ZÁPADOČESKÁ UNIVERZITA V PLZNI

FAKULTA STROJNÍ

KATEDRA ENERGETICKÝCH STROJŮ A ZAŘÍZENÍ

ZÁPADOČESKÁ UNIVERZITA V PLZNI

POWER SYSTEM ENGINEERING, THERMODYNAMICS & FLUID FLOW

2013

JEDNOTLIVÝ PŘÍSPĚVEK ZE SBORNÍKU

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

TURBULENCE MODELS SUITABLE FOR CFD SIMULATION OF PRESSURE LOSSES IN THE COMPRESSOR'S FILTER

RUMAN Radovan, ŠUSTEK Jaroslav, TOMLEIN Peter

The paper deals with the selection of suitable turbulence model for modeling pressure losses in refrigerant flow of CO_2 through the suction filter of compressor. CFD simulations of flow through the suction filter were carried out with $k-\varepsilon$, $RNG\ k-\varepsilon$ and $k-\omega$ turbulence models. CFD simulation's result of the pressure losses of suction filter are compared with experimental measurements and on the basis of CFD simulation and experimental measurement was chosen the most suitable model for modeling pressure losses with CO_2.

Keywords: pressure losses, carbon dioxide, suction filter

Introduction

Currently, we notice the return of the old refrigerant CO_2 (carbon dioxide), which is the alternative to high-GWP refrigerants due to its favorable physical properties in certain applications and climates, despite the high operating pressures. Therefore, development of compressors is focused on refrigeration circuits using carbon dioxide as a refrigerant. One of the main criteria of the development is efficiency, which is affected by pressure drops in refrigerant flow inside the compressor, especially in the suction side, e.g. in the suction filter. The paper deals with the selection of suitable turbulence model for CFD modeling of pressure losses in the carbon dioxide refrigerant flow through the suction filter of reciprocating compressor (*Fig. 3*). Using carbon dioxide as refrigerant has engaged many researchers in their theoretical and experimental work. Yoon, Kim et al. (2003) in their work presented measured data to construct the characteristics of heat transfer coefficient and pressure drop obtained in the gas cooling process in a horizontal tube [10]. Son and Park (2005) experimentally investigated the heat transfer coefficient and pressure losses in the cooling process with CO_2 in a horizontal tube [9]. Lisboa et al. (2010) investigated heat transfer of carbon dioxide using computational fluid dynamics (CFD) [5]. Jiang et al. (2008) investigated numerically and experimentally convective heat transfer of CO_2 at supercritical pressures in a vertical porous tube [3]. Fard (2009) has developed CFD model to determine the convective heat transfer coefficient of carbon dioxide in two porous tubes [2].

2. Statistical models of turbulence

Set of additional equations and empirical relationships, which together with the equations of motion create solvable system of equations, are called turbulence models. Turbulence models can be divided into several groups, which are near described in literature [7] and [8]. Below are described the most used two-equation turbulence models.

2.1 Two-equation $k - \varepsilon$ turbulence model

Turbulent viscosity ν_t is at two-equation $k - \varepsilon$ model defined by the equation, which includes kinetic energy *k* and speed of dissipation ε [1], [4]:

$$\nu_t = c_\nu \frac{k^2}{\varepsilon} \tag{1}$$

where c_ν is an empirical constant. Distribution of the *k* is given by transport equation. The exact form of the transport equation for *ε* can be derived from the Navier-Stokes equations. The resulting form of equation for dissipation speed can be expressed as follows [1], [4]:

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial \bar{u}_j \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + c_{1\varepsilon}\nu_t\left(\frac{\partial \bar{u}_j}{\partial x_l} + \frac{\partial \bar{u}_l}{\partial x_j}\right)\frac{\partial \bar{u}_l}{\partial x_j} - c_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{2}$$

where σ_ε, $c_{1\varepsilon}$ and $c_{2\varepsilon}$ are empirical constants. This turbulence model is verified for tasks with high Reynolds number. This model is not also suitable for solving problems in curvilinear flow past bodies, because the results do not fully correspond practically proven results.

