OVERVIEW OF NUMERICAL METHODS FOR SIMULATING RANQUE-HILSCH EFFECT WITHIN VORTEX TUBES

Vadim NIKITIN[1], Paulius BOGDEVIČIUS[2], Marijonas BOGDEVIČIUS[3]

Vilnius Gediminas Technical University, Vilnius, Lithuania
E-mails: [1]vadim.nikitin@vgtu.lt; [2]paulius.bogdevicius@vgtu.lt; [3]marijonas.bogdevicius@vgtu.lt

Abstract. The main goal of this paper is to present a summarized overview of the methods used for Ranque–Hilsch vortex tube numerical analysis using computational fluid dynamics (CFD). The overview is mainly focused on the most recently conducted studies that are both backed by experimental data and is evaluated by the authors as being in good agreement with experimental results. Generalized tendencies in computational vortex tube analysis are presented while focusing on researchers' approach towards the geometric model used for the study and the governing equations. A brief introduction to the subject matter is presented followed by a short retrospective of the previous studies and related challenges. The conclusions are formed based on the most recent, as well as previously analysed, published research results.

Keywords: vortex tube, Ranque-Hilsch vortex tube, energy separation.

Introduction

The Ranque–Hilsch effect was first discovered by Ranque in 1933 and was later reintroduced and improved in efficiency by Hilsch in 1947. The Ranque–Hilsch vortex tube is a simple device with no moving parts that is capable of dividing a high pressure flow into two flows of different temperatures. The vortex motion in Ranque–Hilsch vortex tube is created by tangential injection of a compressed gas. When compressed gas is injected through one or more tangential nozzles into the tube, a strong vortex-like flow field is established, giving rise to a non-uniform temperature field. The gas closer to the axis is colder than the incoming feed gas, and the gas near the periphery of the tube is hotter. The difference in the temperature between the two gas streams is referred to as the "temperature separation" effect (Farouk *et al*. 2009). Despite its low efficiency, the simplicity, the low cost and the feature of having no moving parts which translates into no maintenance makes the vortex tube attractive for many low temperature applications (Avcı 2013). This apparatus is utilised in the vast industrial branches such as gas purifying, spot cooling and most commonly in refrigeration processes (Pourmahmoud *et al*. 2012).

The vortex tubes can be classified into two categories, one being the more ubiquitous counter-flow type and the other the uni-flow type, both of which can be observed in Figure 1. Working principle of the counter flow vortex tube

can be explained as follows. Compressible gas, which is tangentially introduced into the vortex tube from nozzles, starts to make a circular movement inside the vortex tube at high speeds, because of the cylindrical structure of the tube,
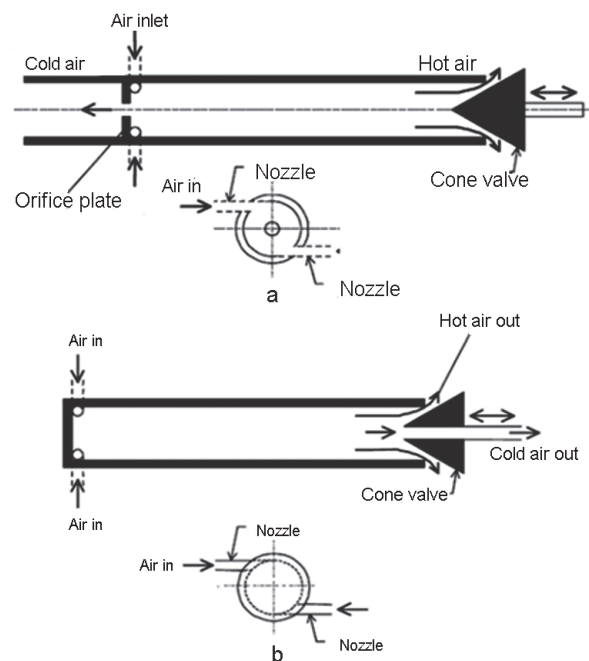

