Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 88-95
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

A Study of the Performance of the Local Exhaust Ventilation System at Welding Station using Computational Fluid Dynamics (CFD) Simulation

Ahmad Redzuan Yahya[1], Hanis Zakaria[1*]

[1]Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.011
Received 02 Aug. 2021; Accepted 27 Nov. 2021; Available online 25 December 2021

Abstract: Local exhaust ventilation (LEV) system is an engineering control widely used in any welding workplace to control hazardous airborne contaminant exposure to an acceptable limit. Hence, this study is performed to analyse the performance of the LEV system by studying the air distribution using computational fluid dynamics (CFD) simulation. The components involved in this study are 14 capture hoods, 14 extraction arms and a ducting system. ANSYS 19.1 CFX software and two turbulence models which are k-ε and shear stress transport (SST), are used in this simulation study. The material assumption in this study is the air properties at 30°C considering the temperature of the LEV system where the welding stations operates in. The findings from the study show the velocity readings obtained by the simulation study are higher compare to the experimental study. By comparing the findings for each turbulent model with previous experimental results, it can be deduced that the k-ε and SST turbulence model of 0.05m element size can produce the minimum average percentage difference of 32.82 % and 33.82 % respectively. The percentage differences between the turbulence models and experimental study are notable when there is the 90° duct junction and sudden difference in the ducting diameter between ducting point 5 and 6. Therefore, a further decrease in the element size with a more refined mesh model needs to be dealt with to select the best element choice for the simulation study.

Keywords: Local Exhaust Ventilation, Computational Fluid Dynamics , K-Epsilon, Shear Stress Transport

**Corresponding author: hanisz@uthm.edu.my*
2021 UTHM Publisher. All right reserved.

1. Introduction

The issues on workers' health and safety must be taken as a priority in every industry. As there are high usage of chemical substances in many industries, controlling hazardous chemicals in the workplace can help in reducing workers' exposure and minimising the risk associated with hazardous chemicals. The ventilation system is a system to control air contaminants by removing them from the workplace. Workers are exposed to air contaminants from many sources and it can affect their health. Therefore, the proper ventilation system is needed to maintain good air quality in the workplace.

Local exhaust ventilation (LEV) system is a ventilation system widely used in many industries to control air contaminants' exposure such as fumes, dust and chemical vapors. [1] LEV system should be installed based on the guidelines in Occupational Safety and Health Regulation 2008 to ensure air ventilation of the system can perform very well. Maintenance of the LEV system must be carried out regularly to ensure the overall performance of the system is good and can control the air contaminants level under allowable limits.

The welding industry is one of the industries that cause many adverse occupational health effects. Respiratory system problem, bronchitis and pulmonary infections are some of the health effects by exposing to fumes and gases produced during the welding process. [2] Exposure to manganese-containing fumes during welding had a significant effect on the performance of neurobehavioral system in the human body.[3] A study stated that after installing the LEV system, the welding fume exposed to the welder did not surpass the standard limit of fume from various metal. [4] Implementing the LEV system in the welding industry helps welders work in a safe environment and reduce their exposure to fumes and gases.

The study will be focused on the use of Computational Fluid Dynamics (CFD) software to analyse the performance of LEV system. CFD is a mechanism to simulate fluid dynamic flows and characteristics. The airflow distribution of LEV system is simulated in CFD software and the result from the simulation is observed in term of its velocity produced.

2. Materials and Methods

2.1 Components Involved in Simulation

The local exhaust ventilation system in this study is modeled based on the design of the welding station. The components involved in this design are the 14 capture hoods, a ducting system and 14 extraction arms.