2.2 RNG $k - \varepsilon$ turbulence model

This model is derived from the classical $k - \varepsilon$ model when we use a mathematical procedure called "Renormalization Group" (RNG) method. Renormalization procedure consists in the gradual elimination of small eddies, where momentum equations (Navier-Stokes equations) will be transformed. Formally, this model is identical to the $k - \varepsilon$ model, but there are other values of empirical constants and turbulent viscosity is defined by different relations. However model does not remove the fundamental shortcomings of standard k-ε model and brings only a small improvement in the calculation. Generally it can be said that the models based on equations k-ε quite well calculate fully developed turbulent flow, but in areas near the rigid wall model accuracy decreases [1].

2.3 Turbulence model $k - \omega$

Model by Wilcox uses to determine the turbulent linear scale so-called specific dissipation rate $\omega = \varepsilon / k$. Time-scale of turbulence is solved by the turbulent kinetic energy as it was in the previous model. Linear scaling is solved by the new above mentioned variable *ω*, which is called the vorticity. Turbulent kinetic energy is solved by equation [1]:

$$\frac{\partial k}{\partial t} + \frac{\partial \bar{u}_j k}{\partial x_j} = \tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma^*\nu_t)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

and vorticity is solved by the following differential equation [1]:

$$\frac{\partial \omega}{\partial t} + \frac{\partial \bar{u}_j \omega}{\partial x_j} = \alpha\frac{\omega}{k}\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma\nu_t)\frac{\partial w}{\partial x_j}\right] \tag{4}$$

Turbulent viscosity ν_t is at this two-equation model given by a simple equation [1]:

$$\nu_t = \frac{k}{\omega} \tag{5}$$

In this case it is possible to determine dissipation speed ε from a simple algebraic equation [1]:

$$\varepsilon = \beta^* \, \omega k \tag{6}$$

This model is suitable for solving the flow with low Reynolds number and for tasks with transient flow regime, where is not fully developed turbulence. The disadvantage of this model is great sensitivity to flow at a considerable distance from the wall. This model thus calculates quite accurately the flow near the wall, but on the contrary its accuracy decreases in the free stream. Its accuracy is therefore reversed due to the k - ε model.

3. Test facility

Pressure losses measurement of suction filter was carried out with the help of Emerson Climate Technologies in their engineering laboratory in Mikulov (Czech Republic). Measuring line consists of a compressor, an oil separator, Coriolis mass flow meter, heat exchanger, control and shut-off valves, and pressure and temperature sensors. Measuring line scheme is shown in *Fig. 1*. Refrigerant in circuit is pumped by 4 cylinder compressor (capacity 38 kW). Refrigerant mass flow rate is measured by Coriolis mass flow meter. Throughout the whole cycle the refrigerant moves only in gaseous form without phase change.

