Engineering Science and Technology, an International Journal xxx (2018) xxx–xxx

HOSTED BY

ELSEVIER

Contents lists available at ScienceDirect

Engineering Science and Technology, an International Journal

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

Full Length Article

Modelling of hot air chamber designs of a continuous flow grain dryer

Lotte Strange Kjær, Mathias Poulsen *, Kim Sørensen, Thomas Condra

Institute of Energy Technology, Aalborg University, Pontoppidanstræde 101, DK-9220 Aalborg, Denmark

ARTICLE INFO

Article history:
Received 9 June 2017
Revised 6 December 2017
Accepted 4 February 2018
Available online xxxx

Keywords:
CFD
Fluid flow
Temperature distribution
Uniformity index
Continuous flow dryer
Grid convergence index

ABSTRACT

The pressure loss, flow distribution and temperature distribution of a number of designs of the hot air chamber in a continuous flow grain dryer, were investigated using CFD. The flow in the dryer was considered as steady state, compressible and turbulent. It is essential that the grain is uniformly dried as uneven drying can result in damage to the end-product during storage. The original commercial design was modified with new guide vanes at the inlets to reduce the pressure loss and to ensure a uniform flow to the line burner in the hot air chamber. The new guide vane design resulted in a 10% reduction in pressure loss and a γ-value of 0.804. Various design changes of the hot air chamber were analysed in terms of pressure loss and temperature distribution with the aim of a temperature variation of ±5 K at the outlet ducts. An obstruction design was analysed, which improved mixing and gave a temperature distribution within the limits. However, the pressure loss was six times larger than the original design with new guide vanes. Finally, The static mixer design resulted in a 23% reduction in the pressure loss, but had a mean absolute deviation of 22.9 K.

1. Introduction

Normally grain is harvested with a moisture content of 18–25% [1]. Therefore, post harvest treatment, in the form of drying, is needed in order to reduce the moisture content to 14% as this ensures safe storage of the grain. Typically the amount of energy used to remove water by evaporation from grain during drying is in excess 4.5 MJ/kg, whereas the amount of energy required to evaporate free water is only 2.3 MJ/kg and this makes the process of drying grain an energy intensive process [2]. Therefore, it is important to ensure that the drying process is as efficient as possible, while grain damage from drying is minimised.

In the current generation of continuous flow dryers, it has been observed that improper mixing of the air flows in the hot air chamber occurs. This causes significant variations in the temperature and flow distributions when the air enters the grain drying column. In Giner et.al. [3] a complex model of mixed-flow grain drying was developed where the two-dimensional flow patterns of the air and grain around the inlet and outlet air ducts in the drying column have been taken into account. As a result, it was found that heating and drying of the grain were more intense in the close proximity of the air ducts leading into the grain column. Thus, it is important to obtain a uniform temperature distribution to avoid over-drying of the grain near the air ducts. A design of the hot air chamber which can ensure a uniform temperature distribution into the grain column is therefore desirable.

There have been several studies on the air flow and temperature distribution in smaller dryer systems such as batch and cabinet dryers [4,5]. However, the studies are either performed on small or on laboratory scale dryers and in the literature the hot air chamber in a full scale continuous grain dryer is seldom investigated.

The purpose of this study is to identify a conceptual design that can be the basis for of further design improvements. These should reduce the variation in the drying air temperature at the outlet ducts. However, it is not desirable to increase the operation cost and so the pressure loss in the new design must be kept as low as possible.

2. Initial dryer design

A new conceptual design for a continuous flow grain dryer has been developed, and the hot air chamber will be the starting point for this study. The geometry of the hot air chamber, in the new design, is shown in Fig. 1.

The hot air chamber, seen in Fig. 1, consists of two ambient air inlets, where the ambient air is led into the hot air chamber by two axial fans. Correspondingly, air that has been used to cool the grain is recirculated into the hot air chamber through a sub-chamber. The recirculated air is then mixed with ambient air, that has been

* Corresponding author.
E-mail addresses: lottestrangekjaer@gmail.dk (L.S. Kjær), mps@et.aau.dk (M. Poulsen), kso@et.aau.dk (K. Sørensen), tc@et.aau.dk (T. Condra).

Peer review under responsibility of Karabuk University.

https://doi.org/10.1016/j.jestch.2018.02.002
2215-0986/© 2018 Karabuk University. Publishing services by Elsevier B.V.

