Available online at www.sciencedirect.com

ScienceDirect

ENERGY

Energy 33 (2008) 930–941

www.elsevier.com/locate/energy

Computational fluid dynamic simulations in thermal waste treatment technology—Design, optimisation and troubleshooting

Jiří Hájek*

Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology–UPEI-VUT, Technická 2, 616 69 Brno, Czech Republic

Received 18 December 2006

Abstract

The present work provides three case studies featuring applications of computational fluid dynamics (CFD) in the area of thermal waste treatment (TWT). The purpose of the paper is to demonstrate the benefits offered by simulations in TWT technology in the design, optimisation and troubleshooting of those systems. The case studies deal with practical problems, namely a performance evaluation of a fabric filter bag with an add-on Venturi nozzle, the design optimisation of flow homogenising vanes in a heat exchanger, and finally troubleshooting in a novel volatile organic compound (VOC) treatment unit.

Each case study includes an outline of the modelling approach and summary of the most important results. The flow analyses are performed using standard methods implemented in the commercial software code FLUENT. The case study on design optimisation combines decision support in the selection of the best conceptual solution and an automatic shape optimisation using the commercial code SCULPTOR. The results clearly show the usefulness and applicability of the CFD computations in the area of thermal waste treatment.

Keywords: CFD; Waste; Thermal treatment; Incineration; Design optimisation; Troubleshooting

1. Introduction

Computational fluid dynamics (CFD) has evolved in the past decades into a formidable field, reaching from numerical methods through the turbulence theory to chemical kinetics and drawing upon many other areas such as multiphase flows or radiative heat transfer. Therefore it is not surprising that CFD finds applications in a field so varied as thermal waste treatment. However, the frequency of applying CFD to simulate parts of thermal waste treatment (TWT) technologies is small compared to other processes employing combustion as glass or metal production, furnaces in chemical and petrochemical industry, etc. Whilst there are a number of reasons for this, it is the author's belief that CFD has significant potential for providing new insights in the design, optimisation and troubleshooting of TWT units. The present work aims to provide real-life case studies that support this statement.

Advanced mathematical modelling provides both physical information about the properties of the flow, important for further optimisation of the design or operating conditions, as well as attractive visualisations that may appeal to customers. In spite of this, CFD analyses of complete TWT units (e.g. municipal solid waste incinerators and rotary kilns) are new emerging. Some examples can be found in Refs. [1–5], but those typically require rather extensive preliminary experimental studies. This need is generally due to the principal characteristic feature of many types of waste—its non-homogeneity and extremely variable properties. Municipal solid waste incinerators are one of the TWT technologies that are most difficult to model with CFD. Therefore, most works published to date use severely simplified boundary conditions on the grate, see e.g. [6,7]. Modelling biomass grate

*Tel.: +420 608 431 413; fax: +420 541 142 177.
E-mail address: hajek@fme.vutbr.cz

0360-5442/$ - see front matter © 2007 Elsevier Ltd. All rights reserved.
doi:10.1016/j.energy.2007.11.010

boilers that are physically similar has been much more successful, as witnessed by detailed predictions including bed processes (e.g. in Refs. [8,9]).

Nevertheless, existing difficulties do not prevent the application of CFD to less-complex analyses concerning individual pieces of equipment or various parts of waste treatment lines. This approach typically circumvents solving intangibly complex tasks such as poorly defined fuel burning or bed combustion processes and concentrates on separate problems, unaffected by such unknown features. The applications of CFD methodology to physically better define problems in the area of TWT are much more frequent than the complex simulations mentioned in the previous paragraph. The spectrum of problems includes, but is not limited to, heat exchangers [10], cyclones [11], electrostatic precipitators [12], filters [13], scrubbers [14] and catalytic reactors [15].

In a similar vein, the present work provides sample case studies based on CFD computations in the TWT area, performed recently by the author and his group, that are believed to be instructive. Three basic motivations for a CFD application are highlighted, namely design, optimisation, and troubleshooting. Typical situations are described, where CFD has provided important information, insight or improvement.

The software system FLUENT (v. 5 and 6) was used in this work unstructured hybrid meshes [16]. FLUENT contains a large array of sub-models for turbulence modelling, reactive flows, radiative heat transfer, etc. It may thus be considered an adequate tool for CFD simulations in the field of thermal waste treatment.

1.1. Accuracy of CFD simulations

It should be stressed that prediction accuracy is a major concern in industrial applications of CFD. Unlike structural analysis, numerical prediction of engineering flows often poses very complex problems, which cannot be solved to a predefined accuracy-level turbulence coupled with chemical reactions, is one of those. Therefore, the goals of CFD analysis have to be chosen cautiously—quite often it is possible to expect only qualitatively correct predictions.

A practically relevant example from the area of reactive flows, where quantitative reliability is not yet available, is computation of NO_x emissions (for a recent review see [17]). Burner and furnace design are two frequently encountered applications of CFD in many industries and they are very important in TWT units as well. In waste incineration plants, low-NO_x gas burners are often used in secondary combustion chambers, where they provide direct heating to ensure sufficient residence time of flue gases under temperatures required by legislation.

A recently launched research activity, documented in [18], aims at the validation of FLUENT software for burner design and optimisation. Simulations are used to analyse an experimental burner during its development, with the focus on prediction of NO_x emission levels produced by various geometric modifications of the burner and fuel nozzles (e.g. their axial and radial position as well as tangential orientation). The numerical exercise will be evaluated using measurements in a new large-scale experimental furnace [19] (presently under construction). Based on previous results [20] it is expected that trends will be well predicted, which would provide a much-needed validation example for the applicability of CFD in this specific area. It should be noted, however, that various researchers found the prediction of diffusion flames stabilised by swirl very challenging (swirl stabilisation is typical for low-NO_x burners), see e.g. [21–23]. This implies that caution is required also in simulations of e.g. secondary combustion chambers, when equipped with such burners.

2. CFD in TWT technology design

The application of CFD in the product design phase is a trend, which has its origin in its cost effectiveness. By being able to analyse performance of a process/product before manufacturing a prototype or even avoiding the necessity of physical prototypes, it is possible to shorten the development phase and often to improve the quality/performance ratio. This is a general concept called virtual prototyping, which is applicable to any product, including TWT units.

The following example of applying CFD in the design phase of a TWT technology is an analysis of an add-on element used in fabric filters, which was hypothesised to improve their cleaning efficiency and decrease operating costs.

2.1. Analysis of add-on venturi nozzle in fabric filter bag

Fabric filters are commonly found in many kinds of plants, and are used for collecting of fine particulate matter contained in the off-gas produced by the plant. The filter bags need to be cleaned periodically and this is done typically by a short (less than 100 ms) pulse of compressed air being applied at the throat of the filter bag, i.e. in a counter-current direction to the flow of off-gas during normal operation. (Of course, cleaning is mostly performed in temporarily closed sections of the baghouse.)

To improve the performance of the cleaning pulses, some baghouse manufacturers equip filter bags with Venturi nozzles, located in the throats of the bags. These inserts are deemed to improve the cleaning, and their negative influence in terms of increased pressure drop during normal filter operation is considered insignificant. This hypothesis has been recently examined by means of CFD modelling [24] and the results are briefly summarised in this section. The concrete application is a municipal waste incinerator, so that the filtered medium is hot flue gas (210 °C).

Cylindrical filter bags (diameter 15 cm, length 5.6 m) are installed vertically and are arranged into a matrix structure