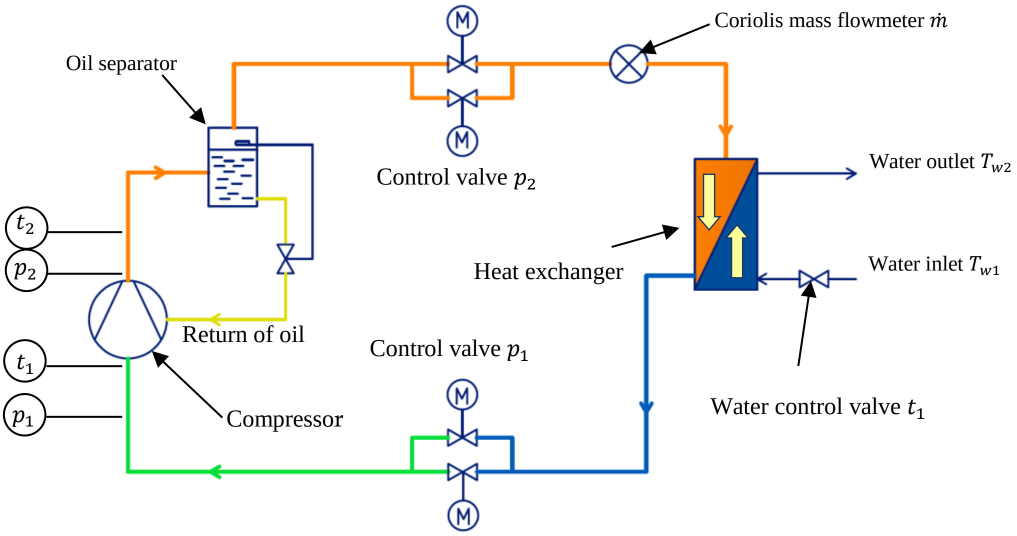


Fig. 1: Scheme of measuring line

Measurement technique has recorded the suction and discharge pressure, suction and discharge temperature, refrigerant mass flow rate, pressure loss in the inlet channel of stator cover with built-in suction filter. Testing compressor, together with the location of the pressure and temperature sensors, can be seen in *Fig. 2*. Pressure loss of suction filter was measured by differential pressure transducer. The first measurement point was connected to the shut-off valve, at the inlet of flow to the filter. The second measurement point recorded the pressure through the drilled hole in the stator, at the outlet of flow from suction filter.

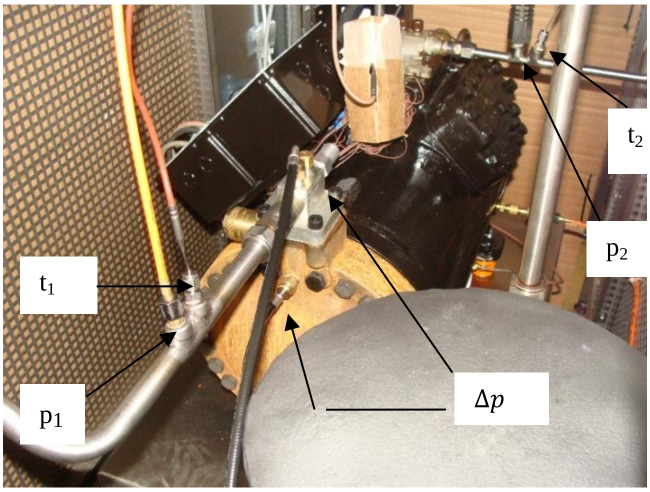


Fig. 2: Measuring line with placing pressure and temperature sensors

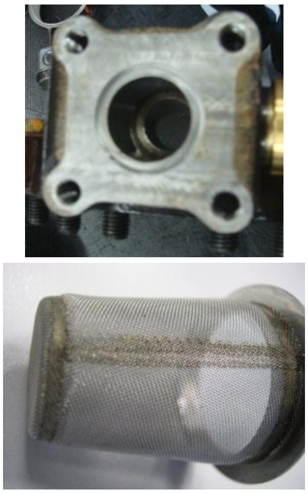
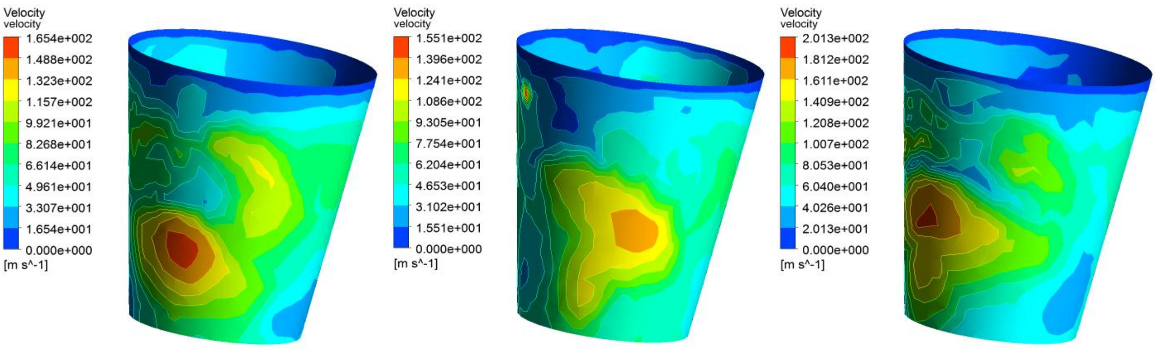


