International Journal of Infrastructure Research and Management
Vol. 11 (1), June, 2023, pp. 53 - 68

COMPUTATIONAL FLUID DYNAMICS (CFD) STUDY OF FLOW DEVELOPMENT IN CONCENTRIC ANNULUS AND ECCENTRIC ANNULUS USING TAGUCHI METHOD

Lau Wei Jen & Mohd Shahrudin Mohd Alias
Infrastructure University Kuala Lumpur, MALAYSIA

ABSTRACT

Computational Fluid Dynamics (CFD) provides an alternative approach to study about fluid flow in various complex geometries, providing options through the simulations and allowing an alternative form for theoretical advances. The purpose of this study is to find out how the type of flow and the type of annulus affects the overall output. The objectives of this study are to study the fluid flow behaviour with different annulus geometries and investigate the optimum effect of different flow regimes with dissimilar annular passages on velocity profiles using the Taguchi method. In this study, a series of simulations are done for fully developed laminar flow and turbulent flow in different annulus geometry and observed the flow behaviour in the annulus. The factors used in the simulations are flow regime and annulus geometry. It is found that the flow development in the concentric annulus is uniform but non-uniform in the eccentric annulus. The concentric annulus with laminar flow combination gave the lowest maximum velocity, which is 0.0989 m/s at the outlet of the annulus, while the eccentric annulus with turbulent flow combination gave the highest maximum velocity of 2.246 m/s. Through the analysis of the Taguchi method and ANOVA technique, the combination of the eccentric annulus and turbulent flow showed the highest fluid flow velocity with flow regime being the most significant factor (P-value of 0.011) based on the significant value (P-value less than 0.05 is statistically significant). The investigation through the Design of Experiment (DOE) and ANOVA technique should provide the reference for future studies in effectively identifying the effect of other factors such as the annulus design and the operating condition of the cylinders. The suggestions for future studies include studying the effect of drilling mud as the type of fluid and studying the effect of inner shaft rotation on the output by using DOE strategies.

Keywords:
CFD, Fluid Flows, Concentric and Eccentric Annuli, Taguchi Methodology, ANOVA

INTRODUCTION

The study of fluid flow is significantly important in various fields such as the oil and gas industry when engineers deal with Newtonian and non-Newtonian fluids in the eccentric annulus or concentric annulus. The eccentricity of an annulus is the amount of displacement of a geometric centre of a rotating or non-rotating part from the true centre. The eccentric annulus will have the same cross-sectional area as the concentric annulus but a different fluid flow profile. A 0.0 pipe eccentricity means that the inner part is perfectly centred in the annulus (100% casing standoff). Vice versa, an eccentricity of 1.0 means that the inner part is in close contact with the annulus (0% casing standoff).

Annular flow performance is vital to be studied as it significantly affects the planning and design of the hydraulic program for a well-drilling operation (Mohammed Nawai, 2011). This is due to pressure may be lost at a major scale in a constricted space. An eccentric annulus will have a lesser frictional pressure drop than a concentric annulus as the gap is wider on one side for fluid to flow. Higher eccentricity allows fluid to flow with lesser energy required. A concentric annulus will have a uniform axial velocity profile, while an eccentric annulus will have a slower fluid flow narrow side of the pipe, which may cause a problem like solid accumulation if not appropriately studied.

One of the major challenges in designing a hydraulic program is to precisely estimate the pressure drop of drilling fluid due to friction in a wellbore annulus. Several parameters cause frictional pressure losses of the drilling fluid, such as the rheology of the drilling fluid, flow regime and pipe

ISSN Print: 2811-3608
ISSN Online: 2811-3705
https://iukl.edu.my/rmc/publications/ijirm/
DOI:10.63646/j.jdsti.2023110105

eccentricity (Mohammed Nawai, 2011). Problems like overestimating pressure drops will occur if the pipe eccentricity is not considered during the designing of the program as eccentric annulus will decrease the frictional pressure losses considerably.

In a well-drilling operation, it is crucial for frictional pressure loss calculations as the calculation allows the determination of horsepower required to operate the drill. The frictional pressure loss calculation will also be used to determine the bottom hole treating pressure and maximum wellbore pressure. Optimising the drilling parameter through the calculation allows engineers to make an applicable evaluation of wellbore hydraulics. It will reduce drilling problems encountered and at the same time, reduce drilling operation costs.

The velocity of fluid flow in different geometries is studied. The concentric annulus and eccentric annulus are studied through the simulation by using Computational Fluid Dynamics (CFD) method in ANSYS Fluent, where the meshing of the annulus is conducted. ANSYS is one of the software used for finite element analysis (FEA) other than ABAQUS and DIANA. According to Zhicheng and Nordin (2022), ANSYS is less employed in papers published in recent years than ABAQUS. Thus, the selection of ANSYS software in the research may further contribute to the FEA study's coverage of the software application. Taguchi method is used to study the effect of different fluid flow regimes (laminar flow and turbulent flow) in different geometries. The simulation runs are set according to the Orthogonal Array Design in the Taguchi method and the data is analysed by using the ANOVA technique with the aid of Minitab software.

Problem Statement

One of the downsides of drilling horizontal and inclined wells is that wellbore eccentricity will occur due to the inclination (Kelessidis & Dukler, 1989). The eccentric annulus differs from the concentric annulus as the inner part is offset to one side of the annulus. The eccentric annulus will have a non-uniform fluid velocity profile. The stagnant zone in the eccentric annulus due to low fluid flow velocity causes the cutting removal process during the drilling operation to be inefficient. The pressure losses in an eccentric annulus are only 40% of the value in a concentric annulus. From an industrial application standpoint, the issue regarding fluid flow in the annular space between two circular cylinders is still encountered in many types of equipment and engineering applications due to a narrow annular space that exists in the clearance (Abou-Ziyan et al., 2021). Well-drilling (the annulus between a borehole and a drill pipe), as well as mixers and heat exchangers, are among the involved equipment as they are based on the same annulus design. Therefore, the similar flow behaviour of fluid in a concentric annulus cannot be implemented in an eccentric annulus. The flow simulation using Computational fluid dynamics (CFD) modelling is required to study this matter.

