© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

Journal homepage: http://penerbit.uthm.edu.my/ojs/index.php/ijie

ISSN : 2229-838X e-ISSN : 2600-7916

The International Journal of Integrated Engineering

Design and Investigate of Flushing System for Electrical Discharge Machining (EDM) Application

Haris Rachmat[1], Rasidi Ibrahim[2*], Kee Hem Ming[2], Al Emran Ismail[2], Zulafif Rahim[2], Worapong Swangsari[1]

[1]Fakulti Rekayasa Industri,
Telkom University, Bandung, 40257, INDONESIA

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

*Corresponding Author

DOI: https://doi.org/10.30880/ijie.2020.12.02.026
Received 19 April 2019; Accepted 9 January 2020; Available online 28 February 2020

Abstract: Electrical Discharge Machining (EDM) is high precision machining process in which no actual contact between the workpiece and electrode during sparking. Dielectric fluid play a role as flushing medium and semiconductor between workpiece and electrode to stabilization and controlled spark gap ionization condition. In real condition, nozzle flushing system in EDM machine not able to complete remove debris formed during machining and affect the machining performance. Improper flushing due to lack of guideline at setup position of nozzle and inlet pressure caused low material removal rate, irregular tool and higher cost on raw material. To overcome this problem, the design and investigate of flushing system in EDM application is required. The design and investigation undergo by simulation of ANSYS Computational Fluid Dynamics (CFD) with a virtual experiment to accurate prediction of flushing performance. The influence of nozzle size and inlet pressure supplied on flushing efficiency were analyzed to avoid improper flushing on die-sinking EDM process. The simulation and experiments clarified that the higher inlet pressure, P=0.20 bar and larger nozzle diameter, D=6mm resulting in higher total pressure which is 2647.16 Pa. Furthermore, the streamline of velocity and eddy viscosity contour in the work tank using to analyze the turbulence zone by nozzle flushing obtained by the CFD analysis. The condition in case 5 (D=5mm, P=0.15 bar) is more efficiency on debris removal rate based on the result of high total pressure on machining zone and eddy viscosity contour showed the turbulence zone only formed area near to outlet of system. The model results have been shown good agreement with experiment and co-relation data.

Keywords: Flushing system, Electrical Discharge Machining, nozzle size, nozzle inlet pressure, CFD simulation

1. Introduction

Electrical Discharge Machining, also known as EDM, is the cutting process in which no actual contact between the workpiece and electrode during the machining (sparking) [1]. The limitation to use a conventional machine to cut hard materials has divert the non-conventional machine such as electrical discharge machining (EDM) as one of the preferred techniques for the hard materials [2]. Nowadays, EDM applied in many field of industries such as in automobile industry, chemical industry and aerospace industry. The reason is manufacturing process on conduct electricity materials was freedom by its hardness or toughness in EDM operation and also for many types of machining [3,4]. During EDM operation, the partial ionization of the workpiece will cause debris formed in electrode to workpiece spacing. The particles resulting purity of dielectric liquid from "pure" to "impure" state and affected to dielectric

**Corresponding author: harisrachmat@telkomuniversity.ac.id*

strength liquid. When the concentration of the debris exists at spark gap was too will cause the damage of the electrode and the machining work if the control system does not stop it instantly.

Flushing is the one of the importance factors in non-electrical parameters in the EDM process. Flushing was generated fluid flow to remove the eroded particles from gap between electrode and workpiece. Basically, flushing methods categories into 4 types which are normal flow, reverse flow, jet flushing and immersion flushing [5]. The simplest and primitive flushing method in EDM operation is jet flushing and the development of flushing system will focus on the jet flushing. The method is ejected dielectric fluid from a nozzle towards the machining area. Nozzle is designed to increase the kinetic energy of flowing fluid at the expense of its pressure and the direction and characteristics of the fluid can be controllable [6]. In jet flushing some of important considerations are distribution of the nozzles, flow rates, angles at which the nozzles are directed at the gap, and layout of the cavities.

However, there is no guideline on the suitable flushing distance to obtain the required flushing pressure and it is not easy to determine the most appropriate position of the nozzle(s). This position is usually decided by operators an dependence on their experience, often without confidence [7]. Most industries divided EDM and other machining process into two phases which are rough machining and finishing machining [8]. By providing optimum flushing condition in EDM process will cause some cons for operation such as time required in planning and providing the best position for the nozzle. But the flushing pressure applied was constant and not present the highest flushing efficiency and because improper flushing occurs in EDM operation [9]. Improper flushing pressure will cause the improper debris removal that consequently reduce material removal rate, increase in electrode wear rate and surface roughness of specimen [10].

