The Influence of the Excitation Frequency of the Cavity Changes of Synthetic Jet Actuator to the Formation of Vortex Ring

by Ramon Trisno_6

The Influence of the Excitation Frequency of the Cavity Changes of Synthetic Jet Actuator to the Formation of Vortex Ring

Ramon Trisno, Harinaldi, and Engkos A. Kosasih
Department of Mechanical Engineering, Universitas Indonesia, Depok, West Java, Indonesia 16424
Email: ramontrisno@gmail.com, harinald@eng.ui.ac.id, kosri@eng.ui.ac.id

***Abstract* – This research is a basic study of the development of turbulent flow separation controlling the aerodynamic phenomena, especially in the design of the vehicle body. The purpose of this study was to analyze the performance of the synthetic jet (SJA) as one of the tools in reducing the flow controller separation area on the bluff body model of the vehicle. To get the maximum results of the performance of the synthetic jet actuator, the research starts with the actuator characterize. Its characteristics include changes in shape of the cavity and orifice diameter. Its cavity forms, namely half balls, tubes and conical, while its orifice diameter is 3, 5 and 8 mm. The study was conducted using computational and experimental approaches. Both types of research methods was compared and displayed the results in graphical form. This result is a reference in determining future research. The experimental results in the form of the flow rate for each type of cavity can determine the formation of vortex ring, whereas in CFD simulations, the formation of vortex ring can be seen from the visualization of the flow contour. Vortex ring cavity can occur at B3, T3, T5, K3 and K5. Vortex ring is not formed on any type of cavity with a diameter of 8 mm orifice.**

***Index Terms*—synthetic jet actuator, reverse Ahmed body, vortex ring, bluff body**

I. Introduction

Global warming is one of the main problems in recent years. The International Energy Agency in World Energy Outlook 2007, concluded that emissions of greenhouse gases will increase by 57% by 2030 [1]. It was submitted by the Intergovernmental Group of Experts on the Evolution of the Climate (GIEC) in 2001, which states that human activity was first on the causes of the greenhouse effect and the increase in temperature in the 20th century [2]. One of the human activities that cause these problems are in the areas of transport, such as the increase in the number of vehicles significantly.

In this regard, the latest research in the field of vehicle aerodynamics, carried out with a design standpoint efficient and able to impact fuel economy. For this reason, ground vehicle aerodynamics have been studied experimentally and numerically by many researchers. Most previous studies have used simple models of vehicles that can generate relevant features of the flow around a real vehicle [3-8] the flow field around the eddy is marked by a pair of horseshoe vortices (a pair of horseshoe vortices) and trailing vortices emanating from the edge of the sloping side of the body.

Experiments passive flow control in the wind tunnel model or prototype vehicles have been many studies done on various [9, 10]. However, passive flow control in fact give a weak effect in aerodynamic vehicle involved because of the nature of the flow is turbulent and the contribution of friction on aerodynamic barrier is still small, only about 10% [11].

Flow control on the bluff body in order to reduce drag and noise is one of the main issues in aerodynamics. The difference in pressure between the front and the back of the bluff body is a major contributor to the overall drag, the difference is primarily due to flow separation at the rear of the body [12].

The need for a reduction in the drag force is more effectively encourage the automotive design more creative in developing innovative active control model. Active control method makes it possible to modify the topology of the flow without changing the shape of the vehicle. Within the scope of academic and industrial laboratories, active control methods have been and still are developed with computational and experimental methods, and significant results have been obtained in the academic framework [9]

Active flow control in the form of a blowing continuous of placed on the back of a generic square back bluff body gives a drag reduction of 20% [13]. The same is done Bruneau C.H. et al [14], in which the active flow control used are three combination of a placement suction and blowing on the back of Ahmed models, which result the best drag reduction of 13%. Numerically study conducted by Kuorta and Gillieron [15] by using active control of synthetic jet is placed on top of the rear window Ahmed model where the drag reduction of 13% was obtained.

