ASM Sc. J., 14, Special Issue 1, 2021 for ICSTSS2018, 143-147

Turbulent Model Selection for Synthetic Jet Characteristic Study

Sh Mohd Firdaus Bin Sh Abdul Nasir [1,2*], Mohd Zulkifli Abdullah[2] and Mohd Khalil Abdullah[3]

[1]*Faculty of Mechanical Engineering, Universiti Teknologi MARA, Permatang Pauh, Pulau Pinang, Malaysia*

[2]*School of Mechanical Engineering, Universiti Sains Malaysia, Nibong Tebal, Pulau Pinang, Malaysia*

[3]*School of Materials and Mineral Resources, Universiti Sains Malaysia, Nibong Tebal, Pulau Pinang, Malaysia*

Rapid change in electronic industries product rise the device temperature which required cooling to avoid device failure and provide longer life span. Thin package device no longer able to use fan or pipe-based cooling due to space constraint. The zero net mass flux with thinner device dimension become major advantages for future electronic cooling. In this research, synthetic jet has been modeled to study the characteristic of temperature reduction and velocity produced during cooling. ANSYS FLUENT® software has been used to characterize the fluid and heat temperature behaviour. Three turbulent models in ANSYS FLUENT® have been chosen: k-ε, k-ω and SST k-ω. The comparison on heater surface temperature and velocity at 1 cm distance has been compared with experimental data. The results show that SST K-ω contour produces 91% result similarity on temperature and velocity compared to other turbulent models.

Keywords: ANSYS FLUENT®; cooling; heat transfer coefficient; synthetic jet; turbulent model

I. INTRODUCTION

Synthetic jet used convective heat transfer for electronic cooling applications. It is critical for current electronic devices that power consumption increases along with the downscale of device dimension that result to low space availability for heat to dissipate. Synthetic jets are jets with zero-net-mass-flux, directly develop from the fluid system in which the jet actuator is embedded (McQuillan *et. al.*, 2014; Silva-Llanca & Ortega, 2017; Greco *et al.*, 2017).

A synthetic jet working principal from the oscillating diaphragm in a cavity, which produces changes in swept volume which entrained into and ejected through the nozzle as shown in Figure 1. Mahalingam and Glezer (2013) and Mangate and Chaudhari (2016) used synthetic jet to reduce surface temperature from 70°C to 36°C which has higher power dissipation 20-40% compared to fan with heat sink. The diaphragm motion is practically achieved by a piston type motion to obtain the desired amplitude or frequency (Firdaus *et. al.*, 2018; Chaudhari *et. al.*, 2011; Tan *et al.*, 2015). Numerical studies on thermal performance of synthetic jet very limited in published literature that focusing the turbulent model comparison with experimental result using 3D structured meshing type due to meshing form complexity behaviour during fluid and heat simulation that commonly not able to match integrate.

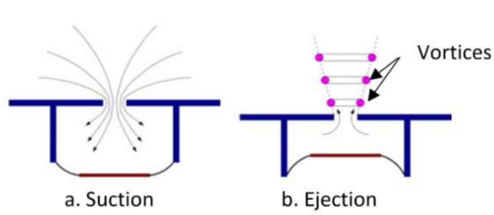


Figure 1. Synthetic Jet motion; (a) suction stage and (b) ejection stage (Firdaus *et al.*, 2018)

In numerical study, User Define Function (UDF) is used for the diaphragm motion using dynamic layering mesh. The UDF produces the sinusoidal periodic diaphragm motion from Equation (1):

*Corresponding author's e-mail: sh.firdaus@uitm.edu.my

$$y = A \sin(\omega t) \quad (1)$$

where A is the diaphragm amplitude, ω is the angular frequency and t is time.

When the frequency drive at resonance, lower power effort required which make higher cooling rate as stated by Tan *et al.* (2015) and Firdaus *et al.* (2018). The heat transfer coefficient value, *h* can be determined using Equation (2):

$$h = q_{conv} / (T_s - T_\infty) \quad (2)$$

where T_s is the heated surface data temperature, q_{conv} is the total net heat flux and T_∞ is the ambient data temperature. Turbulence model in ANSYS FLUENT® has widely used to predict the effect of computational fluid dynamic (CFD) characteristic environment.

In this paper, there are three types of turbulent models; k-epsilon (k-ε), k-omega (k-ω) and shear stress transport k-omega (k-ω SST) that will be studied for characterizing the heat transfer in ANSYS FLUENT® software (Tang & Zhong, 2005; Wang *et al.*, 2006), which suitable to show the fluid and heat behaviour at given parameter. Turbulent model selection is a vital concern to simulate better output which response nearly similar characteristic with experimental results pattern (Ramon *et al.*, 2017). The fabricated model was tested for heat transfer performance. The results obtained from the simulation and experiment were compared for heat removal performance pattern similarities (Harinaldi, 2012). Currently, numerical studies on thermal performance of synthetic jet in published literature for the turbulent model comparison with experimental result using structured meshing type was very limited.

