CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong

ISBN 978-88-95608-30-3; **ISSN** 2283-9216 DOI: 10.3303/CET1439154

Modelling and Research of Heat Transfer in Fins of the Pneumatic Pulsator

Jacek Wernik*, Krzysztof J. Wolosz

Department of Process Equipment, Warsaw University of Technology, Plock Branch, Al. Jachowicza 2/4, 09-402 Plock, Poland
wernik@pw.plock.pl

Loose material stored in silos can create vaults and bridges. The situation takes place while an adverse operating conditions occur. Hence, restriction of transportation or even blockage of the loose material could be. Various solutions are taken into account in order to prevent situations mentioned above. One of them is a destruction of loose material structure by the compressed air shockwave. This phenomenon is applied in equipment such as, for example pneumatic pulsator.
The methods of CAE, particularly CFD are commonly utilized during the design of the process equipment. The airflow phenomena are the bases of the pneumatic pulsator design, and therefore, the usage of CFD took place. It is crucial to employ heuristic, analytical, and experimental knowledge in the design methodology. Only the application of the three makes possible to create a rational, and subsequently optimal design.
The article presents a chosen set of the design issues considering the head of the pulsator and fins placed on that head. A numerical simulations were carried out and the results obtained showed temperature distribution on head surface and in the fins, stress distribution, safety factor, displacement in the head, and the temperature and velocity field of the flowing around air. The numerical results were validated and compared with the analytical and experimental results, which confirmed their value. The results confirm the head design of reason. The temperature distribution calculated with CFD are approximately equal to the values measured during the exploitation investigations. The CFD utilization helped to launch a new product to the market, but it cannot eliminate the exploitation investigations, nevertheless.

1. Introduction

Storage of loose materials, which attract and arrest moisture, in silo may cause various problems such as vault creation above silo outlets. This can limit or even totally disable the downward movement of the material. The preventive measures against vaults creation are following:
1) Non-symmetrical shape of the vessel,
2) Material outlet positioned close to a vertical vessel wall,
3) Baffles placed above the outlet,
4) Rotating flat bottom (in cylindrical vessels),
5) Screw conveyor placed in the material outlet,
6) Vibratory, striking or pneumatic equipment, to act on the material bed, attached to the vessel.

To prevent vault creation or to destroy vault by aeration and separation of the loose material from encasing walls, a pneumatic pulsator can be applied. Design studies leading to the selection of the optimal shape of the head of pneumatic pulsator provided with pressure accumulator was presented by Wernik and Wolosz (2008). The next step on CFD calculations and optimization of pneumatic pulsators was reported by Urbaniec et al. (2009). During air flow in the channels of pneumatic pulsator due to frictional heat is produced, which is discharged to the environment. The first attempt to make a research on this problem has been undertaken recently and the results of the study have been described by Wernik and Wolosz (2013).

Please cite this article as: Wernik J., Wolosz K.J., 2014, Modelling and research of heat transfer in fins of the pneumatic pulsator, Chemical Engineering Transactions, 39, 919-924 DOI:10.3303/CET1439154

Nowadays, sophisticated numerical techniques based on the solution of the Reynolds-averaged Navier–Stokes equations are being used to predict heat transfer in many devices and apparatuses. Good practice in investigation of fins can be read in (Wu et al., 2014)

Numerical simulations of heat transfer in pneumatic pulsator has not been considered. Heat transfer enhancement by means of various techniques is an important task for research. Enhancement of convective heat transfer are well reported in the literature, see Bergles at al. (1995).

2. Finned head of pneumatic pulsator

A pneumatic pulsator consists of two main components: a head to steer air flow and a cylindrical vessel to act as pressure accumulator. Vessel is made of a cylindrical part and two ellipsoidal bottom heads conforming to pressure vessel standards. The vessel volume depends on the silo capacity and type of loose material, typically 0.1 m^3. The head includes three channels: inlet channel through which air flows from the pressure accumulator, outlet channel directing air flow to the silo, and control channel to control the air flow. The air flow from the inlet to outlet channels is controlled by valve which opening depends on the pressure difference between inlet and control channels. In order to optimise the head design, the flow phenomena were numerically simulated and the channels were designed (Wolosz and Wernik, 2012). The data obtained from simulation were also used as a basis for setting up an experimental stand to validate simulation results. Pneumatic pulsator is shown in Figure 1.