Due to the geometry of the concentric annulus and eccentric annulus being different, approximation of the flow regime in both the concentric annulus and eccentric annulus may be inaccurate in the calculation of heat transfer in designing the drilling program. Hence, many studies do not actually provide a solution as nearly all analysis focuses on only one type of flow regime on a specific geometry. Besides, a good hole cleaning allows the operation to avoid drilling issues. If the hole cleaning is failed to be ensured, major costly drilling problems such as cutting bed formation damage, fluid loss, lost circulation, high torque and drag, or stuck pipe may occur, where the cost may be up to millions of dollars to be covered by the company. Therefore, this study focused on the type of flow regime (laminar flow and turbulence flow) in affecting the velocity profile in both the concentric annulus and eccentric annulus.

Research Objectives

As fluids behave differently in the concentric annular passage and eccentric annular passage, it is important to study the effect of different geometries on fluid flow development. This study is conducted by using Computational Fluid Dynamics (CFD) method.

The flow regime in an annular passage can be laminar flow or turbulent flow, thus it is important to study different types of flow regimes in a different type of annulus. The effect of flow regime on velocity profiles is focused in this study as well.

The objectives of this study are to study the fluid flow behaviour with different annulus geometries using Computational Fluid Dynamics (CFD) and to investigate the optimum effect of different flow regimes with dissimilar annular passages on velocity profiles using the Taguchi Method.

LITERATURE REVIEW

Singh et al. (2021) conducted a CFD analysis on the turbulent flow of power-law fluid in a partially blocked eccentric annulus. They aim to study the turbulent flow of power-law fluid in wellbores and develop a relationship between bed shear stress and the level of blockage. When the blockage level of an annulus increases, the local fluid velocity and cutting bed shear stress are reduced. A series of simulations and analyses found that the bed height, diameter ratio, and fluid behaviour index will eventually affect the dimensionless bed shear stress, which is one of the crucial flow parameters. Besides, the authors stated that the thickness of the cutting bed would affect the annular pressure gradient, which is more noticeable when the dimensionless bed height is greater than 50%. Lastly, the authors used CFD simulation of turbulent flow in this study because CFD simulation provides a satisfactory projection of relevant flow parameters.

Ferroudji et al. (2019) did a numerical study on power-law fluid flowing through the concentric annulus and eccentric annulus using the finite volume method under the conditions of both laminar flow regime and turbulent flow regime with inner shaft rotation. The purpose of the study is to find out the effect of inner rotation on pressure drop in the concentric annulus and eccentric annulus under both types of flow regimes. The study also focused on the variation of flow behaviour index and diameter ratio from low to high values that affects the pressure drop. The study showed that the inner shaft rotation successfully reduces the pressure drop in a concentric annulus, but the effect is negligible if the flow regime is turbulent. The formation of secondary flow will increase the pressure drop slightly for both laminar and turbulent flow. The increase of the flow behaviour index and diameter ratio from a low to a high value will result in a higher pressure drop. They found out that to prevent secondary flow from forming, a low diameter ratio and high flow behaviour index of the eccentric annulus are needed.

Escudier et al. (2000) did a similar study that focus on the effects of inner cylinder rotation on the laminar flow of a Newtonian fluid through an eccentric annulus. The study carried out by Escudier et al. (2000) focuses on the fully developed laminar flow of a Newtonian liquid through an eccentric annulus with combined bulk axial flow and inner cylinder rotation. The flow behaviour in the annulus is analysed by the low, medium, and high eccentricity of the annulus. For low eccentricity ($E < 0.3$) annulus, the flow is rotation dominated. This means that the Fanning friction factor and Reynolds number may increase higher than the value of the non-rotating inner shaft concentric annulus. This is due to the flow being pulled around the inner shaft transferring the peak velocity in the wide gap to the narrow gap, which increases the axial gradient. As for medium eccentricity ($0.3 < E < 0.8$) annulus, the study showed that friction is reduced when eccentricity increases, resulting in a lower frictional pressure drop. However, secondary flow begins to develop, having the maximum axial velocity in the wide gap of the annulus. Lastly, for high eccentricity ($0.8 < E < 1$) annulus, the flow is recirculation dominant, but the authors observed an unexpected flow behaviour. The location of peak velocity is transferred into the widening gap region due to the stronger recirculation. When the flow moved towards the recirculating eddy, it is forced towards the "lower" surface of the inner cylinder, which produces high localised shear stress in the annulus. This reduces the Fanning friction factor and Reynolds number with eccentricity and then further prompts an increase.

Neto et al. (2011) conducted a study on the turbulent flow in the concentric and eccentric annulus with a rotating inner shaft by using Laser Doppler Velocimeter (LDV) technique to get the axial, radial, and tangential mean velocity profiles. By using ANSYS Fluent, the authors managed to carry out a simulation and obtained results with low relative errors. Contours of velocities in an eccentric annulus are included in the study to show the area of the maximum velocity of fluid flow in an eccentric annulus and how the inner rotating shaft affects the fluid flow. The authors used five (5) different turbulent models in the simulation and obtained simulation results that are close to each other. The authors claimed that the flow velocity in the narrow part of the eccentric annulus is much lower than the flow velocity in the wide part of an annulus with a non-rotating inner shaft, which means that cutting beds are easily deposited in the narrow gap of the eccentric annulus. In this study, they showed that the rotating inner shaft aids in preventing cutting bed deposition as the fluid flow is dragged towards the narrow gap of the eccentric annulus, which is similar to what Escudier et al. (2000) obtained from their study for low eccentricity ($E < 0.3$) annulus.

In the recent review summarized by Abou-Ziyan et al. (2021), they tabulated the fluid flow work for concentric and eccentric studies and indicate that this work still receives a major consideration. This includes the study of laminar flow, turbulent flow, stationary inner cylinder and rotating inner cylinder with various radius ratios. Further from the recent research findings, the effect of rotation on the annular space of radii ratios is still lacking in the literature where it becomes the interest in industrial applications including rotating heat exchangers, mixers, agitators and well drilling to work more efficiently (Abou-Ziyan et al., 2021). The study on the effect of radius ratio on eccentricity still needs to be investigated. In addition to the literature summarized by Abou-Ziyan et al. (2021), the author reviewed that various factors could lead to different findings such as the maximum axial velocity impacted by a different annulus, radius ratio, fluid properties and flow characteristics. These factors should be properly studied with a better approach in designing the experimental or simulation run without relying too much on a trial-and-error approach for optimum results. For this purpose, a Design of Experiment (DOE) methodology that emphasizes statistical analysis such as the Taguchi Robust Methodology should be applied.