Fig. 1. Basic operation of vortex tubes: a) The Ranque–Hilsch standard vortex tube or counter-flow vortex tube; b) the uni-flow or parallel flow vortex tube (Eiamsa-ard, Promvonge 2008)
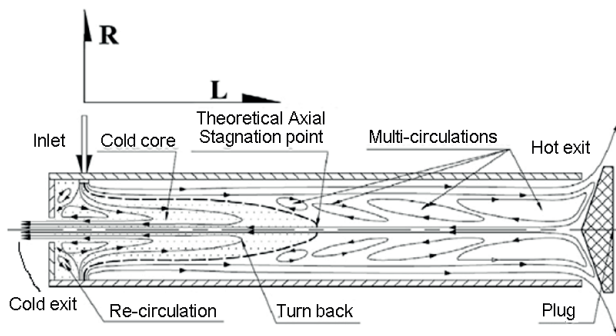
Fig. 2. Structure of a counter-flow vortex tube
(Xue *et al.* 2014)

depending on its inlet pressure and speed. As a result of this, pressure difference occurs between tube wall and tube centre because of the friction of the fluid circling at high speeds. Speed of the fluid near the tube wall is lower than the speed at the tube centre, because of the effects of wall friction (Bovand *et al.* 2014a). The principal flow vectors and vortex tube zones can be seen in Figure 2.

Challenges related to numerical analysis of vortex tubes

The CFD simulation of the flow field in a Ranque–Hilsch vortex tube presents challenging task because of its compressibility, turbulence and high swirl. Moreover, experimental measurement of the internal velocity and temperature fields necessary for the verification are often considered

very difficult to obtain (Secchiaroli *et al.* 2009). Because of the multitude of the obscuring and interacting physical features involved, the actual flow in a vortex tube is very complex, involving recirculation, swirl and temperature separation, the occurrence of which cannot be clearly described (Eiamsa-ard, Promvonge 2007). The most notable data on previous attempts cited from (Eiamsa-ard, Promvonge 2008) is presented in Table 1. The aforementioned data provides a retrospective look at the computational methods used for evaluating the Ranque–Hilsch effect.

As can be deducted from the reported data, the advancements in CFD as well as exponential increase in computational resources provided a basis for extensive numerical analysis of the said phenomenon. Nevertheless, the complex behaviour of the high velocity fluid within a vortex tube as well as lack of complete theory on the subject to this day limits the capabilities to achieve accurately numerical representation of the Ranque–Hilsch effect.

Recent development in vortex tube CFD analysis

Numerous researchers report sufficient correlation between experimental and numerical data even when conventional CFD methodology is used.

Sadeghiazad, Rafiee (2014) have performed experimental and computational studies seeking to determine the effect of using a truncated cone throttle valve and the number of nozzle intakes. Multiple measurements were

Table 1. Retrospective summary of numerical studies on vortex tubes (Eiamsa-ard, Promvonge 2008)

Investigators	Flow considered	Model	Method or software used	Results compared with measeurements
Linderstrom-Lang (1971)	Incompressible	Zero-equation	Stream-function	Poor but just trend
Schlenz (1982)	2D compressible	Zero-equation or mixing length	Galerkin's technique	Poor but qualitative trend
Amitani *et al.* (1983)	2D compressible	Neglected	Finite difference	Fair but assumptions in doubt
Borissov *et al.* (1993)	Incompressible	–	Velocity field induced by helical vortex	Qualitative agreement
Guston and Bakken (1999)	2D compressible	k–ε model	Fluent code	Fairly good
Frohlingsdorf and Unger (1999)	2D compressible	k–ε model	CFX code	Fairly good
Promvonge (1999)	2D compressible	ASM and k–ε model	Finite volume	Good
Behera *et al.* (2005)	3D compressible	k–ε and RNG k–ε model	Star-CD code	Fairly good
Aljuwayhel *et al.* (2005)	2D compressible	k–ε and RNG k–ε model	Fluent code	Fairly good
Skye *et al.* (2006)	2D compressible	k–ε and RNG k–ε model	Fluent code	Fairly good
Eiamsa-ard and Promvonge (2006)	2D compressible	ASM and k–ε model	Finite volume	Good

carried out with varied inlet air pressure, claiming that the CFD methods used for the geometry optimization can be effectively used to assess the vortex tube performance characteristics. The numerical investigation was carried out by full three-dimensional steady state compressible fluid CFD-simulation using Ansys Fluent, the employed model utilized the standard k–ε turbulence model to solve the flow equations. Good correlation between numerical and experimental results was established with deviation of the predicted and measured values for the cold temperature difference being less than or equal to 4.2%.