II. METHODOLOGY

The synthetic jet was modelled in ANSYS FLUENT® numerical simulation for resonance frequency determination and fabricated using rapid prototyping for several experiments' analysis.

A. Numerical Parameter

Structured meshing able to reduce the processing time and increase the fluid behaviours results. Table 1 shows the model dimension and operating range. The numerical simulation was carried out for 1,000 cycles which equal to 10000-time step to obtain steady velocity output and smaller time steps able to capture more details on the changes happen during fluctuation. Each 100-time step per cycle, the iteration was continued up to the residual of mass and momentum decreasing below 10^{-3} and energy residual were reduced below 10^{-6}, which is the the computation criteria of convergence. Temperature results were taken every 45th time step which the position of diaphragm during the ejection stage and 85th time step at suction stage for 1 cycle.

Table 1. Numerical setup parameter

Parameter	Unit
Frequency	500 Hz
Volume Chamber	5 mm
Nozzle diameter	2mm

There are three different turbulent models selected; k-epsilon (k-ε), k-omega (k-ω) and shear stress transport k-omega (k-ω SST) and tested. Structured meshing has been used throughout this research in order to reduce computing processing time and organize results pattern. The detail boundaries in the computational domain were shown in Figure 2. The boundary and initial conditions are as follows:

a) Diaphragm sinusoidal motion; (Eq.1)
b) Wall: x = y = z = 0 (No motion)
c) Outlet Pressure, P= 0 Pa (No Pressure Different)

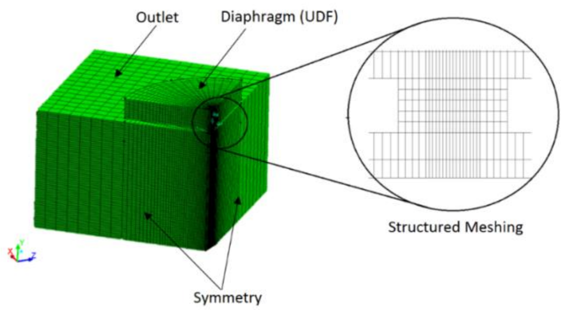


Figure 2. Computational domain boundary conditions

B. Experimental Setup

Figure 3 shows the experiment setup where 20Watt heater were used as the heat source. The synthetic jet was fixed at 50 mm distance from the heated surface. 500 Hz sinusoidal frequencies signal was supplied and monitored with external oscilloscope. Data logger system was used to record the ambient temperature and heater surface temperature. The data logger start to record the temperature data as the experiment started. The heater temperature fluctuate when the synthetic jet start cooling and it takes 15 minutes to stabilize in order to maintain the cooling temperature at the heater surface.

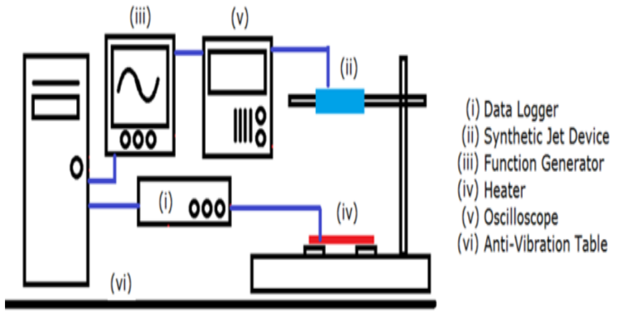


Figure 3. Thermal experimental setup

III. RESULTS & DISCUSSION

Numerical results show that the k-ω SST has nearly similar pattern compared to other turbulent models as plotted in Figure 4. The K-w and K-e models have more than 100% different far from the experimental results while only 3% of differences between experimental velocity output result and k-ω SST turbulent model compared to other turbulent models. The k-w SST model able to predict close to experimental output velocity as mentioned by Zschirnt *et al.* (2013) and SST k-w model was the best predicting near field heat transfer.