Reynolds Number Equation

The higher the Reynolds number of a fluid type, the more turbulent the fluid is. If the fluid has a Reynolds number below 2100, the fluid flow is laminar. In the other hand, if the Reynolds number is more than 4000, the fluid flow is turbulent. If the Reynolds number is between 2100 and 4000, the fluid flow is considered transition flow. The Reynolds number can be determined by using the formula shown below:

$$Re = \frac{\rho v D}{\mu} \tag{1}$$

Where, Re = Reynolds number, ρ = Density of fluid, v = Velocity of fluid flow through the pipe,
D = Diameter of pipe (also referred to as hydraulic diameter, dh) , μ = Dynamic viscosity of the fluid

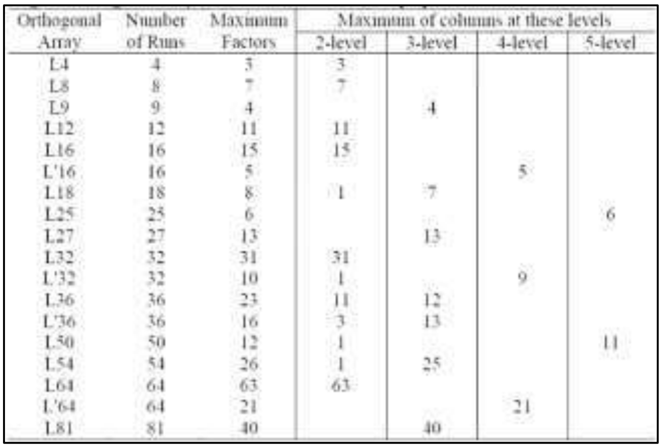
Taguchi Method and Analysis of Variance (ANOVA)

Taguchi method is also known as Taguchi design, which is a designed experiment that allows the choice of factors that function consistently in an actual operating environment (Minitab, 2021). Not every factor can be controlled; thus, these factors are known as noise factors. By using the Taguchi method, control factors are identified, and noise factors are minimised. A process output with the least noise factors will produce a consistent output and performance regardless of environmental conditions.

Taguchi method uses orthogonal arrays, which is a combination of factors to estimate the effects of factors on the response mean and variation. An orthogonal array produces a balanced design in that all factor levels are equally weighted. Due to the orthogonal array design, each factor is assessed independently so that one factor does not affect the approximation of a different factor.

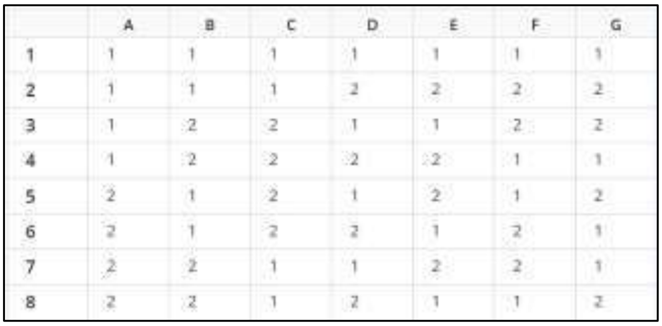
Figure 2 shows an example of an orthogonal array design, which is $L_8(2)^7$. The L_8 in the orthogonal array means that there are eight runs to be conducted, and $(2)^7$ means that there are seven factors with two levels per factor. As previously mentioned, the orthogonal array has all the factor levels equally weighted. The table columns are the control factors, and the table rows are the number of run, which is the combination of factors. The numbers in the table represent the factor levels to be used in the experiment or simulation.

Analysis of variance (ANOVA) is the method to evaluate the importance of factors by comparing the responding variable means at different factor levels (Minitab, 2021). Executing the ANOVA technique requires one responding variable and at least 1 factor with two or more levels. ANOVA requires data that are almost normally distributed to populations with equal variance between factor levels. ANOVA is the approach where variances are used to determine whether the means are different.



Orthogonal Array	Number of Runs	Maximum Factors	Maximum of columns at these levels			
			2-level	3-level	4-level	5-level
L4	4	3	3			
L8	8	7	7			
L9	9	4		4		
L12	12	11	11			
L16	16	15	15			
L'16	16	5			5	
L18	18	8	1	7		
L25	25	6				6
L27	27	13		13		
L32	32	31	31			
L'32	32	10	1		9	
L36	36	23	11	12		
L'36	36	16	3	13		
L50	50	12	1			11
L54	54	26	1	25		
L64	64	63	63			
L'64	64	21			21	
L81	81	40		40		

Figure 1: Taguchi Orthogonal Arrays Selection Table Based on Factors (Balisnomo, 2008)



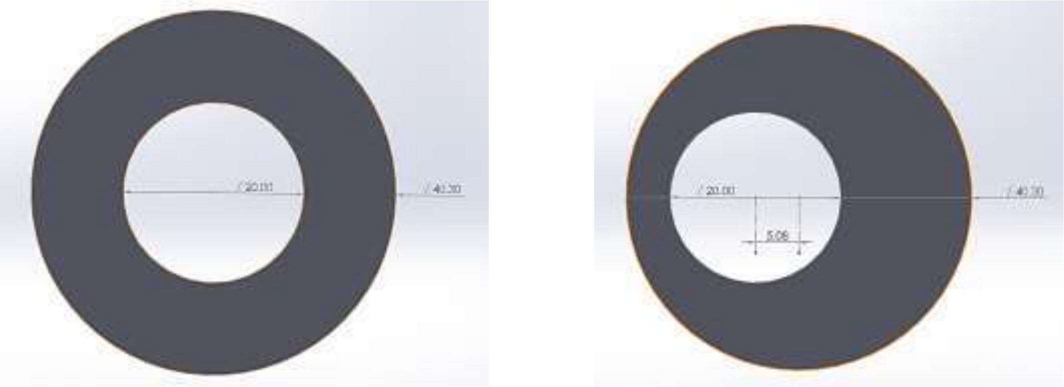
	A	B	C	D	E	F	G
1	1	1	1	1	1	1	1
2	1	1	1	2	2	2	2
3	1	2	2	1	1	2	2
4	1	2	2	2	2	1	1
5	2	1	2	1	2	1	2
6	2	1	2	2	1	2	1
7	2	2	1	1	2	2	1
8	2	2	1	2	1	1	2