Developments in the active control of the synthetic jet lately, mostly resulting from the research conducted on the use of active control in the field of synthetic jet cooling and characterization of the active control. Fluid jet in the form of air or other liquids used for the cooling of electronic equipment on a small scale. While the

Manuscript received August 21, 2017; revised February 7, 2018.

doi: 10.18178/ijmerr.7.3.279-284

application of synthetic jet on Ahmed's body has also been done to study the performance of the synthetic jet drag reduction, but this research has not been completely carried out in depth, just merely knowing that with the installation of synthetic jets can reduce aerodynamic drag on the vehicle.

While research on the application of synthetic jet mounted on a condition of reverse Ahmed body has never been done previous research. Reverse Ahmed body is to change the orientation of the flow direction before as installed on Ahmed body. This model describes the shape of the vehicle is often used in Indonesia. The synthetic jet will be installed on the top of the curve at the Ahmed body. Before the synthetic jet is mounted on a vehicle is necessary to do research on the characteristics of synthetic jet especially in terms of influence of frequency on the performance SJA (synthetic jet actuator) due to changes in the shape of its cavity and orifice diameter. The purpose of this study was to obtain the characteristics of the SJA by varying the shape of the cavity and the size of the orifice diameter by giving the frequency of oscillation to the membrane in order to obtain a vortex ring of such changes.

II. Experimental Setup

Before starting the study of synthetic jet performance to the reduction of drag on the vehicle, to know about the performance of SJA itself, in order to get the maximum results for drag reduction. Selection is based on the SJA actuator cavity shape, orifice diameter and comparison of performance through experiments and CFD simulations. In this study, its cavity form is a type of half ball, tube, and cone with a diameter of each 3 mm, 5 mm and 8 mm. (Fig. 1)

Fig. 2 is a schematic diagram of the synthetic jet performance testing through experimentation. Component consists of synthetic jet cavity and piezo electric membrane components, installed as a whole is driven by means of the function generator. This tool unleashed a wave of the desired signal.

In this experiment a signal used is a sinusoidal wave. Movement is what causes the membrane of the air pushed out of the cavity and sucked into the cavity back. This tool has a mechanical system with a very small mass of the stem and the spring move. This mass was hanged just two microns (two millionths of a meter) from the electronic circuit.

Furthermore, velocity air that comes out of this cavity through the orifice, measured using a hot wire probe type 55P14 Gold Plate. These probes are connected to the Data Logger with brand Dantec CTA Stream Pro type 91C10 module that serves to record the incoming data from hotwire in the form of low voltage. Then the data is entered into the Data Acquisition Board from National Instrument Type BNC-2110. Furthermore, the tool is connected to the computer via a data cable. This computer is also connected to a data logger which serves to control the speed of recording desired data at the time of recording data. Prior to recording the data, the probe must first be calibrated using a calibration unit Dantec Streamline Pro Automatic Calibrator System S/N 9091H0013445 and King's Law is suitable for the job to interpret the data speed below the calibration range. The range of the calibration performed at 0:02 ÷ 30 m / s before each experiment done.

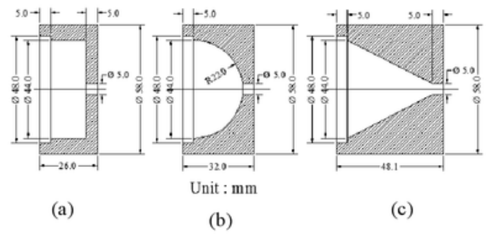

Figure 1. Shape of cavity: type of (a) tube, (b) half ball and (c) cone

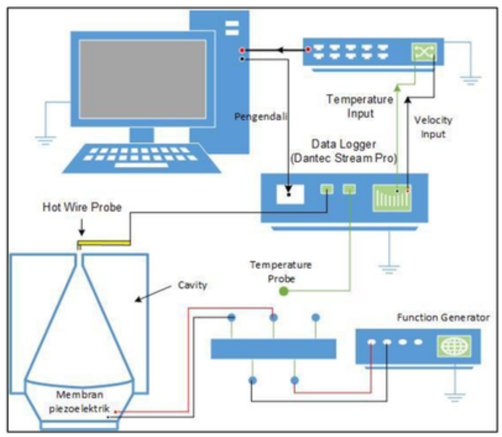

Figure 2. Schematic of testing the performance of the synthetic jet

The calculation result of uncertainty due to calibration for all data retrieval velocity is about 2.5%. Profile across the radial velocity measured with a hot wire probe in the hole using the tool transfer (vise) by the displacement distance is measured using a dial indicator with 0:01 mm accuracy.