To increase the heat transfer in pneumatic pulsator, the external surfaces of channel are finned, as shown in Figure 2. The rectangular fins are placed on the external surface of the inlet channel to the silo in order to increase the heat transfer from the solid body to the environment. The finned head of pneumatic pulsator were made of aluminium alloy - AlSiMg.

3. Calculations of fins

A common simplification made when analyzing an extended surface is that the temperature within the fin is a function of one dimension, measured directly away from the root (Kraus at al., 2001). The characteristic dimension of the fin in the transverse direction is taken to be uniform cross-sectional area / perimeter, i.e.,

Figure 1: Final construction of the pneumatic pulsator

Figure 2: Quarter of finned channel

$0.0003\ m^2$ / 0.13 m for fins pneumatic pulsator. In order to simplify the calculation, temperature distribution in a one-dimensional fin with the tip insulated can be written as:

$$t - t_e = (t_r - t_e) \cdot \frac{cosh[m(L - x)]}{cosh(mL)} \tag{1}$$

where:

t –fin temperature, K,
t_e – ambient temperature, K,
t_r – root temperature, K,
m – fin parameter, m^{-1},
L – fin length, m,
x - independent variable.

The temperature distribution fins analytically determined according to Eq(1) shown in Figure 3. The insulated-tip approximation is adequate for the computation in this case.
Two basic measures of fin performance are particularly useful in a fin design: efficiency and effectiveness. For fins pneumatic pulsator efficiency η_{fin} was calculated according to the equation Eq(2).

$$\eta_{fin} = \frac{\text{actual heat transferred by a fin}}{\text{ideal heat that would be transferred if the entire fin were at } t = t_r} = \frac{tanh(mL)}{mL} \tag{2}$$

The calculated efficiency of the fins is 0.85. However, it is worth pointing out that the efficiency is useful in characterizing fins with more complex shapes. At the end should be add that limit the applicability of the fins is a condition described by Eq(3). Left side is reciprocal of the Biot number.

$$\frac{k}{h \cdot \delta} \rangle 2.5 \tag{3}$$

where:

k - the thermal conductivity of the material, W/(m·K),
h - heat transfer coefficient, $W/(m^2 \cdot K)$,
δ - thickness of the fin, m.

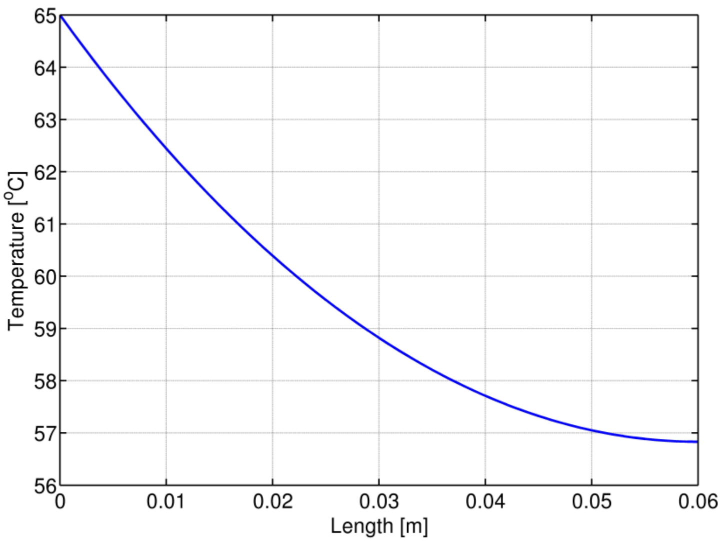

Figure 3: The temperature distribution one-dimensional fin with the tip insulated

4. Modelling of heat transfer using numerical simulations

4.1 Theoretical basis

The subjects of the heat transfer theory in the air flow are stationary, one dimensional, compressible fluid flows, which experience the state conversion - in most cases adiabatic one. Therefore, basic equations mathematically describing the flow (continuity equation, energy equation) are completed with the equation of the conversion of the fluid. The momentum equation considered in the fluid mechanics is not taken into account because there is no need to know the velocity distribution along the cross section of the channel (the knowledge of the mean velocity is sufficient). Kinetic energy in two different cross-sections are only approximated. Although, the approximation is good enough and commonly accepted in practical applications. To determine the exact value of the kinetic energy it is necessary to integrate the particular stream kinetic energy in which local velocity is taken into account. While compression the kinetic energy of the stream is converted into the thermal energy of the fluid and this phenomenon is manifested by the temperature and the pressure increase. At high temperatures the components of the air are dissociating and at the higher they are ionizing, so that the results obtained with the assumption of the perfect gas are not adequate to the real behaviour. The air flow is used in pneumatic pulsators. In order to develop the pneumatic pulsator, it was necessary to combine engineering design with applied research. The knowledge about the thermo-hydrodynamic parameters of the flow (e.g., temperature and velocity fields) in the channels of pneumatic pulsator is important to improve intensity of heat exchange with the environment. In the final design, account was taken of the influence of harsh industrial environment and heat generation in the pulsator head due to rapid flow of compressed air though the channels. To avoid overheating, rectangular fins were attached to the outer surface of the pulsator head. The optimization of shape of pneumatic pulsator therefore always has to be aimed at an increase of the heat transfer simultaneously with improving also the stiffness of pulsator casing.