Figure 2: Example of $L_8(2)^7$ Orthogonal Array

METHODOLOGY

As the first objective of this study is to study the fluid flow behaviour in different annulus geometries, the simulation used the same velocity and geometry parameter (dimension) to observe the development of flow in the different annulus. In this case, the fluid used in the simulation is water as water is a Newtonian fluid and in most drilling operations, water or drilling mud is used as the drilling fluid due to high heat capacity. The water is assumed to be in the condition of no heat generation, no-slip condition, and steady-state fluid. The density of the fluid (ρ) is 998.2 kg/m3, dynamic viscosity (μ) of 0.001003 kg/ms, specific heat capacity (C_p) of 4.182 kJ/kg·K and thermal conductivity of 0.6 W/m·K.

The length of pipe for both the concentric annulus and eccentric annulus is set to constant to determine the variation of results obtained in the simulation. The dimension for geometries in the present simulation work is selected based on the geometry used by Mohammed Nawai (2011) and Neto et al. (2011). According to Neto et al. (2011), the dimension of the annulus was extracted from the previous experimental parameters for concentric and eccentric annular gaps studied by Nouri & Whitelaw (1994). The outer diameter of the pipe (*do*) is 40.3 mm, while the inner diameter of the pipe (*di*) is 20 mm. The eccentricity of the concentric annulus is E = 0.0, while the eccentricity of the eccentric annulus is E = 0.5. As the hydraulic diameter (*dh*) equals to the outer diameter of the pipe minus the inner diameter of the pipe ($dh = do - di$), hence the hydraulic diameter of the pipe in this study is 20.3 mm. The computational length of the pipe is 101 mm. The drawing of the annulus is made by using Solidworks software. All units of the drawing are in millimetre (mm) units. The schematic diagram of the concentric and eccentric annuli with dimensions are shown in Figure 3 (a) and Figure 3 (b) respectively.



(a): Concentric Annulus

(b): Eccentric Annulus

Figure 3: Schematic diagram of (a) Concentric Annulus, (b) Eccentric Annulus

The meshing of the annulus is an important factor in conducting CFD analysis as it improves the accuracy of the simulation. The meshing of the annulus is conducted before the simulation in ANSYS Fluent by importing the Solidworks annulus drawings into the software. By referring to references, the annulus meshing is done using Patch Conforming Method to mesh the geometry. The element of the meshing is Tetrahedron type. The body mesh sizing is 1.5×10^{-3} m, while the geometry faces mesh with sizing of 5×10^{-4} m, which is smaller than body sizing, to improve the accuracy of the simulation.

To study the effect of different flow regimes with dissimilar annular passages on velocity profile, laminar flow regimes and turbulent flow regimes are used in the simulation. The Reynolds number of the laminar flow regime is set to 1600 ($Re = 1600$), while the turbulent flow regime Reynolds number is set to 43234 ($Re = 43234$). For laminar flow simulation runs, the Reynolds number is set to 1600, and the inlet velocity is calculated by using Eqn (1) together with water properties, giving us an inlet velocity of 0.0792 m/s. The inlet velocity for turbulent flow simulation runs is maintained at 2.14 m/s, which is the same inlet velocity as the reference. Using 2.14 m/s inlet velocity, the Reynolds number for turbulent flow simulation runs is calculated using Eqn (1) as well together with properties of water to obtain the Reynolds number for turbulent flow simulation runs of 43234. The turbulent model of the k-ε Standard is selected for the simulation. The simulation is carried out by using ANSYS Fluent software.

For the Taguchi design of this simulation, there are two factors with two levels in conducting the simulation. The orthogonal array in this study is $L_4(2)^2$. The first factor is the flow regime. The two levels of the first factor (flow regime) are laminar flow and turbulent flow. The second factor is the geometry of the annulus. The two levels of the second factor (annulus geometry) are concentric annulus and eccentric annulus. The orthogonal array used in this study is presented in the table below. The Taguchi design analysis is carried out using Minitab software to analyse the S/N ratio. ANOVA technique is used to analyse the factors by plotting means to determine the most affecting factor and ranking the factors in this simulation.

Table 1: $L_4(2)^2$ Orthogonal Array

Trial Number	Flow Regime	Annulus Geometry
1	1	1
2	1	2
3	2	1
4	2	2

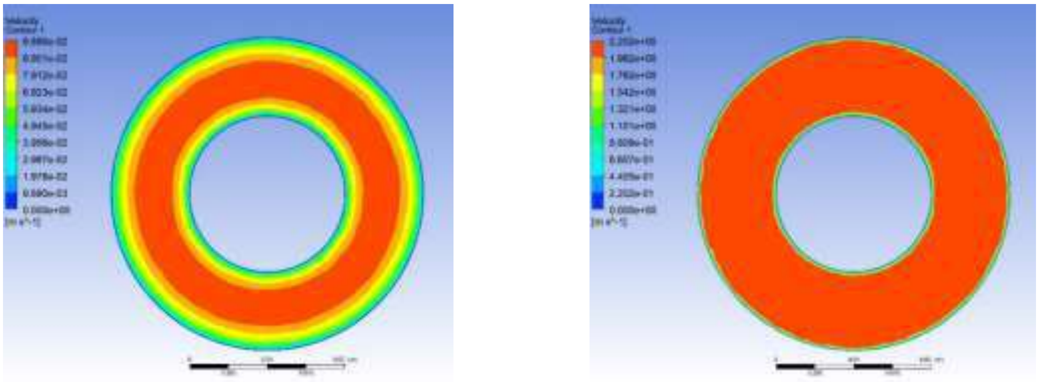
Table 2: Factors with Chosen Levels

Factors	Level of Factors	
	1	2
Flow Regime	Laminar Flow	Turbulent Flow
Annulus Geometry	Concentric Annulus	Eccentric Annulus
Note: Flow Regime:- Level 1 (Laminar Flow, Re = 1600), Level 2 (Turbulent Flow, Re = 43234)		

It is important to be highlighted that this study focuses on studying the effect of different Re with a different flow regime on the axial velocity without the rotating condition. The Taguchi method is used in designing the trial combinations and the flow results are then analysed to select the optimum parameters based on the highest S/N ratio (Larger-the-Better). The data is further analysed by using ANOVA to statistically identify the most significant factor that affects the response based on the significant value (P-value). These steps are important at this early stage to set the reference for future testing with more factors and conditions.