At this stage of computational simulations using CFD from ANSYS software. The purpose of this simulation is to get an initial overview of the performance of the synthetic jet actuators to be studied and compare the CFD simulation results obtained with the results of experiments conducted as well as visualize the vortex ring happens to SJA. The criteria used in the experiment equated with the criteria used in this CFD simulation.

III. Result and Discussion

A. Testing Frequency and Mesh Independencies

The early stages of the experiment is to determine the frequency of which can produce the maximum flow rate that the SJA's performance as a reference in subsequent experiments. This testing is done by varying the frequency of the wave function generator tool used is a sinusoidal wave.

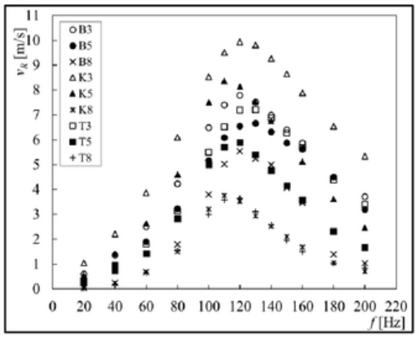

Figure 3. Average velocity graph of experiment results on some of the synthetic jet actuator cavity with amplitude sinusoidal types.

Fig. 3, shows graphics performance testing synthetic jet performed on different frequencies with each cavity and the size of the orifice diameter different. The maximum frequency of synthetic jet actuators are selected based on the velocity of air flow out of the orifice reaches maximum conditions. The maximum jet velocity can be produced by each SJA range at a frequency of 110 Hz ÷ 130 Hz, where the cavity with a conical shape and orifice diameter of 3 mm, can produce the maximum jet velocity average of 9.9249 m / s.

The frequency test is then used as a reference frequency in the simulation method. The maximum frequency is used in the UDF. The results of this simulation aims to establish the meshing obtain simulation results closer to the experimental results.

In the computational method by means of CFD simulations, manufacture meshing is done using software Gambit, while meshing testing performed using FLUENT software. This meshing of testing aims to get meshing that can produce residues of CFD simulation calculation smaller and coherent, so that the meshing forms obtained are meshing unchanged residue again. The form of meshing being tested is then used as a test cavity meshing on the use of software Fluent 6.3.26. Each type of cavity has the shape and number of meshing different. Cavity shape and size adapted to the conditions that made the experimental method. CFD simulation results illustrate the speed of the jet at the position X/D = 0 and Y/D = 0 with the speed of the data record data at 0.0001 / s.

B. Uncertainty in Experimental Methods and Simulation CFD

The measurement results is the estimated value of the quantity being measured. Uncertainty (uncertainty of quantitative measurements) are not reported as a single value but is reported with a range of values estimated true value resides in the value. Measurement results vary reflecting the deviation caused by the performance factor of tools, methods of measurement, environmental conditions, and so forth.

Uncertainty aims to determine the extent of truth in taking the amount of data. Collecting data in this experiment carried out as many as 60,000 data during 6 seconds (10,000 data / s). As for the simulation data is also taken as many as 10,000 data is also for 1 s, applies to the 95% confidence level. namely:

$$U_x \propto \sigma_{\bar{x}} \Rightarrow U_x \approx k(\text{convidence limit})\sigma_{\bar{x}} \qquad (1)$$

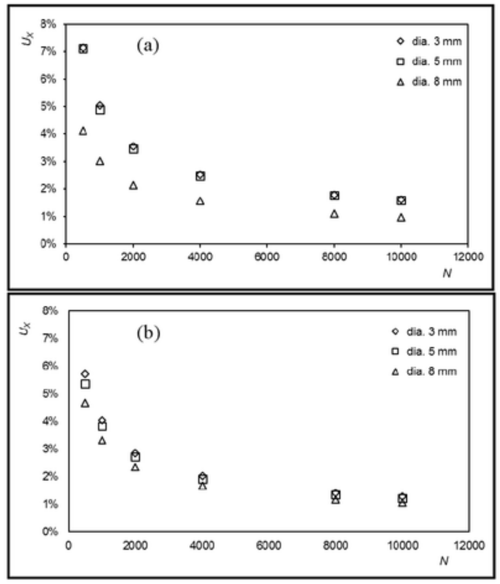

Figure 4. Graph uncertainty for flow velocity in the cavity a half ball shape, (a) experiment, (b) CFD simulation

Graph the results of calculation of uncertainty in each cavity are taken based on the number of sampling of data per second, which is 10,000 data. The processed data is data that occurs on the condition Y/D = 0 and X/D = 0. The point (0,0) is 0.5 mm above the orifice hole, this is due so that the movement of the probe is not in contact with the surface of the SJA.