Due to the above scenario, the design and investigate of flushing system in EDM application is required in order to observe on the limitation and performance of the EDM operation as well as the design and investigate of the flushing system. The system will be simulated by ANSYS Computational Fluid Dynamics (CFD) with a virtual experiment to accurate prediction its performance. CFD methods enable the exploration of many design and operational parameters that are difficult to assess experimentally [11]. ANSYS Computational Fluid Dynamics (CFD) is computer software used to solve and analyze problems that involve fluid flow. ANSYS CFD is able to provide comprehensive graphical representation of the phenomena that occur in the spark gap, including total flushing pressure, velocity streamline and eddy viscosity of fluid in work tank. The simulation and analyzed result can be validated in real EDM application and other non-traditional process [12]. Therefore, this study is initiated with the aim of design and investigating the flushing system for EDM application to avoid improper flushing and improve the performance of EDM by simulate, compare and validate the flushing pressure element.

2. Methodology

2.1 Experiment Setup

Total pressure in fluid medium at any reference point could be captured with miniature pressure transducers. The installation of pressure transducers was shown in Fig.1. The position of the pressure transducer gripped by metal holder with a height 0.5cm from top surface of the workpiece. The pressure transducer was setup at eleven position, the first position at the center point of the workpiece and called as x=0 cm. Another ten positions of pressure transducer were 2.5 cm each distance along the x-axis toward outlet of the system as shown in Fig. 2.