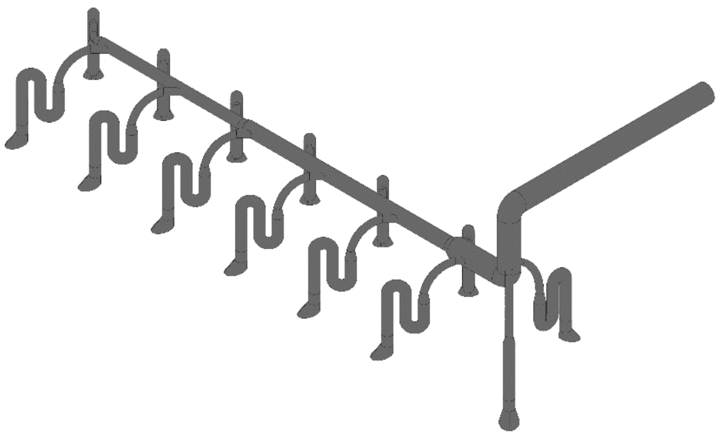

Figure 1: Geometry of LEV system

2.2 Data collection

Data collection for the measurement of the LEV system components is done by referring to the previous experimental study. The velocity data at the capture hood and ducting point used are also from the previous experimental study.[5]

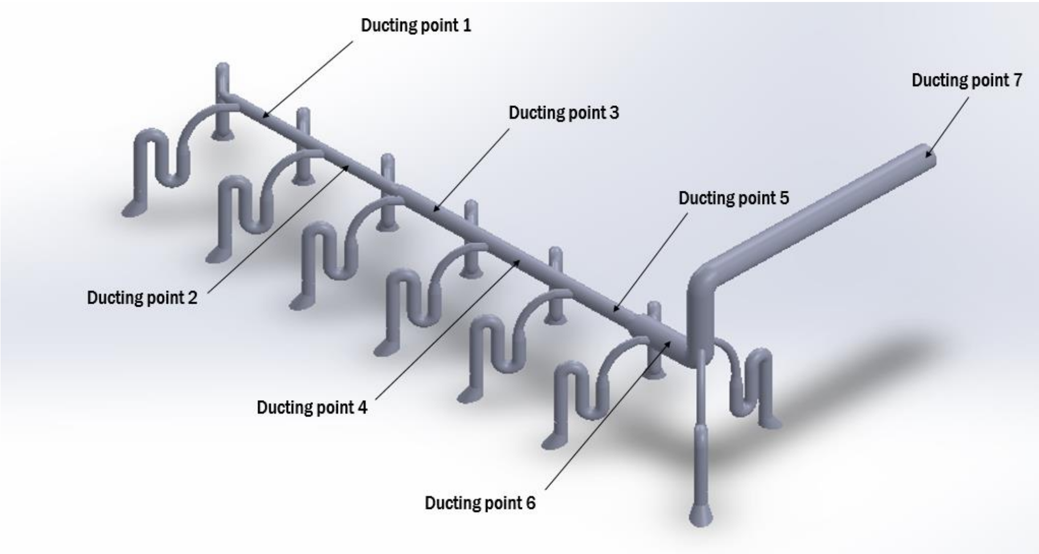

Figure 2: Assigned number for ducting point

2.3 Type of Turbulence Model

There are two type of turbulence models used in this study. There are k-epsilon turbulence model and shear stress transport turbulence model.

2.4 Parameters assumptions

The analysis of environmental conditions in the LEV system is conducted to ensure the parameter assumptions are close enough to the actual environment. Two parameters are assumed in this study. First, the operating temperature is assumed to be at 30ºC and 1 atmospheric pressure. The properties of air at 30ºC and 1 atmospheric pressure are referred from properties table in the book of 'Thermodynamics: An Engineering Approach' by Yunus Cengel and Michael A. Boles.[6]

Table 1: Parameters for the simulation

Item	Parameter	Value	Unit
1	Temperature	30	ºC
2	Density	1.164	kg/m3
3	Specific heat	1007	J/kg·K
4	Thermal conductivity	0.02588	W/m·k
5	Dynamic viscosity	1.872 x 10-5	kg/m·s

The next one is determine the type of fluid flow within the LEV system by calculating Reynold's number from the lowest velocity from the captured hood number 2, which is 1.27 m/s. Reynold's number for the lowest average velocity is determined by using the Eq. 1.

$$Re = \frac{\rho\, Vavg\, D}{\mu} = \frac{1.164\, \frac{kg}{m^3} \times 1.27 \frac{m}{s} \times 0.1524m}{1.872 \times 10^{-5} \frac{kg}{m.s}} = 12\ 034.72\ Re \qquad \text{Eq. 1}$$

Since the Reynold number is Re≥ 4000, hence, the type of flow within the entire system is considered as turbulent flow.