Fig. 4 show charts the results of calculation of data uncertainty is calculated based on the equations that exist in each diameter of orifice. Based on data obtained from experiments and CFD simulations, the B3 value greater uncertainty compared to other types of cavity. Uncertainty of the experimental methods give 1.6081% and the value of CFD simulation results amounted to 1.2789%. While the value of the smallest uncertainty occur at B8, respectively uncertainty of its value is 0.98% for the experimental methods and 1.0632% for CFD simulation method. The value of this uncertainty needs to be aware of magnitude, so that the amount of data captured on the experimental conditions and CFD simulations can qualify a data confidence level of 95%, then the uncertainty of its value must be below 5%. In both these methods, it has been fulfilled.

C. Determining the Vortex Ring

Characteristics of the flow velocity at the SJA taken from velocity in the center line, U_{CL}, as the assumption of the highest velocity profile at the exit orifice. Spatial velocity profile can show the standard deviation of the highest form, some of the speed scale is time for the average flow rate through the expulsion stroke [16], namely:

$$U = \frac{1}{T/2}\frac{1}{A_n}\int_{A_n}\int_0^{T/2} U(t, A_n)dtdA_n \qquad (2)$$

Where U is velocity the average axial phase, T is the period of excitation and A_n is the cross-sectional area of the exit orifice. The average velocity can be used to determine the jet Reynolds number. The important thing at the length scale synthetic jet stream is the stroke length [17-19] L_o/D, namely:

$$L_o = \int_0^{T/2} U_{ave}(t)dt \qquad (3)$$

U_{ave} is an average flow velocity spatially. Stroke length is the distance fluid traveled during the process of pushing the fluid from the cycle. Excitation frequency can be no dimensional as numbers Stokes, S, namely:

$$S = \sqrt{\frac{2\pi f D^2}{\nu}} \qquad (4)$$

The non-dimensional parameter uniquely determine the operating conditions of the SJA and so affects its ability to transfer linear momentum. In particular, Holman et al. [20] shows that in order to reach the exit after expulsion stroke vortex is defined as synthetic jet formation, in which:

$$\frac{\mathrm{Re}_U}{S^2} > C \qquad (5)$$

where C is a constant equal to 0.16 for the axis symmetric and 1 for rectangular nozzles. According Utturkar et al [20], condition of the occurrence of vortex ring is $Re_U/S_2 > 1$. Meanwhile, according to Smith and Swift [19] found the ring vortex can occur if:

$$\frac{L_0}{D} = St > 6 \qquad (6)$$

The results of the calculations using equations (5) and (6) to determine the formation of vortex rings can be seen in Table I, wherein the red box indicates that the cavity can form a vortex ring with a strong vortex structure.

TABLE I. THE RESULT OF THE CALCULATION OF THE OCCURRENCE OF VORTEX RINGS ON THE EXPERIMENTAL METHOD.

Cavity	Re_U	Re_U/S^2	L_o/D
B3	1201.32	2.768	16.248
B5	1092.08	0.836	5.25
B8	1367.35	0.443	2.782
T3	1837.03	3.92	12.31
T5	2469.24	2.055	6.453
T8	2464.2	0.801	2.516
K3	1418.34	3.05	19.20
K5	1711.13	1.44	9.09
K8	1091.83	0.36	2.26

In addition to the calculation of the equation by incorporating existing experimental results, the ring vortex can also be proved by means of CFD simulations. The simulation results also will be matched with the results of the calculations have been done before. The results of this simulation can be seen in the image below.