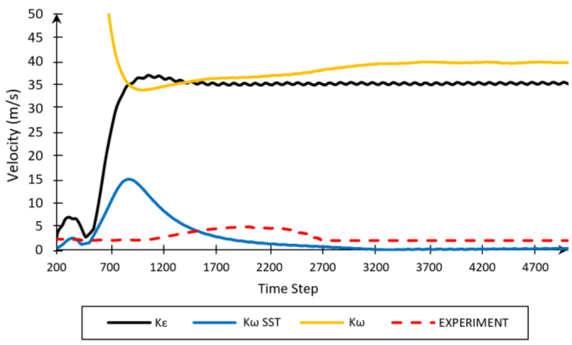
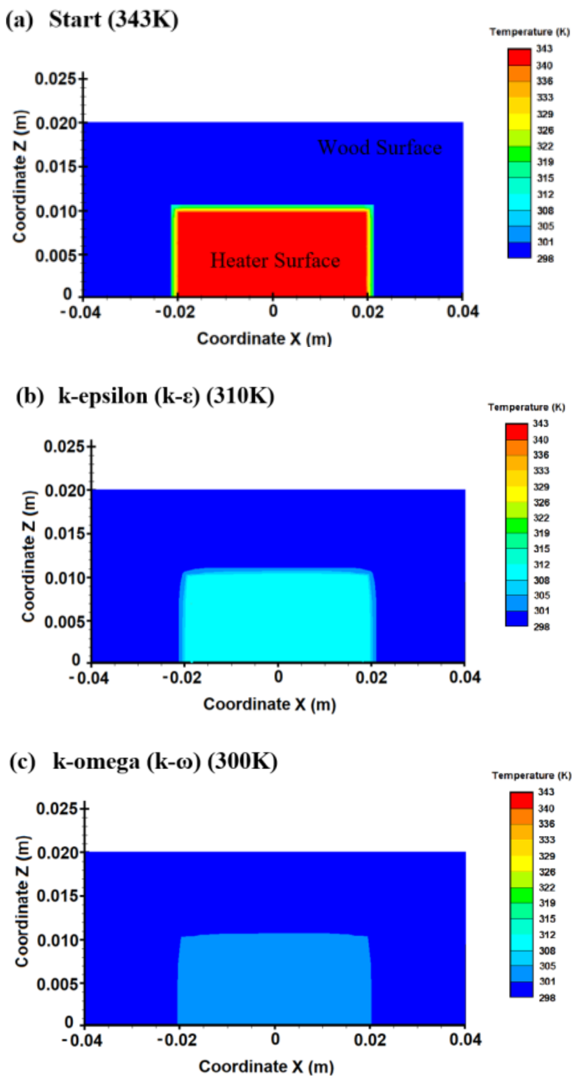


Figure 4. Turbulent model velocity results comparison ANSYS FLUENT®

Figure 5 shows the temperature contour at the heater surface for different turbulent models. The heater was set to 343K temperature as shown in Figure 5(a) before the diaphragm start to fluctuate, the air flow out from the nozzle will reduce the heater temperature. Wood has been used to avoid heat distributed outside the heater surface. All turbulent models able to show the same surface contour where constant temperature was observed for the wood area and rainbow temperature different at the heater surface area. The temperature reduction exhibits like the experimental characteristic. The k-w SST able to show more temperature contour compared to other turbulent models (Chaudhari *et al.*, 2011). The efficiency and accuracy, the shear stress transport k-ω (SST) turbulence model combining the advantages of the standard k-ω model and k-w (Zhang & Xie, 2018).



(d) k-omega (k-ω) Shear Stress Transport (SST) (328.4K)

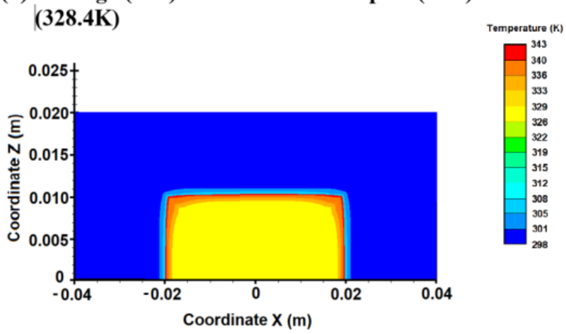


Figure 5. Heater surface contour results comparison for different Turbulent model ANSYS FLUENT®

Table 2 shows the heater surface comparison from the experimental and turbulent model results. Heater surface temperature result for k-w SST has the lowest different to experimental data. Other turbulent models also have good temperature result which below 10% different. Based on the velocity output and heater surface temperature results, the k-ω SST turbulent model predict more closer to experimental results which later will be for further synthetic jet study.

Table 2. Parametric range for numerical setup

	Heater Surface Temperature (K)	**Different (%)** (Model – Experiment) /(Experiment)
Experimental	328.4	-
k-epsilon (k-ε)	300.5	8 %
k-omega (k-ω)	298	9 %
k-omega (k-ω) Shear Stress Transport (SST)	325	1 %

VI. REFERENCES

IV. CONCLUSION

Structure meshing synthetic jet model was designed to select the suitable turbulent model that exhibits near to experimental data. The use of k-ω SST turbulence model has 5% different compared to other models for predicting the velocity and surface heater temperature characteristics. Hence, k-w SST turbulence model can be used for future research on the effect of synthetic jet volume chamber on cooling.