Alekhin *et al.* (2014) have conducted an extensive numerical investigation of a double-circuit Ranque–Hilsch vortex tube by applying and testing multiple turbulence models (k–ε, k–ε RNG, SAS-SST, RSM-LRR and LES) using a three-dimensional model for solving steady state compressible flow in Ansys CFX. The results show that LES turbulence model describes the vortex airflow in a better way with respect to the other turbulence models taken into account whereas SAS-SST turbulence model might be seen as a compromise between computational efficiency and accuracy. It was also determined that the standard k–ε and k–ε RNG models that are often used for numerical analysis of vortex tubes have the worst accuracy of temperature separation prediction.

Khait *et al.* (2014) have suggested the use of a semi-empirical turbulence model to increase the accuracy of vortex tube flow simulation. The proposed numerical model was shown to improve the qualitative and quantitative convergence of the simulated energy efficiency coefficient of the vortex tube with experimental data, and to increase the accuracy of the Ranque–Hilsch energy separation effect prediction. However, the calibration of empirical coefficients is necessary in order to demonstrate the applicability of the model for different vortex tubes working at different conditions. Meeting all requirements, the model is recommended for the simulation of two-dimensional or three-dimensional compressible flows. Using the standard k–ε turbulence model, the proposed numerical model requires the lower computational power as compared to LES and RSM models. This makes it possible to perform a large number of simulations and optimization computations.

Bovand *et al.* (2014a, 2014b) have performed a numerical investigation of the effect of curvature on the performance of vortex tubes for curvature angles of 0 (straight vortex tube) and 110°. The numerical study results were verified through comparison with experimental data. For the numerical study a three-dimensional model was used and the RNG k–ε turbulence model was utilized. Fluent program code was used to solve the governing equations.

The results show that the efficiency of straight vortex tube is higher than the curved vortex tube with angle of 110°. Authors claim that the actual values and CFD model results indicated that CFD model can be successfully used for the determination of heating and cooling performances of curvature effects on Ranque–Hilsch vortex tube.

Liu, X. and Liu, Z. (2014) have performed a thorough numerical research with the aim to reveal the flow characteristic and locate the dominant reasons for the temperature separation inside the vortex tube. A three-dimensional computational simulation was carried out to analyse the mechanism and display the internal flow inside the vortex tube. For this purpose, the CFD was first validated against the measured data gathered from the test rigs. Multiple turbulence models (RSM, Realizable k–ε, RNG k–ε, Standard k–ε, SST k–ω) were applied during the numerical study with the results documented and evaluated. Realizable k–ε turbulence model was chosen as the most rational approach in regards to accuracy and computational cost. The computational results came in close agreement with experimental data, therefore authors claim that the CFD model can be used as a power tool to not only optimize vortex tube design, but also reproduce the complex flow inside the vortex tube to gain detailed insight of the swirling behavior.

(Mohammadi, Farhadi 2014) have presented the results of the experimental and numerical study of separation efficiency in a vortex tube. The main objective of the research was to investigate the separation performances of a vortex tube (VT) for a hydrocarbon mixture. A two-dimensional model was used for the numerical study that used the standard k–ε turbulence model. The governing equations of the steady state simulation for compressible flow were solved using the Fluent code. The working fluid had the characteristics of a gas mixture with relatively small molecular weight difference between components. The authors claim that the results of CFD simulation come in good agreement with the experimental data.