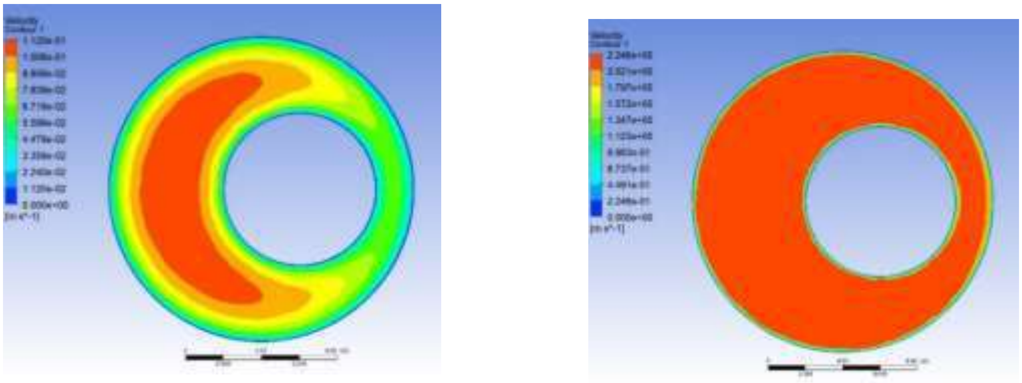
RESULTS AND DISCUSSION

The colours in the velocity contour represent different fluid flow magnitudes. The blue colour contour around the annulus wall shows that the velocity at that location is zero. This is due to the internal resistance of fluid flow, in other words, no-slip condition around the wall. Through the analysis of four simulation runs, it is proven that fluid flow velocity is not uniform when flowing through any surface. Simultaneously, any fluid will have zero velocity at the solid surface due to the no-slip condition. However, the fluid layer that flows far from the solid surface will eventually reach free stream velocity and shows a red-coloured contour in the velocity contour. As shown in Figure 4(a), Figure 4(b), Figure 4(c), and Figure 4(d), the high-velocity magnitude flow shows a red contour, while the flow around annulus walls shows a blue contour.



(a) Laminar Flow + Concentric Annulus

(c) Turbulent Flow + Concentric Annulus



(b) Laminar Flow + Eccentric Annulus

(d) Turbulent Flow + Eccentric Annulu

Figure 4: Velocity Contour of (a) Laminar Flow + Concentric Annulus combination, (b) Laminar Flow + Eccentric Annulus combination, (c) Turbulent Flow + Concentric Annulus combination, (d) Turbulent Flow + Eccentric Annulus combination

Dynamic pressure is the pressure produced by the flowing of fluids. Theoretically, dynamic pressure is higher when the fluid flow velocity is high than in slow-flowing fluids with constant fluid density. In this case, the dynamic pressure is linearly proportional to fluid flow velocity. The relationship between dynamic pressure and fluid velocity is proved in Figure 5(a) and Figure 5(b). In Figure 5(a), the walls of the concentric annulus are positioned at around -0.02 m, -0.01 m, 0.01 m, and 0.02 m. It is observed that the velocity magnitude reduces around the wall of the annulus due to the no-slip condition. Comparing with Figure 5(b), it is observed that the dynamic pressure drops at the same location, which are the walls of the annulus, proving that the dynamic pressure is linearly proportional to the velocity magnitude of flow. This phenomenon is observed to be the same even if the flow regime is turbulent flow, which is proven in Figures 6(a) and 6(b). Besides, based on the graphs obtained through simulation, it is observed that the fluid flow across concentric annulus annular passage is uniform.

Due to the higher fluid inlet velocity used in turbulent flow regime simulation runs, the resulting velocity magnitude in the annulus is much higher compared to laminar flow regime simulation runs. This is due to the higher Reynolds number used in turbulent flow regime simulation runs. As dynamic pressure is linear to velocity magnitude, the dynamic pressure in the annular passage for turbulent flow regime simulation runs is much higher than laminar flow regime simulation runs.

In the case of eccentric annulus simulation runs, it is also observed that the velocity magnitude reduces around the wall of the annulus due to the no-slip condition and the dynamic pressure drops at the same location. However, due to the smaller gap on one side of the eccentric annulus, the fluid flow velocity of the smaller gap is lower than the large gap of the annulus. The lower fluid flow velocity results in lower dynamic pressure as well. This indicates that the fluid in the smaller gap has a lesser ability to carry any solid particles in terms of the drag force applied on the particles, which may cause solid formation and result in annulus blockage. Based on CFD simulation by Hajipour (2020) on the drill cuttings and parametric studies on the eccentric annulus, the effect of particle size on hole cleaning is more obvious at higher drilling fluid flow velocity. The concentration for all particle sizes is transferred easily in the annulus with the increase of drilling fluid flow rate which also causes higher drag force exerted on the solid particles. This issue also happens when the turbulent flow regime is used in the simulation run but with higher fluid flow velocity and dynamic pressure values. The simulation results show that the flow experiences higher pressure losses in the narrow gap of eccentric annulus due to the flow resistance between the gaps.

