Journal of Cleaner Production 111 (2016) 117–124

Contents lists available at ScienceDirect

Journal of Cleaner Production

journal homepage: www.elsevier.com/locate/jclepro

The influence of the fabric filter layout of in a flow mass filtrate

Thomas Wilis Cândido Pereira [a], Felipe Bezerra Marques [a], Fábio de Assis Ressel Pereira [a], Daniel da Cunha Ribeiro [a], Sandra Mara Santana Rocha [b, *]

[a] DETEC–Departamento de Engenharias e Tecnologia, Universidade Federal do Espirito Santo, Rodovia BR 101 Norte, Km 60, CEP: 29932-540 São Mateus, ES, Brazil

[b] DECH–Departamento de Educação e Ciências Humanas, Universidade Federal do Espirito Santo, Rodovia BR 101 Norte, Km 60, CEP: 29932-540 São Mateus, ES, Brazil

ARTICLE INFO

Article history:
Received 21 December 2014
Received in revised form
14 September 2015
Accepted 17 September 2015
Available online 8 October 2015

Keywords:
Computational fluid dynamic
Design
Fabric filter
Gas filtration

ABSTRACT

Nowadays, more and more companies are forced to control particulate emissions to the atmosphere, due to economic reasons, legal aspects or simply the change in policy that favors ecological commitment. This new trend induces the necessity for better particle capture equipment and has made the environmental control equipment market increase by 20% in the last few years (Rocha et al., 2012). The dry capture of fine particles is carried out by filtration, and the fabric filter has become a very popular equipment because of its high efficiency and low operational cost. In the other hand, this efficiency is dependent upon proper design and layout, and the particle–filter interactions greatly influence the filtration results and the overall efficiency.

In this context, this article presents the computational fluid dynamic (CFD) simulation of the airflow inside an industrial fabric filter with a set of 49 bags and a total surface area of 75 m^2. The simulations were performed using the ANSYS FLUENT v13.0. This package applies the Finite Volumes Method, using the URANS approach, as well as the realizable k–ε model for the turbulence and the SIMPLE algorithm for the pressure–velocity coupling (Miltner et al., 2015). Four inlet positions were analyzed in order to determine how the internal flow profile and the equipment filtering performance depends on the inlet position. The simulations allowed concluding that the simple and double inverted feed positions led to a more satisfactory flow, considering not only the overall efficiency but also the bag's useful lifespan.

1. Introduction

The increase in the global population has caused a rise in the exploration of natural resources and in the global pollution levels. This called the attention of environmentalists, government policy makers and academic institutions that have pressured industries to invest in better pollution control equipment. Because of this, cleaner production programs have been numerously implemented around the world, such as in the mining, electronics, base metal industries among others.

According to Yilmaz et al. (2014), in order to choose the right equipment for pollution control, the advantages and the environmental impacts must be analyzed, along with operational costs. Kubota and Rosa (2013) points out that environmental concern taken into account during the development phase result in better environmental performance, cost reduction and process optimization.

As far as the control of particulate material is concerned, the fabric filter has been consistently shown to be the most efficient and to have the best cost-benefit ratio for dry collection. If designed correctly it can also have the less impact in the environment, since the particles can be fed back into production as raw material (in most cases). A drawback is that this high efficiency is very dependent on the choice of filter medium, the equipment geometry and the process variables. In this sense, it becomes clear that the fabric filter should be specifically designed for each application, as opposed to a generic design that tries to accommodate most applications.

Unfortunately due to the high costs and complexity in the project of industrial equipment, the design of the fabric filter is mostly based on empirical prototypes, which can cause under or

* Corresponding author. Tel.: +55 2733121595.
E-mail addresses: twcp0104@hotmail.com (T.W.C. Pereira), felipebmarques2006@hotmail.com (F.B. Marques), f.ressel@gmail.com (F.A.R. Pereira), daniel.ribeiro@ufes.br (D.C. Ribeiro), rochasms@gmail.com, sandra.m.rocha@ufes.br (S.M.S. Rocha).

http://dx.doi.org/10.1016/j.jclepro.2015.09.070
0959-6526/

Nomenclature

ρ	density (kg m^{-3})
$\overrightarrow{v}$	flow velocity vector (m s^{-1})
$\overline{\tau}$	stress tensor (kg m^{-1} s^{-2})
μ_{eff}	effective viscosity (kg m^{-1} s^{-1})
$\overrightarrow{v^T}$	terminal velocity tensor (m s^{-1})
k	turbulent kinetic energy (kg m^2 s^{-2})
μ_t	eddy viscosity (kg m^{-1} s^{-1})
σ_k	turbulence model constant
G_k	turbulence model constant
ε	turbulent dissipation rate (m^2 s^{-3})
S, S_{ij}	mean strain rate (s^{-1})
ν	turbulence model constant
C_μ	rotation effect constant
Ω_{ij}	mean rate of rotation tensor
ω	angular velocity (radians s^{-1})

oversizing problems. The advent of numerical simulations of fluid flow in complex geometries makes it possible to predict the performance of this equipment with a high degree of reliability, and reduced time and cost. The technology that delivers such prediction is called computational fluid dynamics (CFD).