Bej, Sinhamahapatra (2014) published a paper that presents results of a series of numerical simulation carried out using standard k–ε turbulence model focusing on exergy analysis on second stage of the vortex tube for different cold fractions. The numerical investigation was set up to correspond the experimental data in order to ensure the means to verify the numerical model validity. A two-dimensional axisymmetric model was used for the numerical analysis in Ansys Fluent. Authors claim that the model predictions compare favourably with experimental data. Hence, it can be confidently used to further investigate the performance of multiple vortex tubes used in different combinations.

Kandil, Abdelghany (2015) have presented the results of a computational study of the Ranque–Hilsch vortex tube. The study was performed in correlation with the experimental data presented in a separate paper with a goal of optimizing the performance of a vortex tube using multiple approaches. The study was carried out with an axisymmetric model using the Ansys Fluent software. The results of the CFD model of the vortex tube using the standard k–ε turbulence model showed the best agreement with the experimental results. The optimization was performed in regards to the cold orifice to tube diameter ratio as well as the effect of the additional cooling fins located on the wall of the vortex tube. The results to showed good agreement with the experimental measurements, as claimed by the authors.

Thakare, Parekh (2014) have published a paper aimed towards presenting the CFD study of vortex tube carried out to gain an understanding about influence of thermophysical properties such as thermal diffusivity, Prandtl number, specific gas constant and thermal conductivity of different gases and turbulence models on its performance. The two-dimensional axisymmetric model is claimed to be advantageous because it reduces the computational efforts required; still, it preserves the three-dimensional features of the computational domain. The energy separation has been observed for eight different gases as working fluid. Numerical simulation of vortex tube was attempted in order to understand the effect of different working fluids and different turbulence models on energy separation inside the vortex tube. In order to understand the complex nature of highly compressible, turbulent and swirling flow within the vortex tube, different turbulent models, namely, one equation Spalart–Allmaras, two equations Standard k–ε and Standard k–ω model were used. Results of the numerical simulation are first compared against available experimental data and corresponding CFD data. It had been observed that the deviation in prediction of total temperature of air in cold region as well other parameters of flow physics in case of Standard k–ε turbulence model used with QUICK scheme is within the acceptance range, which makes it suitable for the purpose of design and analysis of vortex tube.

Secchiaroli et al. (2009) have published the results of the numerical simulations of the internal flow in a commercial model of a Ranque–Hilsch vortex tube operating in jet impingement. Simulation of the turbulent, compressible, high swirling flow was performed by implementing both RANS and LES techniques. RANS simulations were performed on an axial-symmetric computational domain, while LES were performed on a three-dimensional computational model in order to avoid symmetry imposing in turbulent

structures. All of the calculations were performed using Fluent. The effect of different turbulence closure models have been tested in RANS simulations using a first order closure RNG k–ε and, for what is claimed to be the first time in this kind of flow, a second order RSM (Reynolds Stress differential Model) closure. RANS computations have been executed on an axis-symmetric two-dimensional mesh and results have been compared with LES ones, obtained over a three-dimensional computational grid. Results in different sections of the tube show significant differences in the velocity profiles, temperature profiles and secondary vortex structures, varying turbulence model.

Pourmahmoud et al. (2012) present the result of the investigation of the effect of helical nozzles on both energy separation and refrigeration phenomena in the Ranque–Hilsch Vortex by implementing the methods numerical analysis. The standard k–ε turbulence model was introduced to the governing equations for analysing highly rotating complex flow field. The CFD models were based on the separately obtained experimental results. The considered vortex tube had been equipped with six straight nozzles and the system was investigated with respect to various geometrical parameters such as tube length. In all of three-dimensional CFD models, compressible and fully turbulent fluid flow inside the vortex tube were simulated in order to realize the flow patterns as well as heating or cooling functions of the system. Some of numerical results were confirmed by the experimental data of vortex tube, which shows good agreement with them.

