Numerical Analysis of the Flow Phenomena Inside the Vortex Tube with Different Turbulence Models

Osama Ali Ahmed Awan, Robert Sager, Thomas-Sebastian Gier, Manfred Wirsum and Ekachai Juntasaro

June 11, 2024

Numerical Analysis of the Flow Phenomena inside the Vortex Tube with Different Turbulence Models

Osama Ali Ahmed Awan[1,2][0000-0001-7735-8631], Robert Sager[2][0009-0002-5762-1257], Thomas Gier[2], Manfred Wirsum[2], Ekachai Juntasaro[1][0000-0001-7986-5856]

[1] The Sirindhorn International Thai-German Graduate School of Engineering, KMUTNB, Bangkok, Thailand

[2] Institute of Power Plant Technology, Steam and Gas Turbines, RWTH Aachen University, Aachen, Germany

Abstract. To accelerate progress towards sustainability, a fundamental shift from conventional to renewable energy sources is necessary, but insufficient on its own. Improving the efficiency of existing energy systems is equally important. One promising avenue for achieving this goal involves integrating previously overlooked devices, such as the vortex tube, into the current systems. Thus, there is a growing emphasis on overcoming its limitations, including limited development and suboptimal efficiency, to unlock its potential in diverse applications. The current focus is primarily on computational research rather than experimental approaches due to the advantages in terms of cost and time. Nevertheless, computational studies present their own set of challenges, with two prominent hurdles being the attainment of acceptable mesh and the selection of an appropriate turbulence model. This study aims to address these challenges. An acceptable mesh has been obtained by optimization, which involves refinement of mesh at the inlet and outlet regions, followed by a comprehensive assessment of mesh independence at each stage. Furthermore, achieving a y+ value of 1 in the most important regions of the vortex tube is crucial, particularly when employing low Re turbulence models, to accurately predict boundary layer behavior. Building on the mesh studies, the performance of different turbulence models is evaluated with reference data. Among the considered models, the standard k-ε turbulence model has the best performance, aligning closely with experimental results for almost the same geometric setup. As a result, the standard k-ε turbulence model is selected for further numerical investigations.

Keywords: Ranque-Hilsch vortex tube, CFD, heat transfer, turbulence modelling, temperature separation effect.

1 Introduction

Energy is the backbone for urbanization and technological advancement of the world. However, the current challenge for the world is to produce sustainable energy to address environmental concerns. At the same time, there is a need for greater efficiency and a reduction in energy waste. This can be achieved by inclusion of new devices into existing systems. One such device is the vortex tube, which is a simple thermal device with no moving parts. The first idea of a vortex tube was given by a French scientist Georges J.

Ranque in 1933. More than ten years later, in 1947, the German physicist Rudolf Hilsch gave an adequate explanation of the temperature separation effect (TSE) inside the vortex tube. For this reason, the vortex tube is commonly referred to as the Ranque-Hilsch vortex tube (RHVT) [1]. The RHVT is an expansion device that separates the incoming high pressure stream into hot and cold streams. The resulting hot stream has a higher temperature than the incoming high pressure stream, while the subsequent cold stream has a lower temperature than the incoming high pressure stream. Although the RHVT holds great potential for different applications, it has certain limitations such as suboptimal efficiency and limited development. Therefore, much research is still being carried out to find the optimum geometric and operating parameters to exploit the advantages of the RHVT.

Primarily, there are three different types of research in the field of vortex tubes: experimental, numerical, and analytical. Computational research has the advantages of flexibility, cost-effectiveness, time-efficiency, and adaptability over the experimental and analytical approaches. Therefore, akin to other fields, more emphasis has been put on computational research of the RHVT. While several researchers have explored the interesting advances and flow phenomena within the RHVT, the answer to one question remains vague: Which is the best turbulence model for the numerical analysis of the RHVT? Different authors have different opinions on this, which makes it difficult to answer. Table 1 shows the turbulence models considered and the best performing ones by various researchers in the recent past. The choice of a suitable turbulence model is of significant importance for the accurate numerical prediction of the flow within the RHVT. Therefore, the authors of this paper have attempted to address this issue by considering Standard k-ε (k-ε), Standard k-ω (k-ω), Shear Stress Transport k-ω (SST k-ω), and Renormalization Group k-ε (RNG k-ε) in the current study. Several additional turbulence models, namely Scale Adaptive Simulation Shear Stress Transport (SAS SST), Spalart-Allmaras (SA), Reynolds Stress Equation Model (RSM), and Realizable k-ε, have been considered by various authors. However, these were not included in the current study as they did not perform well based on the available literature.