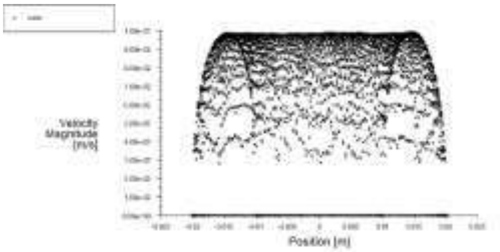


Figure 5 (a): Graph of Velocity Magnitude versus Annulus Position (Laminar Flow + Concentric Annulus)

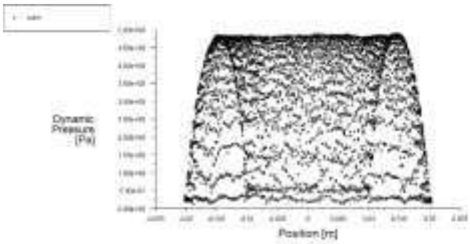


Figure 5 (b): Graph of Dynamic Pressure versus Annulus Position (Laminar Flow + Concentric Annulus)

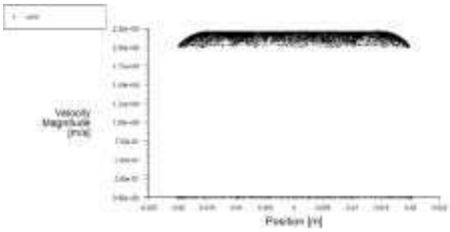


Figure 6 (a): Graph of Velocity Magnitude versus Annulus Position (Turbulent Flow + Concentric Annulus)

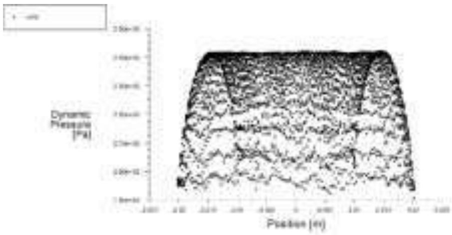


Figure 6 (b): Graph of Dynamic Pressure versus Annulus Position (Turbulent Flow + Concentric Annulus)

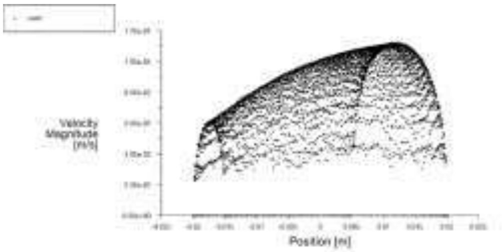


Figure 7 (a): Graph of Velocity Magnitude versus Annulus Position (Laminar Flow + Eccentric Annulus)

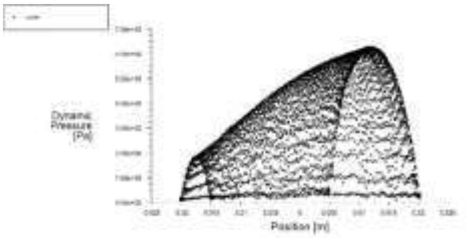


Figure 7 (b): Graph of Dynamic Pressure versus Annulus Position (Laminar Flow + Eccentric Annulus)

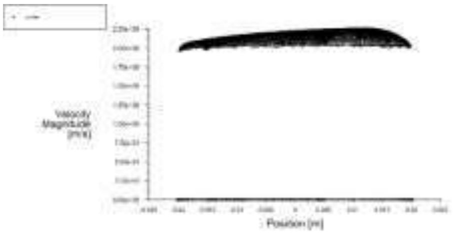


Figure 8 (a): Graph of Velocity Magnitude versus Annulus Position (Turbulent Flow + Eccentric Annulus)

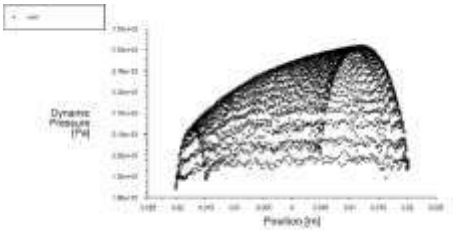


Figure 8 (b): Graph of Dynamic Pressure versus Annulus Position (Turbulent Flow + Eccentric Annulus)

Based on the simulation results, the maximum velocity of fluid flow in each simulation runs are recorded based on the generated velocity contour. From the results, the maximum velocity of each simulation run is tabulated in Table 3.

Table 3: Maximum Velocity at Outlet for Each Simulation Run

Trial Number	Flow Regime	Annulus Geometry	Max Velocity
1	1	1	0.0989 m/s
2	1	2	0.112 m/s
3	2	1	2.202 m/s
4	2	2	2.246 m/s

Model comparison

The prediction of the CFD model is compared against results reported by Mohammad Nawai (2011) for the validation of concentric and eccentric annuli with the eccentricity, E of 0.0 and 0.5 respectively. The radius ratios, η for both present and Mohammad Nawai (2011) models are 0.5. A comparable result is obtained for Re 1600 where the axial velocity for the smaller gap in the eccentric annulus decreases in contrast with the concentric case. This is mainly the slight percentage of water flows in the small space due to a larger resistance to flow than that of the wider gap (Abou-Ziyan et al.,2021; Neto et al.,2011). The other comparison is made to the axial velocity of the concentric annulus where the velocity distribution is approximately symmetry, and the maximum axial velocity is near the centre of the cross-sectional hydraulic area (r/s = 0.5). X-axis (r/s) is the distance between the inner and outer cylinder with 0.0 at the outer cylinder surface and 1.0 at the inner cylinder surface. For a higher Re, a reasonable agreement in the velocity distribution is obtained for Re over than 2000 (Re > 2000) to that of the result of the stationary inner cylinder by Abou-Ziyan et al. (2021).

Analysis of Variance (ANOVA)

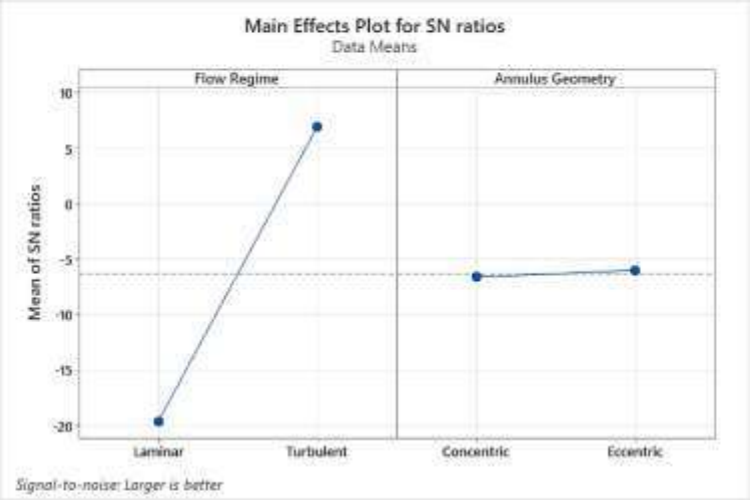


Figure 9 (a): Main Effects Plot for S/N Ratios for Flow Regime and Annulus Geometry