4.2 Simulating heat transfer

The following assumptions are made to model the air flow problem: Newtonian fluid, compressible and turbulent flow, fluid is considered as an ideal gas. The applied governing equations are the 3D Reynolds-averaged Navier-Stokes equations, which were solved by commercial Computational Fluid Dynamics software. The governing equations solved by the model are: continuity equation, momentum equation, energy equations, turbulence model and equation of state. The computational model and mesh were generated in the SolidWorks software. In order to avoid many simplifications in the computational model the computational model was generated using the total pneumatic pulsator geometry.

Two different geometries of fin configurations were studied – rectangular and triangular. Due to complexity of the geometry, different computational grid sizes are required. These grids are not uniform in all directions and were structured by mixing different types of cells. For the grid used in the computational model the mesh density is high in the near-wall region of the channel. Air flow and heat transfer analysis of the first stage pneumatic pulsator with different fins configurations placed in the channel were realized using CFD software (FLUENT, Solid Works). This code allows to solve the Reynolds averaged Navier-Stokes and the transport equations of the turbulent quantities for the compressible viscous flow. This CFD code solves the equations using the finite element method to discretize the governing equations inside the computational domains. The standard k- ε model was used for all numerical simulations. This model is a semi-empirical linear eddy viscosity model based on the transport equations for the turbulent kinetic energy and dissipation rate. Due to the fact that the governing equations are non-linear and coupled, several iterations were needed to reach a converged solution. Different variations of thermal boundary condition on the fins have been considered in study. In all numerical studies that have been carried out, constant heat flux was employed as a boundary condition on the channel walls. Figure 4 shows the temperature distribution determined numerically on the segment of finned channel and two opposing fins. The highest and lowest temperature value is respectively 64 °C and 55 °C.

Surface temperature distribution along the height of fin is shown in Figure 5. The temperature values correspond to the seven selected nodes.

The results of the thermal study can be used as an input for the analysis of demand - determine the stress, displacements and deformations of the thermoplastic. This will be the stage further study. The model of pneumatic pulsator was used to represent the air flow around a specified temperature.