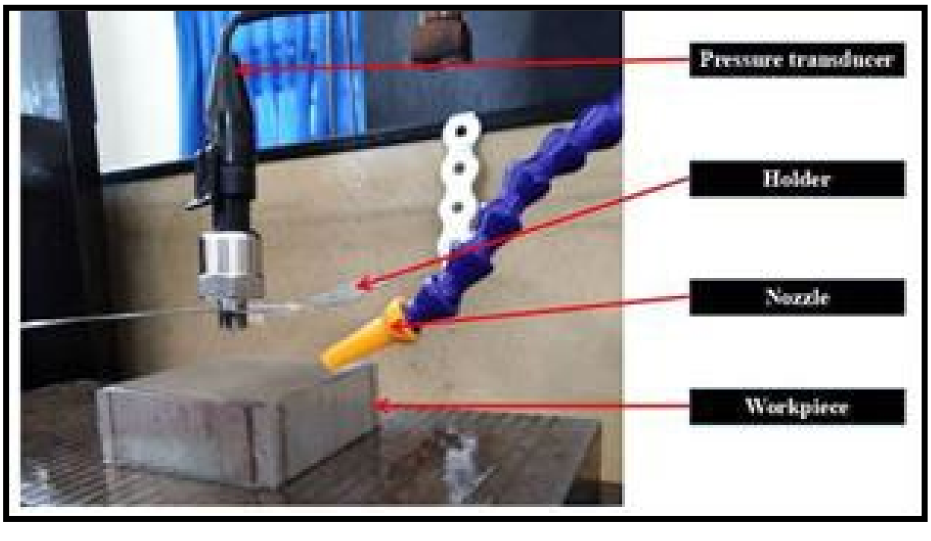


Fig. 1 - Setup of pressure transducers

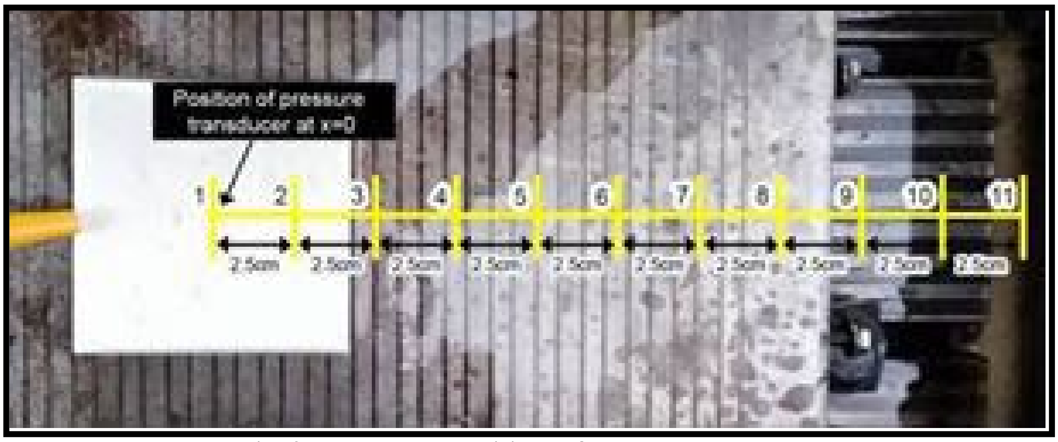


Fig. 2 - The eleven positions of pressure transducer

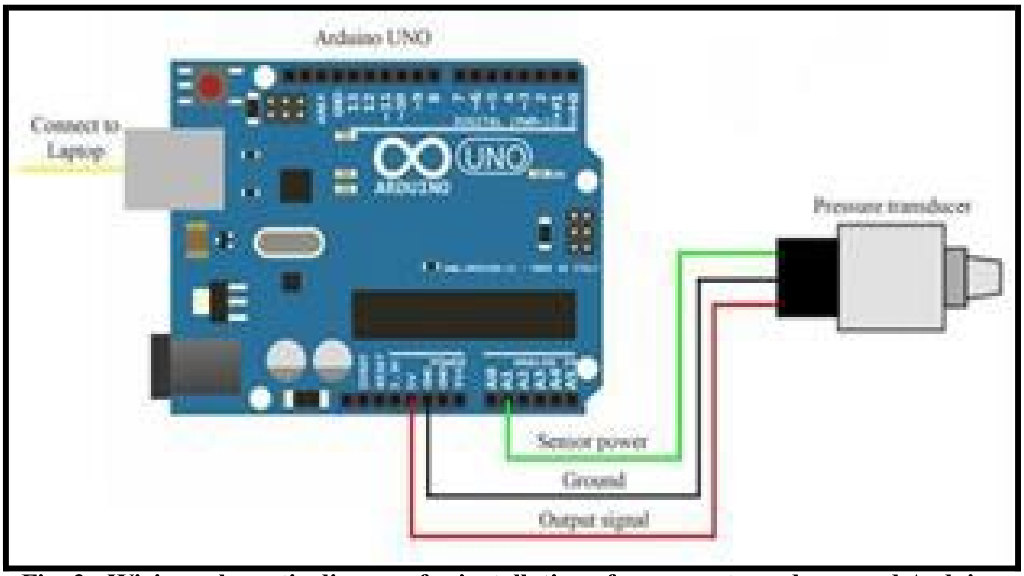


Fig. 3 - Wiring schematic diagram for installation of pressure transducer and Arduino

2.2 Pressure Measurement Apparatus

The instrument connected to Arduino Compatible Atmel DIP and linked to Laptop and the wiring schematic diagram of showed in Fig. 3. The software and coding were used to calculate the differential pressure at the eleven positions.

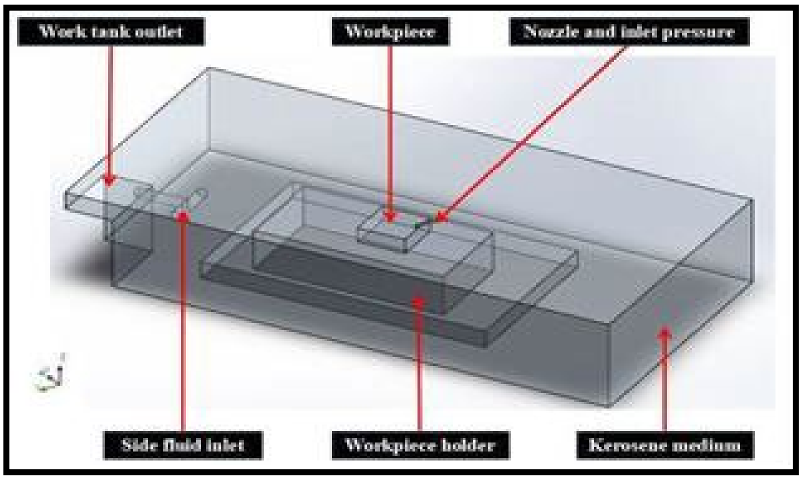
3. Simulation Calculation

3.1 Physical Model

The simulation objects is the flow field of the nozzle jet issuing in to a rectangular work tank which fully fill with kerosene. Coordinate system associated with the nozzle is simulated. The structure model is simplified based on the structure of nozzles and rectangular tank. The model is built according to the Roboform 100 EDM machine size and mesh is generated as shown in Fig. 4.