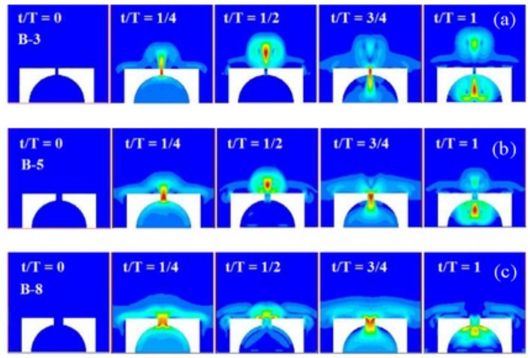

Figure 5. The formation process of vortex rings on (a) B3, (b) B5 and (c) B8.

Fig. 5 are in the simulation process of formation of vortex ring cavity with half ball orifice diameter of 3 mm, 5 mm and 8 mm using FLUENT software, while for the other cavity is not shown here. Fig. 8(a) shows the contour of flow velocities caused by the movement of the membrane on condition B3. If the membrane moves and is at $t/T = 1/4$, then in this position the membranes doing blowing, so that the visible presence of air flow out through the orifice. The contour of flow velocities is beginning to show the formation of vortex rings. The position of $t/T = 1/4$ is the peak expulsion stroke undertaken by the movement of the membrane. After reaching the position of $t/T = 1/2$, then start this vortex ring section broke away and moved further away from the end of orifice. Vortex ring that has been formed remain intact and further intensified in the position of $t/T = 3/4$. At the position of $t/T = 1$, vortex ring looks have released themselves from the end of orifice. Fig. 8(b) shows contours of velocity jet produced by SJA type B5. Fig. 8(c) shows contours of velocity jet produced by SJA type B8. Cavities with 3 mm orifice diameter can form vortex rings with a stronger structure than any other cavity

D. *Effect of Vortex Ring Formation Against Excitation Frequency*

In the vortex ring formation, the influence of the excitation frequency often affect the magnitude of velocity jet spread out of the SJA. This phenomena has an impact on the performance of SJA in forming a vortex ring. In Fig. 6, we will see the effect of changing the frequency formation of vortex rings on each type of cavity used in this experimental method.This Fig. shows the results of calculations using the equations of vortex ring proposed by Smith and Swift, to get *Lo/D* of different frequencies. From the results of this calculation shows the difference in the results shown by each cavity. The red line on the *Lo/D* = 6 distinguish between areas that formed vortex ring with areas that cannot form a vortex ring. Generally the cavity with orifice diameter of 3 mm included in the region of vortex ring formation and decreased due to higher excitation frequency. As for the cavity with orifice diameter of 5 mm, only a portion of a frequency that can form a vortex ring, whereas at high frequencies, most of SJA can no longer form a vortex ring. Vortex ring formation occurs around a frequency that can produce a maximum jet velocity. While in the cavity with a diameter of 8 mm orifice, all cavity cannot form a vortex ring at different frequencies condition. Only cavity type B8, could be expected to form a vortex ring at the frequency of the condition can produce the maximum jet speed, because in these conditions the value of *Lo/D* was already close to 6, this value as required by Smith and Swift.

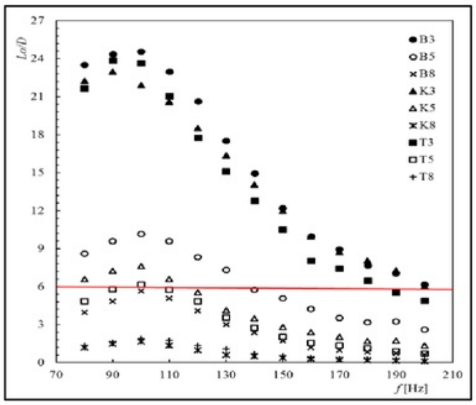

Figure 6. Vortex ring calculation result by Smith & Swift

In addition, to determine the occurrence of vortex ring can also be determined from the equation (5) raised by Utturkar. From the calculation using this equation, the results are not much different from the equation (6) in Fig. 6.