2.5 Meshing

The meshing method, a tetrahedrons-patch conforming method, is selected to be used in this study. It is due to its suitability to handle the meshing process of the LEV system geometry. The capture curvature setting is turned on to control the element size at the edges and faces of the geometry. For inflation, automatic inflation is used by set it to the option of ‘program controlled’. It is essential to use the inflation setting for this study to capture the boundary layer region in the LEV system.

For each simulation, the skewness ratio mesh metric is used to analyse the quality of mesh produced. Table 1 shows the summary of mesh quality for every element size.

Table 2: Mesh quality

GIT	Element Size	Maximum Skewness Ratio	Quality Condition
1	0.09	0.89999	Acceptable
2	0.08	0.89995	Acceptable
3	0.07	0.89997	Acceptable
4	0.06	0.90898	Acceptable
5	0.05	0.89000	Acceptable

2.6 Boundary conditions

Three types of boundary conditions are being used. The first boundary condition is an inlet at every capture hood. The following boundary condition is an outlet which is at the ducting point 7. The last boundary condition is the inner ducting wall. It is crucial to set this boundary condition to ensure that the fluid only flows inside the system. The ‘No slip condition’ option is set in this simulation to consider the viscous layer near the wall.

2.7 Interpretation of result

The velocity streamline of the LEV system is set and observed after the simulation has been completed. The cross-sectional line consists of an X-axis and Y-axis constructed at each ducting point where the velocity needs to be observed. At each of the two axes, seven samples are set to obtain the velocity value that passes through them. 14 velocity values are obtained at every ducting point.

After collecting all the velocity value at each location, the average velocity at each ducting point can be calculated by using Eq 2.

$$Average\ velocity = \frac{\sum Velocity\ at\ X - axis + \sum Velocity\ at\ Y - axis}{14} \qquad \text{Eq. 2}$$

3. Results and Discussion

Grid independence test is conducted by comparing the velocity changes between two turbulence models, k-epsilon and shear stress transport at ducting point 1 as the element size is changed. From the graph of velocity at ducting point 1 in Figure 2, it can be observed that both k-epsilon and shear stress transport turbulence model produced an almost similar pattern of velocity for each element size. The graph also shows the velocity produced by the k-epsilon turbulence model is a bit lower compared to the shear stress turbulence model. Based on the graph pattern, it is shown that the velocity at ducting point 1 is increased from element size 0.09m to 0.08m. However, starting from element size 0.08m until 0.05m, the velocity for both turbulence models decreases as the element size is reduced and the number of elements increases.