Table 1 - Mesh result by using ANSYS FLUENT

Nozzle diameter (mm)	Volume (m^3)	Nodes	Elements
4	0.047359	48314	256340
5	0.047359	45106	237993
6	0.047359	43036	226492



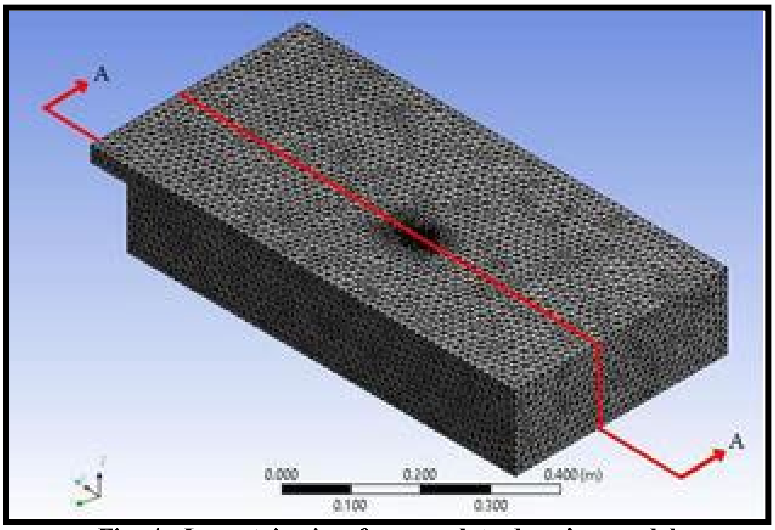


Fig. 4 - Isometric view for complete drawing model

Fig. 5 - Mesh Generation

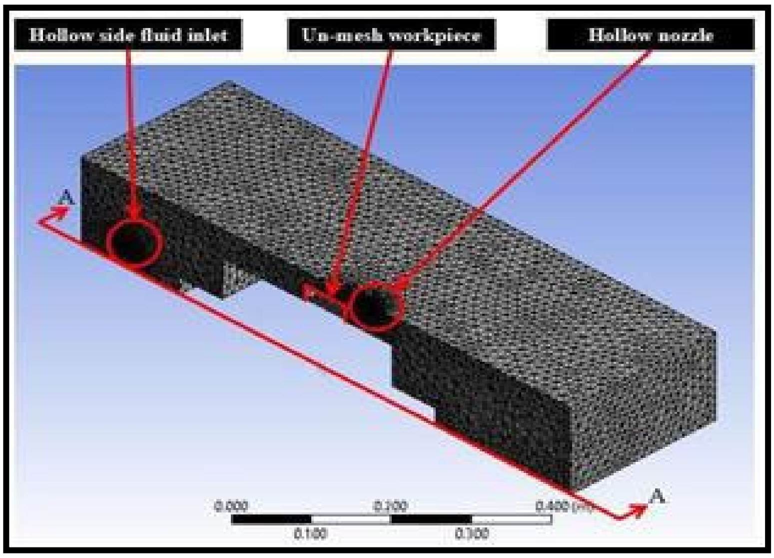


Fig. 6 - A-A section plane vi of model after mesh

3.2 Boundary Condition

Boundary conditions are specified as follows. Firstly, the inlet boundary condition is the nozzle inlet pressure which is equal to 0.1, 0.15 and 0.2 Bar in each follow by experiment. Other constant inlet boundary condition is the side fluid inlet which is 0.1 Bars in every case. Next is the outlet boundary condition is the pressure outlet which is equal to surrounding flow field pressure. For the axial boundary condition, the radial velocity is 0. Another boundary conditions is the solid wall surface condition, the wall satisfies no slipping condition, closing the wall area adopting wall function method. Standard Viscous Model k-epsilon (2eqn) model is used as solver setup.

4. Result and Analysis

Fig. 7 shows the result of total pressure distribution at the reference line of the fluid model. The graph comparing all the case study which shows the total pressure of the system versus the distance from 0 to 5cm on the reference line. All the case forms a decline trend or pressure drop from the initial point. When P=0.20 bar and D=6mm, the case shows the highest total pressure and produce a good material removal process from the machining zone. The influence of nozzle size and nozzle inlet pressure on total pressure on workpiece surface was showed in the diagram. The higher the nozzle size in the same nozzle inlet pressure, the higher the total pressure on workpiece surface. In addition, the higher nozzle inlet pressures in same nozzle size, the higher the total pressure on workpiece.

