Thermo Fluid-dynamics of Combustion Products in Heat-Accumulation Stoves: A Study Case

Daniele Rossi[1], Paolo Scotton[1]

[1]University of Padova, Department of Geosciences, Padova, Italy

Abstract

The research regards a particular kind of heat accumulation stoves built in ceramic and refractory. They are a traditional element of the European Alpine regions, whose history began in the fifteenth century. They consist of a combustion chamber, where woody material is burned, followed by a twisted conduct where the high temperature products of combustion flow, giving off heat to the refractory. The stored heat is returned slowly to the environment in the form of radiant heat. The way the heat exchange happens depends mainly on gas flow conditions and on conduct physical properties. The attempt to describe this phenomenon is the goal of the present research, in order to provide increased awareness in the definition of design criteria, considering the need of energy saving and pollution control. In this perspective an instrumented laboratory physical model has been realized, measures of temperature, and pression and velocity have been taken inside the conduct and on the outer surface of the conduct (Figure 1). Comsol Multiphysics has been used to simulate the thermofluidodynamics of combusted gases within the physical model composed by a combustion chamber, a refractory conduct, and the chimney. In order to better represent the real case, the model has been implemented at unsteady boundary conditions and considering almost all the physical phenomena that we consider to have a significant influence on the fields of velocity and temperature. The physical phenomenon of natural combustion of woody material is a chemical reaction that, once provided the activation energy, continues initially increasing temperature and velocity of gasses and then decreasing the two variables until the end of reaction (Figure 2). This cause a continuous variation in the motion conditions of combusted gases and that is what we are trying to reproduce numerically. Verified that the minimum Reynolds number was greater than 4000, it was decided to use the k-e turbulent model. The variation in time of temperature at some positions along the tube, both inside and at its outer surface, have been compared with the temperatures measured during the laboratory experiments. In Figure 3 one can see the comparison between laboratory temperature (continues line) and the temperature at the same location from the numerical simulation (dotted line). It has been found that, in case of non isothermal flow, the volume forces due to gravity are important in the simulation of the fluid motion inside the tube. Still remains to be investigated the role of the radiation component inside the conduct and the role of the combusted gasses in their absorption and transmission. Dimension and type of mesh has been proved to be very important in order to obtain a good representation of the physical results at a reasonable computation time. The numerical methods and features available in COMSOL Multiphysics version 4.2a allow a reasonable representation of the motion and thermodynamics of combusted gases. Some significant limitations remain in the software, the first of which is the inability to represent the wall roughness that can have an important role in the studied physical

phenomena.

Reference

1. A., Ghetti, Idraulica, 1987, Edizioni Libreria Cortina, Padova.
2. R. J. Garde, 2010, Turbulent Flow, New Age Science Limited, UK.
3. L. Ignat, D. Pelletier, and F. Ilinca, “A Universal Formulation of Two-equation Models for Adaptive Computation of Turbulent Flows,” Computer Methods in Applied Mechanics and Engineering, vol. 189, pp. 1119—1139, 2000.
4. K. Abe, T. Kondoh, and Y. Nagano, “A New Turbulence Model for Predicting Fluid Flow and Heat Transfer in Separating and Reattaching Flows–I. Flow field calculations,” International Journal of Heat and Mass Transfer, vol. 37, no. 1, pp. 139—151, 1994.
5. G. La Casa, 2007, Analisi sperimentale e numerica della convezione mista in aria in canali uniformemente riscaldati in presenza di una parete mobile, Tesi di dottorato in Ingegneria dei Sistemi Meccanici, XX ciclo, Università degli studi di Napoli Federico II
6. Scotton, P., Rossi, D., 2011, Study of Gas Dynamics in the Heat-accumulation Stoves, Comsol Conference 2011 Stuttgart Proceedings, ISBN 9780983968801.
7. Lienhard IV, J. H., Lienhard V, J. H., 2006, A Heath Transfer Texbook, Third Edition, Phlogiston Press, Cambridge, Massachussets, U.S.A.

Figures used in the abstract

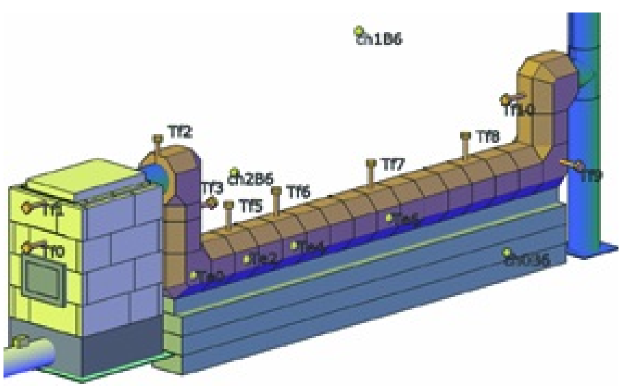

Figure 1: A view of the instrumented physical model of a simplified heat accumulation stove.

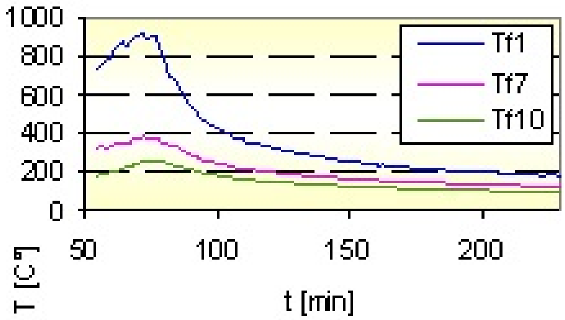

Figure 2: Laboratory temperature taken at three points of the phisical model: Tf1 is the temperature inside the combustion chamber, Tf7 is the temperature measured roughly at the middle of the refractory conduct and Tf10 is the temperature just before the chimney.

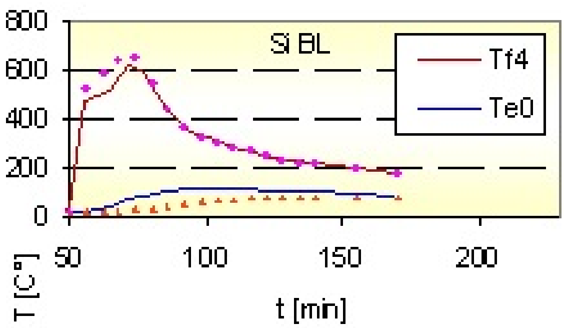

Figure 3: Comparison between experimental temperature (continues line) and numerical one (dotted line) at the same position. The thermocouple Tf4 is placed inside the tube, the Te0 measures the temperature at the outer surface of conduct.