Dutta et al. (2010) have performed a comparison of turbulence models using an axisymmetric computational fluid dynamics model with the objective to select an appropriate turbulence model for the simulation of the flow phenomena in a vortex tube with optimum computational expense. Numerical methods were used to compare the influence of different Reynolds Averaged Navier–Stokes (RANS) based turbulence models in predicting the temperature separation in a Ranque–Hilsch vortex tube. The standard k–ε, RNG k–ε, standard k–ω and SST k–ω turbulence models were used in the presented study. The performance curves (hot and cold outlet temperatures versus hot outlet mass fraction) obtained by using these turbulence models were compared with the experimental results. The performance analysis showed that among all the turbulence models investigated in this study, temperature separation predicted by the standard k–ε turbulence model is closer to the experimental results. However, it is noted that none of the turbulence models could fully resolve the intensely turbulent swirling flow inside the vortex tube.

Table 2. Numerical methods for modelling of Ranque–Hilsch effects suggested by the researchers

Investigators	Software used	Geometric model used	Suggested computational approach
Secchiaroli *et al.* (2009)	Fluent	Two-dimensional axisymmetric	RNG *k*–ε
Dutta *et al.* (2010)	Fluent	Two-dimensional axisymmetric	Standard *k*–ε
Pourmahmoud *et al.* (2012)	Fluent	Full three-dimensional	Standard *k*–ε
Bovand *et al.* (2014a)	Fluent	Full three-dimensional	RNG *k*–ε
Khait *et al.* (2014)	Fluent	Full three-dimensional	standard *k*–ε (semi-empirical model with additional calibration)
Rafiee, Sadeghiazad (2014)	Fluent	Full three-dimensional	Standard *k*–ε
Liu, X., Liu, Z. (2014)	Fluent	Full three-dimensional	Realizable *k*–ε
Mohammadi, Farhadi (2014)	Fluent	Two-dimensional	Standard *k*–ε
Bej, Sinhamahapatra (2014)	Fluent	Two-dimensional axisymmetric	Standard *k*–ε
Thakare, Parekh (2014)	Fluent	Two-dimensional axisymmetric	Standard *k*–ε with QUICK scheme
Alekhin *et al.* (2014)	CFX	Full three-dimensional	SAS-SST
Kandil, Abdelghany (2015)	Fluent	Two-dimensional axisymmetric	Standard *k*–ε

The presented information has been summarized and presented in Table 2.

The data within the table above shows general tendencies in the approach towards numerical evaluation of vortex tube effects. Two-dimensional axisymmetric and full three-dimensional models are both commonly used with similar results and the standard *k*–ε turbulence model appears to be the most commonly implemented for such cases. While RNG *k*–ε turbulence model is capable of accounting for the effects of smaller scales of motion within the turbulent flow, it is often noted that for high vorticity flows the use of RNG *k*–ε model provides marginal improvements in model accuracy.

When applying standard *k*–ε turbulence model the following transport equations for turbulent kinetic energy *k* and turbulent dissipation ε are implemented (Launder, Spalding 1972):

$$\frac{\partial}{\partial\tau}(\rho\kappa)+\frac{\partial}{\partial\xi_\iota}(\rho\kappa\upsilon_\iota)=\frac{\partial}{\partial\xi_\varphi}\left[\left(\mu+\frac{\mu_\tau}{\sigma_\kappa}\right)\frac{\partial\kappa}{\partial\xi_\varphi}\right]+$$
$$\Gamma_\kappa+\Gamma_\beta-\rho\varepsilon-\Psi_M+\Sigma_\kappa;$$

$$\frac{\partial}{\partial\tau}(\rho\varepsilon)+\frac{\partial}{\partial\xi_\iota}(\rho\varepsilon\upsilon_\iota)=\frac{\partial}{\partial\xi_\varphi}\left[\left(\mu+\frac{\mu_\tau}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial\xi_\varphi}\right]+$$
$$X_{1\varepsilon}\frac{\varepsilon}{\kappa}(\Gamma_\kappa+X_{3\varepsilon}\Gamma_\beta)-X_{2\varepsilon}\rho\frac{\varepsilon^2}{\kappa}+\Sigma_\varepsilon;$$

where: u_i – velocity component for corresponding direction, E_{ij} – component for rate of deformation, G_k – generation of turbulence kinetic energy due to the mean velocity gradients, G_b – generation of turbulence kinetic energy due to buoyancy, Y_M – contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, σ_k, σ_ε – turbulent Prandtl numbers for *k* and ε, μ_t – eddy (tur-

bulent) viscosity, determined from the following equation:
$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon};\ C_{1\varepsilon},C_{2\varepsilon},C_{3\varepsilon},C_\mu\ ,\ \text{are constants.}$$