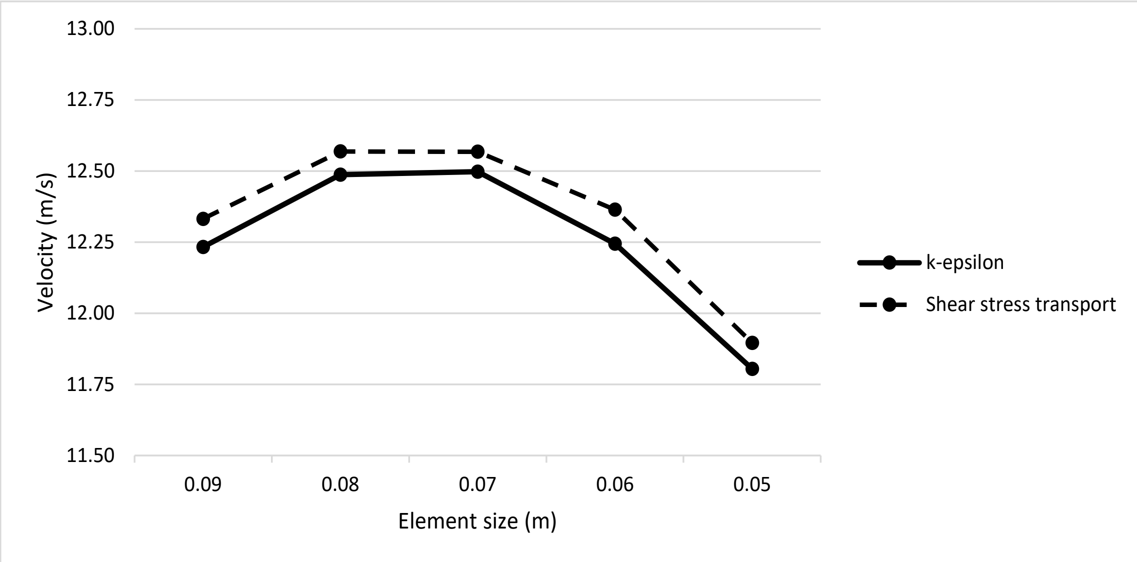

Figure 3: Graph of velocity at ducting point 1

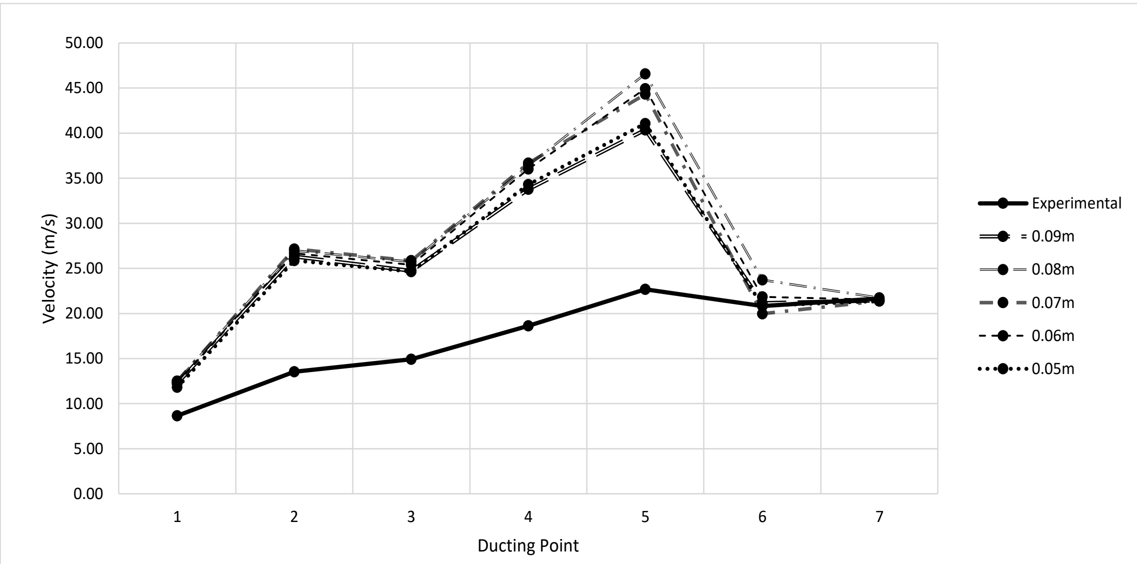

Figure 4: Graph of velocity distribution for k-epsilon turbulence model

For all the simulation studies for k-epsilon turbulence model, every element size shows the velocity distribution from ducting point 1 (DP 1) to ducting point (DP 6) is higher than the experimental study as shown in Figure 3. However, all the element size produce almost the same pattern of the graph. The graph also shows the maximum velocity is generated at DP 5, where the point is located at 8 inches diameter duct. Besides, the minimum velocity generated is at DP 1. It is the location where the point is the furthest from the outlet sources (DP 7) and is located at 6 inches diameter duct.