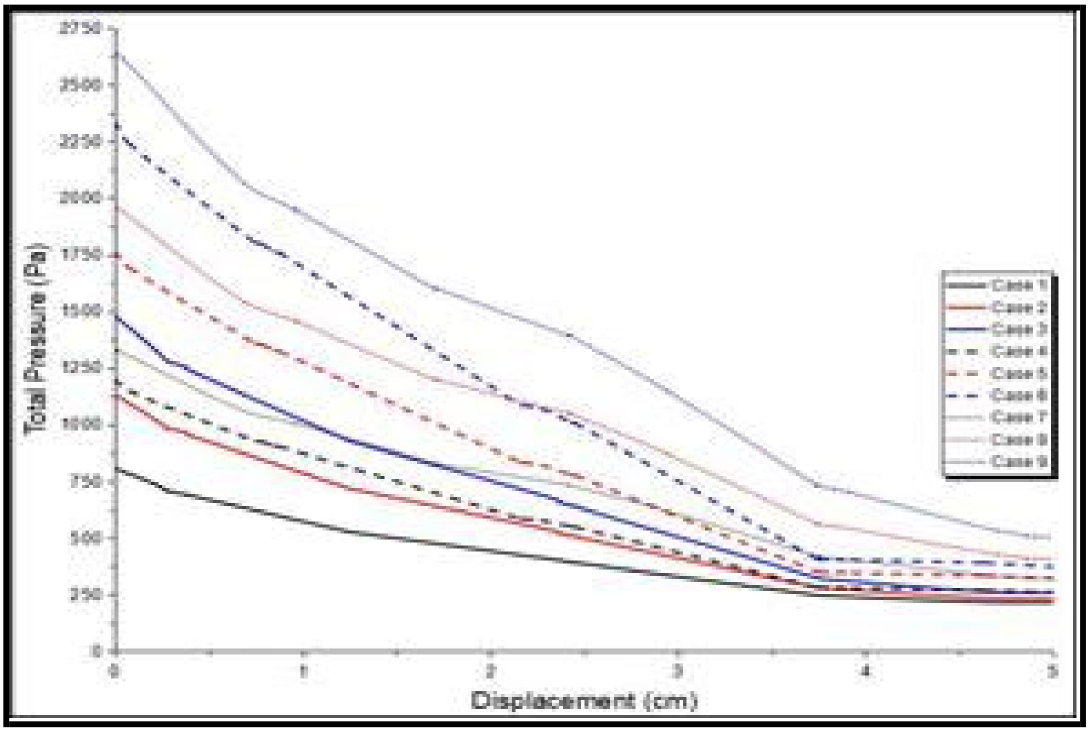


Fig. 7 - Pressure distribution for all cases at reference line of geometry model

Eddy viscosity was known as the turbulent transfer of momentum by eddies giving increase to an internal fluid fiction, in a similar pattern to the action of molecular viscosity in laminar flow but replaced on a much larger scale. It was also called as turbulent viscosity. Viscosity cause shear stress in response to shearing of the flow and eddies cause a similar effect. However, they do it in different conditions which were physically viscosity was moving faster fluid into slower regions and eddies moving slower fluid into faster regions [13]. Lots of little eddies make the fluid become more viscosity. In other word, the turbulence would increase when eddy viscosity rise. Fig. 8 showed the eddy viscosity contour in each case generated by ANSYS FLUENT. In the figure, all the contours were compared and their major colour were showed in green colour (higher eddy viscosity) and blue colour (low eddy viscosity). The circle zone was consisting higher value of eddy viscosity in the tank, at the area turbulence will be formed [14].

All the cases showed 4 to 6 turbulence zones in the fluid medium tank and high possibility debris will be stay in the turbulent zone. So that the turbulence zone located at the right-hand side of the tank (near to outlet) would increase the removal rate of debris from the tank [15]. When turbulence zone created at left hand side or middle of the tank, the debris remained in the fluid medium and resulted the purity of kerosene into impure condition. In cases D=5mm, most of the high eddy viscosity zone located at right hand side compared to cases D=4mm or D=6mm. Therefore, this nozzle diameter setting can have generated turbulence flow near to outlet of the flushing system and let removal rate of debris from tank increased. When pressure inlet of nozzle setup as 0.15 bar, the green color zone (higher eddy viscosity) were lesser comparing to cases P=0.10 bar and P=0.20 bar. In this situation, the flow of the fluid mainly in laminar flow, the regular flow direction to strengthen the kinetic energy to carried up debris in the tank. According to both conditions, the ascending sequence of material removal rate from for the flushing system for all case which were case 1/ case 3/ case 7/ case 9, case 4/ case 6, case 2/ case 8 and case 5. The case showed best flushing performance by compared turbulence zone location will be case 5.