Fig. 3: Placing of suction filter in the stator cover canal (up) of compressor

4. Results and discussion

CFD simulation of pressure losses with different turbulence models in refrigerant flow of CO_2 through intake canal in stator cover with built in suction filter was carried out. The simulation results were then compared with measured values in the outer points of working envelope of compressor. There were investigated three points, HDF (High Density Flow) with parameters p_1= 3.97 MPa, p_2 = 5.90 MPa and t_1= 17 °C, point B with parameters p_1= 2.65 MPa, p_2 = 5.09 MPa, t_1= 2 °C and point HCR (High Compression Ratio) with parameters p_1= 1.97 MPa, p_2 = 10.00 MPa, t_1= -8 °C. The behavior of flowing medium was investigated in the inlet canal of stator cover, where suction filter is placed. Carbon dioxide was set as flowing medium and physical properties were calculated from Peng-Robinson real gas model. Flow was set as isothermal, boundary conditions for fluid were defined for inlet and outlet with pressure and mass flow rate. Physical model for suction filter was set as porous medium with isotropic loss model and with value of medium porosity equals 0.21. In the *Fig. 4* are displayed velocity fields of suction filter, where $k-\varepsilon$, $RNG\ k-\varepsilon$ and $k-\omega$ turbulence models in HDF condition are used.

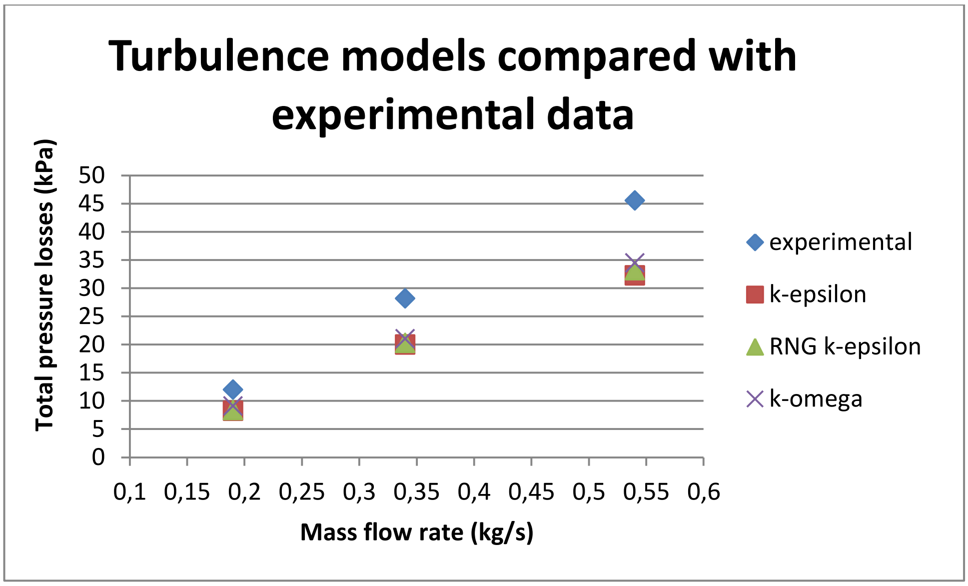


$k-\ \varepsilon$ *model* *RNG* $k-\ \varepsilon$ *model* $k-\ \omega$ *model*

Fig. 4: Displaying velocity fields on the surface of suction filter in HDF condition

Each investigated condition (3 points) had different value of mass flow rate according to the position in working envelope of compressor. In HDF condition mass flow rate equals to 0.54 kg/s, in B condition 0.34 kg/s and in HCR condition 0.19 kg/s. The values of pressure losses in the CFD simulation were obtained in the plains where the pressure sensors were placed.

Graph 1 shows comparison of the values of pressure losses gained from CFD simulation for three different turbulence models with values from experimental measurements. The maximum deviation between experimental data and calculated results from CFD simulation was 29.23%. Graph 1 show that the best results according to the experimental measurements, gives $k-\omega$ turbulence model. According to the theory this model gives better results in flow near the wall. So in fluid flowing through the suction filter, which we consider as a porous medium, it is better to use this model than other two models in case study. Differences between $k-\varepsilon$ and $RNG\ k-\varepsilon$ model are relatively small, because RNG model only improves the standard $k-\varepsilon$ model. Other authors, e.g. Lisboa [5] investigated heat transfer of supercritical carbon dioxide under high pressure conditions. The $RNG\ k-\varepsilon$ model gave results that were closer to experimental data (deviation was within $\pm$ 5 %) than standard $k-\varepsilon$ and $k-\omega$ models. Yoon and Kim [10] have measured pressure losses of supercritical carbon dioxide during gas cooling process in a horizontal tube. Experimental data were compared with theoretical values according Blausius' correlation and the deviation between the measured data and this correlation was 5.9%.