IV. Conclusion

From the results of research on the characteristics of synthetic jet actuator, it can be some conclusions. The maximum speed that can be generated SJA in the frequency range 110 Hz - 130 Hz with an uncertainty of 2% to the amount of data 10000. The most high-velocity jet produced by K3 type cavity, which cavity a conical shape with a diameter of 3 mm orifice. Average jet velocity reaches 9.98 m/s. Vortex ring can be determined by the method of experiment and CFD simulation method, where both these methods give the same result. As well as on the experimental method, the second opinion of researchers [19, 20] issuing the equation of the formation of a vortex ring has the same result. Vortex ring cavity can occur at B3, T3, T5, K3 and K5. Vortex ring is not formed on any type of cavity with a diameter of 8 mm orifice.

Acknowledgements

The authors would like to thank the financial support provided by Universitas Indonesia through the PITTA 2017 funding scheme under Grant no. 857/UN2.R3.1/HKP.05.00/2017 managed by Directorate for Research and Public Services (DRPM) Universitas Indonesia

References

[1] IEA, 2007, World Energy Outlook 2007, Executive Summary, China and India insights, International Energy Agency IEA, ISBN : 978-92-64-02730-5.

[2] Groupe d'experts Intergouvernemental sur l'Evolution du Climat (GIEC) (2001). "Bilan 2001 des changements climatiques: Les éléments scientifiques", 3èmerapport d'évaluation 2001, [Online]. Available: http://www.grida.no/climate/ipcc_tar/vol4/french/pdf/wg1sum.pdf.

[3] S. R. Ahmed, G. Ramm, and G. Faltin, "Some Salient Features Of The Time-Averaged Ground Vehicle Wake," 1984.

[4] C. Hinterberger, M. García-Villalba, and W. Rodi, "Large eddy simulation of flow around the Ahmed body," in *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, R. McCallen, F. Browand, and J. Ross, Eds., ed Berlin, Heidelberg: Springer Berlin Heidelberg, 2004, pp. 77-87.

[5] E. Fares, "Unsteady flow simulation of the Ahmed reference body using a lattice Boltzmann approach," *Computers & Fluids*, vol. 35, pp. 940-950, Sept. 1, 2006.

[6] M. Minguez, R. Pasquetti, and E. Serre, "High-order large-eddy simulation of flow over the "Ahmed body" car model," *Physics of Fluids*, vol. 20, pp. 095101, 2008.

[7] V. Uruba and O. Hladík, "On the Ahmed Body Wake," in *Colloquium Fluid Dynamics*, 2009, pp. 21-23.

[8] B. Conan, J. Anthoine, and P. Planquart, "Experimental aerodynamic study of a car-type bluff body," *Experiments in Fluids*, vol. 50, pp. 1273-1284, May 01, 2011.

[9] M. Gad-el-Hak, "Modem Developments in Flow Control," *Applied Mechanics Reviews*, vol. 49, pp. 365-379, 1996.

[10] W Hucho and G. Sovran, "Aerodynamics of Road Vehicles," *Annual Review of Fluid Mechanics*, vol. 25, pp. 485-537, 1993.

[11] A. K. and P. Gilliéron, "Impact of the automotive aerodynamic control on the economic issues," *Journal of Applied Fluid Mechanics*, vol. 2, pp. 69-75, 2009.

[12] W. H. Hucho, "Aerodynamik der stumpfen Körper," *Physikalische Grundlagen und Anwendungen in der Praxis*, Vieweg, 2002.

[13] M. Rouméas, P. Gilliéron, and A. Kourta, "Analysis and control of the near-wake flow over a square-back geometry," *Computers & Fluids*, vol. 38, pp. 60-70, Jan. 1, 2009.

[14] C.-H. Bruneau, E. Creusé, D. Depeyras, P. Gilliéron, and I. Mortazavi, "Active procedures to control the flow past the Ahmed body with a 25° rear window," *International Journal of Aerodynamics*, vol. 1, pp. 299-317, 2011.

[15] M. Rouméas, P. Gilliéron, and A. Kourta, "Drag reduction by flow separation control on a car after body," *International Journal for Numerical Methods in Fluids*, vol. 60, pp. 1222-1240, 2009.

[16] Y. Utturkar, "Numerical investigation of synthetic jet flow fields," University of Florida, 2002.

[17] B. L. Smith and A. Glezer, "The formation and evolution of synthetic jets," *Physics of Fluids*, vol. 10, pp. 2281-2297, 1998.

