30. Setkání kateder mechaniky tekutin a termomechaniky

HYDRO TERMO

22.-24.6. 2011

Špindlerův Mlýn

Jednotlivý příspěvek ze sborníku

TECHNICKÁ UNIVERZITA V LIBERCI

evropský sociální fond v ČR

EVROPSKÁ UNIE

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

OP Vzdělávání pro konkurenceschopnost

INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Characteristics of a mathematical model of the spiral heat exchanger using CFD ANSYS Fluent

Marian BOJKO[1], Adéla Macháčková[2] Zuzana KLEČKOVÁ[3], Radim KOCICH[4],

[1] Ing. Marian Bojko, Ph.D., VŠB-Technical University of Ostrava, Department of Hydrodynamics and Hydraulic Equipment, 17.listopadu 15/2172, 708 00, Ostrava Poruba, marian.bojko@vsb.cz

[2] doc. Ing. Adéla Macháčková, Ph.D, VŠB-Technical University of Ostrava, Department of Thermal Engineering, 17.listopadu 15/2172, 708 00, Ostrava Poruba, adela.machackova@vsb.cz

[3] doc. Ing. Zuzana Klečková, CSc., VŠB-Technical University of Ostrava, Department of Thermal Engineering, 17.listopadu 15/2172, 708 00, Ostrava Poruba, zuzana.kleckova@vsb.cz

[4] Ing. Radim Kocich, Ph.D., VŠB-Technical University of Ostrava, Department of Materials Forming, 17.listopadu 15/2172, 708 00, Ostrava Poruba, radim.kocich@vsb.cz

Abstrakt: *Paper describes a three-dimensional mathematical model of fluid flow exhaust gas in the heat exchanger. It is a spiral heat exchanger in which hot exhaust gases are blown. Then heat of exhaust gas is removed by water flowing through a spiral tube. Article in detail defines the problems of heat transfer due to conduction and convection heat individual areas of exchanger. The mathematical model includes the effect of physical properties of individual flowing media and material properties of walls exchanger. Defined mathematical model is solved in the programming software ANSYS Fluent13.1. The basic current variables are evaluated in the individual sections (velocity and temperature fields).*

1. Introduction

Mathematical modeling of fluid flow includes real now widely used in many applications in the industry. From the metallurgical industry, chemical industry, to environmental. In this paper is characterized a mathematical model flow of combustion products in the heat exchanger. The gaseous combustion products are derived from combustion of natural gas in microturbines, which achieves on outlet of tens of KW. One way to increase the thermal efficiency of energy processes based on combustion of gaseous fuels, the design of various types of heat exchangers that use waste heat produced by combustion products in various stages of the energy process. In this case the spiral heat exchanger is designed, which water flows through, which removes heat from gaseous combustion. The primary content of this paper is the characterization and design of a mathematical model of flow combustion products through heat exchanger including the consideration of heat transfer to the surrounding walls. Numerical modeling of problems of flow in heat exchanger by using CFD has been solved by several authors [1], [2], [3], [4], [5]. This paper defines the possibility of heat transfer solution in a spiral heat exchanger with a detailed characterization of mathematical model of flow combustion products in heat exchanger

2. Characterization of heat exchanger model

Defined mathematical model will be applied to the spiral heat exchanger, which is illustrated in Figure 3. This is a recuperative heat exchanger, however in certain phases behaves like uniflow, because the influence of spiral loops water flows in the direction of gas flow.

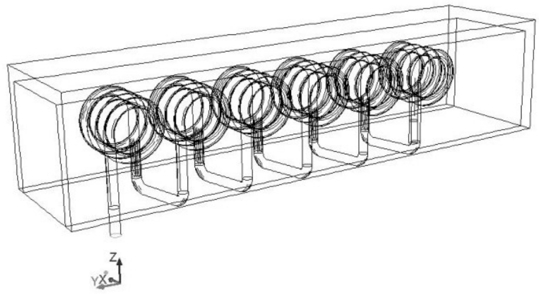

Figure 3: The model of spiral heat exchanger

The heat exchanger consists of three layers of insulating material SIBRAL (top and side walls) and the bottom surface of the heat exchanger is made up of stainless steel sheets. Material of spiral loops is made from copper.

3. Characterization of the mathematical model flow of combustion products through heat exchanger

Based on the characteristics of combustion products flow problems and water in the heat exchanger was defined turbulent Standard k-ε Model with heat transfer. Below the basic balance equations of a mathematical model are defined:

Mass equation:

$$\frac{\partial(\rho \bar{u}_j)}{\partial x_j} = 0 \quad (1)$$

Momentum equations:

$$\frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\frac{\partial \bar{u}_i}{x_j}\right] \quad (2)$$

Equation of turbulent kinetic energy:

$$\frac{\partial(\rho \bar{u}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \quad (3)$$

,where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients:

$$G_k = \mu_t\left(\frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j}\right)\frac{\partial \bar{u}_j}{\partial \bar{x}_i} \quad (4)$$

Equation rate of dissipation:

$$\frac{\partial(\rho \bar{u}_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k -$$

$$-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (5)$$

,where

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \;, C_\mu = 0.09 \,, C_{1\varepsilon} = 1.44$$

$$C_{2\varepsilon} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

Energy equation:

$$\frac{\partial}{\partial x_j}\left[\bar{u}_j(\rho E + p)\right] = \frac{\partial}{\partial x_j}\left[k_{eff}\frac{\partial T}{\partial x_j} + \bar{u}_i(\tau_{ij})_{eff}\right] \quad (6)$$

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \quad \text{total energy} \quad (7)$$

$$h = \sum_j Y_j \int_{T_{ref}}^{T} c_{p,j} dt \quad \text{sensible enthalpy} \quad (8)$$

$$k_{eff} = k + \frac{c_p \mu_t}{Pr_t} \quad \text{effective conductivity} \quad (9)$$

,where k is thermal conductivity, c_p is specific heat capacity and Pr_t is turbulent Prandtl number ($Pr_t = 0.85$).