The application of CFD techniques provides results that allow the analysis of the flow behavior inside a filtration system, taking into account important phenomena as turbulence and correlating them to pressure drop in the filter media. Industrial use of these techniques is relatively new (Damian, 2003). Despite recent arrival, the CFD techniques have proven to be effective in not only designing projects for new equipment, but also in improving existing plants and optimizing the operation of traditional equipment (Rocha et al., 2014).

In this context, this work presents the results of the CFD simulation of the airflow inside an industrial fabric filter. The commercial software utilized was ANSYS FLUENT v13.0. Four different inlet configurations were tested for the same fabric filter. Based on CFD results for mass flow and streamlines, it was possible to judge which configuration would have greater potential for optimal performance.

Errors tied with the numerical simulation of this flow and any CFD simulation are of 3 types. They can be iteration/round-off errors, discretization errors, and modeling errors (Fernández-García et al., 2015). In order to guard against the first of these, the simulations were continued until no significant changes in the residuals could be observed, below the standard value of 10^{-5} (10^{-3} for the continuity residuals). The mesh geometry was chosen in order to be as fast as possible while still being refined enough to guard against discretization errors. Modeling errors are hard to ascertain, but, judging from previous studies, we feel confident that they are minimized in this simulation and that the assumptions are sound.

2. Material and methods

The numerical approach used in this study is based on three main steps: the geometry construction and its discretization, the numerical procedure to solve the governing equations starting from the prescribed boundary conditions, and finally the post-processing the results. Experimental studies in this area are scarce, but according to the results of Damian (2003) these simulations yield satisfactory accuracy and provide a useful real world model.

In regards to the geometry of the industrial fabric filter, the bags followed a 7 × 7 arrangement (49 bags), providing a filtration area of 75 m^2. Each bag has a dimension of 0.3 m diameter and 3.0 m height. The fabric filter main dimensions are: height 5.3 m, length 1.8 m and 2.0 m depth. Four inlet sections were tested independently (Fig. 1). The inlets consist of rectangles with dimensions 36 cm wide and 33 cm high, with the exception of the lateral inlet which has dimensions 6 cm wide and 2.096 m high. The four outlets at the top of the equipment each have dimensions of 50 cm wide and 15 cm high.

The four-inlet setup used was selected in order to represent a wide array of possibilities. The Simple feed (Case A) has the flow entering the equipment from the bottom, and meeting the bags in an ascending flow. The Conventional feed (case B) enters the filter through the middle, impacts the bags head-on, and disperses throughout the rest of the equipment. The Double Inverted feed (case C) is similar to the simple feed, but enters from two different positions and meets the bags in an ascending fashion. The Lateral feed (case D) comes tangentially from the side, and has the airflow going around the filter and through the bags.

• Simple feed	- Case A
• Conventional feed	- Case B
• Double Inverted feed	- Case C
• Lateral feed	- Case D

The domain discretization is based on an unstructured approach. Aiming to determine the optimal mesh size, a group of preliminary tests was conducted to reach mesh independence. The variable tested was the average pressure. The range investigated started with 379,693 and reached 1,614,466 elements. The actual mesh was optimized with 527,609 elements.

As for the material and boundary conditions, the physical air properties considered a density of 1.225 kg/m^3 and a dynamic viscosity of 1.7894e^{-5} Pa s; the feeds used in the air intake with 22.84 m/s velocity. The porous fabric had 0.0025 m of thickness and a permeability of 5.87 × 10^{-10} m^2.

3. Theory

The governing equations are solved using the Unsteady Reynolds Averaged (URANS) approach in which Navier–Stokes and Continuity equations are averaged in a time interval together with the transient term, as shown in Equations (1) and (2).

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\overrightarrow{v}) = 0 \quad (1)$$

$$\frac{\partial(\rho\overrightarrow{v})}{\partial t} + \nabla\cdot(\rho\overrightarrow{v}\overrightarrow{v}) = -\nabla p + \nabla\cdot\overline{\tau} \quad (2)$$

Where: $\overline{\tau} = \mu_{eff}[(\nabla\overrightarrow{v} + \nabla\overrightarrow{v^T})]$ is the stress tensor for the incompressible flow (kg m^{-1} s^{-2}), $\overrightarrow{v}$ is the flow velocity (m s^{-1}), p is the static pressure (kg m^{-1} s^{-2}), and ρ is the gas density (kg m^{-3}). The effective viscosity is the sum of dynamic viscosity (fluid property) and eddy viscosity (flow property, calculated by the turbulence model) (Long and Yao, 2012).

A transient approach was necessary to achieve convergence, since the convergence with a steady state approach was unstable and the flow is inherently transient. This behavior is expected because the inlet is an impingent jet. Considerations made are single fluid flow (air), no gravity effects, and an isothermal hypothesis. The turbulent field is modeled using the realizable k–ε