Nomenclature

Symbol	Description/Unit		
A	Area [m^2]	s	Sign Factor [–]
e	Error [–]	T	Temperature [K]
GCI	Grid Convergence Index [–]	U	Velocity [m/s]
g	Grid size [m]	v	Velocity Magnitude [m/s]
h	Enthalpy [J/kg]	V	Volume [m^3]
k	Turbulent Kinetic Energy [m^2/s^2]	x	Length [m]
$\dot{m}$	Mass Flow [kg/s]	γ	Uniformity Index [–]
N	Number of cells [–]	μ	Viscosity [Pa · s]
r	Refinement Ratio [–]	ρ	Density [kg/m^3]
p	Pressure [Pa]	σ	Prandtl Number [–]
P	Global Order of Accuracy [–]	ϕ	Test Parameter [–]
		ω	Specific Turbulent Dissipation Rate [1/s]

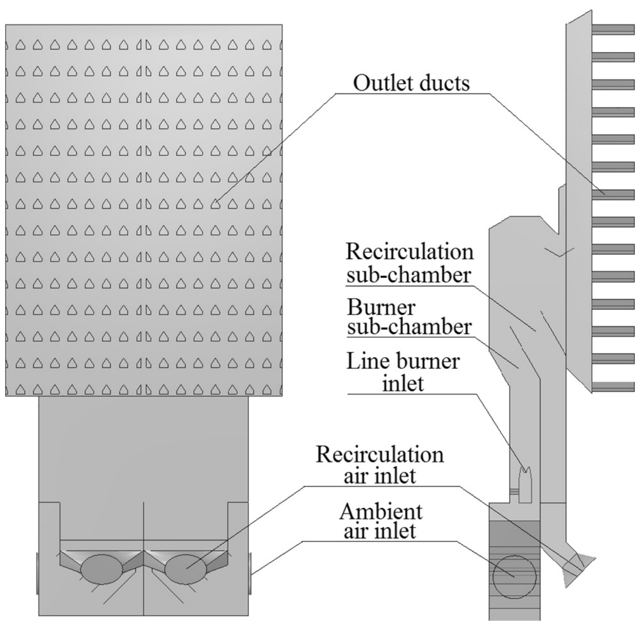

Fig. 1. Geometry of the hot air chamber.

heated by a line burner, at the end of the sub-chamber. The mixing is not carried out upstream of the burner, due to the presence of dust in the recirculation air, which, if passed through the burner, could result in a fire, or a dust explosion. After the two air streams have mixed, the air is led through a manifold, where it is distributed into the outlet air ducts. In the outlet air ducts the air is forced through the grain in the drying column.

3. Computational methodology

3.1. Solution domain

The solution domain has been simplified in order to reduce the number of cells needed to generate a proper mesh. Initially the interior baffles and guide vanes in the geometry were set as plates with a thickness of 2 mm, as this is the material thickness that will be used in the physical construction. However, when generating a mesh on the geometry some low quality elements were formed on the edges of all the baffles and guide vanes. Therefore, it was decided that all of the baffles and guide vanes should be modelled as thin surfaces instead, thereby eliminating poor quality elements. Furthermore, the sub-chamber to the right of the line burner was removed in the model, as no flow is present in the sub-chamber. The last simplification made to the solution domain was the removal of the area above the top baffle, as the flow in this area is not of interest.

3.2. Governing equations

The fluid flow in this study was assumed to be air in steady-state, compressible, and three-dimensional turbulent flow. The numerical calculation of the flow can be considered as mathematical formulations of the conservation laws of fluid mechanics. By applying the mass, momentum, and energy conservation, the fundamental governing equations of fluid dynamics; the mass (1) and momentum (Navier Stokes) (2), (3) equation can be written as follows [6]:

$$\frac{\partial(\rho U_i)}{\partial x_i} = 0 \tag{1}$$

where ρ is the density of the fluid, U_i are the Cartesian velocity component (i = 1, 2 and 3), x_i are the coordinate axes, and the repeated indices imply summation over 1–3.

$$U_j \frac{\partial(\rho U_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial U_i}{\partial x_j}\right) + \frac{\partial R_{ij}}{\partial x_j} \tag{2}$$

where p is the pressure, μ is the dynamic viscosity. The Reynolds stresses can be written as:

$$R_{ij} = \mu_t\left(U_j \frac{\partial(\rho U_i)}{\partial x_j}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{3}$$

where μ_t is the eddy viscosity, k is the turbulent kinetic energy and δ_{ij} is the Kronecker symbol.

3.3. Turbulence modelling

The above equations are supplemented by a turbulence model to account for the turbulence in the flow. In the present case, the Shear Stress Transport (SST) k-ω model has been used, as it combines the advantages of the k-ε model in the far-field region and the k-ω model in the near-wall region [7]. The turbulent flow is computed in terms of two additional parameters, i.e. the turbulent kinetic energy, k, and the specific turbulent dissipation rate, ω. The turbulent kinetic energy and the specific dissipation rate are computed using Eqs. (4) and (5), respectively [6,8]:

$$\frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + \widetilde{G}_k - Y_k + S_k \tag{4}$$

$$\frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{5}$$

Please cite this article in press as: L.S. Kjær et al., Modelling of hot air chamber designs of a continuous flow grain dryer, Eng. Sci. Tech., Int. J. (2018), https://doi.org/10.1016/j.jestch.2018.02.002