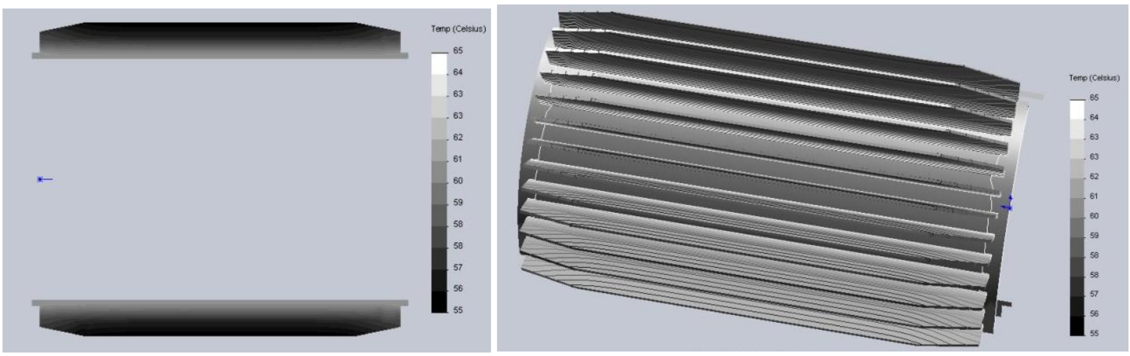

Figure 4: The temperature distribution on the external the channel surface

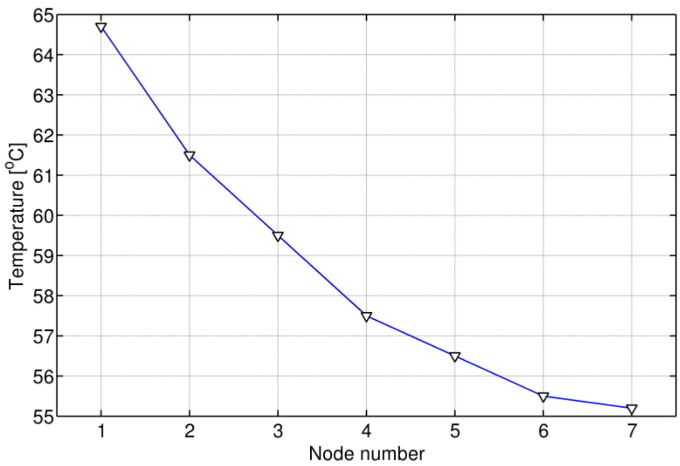

Figure 5: The temperature distribution along the height

4.3 Comparison of numerical simulations results with theory and experimental data

Global data for heat transfer have been obtained by conventional measuring techniques such as thermocouples and pyrometer. It is worth emphasizing that the temperature distribution on the surface of the body and fins obtained numerically roughly correspond to the actual measured temperatures at an industrial plant during the tests. The temperature distribution fins analytically determined according to the Eq(1) shown in Figure 3. It complies with the temperature distribution shown in Figure 5.

5. Conclusions

The design of a fin thus becomes an open-ended matter of optimizing, subject to many factors, e.g. the weight of material added by the fins, the cost and complexity of manufacturing fins. The factor that determines the accuracy of the heat transfer prediction is the thermal boundary condition that is used at the walls of the fins. In thermal study that have been carried out thus far, two boundary conditions have been employed on the channel walls: convection and constant heat flux. The effects of conduction heat transfer in the walls have been considered. In real channels, the heat transfer process that occur is always a combination of forced convection and heat conduction.

Numerical simulations carried out conduction and convection heat transfer in the fins, and the results illustrated the temperature distribution on the surface of the body, the temperature distribution in the fins, stress distribution, the static displacement of the body, the distribution of air temperature along with the speed vectors. The results were compared with experimental and analytical results that confirm their correctness. The results confirm the rational design of the body in terms of thermal. The obtained temperature distribution using CFD agrees approximately with the values measured during the tests.

Although the numerical simulations show quite good results, there is still need for experiments as data base for verification of the codes, considering especially of heat generation in the pneumatic pulsator due to of airflow. Nowadays simulations are used routinely in industry for design purposes. This allowed to

arrive at the final pulsator design within a relatively short time. The basic aim of the project has been reached. The use of fins on the body pneumatic pulsator caused the pulsator body weight increased slightly, to 13.2 kg, however, the heat flux given to the environment has increased almost 4 times.
Future work will focus on the study of circular and hyperbolic fins in the design of the pneumatic pulsator.
Pneumatic pulsators are widely applied for safe and efficient elimination of vaults. If large dimensions of the silos are taken into account, in case of lacking preventive equipment, then vaults' removal may require operation breaks and the intervention of staff authorized to work in dangerous conditions, leading to increased production costs.

References

Bergles A.E., Jensen M.K., Shome B., 1995, Bibliography on Enhancement of Convective Heat and Mass Transfer, Heat Transfer Laboratory Report HTL-23, Rensselaer Polytechnic Institute, Troy, USA

Kraus A., Aziz A, Welty J.R., 2001, Extended Surface Heat Transfer, John Wiley & Sons, Inc., New York, USA

Urbaniec K., Wernik J., Wolosz K.J., 2009, Optimal design of the head of a pneumatic pulsator, Chemical Engineering Transactions, 18, 237-242.

Wernik J., Wolosz K.J., 2008, Numerical Simulations of Air Flow in a Pneumatic Pulsator, 18th Int. Congress of Chemical and Process Engineering, 24-28 August 2008, Prague, Czech Rep., ID: 1546.

Wernik J., Wolosz K.J., 2013, Numercial simulations and research of heat transfer in pneumatic pulsator, 7 th International Conference on Environmental Engineering and Management, 18-21 September 2013, Vienna, Austria, 223-224.

Wolosz K.J., Wernik J., 2012, Pneumatic pulsator design as an example of numerical simulations in engineering applications, Central European Journal of Engineering, 2(1), 76-82.

Wu X., Zhang W., Gou Q., Luo Z., Lu Y., 2014, Numerical simulation of heat transfer and fluid flow characteristics of composite fin, International Journal of Heat and Mass Transfer, 75, 414-424.