Table 1: Different turbulence models used in only recent research studies.

Research Paper	Investigated Turbulence Models								Best Turbulence Model
	k-ε	k-ω	SST k-ω	SAS SST	SA	RNG k-ε	RSM	Realizable k-ε	
Singh et al. [2]	✓	✓	✓	X	X	✓	X	X	k-ε
Dyck et al. [3]	✓	✓	✓	✓	X	X	X	X	SAS SST
Lagrandeur et al. [4]	✓	X	✓	X	X	X	X	X	k-ω SST
Sadeghiseraji et al. [5]	✓	✓	✓	X	X	X	X	X	k-ε
Hu et al. [6]	✓	X	✓	X	✓	X	X	✓	Realizable k-ε
Bazgir et al. [7]	✓	X	✓	X	X	✓	X	✓	RNG k-ε
Chýlek et al. [8]	✓	✓	✓	X	X	X	✓	✓	k-ε

The primary objective of this work is to determine which turbulence model produces the most accurate TSE when compared with reference data. This is accomplished through numerical simulations conducted on a fully three-dimensional model of the RHVT. The details of this research study unfold across the following sections of this paper. Section 2 outlines the geometry under consideration and the reference geometry. It also discusses how to obtain a suitable mesh. The following section 3 presents the results of the research. Finally, the last section encapsulates the conclusions and outlines the future objectives of this research.

2 Numerical Study

2.1 Geometry

In some of the previous studies, a 2D geometric model was considered because its result had a close qualitative resemblance to the 3D analysis results [4]. However, quantitively, it lacks accuracy to predict accurate results. Therefore, a 3D model was selected in this study. The geometric model of the considered RHVT model with dimensions is shown in Fig. 1. The current geometry is very similar to the geometry of Rafiee & Sadeghiazad [9] , and Kaya et al. [10]. Table 2 shows the comparison between the present geometry and the reference geometry.

Table 2: Comparison of geometric parameters of present model with reference models.

RHVT Parameter		This Study	Rafiee & Sadeghiazad [9]	Kaya et al. [10]
Name	Unit			
Length	mm	264	250	100
Diameter	mm	17.6	18	7
L/D	N/A	15	13.89	14.28
Nozzle number	N/A	2	2	2
Nozzle diameter	mm	2.5	1.7	3.38

2.2 Mesh Study

ANSYS 2022 R1 was used for the current numerical study. A 3D CFD mesh was generated on ANSYS meshing. The mesh consists of tetrahedral elements with a size of 0.75 mm. Inflation layers are augmented along the wall to refine the spatial resolution near the boundary walls. The accurate representation of these boundary layers is based on the correct incorporation of these inflation layers. This type of meshing results in robustness of the computational study and leads to more accurate simulations of the problem. As a next step, a grid-independent study was carried out, which is very important to understand how the simulation results change when the grid or mesh resolution is changed. This has a direct influence on the numerical accuracy and optimization of the computational effort. Therefore, the focus of the mesh study in this paper lies on refining the mesh at the inlet and outlet regions of the vortex tube until consistent results are obtained. The sphere of influence function of ANSYS meshing is used to specify the element size. The mesh refinement is carried out until the results remain unchanged. The final element sizes at the inlet, cold outlet, and hot outlet are 0.288 mm, 0.35 mm, and 0.4 mm, respectively. The selected mesh has 8.19 million elements and 2.65 million nodes. Fig. 2 shows the mesh of the vortex tube model used in this study. When considering the low Reynolds number (low Re) turbulence models, the y-plus (y^+) value plays a very crucial aspect of the numerical study. It is the non-dimensional distance of the first cell from the wall and it defines the quality of the mesh in the boundary layer. A y^+ study refers to changing the first layer height of the cell near the wall until a required y^+ value is obtained. In this work, most parts of the RHVT have a y^+ of 1 because this is required for low-Re models. However, the choice of turbulence model does not depend on the y^+ value of the mesh and we do not want to confuse the discussion of the y^+ value with the choice of turbulence model. Therefore, the same mesh is used to evaluate all turbulence models.