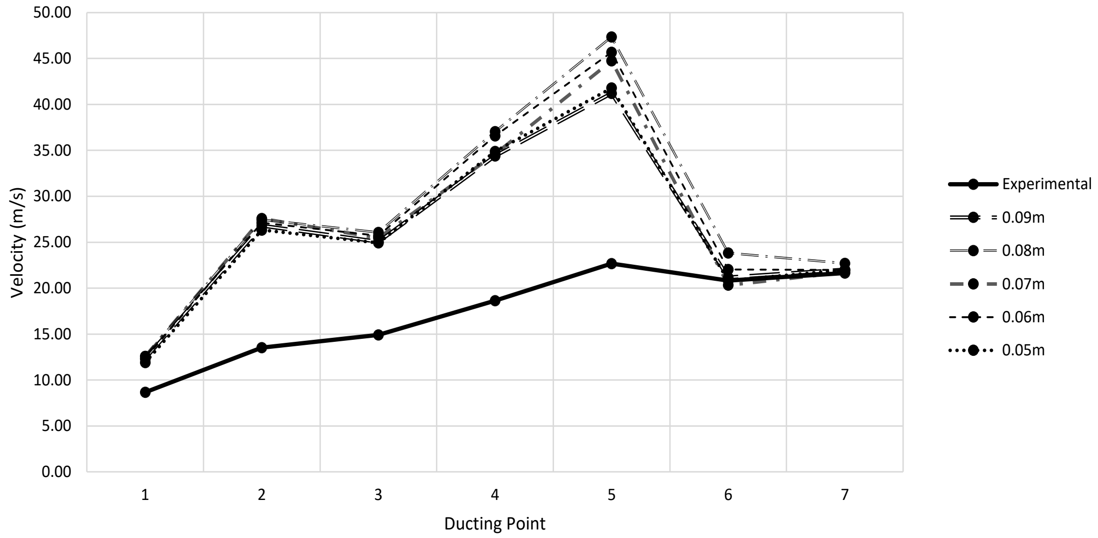

Figure 5: Graph of velocity distribution for SST turbulence model

For shear stress turbulence model, all the simulation study of every element size shows the velocity distribution obtained from DP 1 to DP 6 is higher than the experimental as shown in Figure 4. The graph also shows the maximum velocity is generated at DP 5. It is the location where the ducting point is located at 8 inches diameter duct. Meanwhile, the minimum velocity generated is at DP 1, where the point is located furthest from the outlet sources (DP 7) and is located at 6 inches diameter duct.

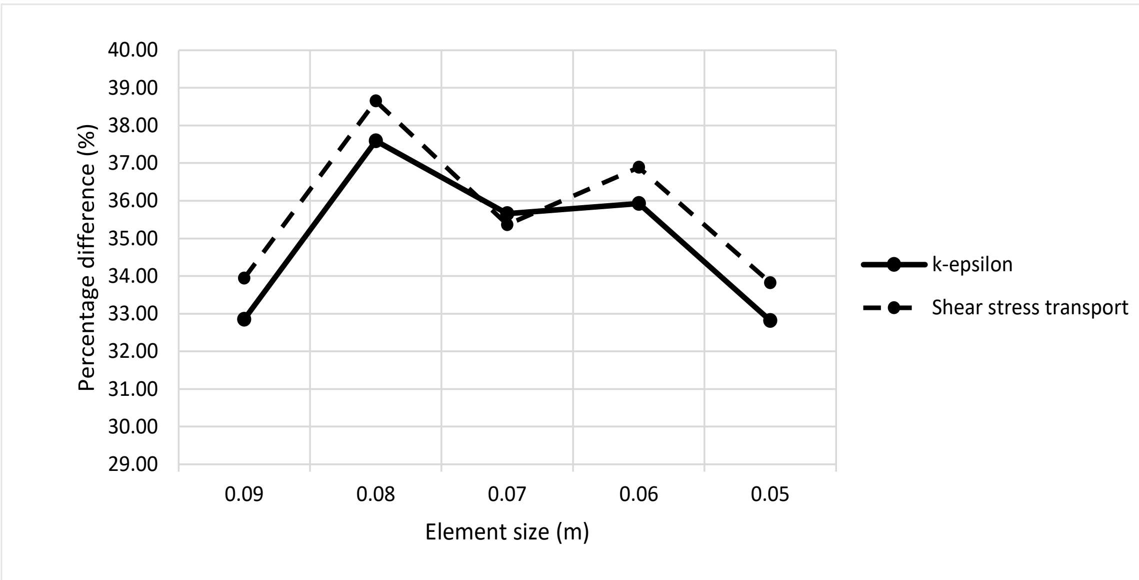

Figure 6: Graph of average percentage difference between turbulence model with experimental

From Figure 6, the 0.05m element size with the highest number of elements produces the lowest percentage difference with experimental study, which is 32.82%. Meanwhile, for the SST turbulence model, the lowest percentage difference with the experimental study is at 0.05m element size with a percentage of 33.82%. The average percentage difference obtained for both turbulent models shows that both simulations have higher velocity than the experimental study as shown in Figure 6. It also shows that the k-ε turbulent model has lower average percentage difference with experimental study than SST turbulence model except for 0.07m element size.

