

Low-NO_x burner design evaluation by CFD

Marek Šarlej, Pavel Petr, and Petr Stehlik

*Brno University of Technology, Faculty of Mechanical Engineering, Institute of Process and Environmental Engineering, Technická 2, 616 69 Brno, Czech Republic,
sarlej@upej.fme.vutbr.cz*

Abstract

This contribution focuses on application of computational fluid dynamics (CFD) in experimental low-NO_x burner design. A set of 27 geometric alternatives were introduced in order to identify an optimum geometry of secondary fuel nozzles of an experimental burner in terms of minimum NO_x production. The most promising alternative was found as a result of the computation process. A detailed analysis of flow, a flame shape and a heat flux through a shell of an experimental chamber, in case of the most promising alternative compared to the less promising one, is subject of this article.

Keywords

CFD, burners, NO_x

1. Introduction

Design of new burners is mostly based on long-time experience. Moreover, burners are tested at testing facilities before putting into operation. However, this approach is limited by economic constraints (costly prototype manufacture and individual tests) as well as technical ones (parameters of the testing facility). Thus, new alternatives of partial and/or complete substitution of physical burner testing are being investigated with support of up-to-date computational tools.

State of the art computational methods for fluid flow prediction including chemical reactions and heat transfer make complex burner simulations possible. On the other hand, these computational methods are not fully reliable. Therefore, good knowledge of their strengths and weaknesses is required [1]. Beside that, prediction accuracy has to be validated as much as possible by experiments.

2. State of the art

This article reports progress of computational modelling of an up-to-date experimental burner installed in a testing facility. The burner has been designed as a generic two stage low- NO_x burner. Due to its sophisticated construction, geometry of fuel nozzles can be easily modified. Thanks to this feature, the burner geometry may be optimised with respect to amount of NO_x emissions. From another viewpoint, it represents an ideal instrument, which can be used for validation of computational models in wide range of many different operation and construction parameters.

3. Computational modeling

The computational model of the experimental burner and the combustion chamber was set up in commercial CFD software FLUENT version 6.2.16 [2] using its built-in sub models and algorithms. The computational mesh was created in GAMBIT 2.2.30 [3], as well as the geometrical model (see Fig 1).

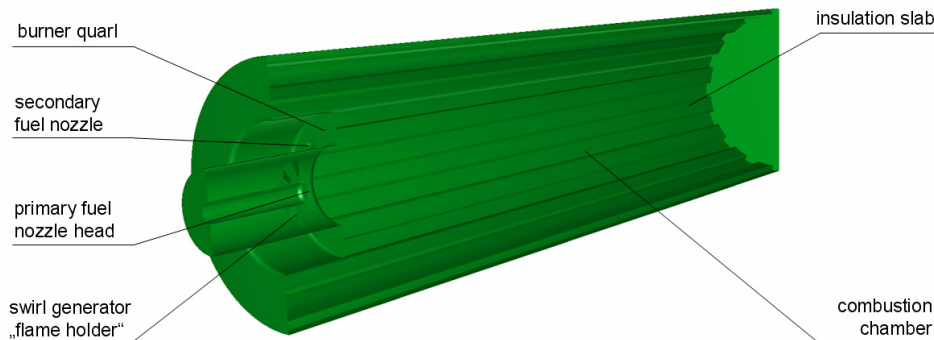


Fig 1- Geometrical model (180° cut for better view)

The entire mesh consists of about 110 000 cells. The mesh surrounding the primary/secondary nozzles and flame area is finer in order to help resolve the expected large gradients of the independent variables. On the other hand, remainder of the combustion chamber was meshed with coarser mesh cells as no significant gradients were expected here.

In order to identify optimum arrangement of the secondary fuel nozzles in terms of minimum NO_x production, a set of 27 (9×3) alternative arrangements of the secondary nozzle heads were introduced. Numbered spots in Fig 22a represent nine (9) alternative locations of one nozzle head in axial and radial direction. Moreover, three alternative slewings 0°, 45° and 90° (tangential directions) of the nozzle heads (see Fig 22b) were introduced to each of the above mentioned alternatives.