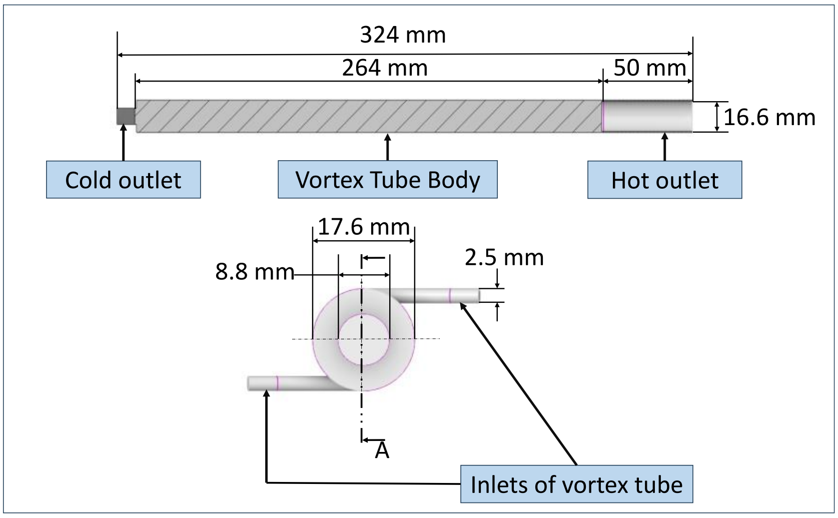

Fig. 1. Dimensions of the vortex tube model considered in this paper.

2.3 Boundary Conditions and Numerical Solution

The hot and cold outlets are pressure type outlets with a value of 1.01 bar, while at the inlet, relative pressure of 2.5 bar is specified. The static temperature at the inlet is set to 294.2 K. Adiabatic no-slip conditions are applied to the inner walls of the vortex tube. These values are the same as those used in the research work of Rafiee & Sadeghiazad [9]. Furthermore, the steady-state simulations are carried out on ANSYS CFX. Air as an ideal gas is chosen as the working fluid. The convergence limit for the residuals is set to 1e-6. In this study, k-ε, k-ω, SST k-ω, and RNG k-ε are considered based on the results of previous research studies as indicated in Table 1.

3 Results and Discussion

This section discusses the results obtained from different turbulence models. The experimental results of Rafiee & Sadeghiazad [9] are used to validate the results of this study. The present comparison is mainly based on the temperature distribution inside the RHVT. Fig. 3 shows the total temperature contours, at the mid-plane of the vortex tube, obtained by different turbulence models. It was observed that all turbulence models were able to generate hot and cold sections inside the RHVT. It can be concluded from the results that the choice of turbulence model does not qualitatively affect temperature separation. In addition, it may be noted that the regions closer to the wall have higher temperatures than the region at the core of the RHVT. Moreover, it may be noted that the values of the maximum and minimum temperatures are substantially influenced by the choice of the turbulence model. If the local contours of the total temperature, at the mid-plane, are qualitatively compared with the experimental results of Rafiee & Sadeghiazad [9], the k-ε and SST k-ω turbulence models have the closest resemblance.

The obtained results are compared with the reference data to find out which turbulence model predicted results closest to the reference data. To compare the results obtained from different turbulence models, the percentage deviation of the cold outlet temperature difference ($\triangle T_c$) of numerical results from experimental results is used. Table 3 shows the average deviation of the results obtained by different turbulence models from that of reference results. Fig. 4 represents the comparison of the experimental and computational results of Rafiee & Sadeghiazad [9] with the results of the present study.