Graph 1: Results from CFD simulations compared with experimental measurement

Conclusion

The paper dealt with the selection of suitable turbulence model for modeling pressure losses in the refrigerant flow in the suction filter of compressor using CFD simulation tool. The maximum deviation between experimental data and results calculated from CFD simulation was 29.23%. Turbulence model $k-\omega$ gives the best results of pressure losses according to the experimental measurements. But difference between investigated turbulence models was not very significant. Future work will be focused on designing geometric modifications of suction filter to ensure reduction of pressure losses.

Literature

[1] BLEJCHAŘ, T. 2010. Turbulence „Flow modeling" - CFX. Ostrava: *VŠB – Technical University*. 2010, 259 p. [online] Available on internet: http://www.338.vsb.cz/studium9.htm

[2] FARD, M. H. 2010. CFD modeling of heat transfer of CO_2 at supercritical pressure flowing vertically in porous tubes. In *International Communications in Heat and Mass Transfer*. 2010, vol. 37, p. 98 – 102.

[3] JIANG, P. X. et al. 2008. Experimental and numerical study of convection heat transfer of CO_2 at supercritical pressures in vertical porous tubes, In *International Journal of Heat and Mass Transfer*. 2008, vol. 51, p. 6283 – 6293.

[4] KOZUBKOVÁ, M. 2008. Fluid flow modeling FLUENT, CFX. Ostrava: *VŠB – Technical university*. 2008, 153 p.

[5] LISBOA, P. F. et al. 2010. Computational fluid dynamics study of a Kenics static mixer as a heat exchanger for supercritical carbon dioxide. In *Journal of Supercritical Fluids.* 2010, 9 p.

[6] MASARYK, M. – FERSTL, K. 2011. *Heat transfer*, Bratislava: Slovak University of Technology, 2011, 424 p. ISBN 978-80-227-3534-6.

[7] MOLNÁR, V. 2009. Computational Fluid Dynamics. Bratislava: *Slovenská technická univerzita*. 2009, [online] Available on internet: http://www.cfd.sk/cfd-book/

[8] PŘÍHODA, J. – LOUDA. P. 2007. *Mathematical modeling of turbulent flow*. Czech Technical University in Prague, 2007, 111 p. ISBN 978-80-01-03623-5.

[9] SON, CH. H. – PARK, S. J. 2006. An experimental study on heat transfer and pressure drop characteristics of carbon dioxide during gas cooling process in a horizontal tube. In *International Journal of Refrigeration*. 2006, vol. 29, p. 539 – 546.

[10] YOON, S. H. – KIM, H. J. et al. 2003. Heat transfer and pressure drop characteristic during the in-tube cooling process of carbon dioxide in the supercritical region. In *International Journal of Refrigeration*. 2003, vol .26, p. 857 – 864.

Acknowledgment

This work and experimental measurements were supported by Emerson Climate Technologies in Mikulov (Czech Republic). Financial support was secured thanks to the help of the "V4 Green energy platform", which is supported by the Visegrad Fund and thanks to the financial grant under the “Program for supportig young researchers from Slovak University of Technology in Bratislava”.

Visegrad Fund
www.visegradfund.org

EMERSON
Climate Technologies

Ing. RUMAN Radovan, Slovak University of Technology in Bratislava, Faculty of Mechanical Engineering, Institute of Thermal Power Engineering, Námestie slobody 17, 812 31 Bratislava, Slovak Republic, Email: radovan.ruman@stuba.sk
Ing. ŠUSTEK Jaroslav, Slovak University of Technology in Bratislava, Faculty of Mechanical Engineering, Institute of Thermal Power Engineering, Námestie slobody 17, 812 31 Bratislava, Slovak Republic, Email: jaroslav.sustek @stuba.sk
doc. Ing. TOMLEIN Peter, CSc., Association for Cooling and Air Conditioning Technology, Hlavná 325, 900 41 Rovinka, Slovak Republic, szchkt@szchkt.org