For all the simulations of every element conducted for both k-epsilon and shear stress turbulence models, the velocity slightly decreases from DP 2 to DP 3 due to the difference in ducting diameter between DP 2 and DP 3. It is because DP 3 has a bigger diameter than DP 2. From DP 3 to DP 5, the velocity increase as they become closer to the 'outlet' boundary condition. It causes the flow rate at this region to increase. Meanwhile, at DP 5 to DP 6, the velocity decreases since 90° ducting junctions cause some minor losses. Apart from that, changes in fluid direction could affect the fluid flow at that ducting region. The velocity at DP 7 for every element size is not similar to the 'outlet' boundary condition set during the simulation setup but the results show that the velocity value for each element size is slightly different from the velocity that was set. It shows that the DP 7 does not fully capture the velocity at that region.

The velocity reading at every ducting point in the k-ε turbulence model is lower than the SST turbulence model for most of the elements. It is due to its suitability to simulate the higher accuracy of the air distribution within the LEV system. The k-ε turbulence model can also simulate the air velocity within the LEV system better as it uses wall functions and has good convergence.

From this study, the k-ε turbulent model is more capable of simulating the air distribution and giving a better analysis of the air properties within the LEV system. It shows a good analysis of the air properties within the LEV system regarding the air velocity to study the system's performance by using CFD simulation. Therefore, the k-ε turbulent model should be used to get the findings as close as the existing system. It can be concluded that the result from the k-turbulent model is more valid to evaluate the air distribution and velocity produced within the LEV system in the CFD simulation.

4. Conclusion

Based on this study, the air distribution within the LEV system is successfully determined by using the CFD software, ANSYS CFX 19.1. The results showed that all the simulation conducted by using k-epsilon and shear stress turbulent model have higher velocity value than the experimental study. The lowest average percentage difference from each turbulence model are analysed and it is shown that the k-ε turbulence model produced the closest value with the experimental study with an average percentage difference of 32.82%. Meanwhile, shear stress transport was only able to achieve a 33.82% average percentage difference. There is a large difference in the velocity value between simulation and experimental study take place at the ducting point 5. The difference could probably cause by the minor energy losses at 90° ducting junctions and different in ducting diameter between ducting point 5 and 6. The energy losses in the ducting could potentially affect the pressure gradient before and after losses. It can be concluded that the computational fluid dynamics approach is capable of providing a better analysis of air characteristics within the LEV system. By conducting CFD simulation, the performance of the LEV system can be studied better than the experimental method. A full-scale grid independence testing can be done to have better analysis and results as long as there are no significant changes in velocity value as the element size is reduced.

Acknowledgement

The authors wish to thank to the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia that has supported on the accomplishment of research activity.

References

[1] Department of Occupational Safety and Health, "Guidelines on Occupational Safety and Health for Design, Inspection, Testing and Examination of Local Exhaust Ventilation System," pp. 1–53, 2008.

[2] M. Jafari and B. Shafiei, "Occupational Exposure to Welding Fumes Using Different Ventilation Scenarios," *Int. J. Occup. Hyg.*, vol. 2, no. 1, pp. 1–5, 2010, [Online]. Available: http://ijoh.tums.ac.ir/index.php/ijoh/article/viewArticle/1.

[3] Y. Mehrifar, M. Bahrami, E. Sidabadi, and H. Pirami, "The effects of occupational exposure to manganese fume on neurobehavioral and neurocognitive functions: An analytical cross-sectional study among welders," *EXCLI J.*, vol. 19, pp. 372–386, 2020.

[4] S. Zare, Y. Sahranavard, H. Ali Hakimi, M. Bateni, M. Karami, and R. Hemmatjo, "Designing, Constructing and Installing a Local Exhaust Ventilation System to Minimize Welders' Exposure to Welding Fumes," *Arch. Hyg. Sci.*, vol. 6, no. 4, pp. 356–362, 2017.

[5] M. A. Mario, "Baseline Inspection of Local Exhaust Monitoring," 2017.

[6] Y. A. Cengel and M. A. Boles, Thermodynamics: An Engineering Approach, vol. Eight Edit. 2015.