Validation for this study was the comparison between the experimental data and the simulation result. In experimental, the density of the fluid (kerosene) was not actually as the value set in FLUENT because in real case the fluid had mixing with some debris and some components had been ignored when simulation geometry model built. The validation results only compared the result of the simulation by FLUENT and the experimental for case 5 (P=0.15 bar, D=5mm) due to the cases showed good result in flushing efficiency. The result of the validation shown in Fig 9. The trend of the graph shows a similar pattern of the total pressure result for experimental and simulation by Computational Fluid Dynamics FLUENT in both cases. However, the value of was different for both cases. As an example, in case 5 at the initial values of the total pressure at the distance x=0 is 1952.75 Pa in experimental result and 1764.0 Pa in ANSYS FLUENT result, the percentage of the error is about 9.67%.

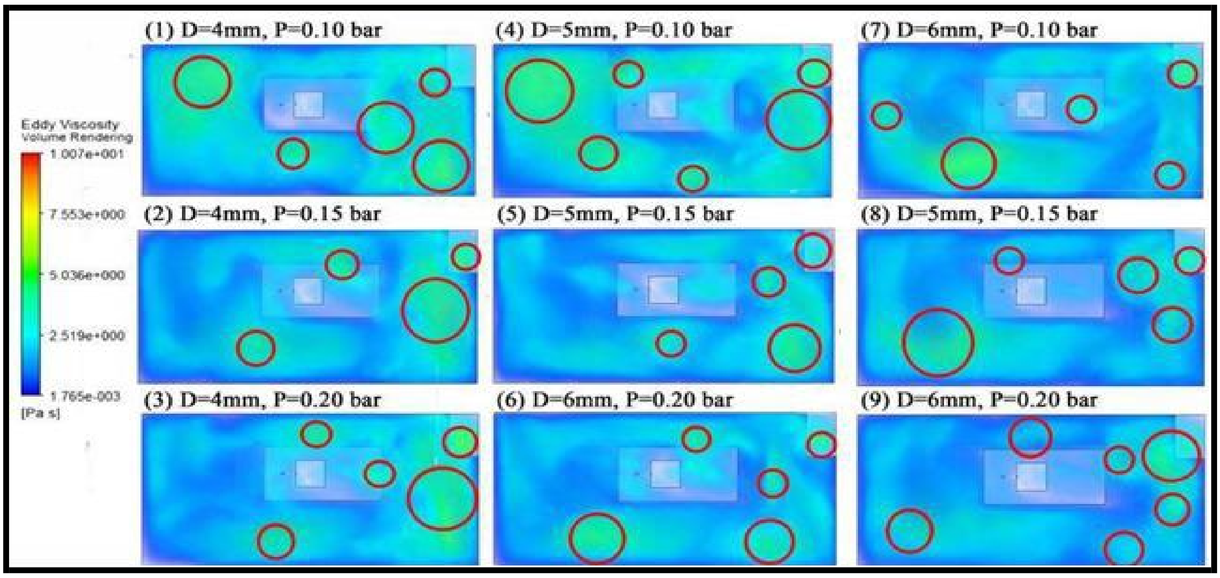


Fig. 8 - Eddy viscosity diagram of P=0.10 bar, P= 0.15 bar and P=0.20 bar for D=4mm, D=5mm and D=6mm