[18] A. Glezer, "The formation of vortex rings," *The Physics of Fluids*, vol. 31, pp. 3532-3542, 1988.

[19] B. Smith and G. Swift, "Synthetic jets at large Reynolds number and comparison to continuous jets," in *15th AIAA Computational Fluid Dynamics Conference*, ed: American Institute of Aeronautics and Astronautics, 2001.

[20] R. Holman, Y. Utturkar, R. Mittal, B. L. Smith, and L. Cattafesta, "Formation criterion for synthetic jets," *AIAA Journal*, vol. 43, pp. 2110-2116, Oct. 1, 2005.

Ramon Trisno, S.T., M.T.. Born in Matur, Agam District, West Sumatera on November 16, 1971. He started to study in Mechanical Engineering at Andalas University in 1990 and obtained his Bachelor of Engineering (S.T.) in 1997. Appointed as a permanent lecturer at the Department of Mechanical Engineering at Pancasila University, Jakarta in 1998. Continued

education to Master degree in 2000 at the Department of Mechanical Engineering, Universitas Indonesia and obtained his Master of Engineering (M.T.) degree in 2003. Currently, he is continuing his doctoral degree in Doctoral Program of Mechanical Engineering at Universitas Indonesia in 2013. Currently active in research in the field of fluid mechanics especially regarding the control of the flow on the vehicle.

Prof. Dr. Ir. Harinaldi, M.Eng. Born in Jakarta on October 30th, 1968. He earned his bachelor degree in Mechanical Engineering at Universitas Indonesia in 1992, Master of Engineering degree in Combustion Engineering in 1997 at Keio University, and Doctor of Engineering in the field of thermophysical engineering in 2000 at Keio University, Japan. During his post-graduate studies in Japan, sponsored by The Hitachi Scholarship Foundation, he did many part-time work in various companies in Japan with a variety of fields, ranging from automotive industry in Isuzu Motor Company, Kawasaki, freelance translator in several publishing/translation companies until became a presenter on Radio Nippon Hosou Kyouku (NHK-World), Japan for more than 5 years (1996-2000). The main activity since 1993 is as a permanent lecturer in the Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, and has served as Head of Fluid Mechanics Laboratory since 2001 until now. His scientific works on the thermofluidics have been published in various proceedings of national and international scientific seminars and journals since 1998. The most recent publications are in the International Journal of Heat and Mass Transfer (2001), JSCE International Journal (2001), International Symposium of Transport Phenomena (2001-2005), National Seminar of Mechanical Engineering (2002-2003), etc. Some of the published scientific publications are: (1) Coherent structures in a turbulent layer with a transverse jet flow perturbation, (2) Flame stability and fuel efficiency of the propane and butane in the recirculation flow on the backward facing step, (3) Stabilization On Flame Diffusion LPG With Jet Slot Type Injection in Recirculation Flow Field, (4) Effect of Airflow Variation on High Lifted-Flame on Propane Propusion Burning, etc.

Dr. Ir. Engkos A. Kosasih, M.T. Born in Tasikmalaya on June 6, 1959, earned his bachelor degree in Mechanical Engineering at Universitas Indonesia in 1991. Then, he completed his Master's degree in Mechanical Engineering at Bandung Institute of Technology in 1997 and Doctoral degree at Universitas Indonesia majoring in mechanical engineering in 2006. Currently, he is working as a lecturer in Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia. He started leccturing in 1992. He is also active in writing scientific publications, some of his publications are: (1) Comparison of Evaporation Rate of Aquades and Sugar Drops, (2) Effect of Downcomer on Outside Side Separator Type of Cyclone: A CFD Simulation, (3) Testing of Air Heaters Control System & Simulation Effect of Thermal Constraints Heating Elements Response, (4) Evaporation of water film (falling film) in a vertical cylinder, etc.

The Influence of the Excitation Frequency of the Cavity Changes of Synthetic Jet Actuator to the Formation of Vortex Ring

ORIGINALITY REPORT

18%	13%	8%	7%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

MATCH ALL SOURCES (ONLY SELECTED SOURCE PRINTED)

5%

★ uad.portalgaruda.org

Internet Source

Exclude quotes	On	Exclude matches	< 10 words
Exclude bibliography	On		