA, Wessel RA. 1988. ASME *Journal of Engineering for Gas Turbines and Power* **110**:117–126.

Görner K, Zinser W. 1986. *ASME 107th Annual Meeting*, Anaheim, California, U.S.A.

Hammersley JM, Handscomb DC. 1964. *Monte Carlo Methods*. Wiley: New York.

Hottel HC, Cohen ES. 1958. *A. I. Ch. E. Journal* **4**(3).

Hottel HC, Sarofim AF. 1967. *Radiative Transfer*. McGraw Hill: New York.

Howell JR. 1968. *Advances in Heat Transfer* **5**.

Khalil EE. 1982. *Modelling Furnaces and Combustors*, Abacus Press: Tunbridge Wells, U.K.

Launder BE, Spalding DB. 1974. *Computational Methods in Applied Mechanical Engineering* **3**:269–289.

Lockwood FC, Shah NG. 1981. *Eighteenth Symposium (International) on Combustion*. The Combustion Institute, Pittsburgh, PA, 1405–1414.

Lockwood FC, Papadopoulos C, Abbas AS. 1988. *Combustion Science and Technology* **58**:5–24.

Patankar SV. 1980. *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation: New York.

Robinson GF. 1985. *Journal of the Institute of Energy* 116–150.

Viskanta R, Menguç MP. 1987. Radiation Heat Transfer in Combustion System. *Progress in Energy and Combustion Science* **13**:97–160.

Xu MH, Yuan JW, Ding SF, Cao HD. 1996. *Proceedings of the Chinese Society of Electrical Engineering* **16**:266–270.

Xu M, Yuan J, Ding S, Cao H. 1998. *Computational Methods in Applied Mechanical Engineering* **155**:369–380.

Yuan JW, Xu MH, Ding SF, Cao HD. 1995. *Proceedings of the Third International Symposium on Combustion*, Beijing, China, 234–241.