Conclusions

It is apparent that numerical investigation of the fluid dynamics within the vortex tube still presents a challenge when it comes to accurate representation of complex swirling fluid flow. That, however, is partially compensated for due to the higher availability of computational resources, which, combined with more time consuming yet more accurate CFD set-ups can result in better correlation between experimental and simulated data. Nevertheless, lack of reliable theoretical background related to Ranque–Hilsch vortex effect limits the ability for analytical evaluation of the gas flow characteristics.

The overview of the most recent research data obtained from the numerical studies yields the following conclusions:

1. While the purely two-dimensional model analysis is less common due to being relatively inaccurate, both three-dimensional and two-dimensional axisymmetric models are commonly used with axisymmetric models benefitting from being less resource demanding and the full three dimensional models having higher overall accuracy and flexibility.

2. It was concluded on multiple occasions that no one of the readily available turbulence models can be implemented to accurately represent the complex fluid dynamics (such as secondary swirls) within the vortex tube. However, the sufficiently accurate generalised representation of key parameter distribution can be achieved with relative consistency without a need

for additional model calibration. For such cases, the standard k–ε turbulence model is often implemented with success and can be considered a go-to model for general evaluation of a vortex tube performance, although at a cost of calculation productivity better accuracy can be achieved with more complex models, such as RNG k–ε. Regardless, a more accurate turbulence model is still required in order to fully simulate the flow within the vortex tube.

References

Alekhin, A.; Bianco, V.; Khait, A.; Noskov, A. 2014. Numerical investigation of a double-circuit Ranque–Hilsch vortex tube, *International Journal of Thermal Sciences* 89: 272–282. http://dx.doi.org/10.1016/j.ijthermalsci.2014.11.012

Avcı, M. 2013. The effects of nozzle aspect ratio and nozzle number on the performance of the Ranque–Hilsch vortex tube, *Applied Thermal Engineering* 50(1): 302–308. http://dx.doi.org/10.1016/j.applthermaleng.2012.06.048

Bej, N.; Sinhamahapatra, K. P. 2014. Exergy analysis of a hot cascade type Ranque-Hilsch vortex tube using turbulence model, *International Journal of Refrigeration* 45: 13–24. http://dx.doi.org/10.1016/j.ijrefrig.2014.05.020

Bovand, M.; Valipour, M. S.; Dincer, K.; Tamayold, A. 2014a. Numerical analysis of the curvature effects on Ranque–Hilsch vortex tube refrigerators, *Applied Thermal Engineering* 65(1–2): 176–183. http://dx.doi.org/10.1016/j.applthermaleng.2013.11.045

Bovand, M.; Valipour, M. S.; Dincer, K.; Tamayold, A. 2014b. Numerical analysis for curved vortex tube optimization, *International Communications in Heat and Mass Transfer* 50: 98–107. http://dx.doi.org/10.1016/j.icheatmasstransfer.2013.11.012

Dutta, T.; Sinhamahapatra, K. P.; Bandyopdhyay, S. S. 2010. Comparison of different turbulence models in predicting the temperature separation in a Ranque–Hilsch vortex tube, *International Journal of Refrigeration* 33(4): 783–792. http://dx.doi.org/10.1016/j.ijrefrig.2009.12.014

Eiamsa-ard, S.; Promvonge, P. 2007. Numerical investigation of the thermal separation in a Ranque–Hilsch vortex tube, *International Journal of Heat and Mass Transfer* 50(5–6): 821–832. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2006.08.018