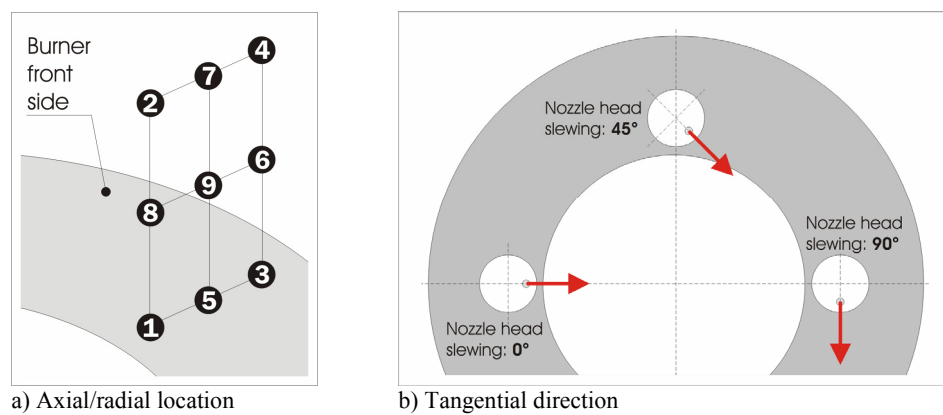


Fig 2 - Secondary nozzles arrangement

As required by the problem physics, the following equations were introduced and solved using second order upwind discretization scheme:

- continuity equation,
- three momentum transport equations,
- energy transport equation,
- five species transport equations,
- two transport equations of the turbulence model.

Turbulent flow was modelled by two-equation SST $k-\omega$ model. This model is advantageous for due to its stability during computation and in comparison with RSM and $k-\epsilon$ models, it offers comparable accuracy of NO_x formation prediction in modelling similar swirling diffusion gas flames [4].

Chemical reactions were modelled by the simple two-step model for methane combustion.

Rates of chemical reactions were calculated using well-known eddy-dissipation model of Magnussen and Hjertager [5], which relates the rate of reactions to the local turbulence properties (representing intensity of turbulent mixing).

Radiative heat transfer was also considered. This was modelled by the discrete ordinates model [6], which provides accurate predictions at moderate CPU costs and is adequate for representing the following typical features of the present diffusion flames:

- participating fluid (flue gas in our case),
- low or fluctuating optical thickness of the flue gas
- localized heat source.

Boundary conditions were set up as follows:

- mass flow inlet for primary/secondary fuel
- mass flow inlet for primary/secondary combustion
- pressure outlet at the flue gas outlet
- constant temperature wall for the cooled outer surface of the furnace shell.

The slabs of internal furnace insulation have been modelled as heat conducting solid.

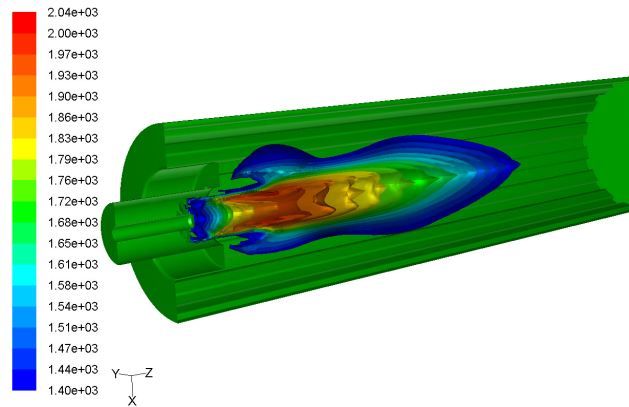
Several alternatives yielded a stable and fast convergence, however, computation process of most of the alternatives had to be interrupted after about 10 000 iterations, when an acceptable level of convergence had been achieved. After that, based on the obtained results, computation of NO_x production took place. Thanks to the fact that the fuel does not contain any chemically bounded nitrogen, all of the NO_x produced is accounted for by the two following formation mechanisms:

- thermal (Zeldovich) mechanism and
- prompt mechanism.