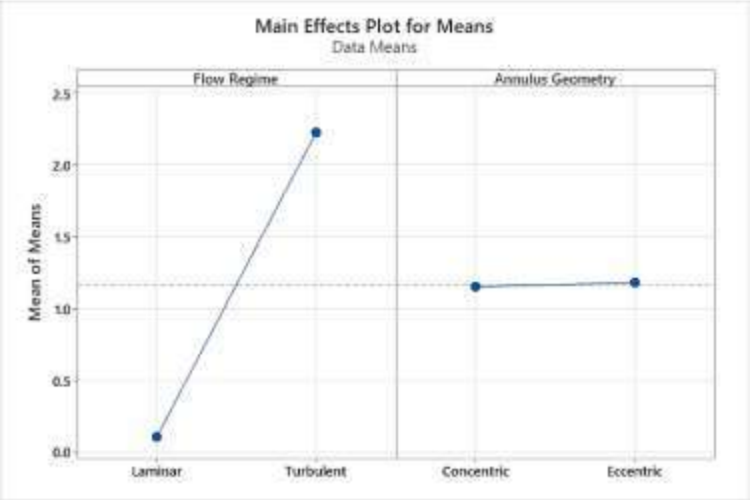


Figure 9 (b): Main Effects Plot for Means for Flow regime and Annulus Geomet

Table 4: S/N Ratio Results

Factors	Levels	S/N Ratio
Flow Regime	1	-19.5559
	2	6.94227
Annulus Geometry	1	-6.61986
	2	-5.99372
Note: Flow Regime:- Level 1 (Laminar Flow, Re = 1600), Level 2 (Turbulent Flow, Re = 43234) Annulus Geometry:- Level 1 (Concentric Annulus), Level 2 (Eccentric Annulus)		

Table 5: Results of ANOVA

Factors	Sum of Square (SS)	Total Sum of Square (Total SS)	Contribution	P-value
Flow Regime	702.151	702.749	99.9149 %	0.011
Annulus Geometry	0.392	702.749	0.0558 %	0.400

Table 6: Response Table for S/N Ratios

Level	Flow Regime	Annulus Geometry
1	-19.556	-6.620
2	6.942	-5.994
Delta	26.498	0.626
Rank	1	2
Note: Flow Regime:- Level 1 (Laminar Flow, Re = 1600), Level 2 (Turbulent Flow, Re = 43234) Annulus Geometry:- Level 1 (Concentric Annulus), Level 2 (Eccentric Annulus)		

Table 7: Prediction of Response Value using Optimum Parameters

Optimum parameter:		Actual Max Velocity	Predicted Max Velocity	Error (%)
Flow Regime	Turbulent Flow	2.2460	2.23828	0.3437
Annulus Geometry	Eccentric Annulus			
Note: Turbulent Flow, Re = 43234				

Table 4 shows the S/N ratio for each factor with respective levels computed in the Taguchi process based on the outcome of Table 3. From Table 4, the combination of the maximum S/N ratio from each factor level will result in the best outcome of the experiment due to the larger the S/N ratio, the better the response value we obtained. The S/N ratio graph shows the graphical representation of response variable change as a function of influencing factor level variation. Figure 9 (a) shows the S/N ratio response curve of this Taguchi experiment. Figure 9 (b) shows the trend of fluid flow velocity, where the velocity increases when the flow regime is turbulent flow, and the annulus geometry is the eccentric annulus.

Based on Figure 9 (a), the highest point of each S/N ratio graph is located at Turbulent Flow and Eccentric Annulus for the 'flow regime' factor and 'annulus geometry' factor, respectively. There is a large variation between the two points for the 'flow regime’ factor graph, but the variation is small for the 'annulus geometry' graph. However, both graphs show a constant trend where the S/N ratio increases when the factor level is level two, which is Turbulent Flow and Eccentric Annulus. This indicates that turbulent flow and eccentric annulus will result in the highest velocity at the outlet. When the flow regime is turbulent, the high Reynolds number also results in high fluid flow velocity. Dynamic pressure increases simultaneously due to higher fluid flow velocity. The high dynamic pressure allows fluid to carry more solids during the flow, which removes the bed cuttings from the annular passage. Eccentric annulus aids in providing higher outlet velocity as well due to the wide gap on one side of the annulus. The flow friction in the eccentric annulus is reduced when eccentricity increases, resulting in a lower frictional pressure drop (Escudier et al., 2000). However, the downside of the eccentric annulus is that solid bed formation may occur at the narrow gap due to lower fluid flow velocity than in the wide part of the annulus with a non-rotating inner shaft. According to a previous study, the solution to prevent solid bed deformation is to use a rotating inner shaft. The inner rotating shaft will drag the fluid flow towards the narrow gap of the eccentric annulus (Neto et al., 2011).

In order to determine the effect of factors on the response variable, the ANOVA technique is carried out and results are recorded in Table 5. As shown in Table 5, the contributions in the percentage of flow regime factor and annulus geometry factor are 99.9149% and 0.0558%, respectively. The percentage contribution is to determine which factor significantly affects the maximum velocity of fluid flow in the annular passage. The larger value of the percentage contribution, the greater the influence of the factor on the response variable. The results of ANOVA show that the most influencing factor is the flow regime, which holds 99.9149% of the contribution to the response variable. In other words, the Reynolds number of a fluid affects the maximum velocity. Besides, the flow regime and annulus geometry factors have a P-value of 0.011 and 0.4, respectively. The P-value of the flow regime factor is lower than the significance level of 0.05 (Minitab, 2022; Hamdan & Yusof, 2014), meaning that the flow regime factor is statistically significant to the changes in the response value. As for the annulus geometry factor, it is not statistically significant to the changes in response as the P-value is 0.4, which is higher than 0.05.