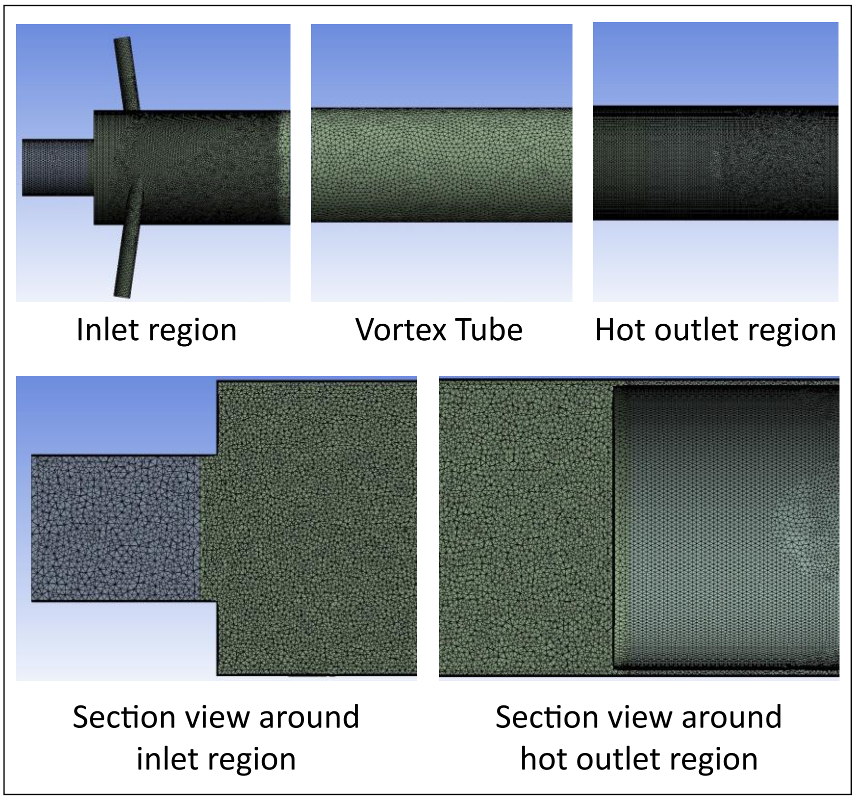

Fig. 2. Regions of mesh refinement and mesh at different regions of vortex tube.

It can be seen that the results of the k-ε turbulence model have close resemblance with the reference results. This is followed by the k-ω turbulence model which also shows good agreement with the reference results. While there is a significant difference in the results of the other two models when compared with reference results. The close resemblance of the results of the k-ε model to the reference data supports the conclusions drawn in previous studies on the same topic, as demonstrated in Table 1.

In addition, Rafiee & Sadeghiazad [9] have used the Reynolds stress model (RSM) model in their study. They were able to capture almost similar results to the experimental results. However, the RSM model is expensive in terms of computational cost. Nevertheless, if the computation cost allows, then the RSM model might be a more suitable choice for numerical investigations of the RHVT. Furthermore, the hot end temperature difference is also an important parameter to evaluate which turbulence model is the best. However, as Rafiee & Sadeghiazad [9] have only considered the cold end temperature difference to evaluate the performance of the vortex tube. This is why only the cold end temperature differences are available for validation. Nevertheless, we will consider the hot end temperature difference as an evaluation parameter in our subsequent numerical investigations.