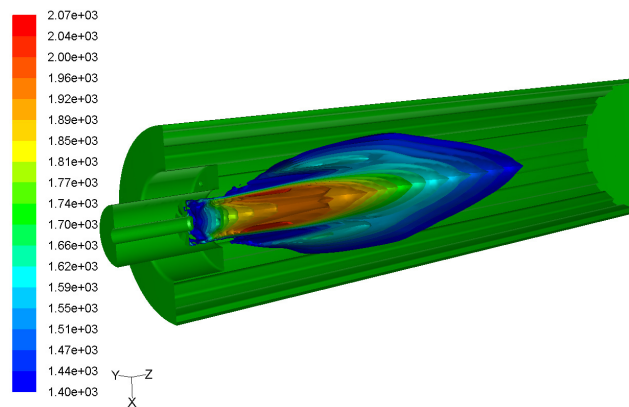
Due to the turbulent nature of the flames, influence of turbulent temperature fluctuations on formation of NO_x also needed to be included. Concentrations of O/OH radicals were calculated using assumption of partial chemical equilibrium.

Analysis of results

NOx outlet volume fraction and flue gas outlet temperature were the primary monitored parameters. The NO_x emissions differ only slightly in dependence on the axial secondary nozzle head position. On the other hand, NO_x production differs significantly in dependence on the radial position as well as on the nozzle head slewing. Finally, it was found out that Alternative 1 (slewing 45°) represents the overall minimum NO_x concentration of 12 ppm. This alternative is compared to alternative Alternative 4 (slewing 90°) that represents the highest NO_x concentration of 33 ppm which is almost three times higher. Furthermore, it was observed that the average concentration of NO_x was highest in case of 90° slewing.



Alternative 1 (slewing 45°)



Alternative 4 (slewing 90°)

Fig 3 - Flame shape

It is obvious from Fig 3 that the shapes of the flames differ significantly. This is due to different slewing of the secondary fuel nozzles. The maximum temperature is 2040 K for Alternative 1 and 2074 K for Alternative 4. Comparing the total heat transfer rate through the combustion chamber shell we get very similar numbers – 632 kW for Alternative 1 and 624 kW for Alternative 4. Distribution of the total heat flux through the combustion chamber shell is also almost identical.

4. Conclusions

The article demonstrates one of the possibilities of using CFD in terms of finding optimum arrangement of secondary fuel nozzles of an experimental burner. A number of alternatives of secondary nozzle arrangement were investigated in respect to NO_x production. Finally, an alternative corresponding to minimum NO_x production was identified and compared to an alternative with highest concentration of NO_x. However, this represents only a first step to a complex optimisation of the experimental burner. The optimisation will investigate influence of various operational parameters such as various values of air equivalence ratio. Moreover, computed results obtained from CFD calculations will be validated by experimental measurements.

Acknowledgements

The article was created thanks to the research project of MSMT in the Czech Republic No.: MSM 0021630502 „Waste and Biomass Utilization focused on Environment Protection and Energy Generation”

References

1. J.D. Jr. Anderson, Computational Fluid Dynamics; The Basics with Applications, McGraw-Hill, Inc., New York, 1995
2. Fluent Inc., FLUENT 6.1 User's Guide, Fluent Inc., Lebanon, 2003
3. Fluent Inc., GAMBIT User's Guide, Fluent Inc., Lebanon, 2002
4. D. Veynante, L. Vervisch, Turbulent combustion modeling, *Progress in Energy and Combustion Science* 28, pp. 193 – 266, 2002
5. B.F. Magnussen, B.H. Hjertager, On mathematical models of turbulent combustion with special emphasis on soot formation and combustion, 16th Symposium (International) on Combustion, The Combustion Institute, 1985
6. J.Y. Murthy, and S.R. Mathur, 'A Finite Volume Method For Radiative Heat Transfer Using Unstructured Meshes', AIAA-98-0860, 1998