Energy equation in solid regions:

$$\frac{\partial}{\partial x_j}(\rho h \bar{u}_j) = \frac{\partial}{\partial x_j}\left(k\frac{\partial T}{\partial x_j}\right) \quad (10)$$

Species transport equation:

$$\frac{\partial(\rho u_j Y_{i'})}{\partial x_j} = -\frac{\partial}{\partial x_i} J_{j,i'} \quad (11)$$

The resulting model of the heat exchanger can be described as a 3D mathematical model of stationary turbulent flow of gas mixture and the water with heat transfer and flow is considered as compressible. Combustion gases flows through the heat exchanger are composed of CO_2, H_2O, N_2, O_2. Calculation of gas density is defined by the ideal gas equation as compressible gas [6],[7]. Next physical properties (viscosity, specific heat capacity, thermal conductivity) of gas mixtures are defined by mixing laws [6], [7].
With regard to the calculation of heat transfer from combustion products to wall spiral pipe into the flowing water is considered the real pipe wall thickness, including the physical properties of wall material. Then in the model of heat exchanger is considered an insulating layer of material Sibral again with defined physical characteristics (density, specific heat capacity, thermal conductivity).

4. Results of numerical simulation

Result of numerical simulation of temperature field in heat exchanger for power of turbine 30kW to burning natural gas is shown in Figure 4.

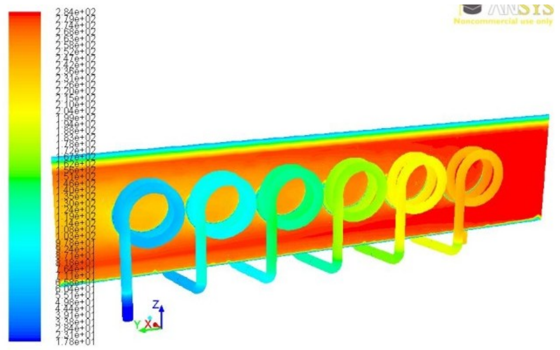

Figure 4: Temperature field in the longitudinal direction and wal surface temperature of spiral pipe (°C)

The figure shows the temperature field in the longitudinal direction and wall surface temperature of spiral pipe, which water flow. From the wall surface temperature of the pipe is visible heating water from inlet. The blue color of wall pipe indicates the inlet of cold water, and combustion products going into the heat exchanger on the opposite side Figure 4.

5. Conclusion

Paper in detail defines the 3D mathematical model of flow of gaseous combustion products and water in the spiral heat exchanger including heat conduction and convection. The introductory chapter are presented the basic balance equations of the mathematical model, including definition of the physical properties of individual flow media and solid materials. Then the mathematical model is applied to the heat exchanger for gas turbine power 30kW. From results is presented the temperature field in longitudinal direction of heat exchanger and the wall surface temperature of the spiral pipe.

The work was supported by project of Ministry of Environment (MŽP ČR) SPII2f1/27/07 „Minimalizace emisní zátěže kogenerační jednotky výzkumem technologických postupů pro využití v komunální sféře“

6. References

[1] PRITHIVIRAJ M., ANDREWS M. J.: Three-dimensional numerice simulation of shell-and-tube heat exhanger, Part I: Foundation and fluid mechanics, *Numerical Heat Transfer*, Part A: Applications, Volume 33, Issue 8, 1998, p. 799-816.

[2] PRITHIVIRAJ M., ANDREWS M. J.: Three-dimensional numerice simulation of shell-and-tube heat exhanger, Part II: Heat transfer, *Numerical Heat Transfer*, Part A: Applications, Volume 33, Issue 8, 1998, p. 817-828.

[3] PRITHIVIRAJ M., ANDREWS M. J.: Comparison of a three-dimensional numerical model with existing methods for prediction of flow in shell-and-tube heat exhanger, *Heat Transfer Engineering*, Volume 20, Issue 2, 1999, p. 15-19.

[4] ANDREWS M. J., MASTER B.I.: Three-dimensional modeling of a Helixchanger heat exchanger using CFD, *Heat Transfer Engineering, Volume* 26, Issue 6, 2005, p.22-31.

[5] ZHANG J., HE Y., TAO W.: 3D numerical simulation on shell-and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles - Part I: Numerical model and results of whole heat exchanger with middle-overlapped helical baffles, *International Journal of Heat and Mass Transfer*, Volume 52, 2009, p. 5371-5380.

[6] KOZUBKOVÁ, M.: *Modelování proudění tekutin FLUENT, CFX*. Ostrava: VŠB-TU, 2008, 154 s., ISBN 978-80-248-1913-6, (Elektronická publikace na CD ROM).

[7] FLUENT: Fluent 13.0 - *ANSYS FLUENT Theory Guide,* ANSYS, Inc. 2010.