Eiamsa-ard, S.; Promvonge, P. 2008. Review of Ranque–Hilsch effects in vortex tubes, *Renewable and Sustainable Energy Reviews* 12(7): 1822–1842. http://dx.doi.org/10.1016/j.rser.2007.03.006

Farouk, T.; Farouk, B.; Gutsol, A. 2009. Simulation of gas species and temperature separation in the counter-flow Ranque–Hilsch vortex tube using the large eddy simulation technique, *International Journal of Heat and Mass Transfer* 52(13–14): 3320–3333. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2009.01.016

Kandil, H. A.; Abdelghany, S. T. 2015. Computational investigation of different effects on the performance of the Ranque–Hilsch vortex tube, *Energy* [in Press, Corrected Proof]. http://dx.doi.org/10.1016/j.energy.2015.02.089

Khait, A.; Noskov, A.; Lovtsov, A.; Alekhin, V. 2014. Semi-empirical turbulence model for numerical simulation of swirled compressible flows observed in Ranque–Hilsch vortex tube, *International Journal of Refrigeration* 48: 132–141. http://dx.doi.org/10.1016/j.ijrefrig.2014.09.006

Liu, X.; Liu, Z. 2014. Investigation of the energy separation effect and flow mechanism inside a vortex tube, *Applied Thermal Engineering* 67(1–2): 494–506. http://dx.doi.org/10.1016/j.applthermaleng.2014.03.071

Mohammadi, S.; Farhadi, F. 2014. Experimental and numerical study of the gas–gas separation efficiency in a Ranque–Hilsch vortex tube, *Separation and Purification Technology* 138: 177–185. http://dx.doi.org/10.1016/j.seppur.2014.10.022

Pourmahmoud, N.; Hassanzadeh, A.; Moutaby, O. 2012. Numerical analysis of the effect of helical nozzles gap on the cooling capacity of Ranque–Hilsch vortex tube, *International Journal of Refrigeration* 35(5): 1473–1483. http://dx.doi.org/10.1016/j.ijrefrig.2012.03.019

Sadeghiazad, M. M.; Rafiee, S. E. 2014. Three-dimensional and experimental investigation on the effect of cone length of throttle valve on thermal performance of a vortex tube using *k*–ε turbulence model, *Applied Thermal Engineering* 66(1–2): 65–74. http://dx.doi.org/10.1016/j.applthermaleng.2014.01.073

Secchiaroli, A.; Ricci, R.; Montelpare, S.; D'Alessandro, V. 2009. Numerical simulation of turbulent flow in a Ranque–Hilsch vortex tube, *International Journal of Heat and Mass Transfer* 52(23–24): 5496–5511. http://dx.doi.org/10.2495/cmem090321

Thakare, H. R.; Parekh, A. D. 2014. CFD analysis of energy separation of vortex tube employing different gases, turbulence models and discretisation schemes, *International Journal of Heat and Mass Transfer* 78: 360–370. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.06.083

Xue, Y.; Arjomandi, M.; Kelso, R. 2014. Energy analysis within a vortex tube, *Experimental Thermal and Fluid Science* 52: 139–145. http://dx.doi.org/10.1016/j.expthermflusci.2013.09.004

SKAITINIŲ METODŲ, SKIRTŲ NAGRINĖTI RANKE-HILČ EFEKTĄ SŪKURINIAME VAMZDYJE, ANALIZĖ

V. Nikitin, P. Bogdevičius, M. Bogdevičius

Santrauka

Straipsnyje apibūdinti moksliniai darbai, kuriuose nagrinėjamas Ranke-Hilč (Ranque–Hilsch) sūkurinis dujų judėjimas, jo skaitiniai sprendimo metodai. Pagal skaitinius metodus gauti rezultatai lyginami su eksperimentiniais tyrimo rezultatais. Pateikti Ranke-Hilč hidrodinaminio efekto privalumai ir trūkumai. Pateiktas *k*–ε turbulentinį dujų judėjimą aprašantis matematinis modelis.

Reikšminiai žodžiai: Ranke-Hilč efektas, sūkurinis vamzdis, energijos atskyrimas, skaitiniai metodai, analizė.