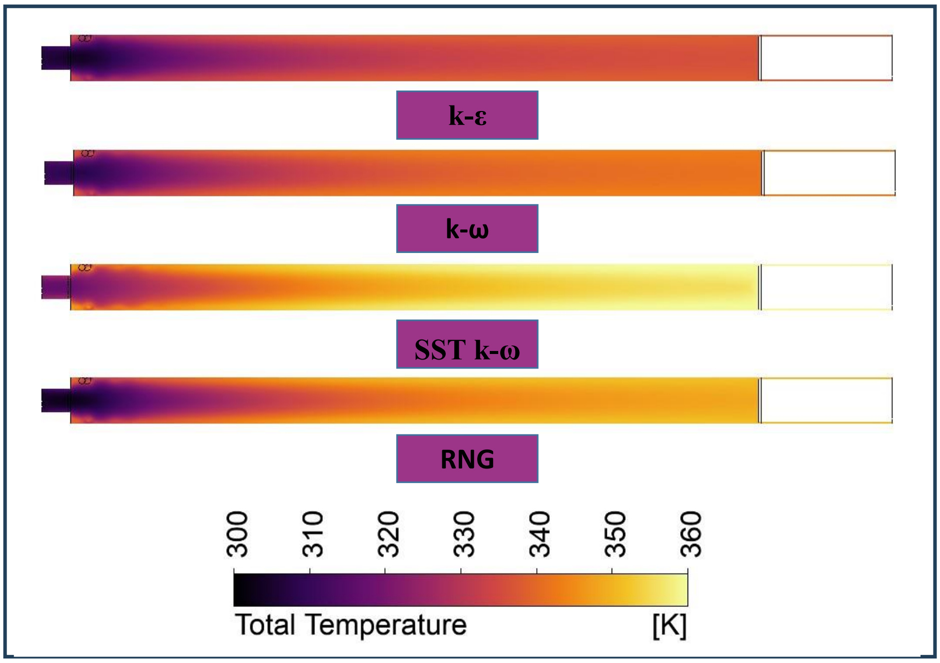

Fig. 3. The obtained local contours of total temperature by different turbulence models at the mid-plane of the RHVT.

Table 3: Deviation of results of present from reference results.

	Turbulence Model			
	k-ε	k-ω	SST k-ω	RNG k-ε
Average % ($\triangle T_c$)	7.18	7.87	22.74	35.7

Moreover, when the velocity vectors were analyzed from the numerical results, a recirculation zone was identified between the hot and cold flows inside the RHVT. Fig. 5 visually depicts this intriguing recirculation zone. To enhance the accuracy of predicted results, it is proposed to refine the mesh around this area by dividing the RHVT into a pair of concentric cylinders. This will be investigated in our next numerical investigations of the RHVT.

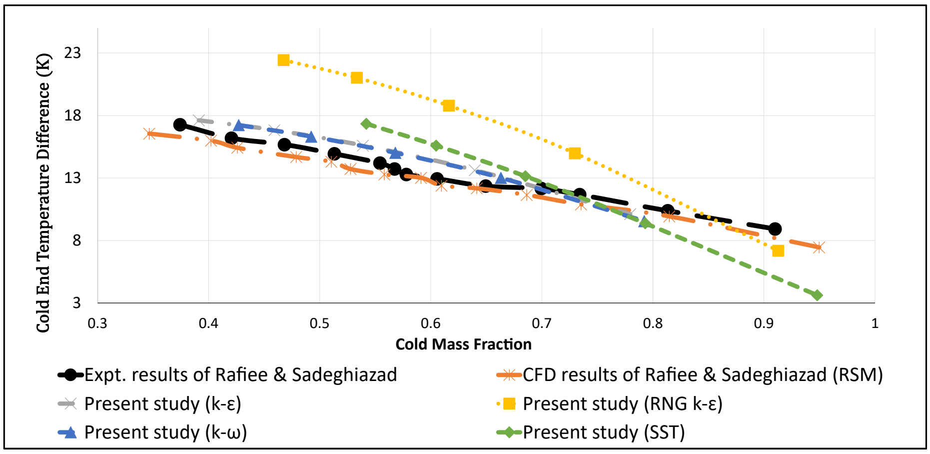

Fig. 4. Comparison of results at different cold mass fractions.

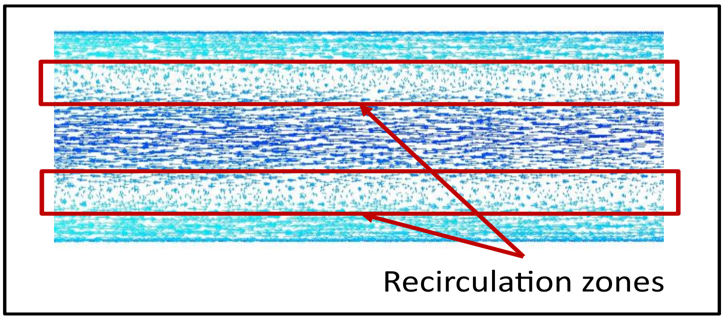

Fig. 5. Recirculation zones visible in plot of velocity vectors at the mid-plane of the RHVT.