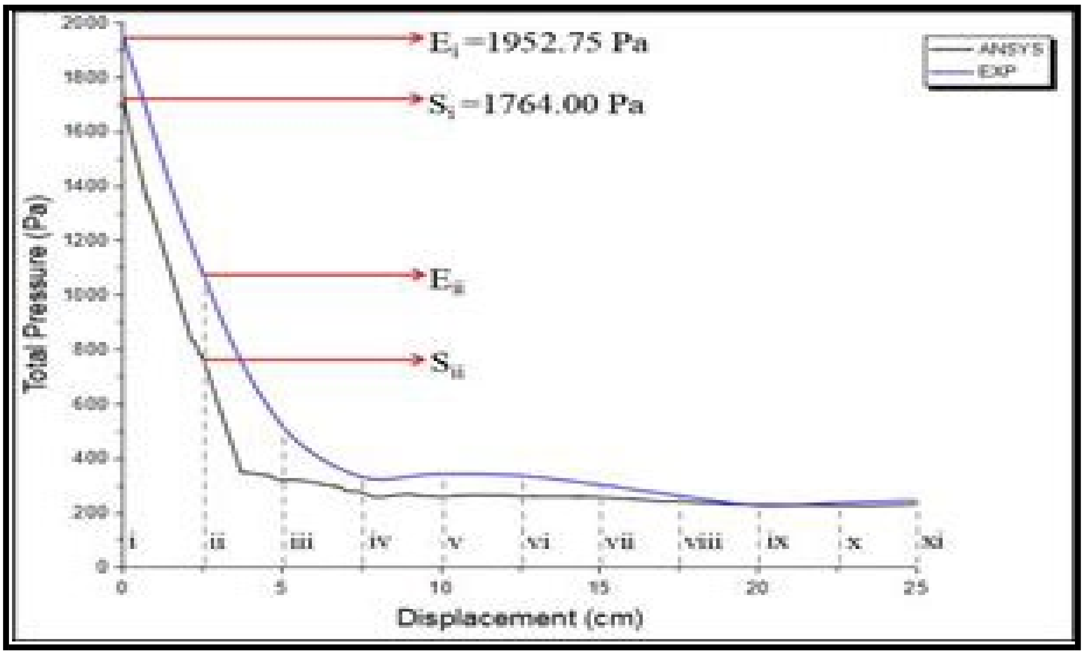


Fig. 9 - Total pressure by using case 5 for validation result by comparing the experimental data and FLUENT result

Table 2 - Calculation of Root Mean Square (RMS) for case 5

Point	Displacement (cm)	Simulation, S (Pa)	Experiment, E (Pa)	Error
i	0.0	1764.000	1952.75	-0.0967
ii	2.5	775.699	1085.98	-0.2857
iii	5.0	316.618	522.42	-0.3939
iv	7.5	273.373	330.78	-0.1736
v	10.0	258.174	340.79	-0.2424
vi	12.5	259.127	334.27	-0.2248
vii	15.0	255.648	301.66	-0.1525
viii	17.5	239.882	261.47	-0.0826
ix	20.0	227.633	225.13	-0.0111
x	22.5	222.996	234.15	-0.0476
xi	25.0	228.994	243.65	-0.0602
		Root Mean Square (RMS)		0.1958
		Percentage		19.58

From the observation, the total pressure was resulted the different value of simulation from the experiment. In this case, the error may cause by highly turbulent flow in the fluid and made the standard k-ε turbulence model cannot deliver the exact predictions for the total pressure result [16]. The sketching of geometry model may not accurate as the exact dimensions and some accessories of the machine in the tank had be ignore in drawing. This it will affect the result obtained from the FLUENT simulation. Besides that, due to the measurement during experiment may occur some random error and parallax error would affect the experimental result. The standard k-ε turbulence model may not suitable for this case. The results are based on using the Renormalization-Group (RNG), the Realizable k-ε model, and the Reynolds stress model (RSM) were quite different and better to predict the flow in the tank compared to k-ε turbulence model. Therefore, the method used the non-suitable turbulence model can give the impacts of the simulation. During the pre-processing (meshing), the process also would affect the results [17-19]. By applying the unsuitable mesh, the calculation will hard to converge. If the setup is not suitable for the curvature angle, it will create a skew that will result in the simulation.

5. Conclusion

In this research, the flushing system in Electrical Discharge Machining (EDM) was developed. The implementation of the system consisted two phases which were the simulation of the developed flushing system and validation the result by real EDM flushing system. In simulation, the fluid flowed in the system had been simulated by ANSYS Computational Fluid Dynamics (CFD) software and followed the exact geometry of Roboform 100 EDM machine's work tank which was the machine run for real experiment. The simulation used three pressure inlet and three nozzle diameter and set at the top of the workpiece with height 0.5 cm and incline angle 30 as the manipulated variables. From the result, the best condition of the flushing system that suitable used in EDM flushing system was the 5mm of nozzle diameter and 0.15 bar inlet pressure supplied to nozzle which results of high total pressure along the reference line, streamline and eddy viscosity contour supported the statement where high total pressure occurred on the top surface of workpiece and also form high turbulence zone close to outlet of work tank. Strong flushing pressure in machining zone when the higher inlet pressure was applied with small nozzle size, the strong axial flow would remove debris from the machining zone. However, high flushing pressure also carried bad influence on the system such as more turbulence flow created at edges and sides of work tank caused the debris remained in the fluid medium and rose the impurity of dielectric fluid. Therefore, the best condition was the case fulfilled with high flushing pressure and less turbulence zone generated in fluid medium. Based on the validation result, Root Mean Square (RMS) method showed 19.58 % error of ANSYS FLUENT simulation with the experimental data in case 5.