Based on the analysis of the S/N ratio graph and the results of the ANOVA technique, it is concluded that the flow regime is the most influencing factor on the maximum velocity of fluid flow in an annular passage. The level of influencing factor can be observed by referring to the rank number in Table 6. This study uses the ranking method to decide the level of influencing factor according to the S/N ratio characteristic based on the Larger-the-Better. The ranking method is a descriptive statistic that ranks the calculated value of the respective factor (Okpala & Roslan, 2019). Turbulent flow has a much higher Reynolds number, which results in higher dynamic pressure. On the other side, an eccentric annulus also contributes to velocity changes due to the wide gap on one side of the annulus, resulting in lower frictional pressure loss compared to a concentric annulus. Figure 9 (b) also shows the main effect of the eccentric annulus on the mean velocity being higher than the concentric annulus. Hence, in this case, the optimum combination for maximum velocity in the annular passage is Turbulent Flow for the 'flow regime' factor and Eccentric Annulus for the 'annulus geometry' factor.

CONCLUSION

The two main objectives of this study are achieved. The concentric annulus has a constant flow development, while the eccentric annulus has non-consistent flow development due to the narrow gap on one side, and it tends to have lower pressure loss towards the inner of the annulus at the wide gap section. This is due to the no-slip condition of the fluid and the resistance of fluid flow between the inner annulus walls decreases. Comparing the wide gap and narrow gap of the eccentric annulus, the fluid flow velocity at the narrow gap is lower. The low fluid flow velocity may result in problems such as solid bed deposition, affecting the overall drilling process. Using the Taguchi method and ANOVA technique, the flow regime factor affects the response value the most, which is the maximum velocity of fluid flow in the annulus. Compared to annulus geometry, the contribution of the flow regime factor is 99.9149%, which is the most significant factor that affects the response value and is ranked number one (1) among the combination of factors. In the four simulation runs, eccentric annulus with turbulent flow results in the highest fluid flow velocity. This shows that the combination of the eccentric annulus with the turbulent flow has a better solid carrying ability due to the higher drag force to be applied on the particles compared to other combinations. This is considered the best combination in this study using water as a fluid. It is important to highlight that this study involves only water. For other types of fluid such as drilling mud, a future investigation must be done to identify its combined effects with the annulus design and cylinder operating conditions.

AUTHOR BIOGRAPHY

Lau Wei Jen is a Bachelor of Mechanical Engineering with Honours candidate at Infrastructure University Kuala Lumpur (IUKL) in the Mechanical and Automotive Engineering, Faculty of Engineering, Science and Technology (FEST*). Email: wjlau6123@gmail.com*

Mohd. Shahrudin Mohd. Alias (Ir.), PEng, MIEM, is a lecturer in the Faculty of Engineering, Science and Technology (FEST) at Infrastructure University Kuala Lumpur (IUKL). He teaches and supervises undergraduate students in mechanical engineering and automotive technology programmes. He is a registered Professional Engineer with the BEM and a corporate member of the IEM. *Email: shahrudin@iukl.edu.my*

REFERENCES

Balisnomo, R. (2008). *Introduction to taguchi method*. Ingersoll Rand. https://www.slideshare.net/rbalisnomo/Introduction-To-Taguchi-Method-05Sep08

Escudier, M., Gouldson, I., Oliveira, P., & Pinho, F. (2000). Effects of inner cylinder rotation on laminar flow of a newtonian fluid through an eccentric annulus. *International Journal of Heat and Fluid Flow*, *21*(1), 92–103.

Ferroudji, H., Hadjadj, A., Haddad, A., & Ofei, T. N. (2019). Numerical study of parameters affecting pressure drop of power-law fluid in horizontal annulus for laminar and turbulent flows. *Journal of Petroleum Exploration and Production Technology*, *9*(4), 3091–3101.

Hajipour, M. (2020). CFD simulation of turbulent flow of drill cuttings and parametric studies in a horizontal annulus. *SN Applied Sciences*, 2 (6), 1-12.

Hamdan, N.S.,Yusof, A. (2014). Determining demographic profiles and tourist' motives for visiting Langkawi Island. *Infrastructure University Kuala Lumpur Research Journal*, 2(1), 96-103.

Kelessidis, V., & Dukler, A. (1989). Modeling flow pattern transitions for upward gas-liquid flow in vertical concentric and eccentric annuli. *International Journal of Multiphase Flow*, *15*(2), 173–191.

Minitab. (2021) *Example of getting and interpreting a p-value*. Minitab LLC. https://support.minitab.com/en-us/minitab/18/help-and-how-to/statistics/basic-statistics/supporting-topics/basics/example-of-getting-and-interpreting-a-p-value/

Minitab. (2022) *Taguchi designs.* Minitab LLC. https://support.minitab.com/en-us/minitab/18/help-and-how-to/modeling-statistics/doe/supporting-topics/taguchi-designs/taguchi-designs/

Minitab. (2021) *What is ANOVA?* Minitab LLC. https://support.minitab.com/en-us/minitab/18/help-and-how-to/modeling-statistics/anova/supporting-topics/basics/what-is-anova/#:%7E:text=Learn%20more%20about%20Minitab%2018,at%20the%20different%20factor%20levels.

Mohammed Nawai, R. (2011). *Computational fluid dynamics (CFD) study of flow development in an eccentric annulus.* (Thesis). Universiti Teknologi PETRONAS.

Neto, J. L. V., Martins, A. L., Neto, A. S., Ataíde, C. H., & Barrozo, M. A. S. (2011). CFD applied to turbulent flows in concentric and eccentric annuli with inner shaft rotation. *The Canadian Journal of Chemical Engineering*, *89*(4), 636–646.

Nouri, J. M., H. Umur, J. H. Whitelaw, (1993). Flow of Newtonian and Non-Newtonian Fluids in Concentric and Eccentric Annuli. *Journal of Fluid Mechanics*, *253*, 617–641.

Okpala, C.A., & Roslan, S.N.A. (2019). The causes of delays and disruption at construction project in Serdang, Selangor Malaysia. *Infrastructure University Kuala Lumpur Research Journal*. 7(1), 1-15.

Singh, R., Ahmed, R., Karami, H., Nasser, M., & Hussein, I. (2021). CFD analysis of turbulent flow of Power-Law fluid in a partially blocked eccentric annulus. *Energies*, *14*(3), 731.

Zhicheng, H.,& Nordin, N. (2022). Finite element analysis on the steel fiber-reinforced concrete beams: A systematic review. *International Journal of Infrastructure Research and Management*. 10(1), 1-15.