4 Conclusions

In this study, a 3D numerical analysis was performed to find the best turbulence model that accurately reflects the real vortex tube behavior. Four different turbulence models are used for the investigation, namely k-ε, k-ω, SST k-ω, and RNG k-ε. The numerical results presented in this study are validated and compared with the experimental data of Rafiee & Sadeghiazad [9]. The geometry in this study and the geometry of Rafiee & Sadeghiazad [9] are almost the same; however, there are differences between them. In terms of the results, all turbulence models were able to qualitatively capture the temperature separation effect. However, the magnitudes of cold and hot temperatures were different for each turbulence model. No turbulence model was able to predict the exact temperature magnitudes when compared to the reference data. The results of k-ε and SST k-ω models were close to the reference data when considering the total temperature contours. However, the k-ε model exhibits the best agreement with the reference data when considering the cold exit temperature difference at different cold mass fractions. The average difference between the results for the k-ε turbulence model is 7.18%. However, it is worth noting that geometry of the RHVT also exhibits a small difference, which could be a contributing factor to the variation in results. Nevertheless, the obtained results for the k-ε turbulence model are almost consistent with the conclusions of previous studies. Hence, it will be used for our future numerical studies. Moreover, a recirculation zone was identified between the hot and cold flows inside the RHVT. The mesh around the recirculation zone will be further refined in the future. This may lead to a more accurate prediction of results.

References

1. Ambedkar, P., Dutta, T.: Analysis of various separation characteristics of Ranque-Hilsch Vortex Tube and its applications-A Review. Chemical Engineering Research and Design, vol. (2023)
2. Singh, R.K., Pramanick, A.K., Rana, S.C.: Computational study of temperature separation for a three-dimensional vortex tube with cold exit diameter and nozzle number variation. International Journal of Ambient Energy, vol. 43, 7046–7060 (2022)
3. Dyck, N.J., Parker, M.J., Straatman, A.G.: The impact of boundary treatment and turbulence model on CFD simulations of the Ranque-Hilsch vortex tube. International journal of refrigeration, vol. 141, 158–172 (2022)
4. Lagrandeur, J., Croquer, S., Poncet, S., Sorin, M.: A 2D numerical benchmark of an air Ranque-Hilsch vortex tube based on a fractional factorial design. International communications in heat and mass transfer, vol. 125, 105310 (2021)
5. Sadeghiseraji, J., Moradicheghamahi, J., Sedaghatkish, A.: Investigation of a vortex tube using three different RANS-based turbulence models. Journal of Thermal Analysis and Calorimetry, vol. 143, 4039–4056 (2021)
6. Hu, Z., Li, R., Yang, X., Yang, M., Zhang, Y.: Numerical simulation for three-dimensional flow in a vortex tube with different turbulence models. Numerical Heat Transfer, Part A: Applications, vol. 77, 121–133 (2020)
7. Bazgir, A., Khosravi-Nikou, M., Heydari, A.: Numerical CFD analysis and experimental investigation of the geometric performance parameter influences on the counter-flow Ranque-Hilsch vortex tube (C-RHVT) by using optimized turbulence model. Heat and mass transfer, vol. 55, 2559–2591 (2019)
8. Vortex Tube: A Comparison of Experimental and CFD Analysis Featuring Different RANS Models, vol. . EDP Sciences (2018)
9. Rafiee, S.E., Sadeghiazad, M.M.: 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, vol. 66, 65–74 (2014)
10. Kaya, H., Günver, F., Kirmaci, V.: Experimental investigation of thermal performance of parallel connected vortex tubes with various nozzle materials. Applied thermal engineering, vol. 136, 287–292 (2018)