V. ACKNOWLEDGEMENT

The authors would like to acknowledge the support of the Ministry of Education (MOE) under the Research Acculturation Grant Scheme (RAGS/1/2014/TK01/UITM/2), Universiti Teknologi MARA (UiTM) and Universiti Sains Malaysia (USM) for their support in undertaking this work.

Chandratilleke, TT & Rakshit, D 2013, 'A synthetic jet heat sink with cross-flow for electronic cooling', International Journal of Thermal Sciences, vol. 49, pp. 504-513.

Chandratilleke, TT, Jagannatha, D & Narayanaswamy, R 2011, Synthetic jet-based hybrid heat sink for electronic cooling, IntechOpen Publisher, Australia.

Chaudhari, M, Puranik, B & Agrawal, A 2011, 'Multiple orifice synthetic jet for improvement in impingement heat transfer', International Journal of Heat Mass Transfer, vol. 54, no. 9–10, pp. 2056–2065.

Firdaus, SM, Abdullah, MZ, Abdullah, MK & Amri, W 2018, 'Synthetic jet cooling diaphragm equation of motion characteristic and verification for computational fluid dynamic modelling', AIP Conference Proceedings, vol. 1974, no. 1, AIP Publishing.

Firdaus, SM, Abdullah, MZ, Abdullah, MK & Fairuz, ZM 2018, 'Heat transfer performance of a synthetic jet at various driving frequencies and diaphragm amplitude', Arabian Journal for Science and Engineering, vol. 44, no.2, pp. 1043-1055.

Greco, CS, Cardone, G & Soria, J 2017, 'On the behaviour of impinging zero-net-mass-flux jets', Journal of Fluid Mechanics, vol. 810, pp. 25–59.

Harinaldi, Engkos, AK, Damora, R, & Rikko, D 2012, 'Forced cooling on a heated wall with impinging flow configuration using synthetic jet actuator under combined wave excitation', Jurnal Teknologi (Sciences Eng.), vol. 58, no. 2, pp. 19–23.

Lv, YW, Zhang, JZ, Shan, Y & Tan, XM 2014, 'Numerical investigation for effects of actuator parameters and excitation frequencies on synthetic jet fluidic characteristics', Sensors Actuators, A Physical, vol. 219, pp. 100–111.

Mahalingam, R & Glezer, A 2013, 'Design and thermal characteristics of a synthetic jet ejector heat sink', Journal of Electronic Packaging, vol. 127, no. 2, pp. 172–177.

Mangate, LD & Chaudhari, MB 2016, 'Experimental study on heat transfer characteristics of a heat sink with multiple-orifice synthetic jet', International Journal of Heat Mass Transfer, vol. 103, pp. 1181–1190.

McQuillan, B, Hertzberg, J & Montoya, LD 2014, 'Flow visualization study of synthetic flow control in the indoor environment', Building and Environment, vol. 73, pp. 239–248.

Ramon, T, Harinaldi & Engkos, AK 2017, 'Vortex ring formation characteristics in synthetic jet due to changes of excitation frequency in the ½-ball cavity actuator', Journal of Physics: Conference Series, vol. 822, no. 1, pp. 3–10.

Silva-Llanca, L & Ortega, A 2017, 'Vortex dynamics and mechanisms of heat transfer enhancement in synthetic jet impingement', International Journal of Thermal Sciences, vol. 112, pp. 153–164.

Tan, XJ, Yong, S & Xie, G 2015, 'An experimental investigation on comparison of synthetic and continuous jets impingement heat transfer', International Journal of Heat Mass Transfer, vol. 90, pp. 227–238.

Tang, H & Zhong, S 2005, '2D numerical study of circular synthetic jets in quiescent flows', Aeronautical Journal, vol. 109, no. 1092, pp. 89–97.

Wang, Y, Yuan, G, Yoon, YK, Allen, MG & Bidstrup, SA 2006, 'Large eddy simulation (LES) for synthetic jet thermal management', International Journal of Heat Mass Transfer, vol. 49, pp. 2173–2179.

Zhang, YP & Xie, Y 2018, 'Numerical investigation of heat transfer characteristics of impinging synthetic jets with different waveforms', International Journal of Heat and Mass Transfer, vol. 125, pp. 1017–1027.

Zschirnt, ME, Lee, A & Yeoh, GH 2013, 'Effects of operating frequency of a synthetic jet and cross flow velocity on the heat transfer enhancement in a micro-channel', in Proceedings of 2013 Fluids Engineering Division Summer Meeting.