References

[1] Abdulkareem, S., Ali Khan, A., Shah, Q. & Konneh, M. (2017). Temperature Distribution in Copper Electrode during Electrical Discharge Machining Process. International Journal of Integrated Engineering, 9(1), 39-43.

[2] Ahmad, S., Lajis, M. A., Abdul Haq, R. H., Tajul Arifin, A. M., Abdol Rahman, M. N., Ho F. H., Abdullah, H. & Hassan, M. F. (2018). Surface Roughness and Surface Topography of Inconel 718 in Powder Mixed Dielectric Electrical Discharge Machining (PMEDM). International Journal of Integrated Engineering: Special issue 2018: Mechanical Engineering, 10(5), 181-186.

[3] Ho, K. H., & Newman, S. T. (2003). State of the art electrical discharge machining (EDM). International Journal of Machine Tools and Manufacture, 43(13), 1287-1300.

[4] Rasidi, I., Rafai, N. H., Rahim, E. A., Kamaruddin, A., Ding, H., & Cheng, K. (2015). An investigation of cutting mechanics in 2-dimensional ultrasonic vibration assisted milling towards chip thickness and chip formation. IOP Conference series: Materials Science and Engineering, 100(1).

[5] Wong, Y. S., Lim, L. C., & Lee, L. C. (1995). Effects of flushing on electro-discharge machined surfaces. Journal of Materials Processing Technology, 48(1-4), 299-305.

[6] Masuzawa, T., Cui, X., & Taniguchi, N. (1992). Improved Jet Flushing for EDM. CIRP Annals, 41(2), 239-242.

[7] Rachmat, H. Mulyana, T. Hassan, S. B. H. & Ibrahim, M. R. (2017). Design selection of In-UVAT using MATLAB fuzzy logic toolbox, Advances in Intelligent Systems and Computing, 549, 538-545.

[8] Ibrahim, M. R., Rahim, E. A., Ghazali, M. H. & Goh, O. (2014). Experimental analysis on ultrasonic assisted turning (UAT) based on innovated tool holder in the scope of dry & wet machining. Applied Mechanics and Materials, 660, 104-108.

[9] Guitrau, E. B. (1997). The EDM Handbook. Hanser Gardner Publications, 23-45.

[10] Munz, M., Risto, M., & Haas, R. (2013). Specifics of flushing in electrical discharge drilling. Procedia CIRP, 6, 83-88.

[11] Mohd Nazrul Aman, N. A., Muchtar, A., Somalu, M. R., Rosli, M. I., Baharuddin, N. A. & Kalib, N. S. (2018). A short review on the modeling of solid-oxide fuel cells by using computational fluid dynamics: assumptions and boundary conditions. International Journal of Integrated Engineering: Special issue 2018: Mechanical Engineering, 10(5), 87-92.

[12] Ibrahim, R., Bateman, R., Cheng, K., Wang, C. & Au, J. (2011). Design and analysis of a desktop micro-machine for vibration-assisted micromachining. Proceedings of the Institution of Mechanical Engineers Part B Journal of Engineering Manufacture, 225(8), 1377-1391.

[13] Ozinsky, A. E. & Ekama, G. A. (1995). Secondary settling tank modeling and design. Water SA, 21(4), 325-332.

[14] Solorio-Diaz, G., Ramos-Banderas, A., Barreto, J. & Morales, R. D. (2008). Modeling Study of Turbulent Flow Effect on Inclusion Removal in a Tundish with Swirling Ladle Shroud. Steel Research International, 80(3).

[15] Chung, T. J. (2002). Computational Fluid Dynamics, Cambridge University Press, New York, London, 106-110.

[16] Launder B. E. & Spalding D. P. (1974). The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3, 269-289.

[17] Yakhot, V. & Smith, L. M. (1992). The Renormalization Group, the Epsilon Expansion and Derivation of Turbulence Models. Journal of Scientific Computing, 7, 35-51.

[18] Ibrahim, R. Rafai N.H., Rahim, E.A., Cheng, K. Ding, H. (2016). A Performance of 2-Dimensional Ultrasonic Vibration Assisted Milling in Cutting Force Reduction, on Aluminium AL6061. ARPN Journal of Engineering and Applied Sciences, Vol. 11, Issue 18,11124-11128.

[19] Nisizima, S. and Yoshizawa, A. (1987). Turbulent channel flow and couette flows using an anisotropic − ε model. AIAA Journal, 25(3), 414-420.