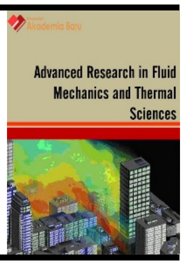
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Erosion prediction of bypass seat in oil well auto-fill equipment using CFD approach

Open Access

Ahmad Rizal Maharam[1,*]

[1] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia

ARTICLE INFO

Article history:
Received 27 February 2017
Received in revised form 26 May 2017
Accepted 18 July 2017
Available online 1 August 2017

Keywords:
auto-fill; CFD; erosion, multi-phase, casing running

ABSTRACT

Throughout the course completion of oil and gas wells, it is important to run casing in a wellbore that is normally laden with wellbore fluids. The wellbore fluid, or mud, serves two essential purposes. First, it helps to keep the inflow of formation fluids into the wellbore by exerting hydrostatic pressure on the formation. The hydrostatic pressure that is applied must be sufficiently high enough to keep fluid inflow but not high enough to crumble down the formation. During casing run, the impacts of surge and swab must be considered when deciding casing running rates to ensure the boundary between the hydrostatic pressure column and formation pressure keep at equilibrium. Second, the mud assists to circulate cuttings from the wellbore during the drilling commencement. Occasionally during running casing in the wellbore, it is important to have dependable auto-fill equipment to help limit the surge and swab effect. The concept is enabling wellbore fluids to enter the casing from the lower end of the casing is called auto-fill or self-fill. Filling the casing consequently from the lower end wipes out the requirement for manual filling the casing from surface while running in hole (RIH) and casing running efficiencies can increased. In the event that circulation time and additionally rate while RIH or well conditioning surpasses pre-determined mud weight and flowrate, erosion on the valve could happen, which lead to higher than anticipated auto-fill deactivation rates. Thus, this study aims to assess flow behavior and erosion characteristics of the auto-fill valve assembly at the highest acceptable predetermined flowrate parameter.

1. Introduction

During the completion of oil and gas wells, it is necessary to run casing in a wellbore that is typically filled with wellbore fluids. The wellbore fluid, or mud, serves two primary functions. First, it exerts hydrostatic pressure on the formation that helps prevent influx of formation fluids into the wellbore. The hydrostatic pressure that is exerted must be high enough to prevent fluid influx, but not high enough to break down the formation. While running casing, the effects of surge and swab must be considered when determining casing running speeds to help ensure the interface between

* *Corresponding author.*
E-mail address: Rizal.Maharam@Halliburton.com (Ahmad Rizal Maharam)

the hydrostatic pressure column and formation pressure maintain a balance. Second, the mud helps to transport formation cuttings from the wellbore during the drilling process [1].

While running casing, it is sometimes critical to have reliable auto-fill floats to help minimize surge and swab while running casing. The process of allowing wellbore fluids to enter the casing from the lower end of the casing is called auto-fill or self-fill. Filling the casing automatically from the lower end eliminates the need for manually filling the casing from the surface while running in hole (RIH) and can increase overall casing running efficiencies.

2. Literature Review

2.1 Erosion Model

There have been a number of theoretical and experimental studies on impinging erosion. Few researchers proposed that erosive wear is a direct consequence of the cutting of surfaces by impacting particles [2-6]. This model sets the main concepts and assumptions for many subsequent single-particle erosion models. The model assumes a hard particle with velocity up impacting a surface at an angle α. The material of the surface is assumed to be a rigid plastic. This model assumes that the specific erosion rate on a surface, e (mass eroded / mass impacting), may be described by:

$$e = K \cdot |u_p|^n \cdot f(\alpha) \tag{1}$$

where α and u_p are the local impact angle and velocity, respectively, K is a scaling coefficient and n is the impact velocity power law coefficient that typically varies between 2 and 3 for ductile materials [6]. *f(α)* is the dimensionless wear function that describes the impact angle effect on the wear rate. This function can take many forms, the following one is used in this work [5].

$$f(\alpha) = \begin{cases} A\alpha^2 + B\alpha & \dots\dots\dots\ \alpha \le \varphi \\ X\cos^2\alpha\sin(W\alpha) + Y\sin^2\alpha + Z & \dots\dots\dots\ \alpha > \varphi \end{cases} \tag{2}$$

A, B, W, X, Y, Z and *φ* are all empirical coefficients. Finnie's model of erosion is considered as the milestone of erosion modelling and provides the basis for modelling of the process.

3. CFD Approach

A computational fluid dynamic instrument has been chosen for the simulation of the flow field inside the auto-fill equipment valve and for the recreation of the molecule directions and their effect on dividers. CFD, at present is of promising methodologies for the investigation of a wide class of problems involving flow domains and in a wide variety of research and modern application fields.

CFD codes are capable of solving the full set of fluid dynamic balance equations, usually in Navier–Stokes formulation for momentum balance. Turbulence can be approximated by different models. In particular, the FLUENT code [7] adopted for this study solves the balance equation set via domain discretization, using a control volume approach to convert the balance partial differential equations (PDEs) into algebraic equations solved numerically.

The FLUENT code has been used in the investigation of solid particle erosion in fluid flow in components of complex geometry [8]. The solution procedure integrates the balance equations over each control volume, thus obtaining discrete equations that conserve primary quantities on a control volume basis. The numerical solution defines the flow field quantities, possibly used by routines implementing models for further flow-related quantities than, e.g. phases transported by a given fluid phase. One of the more important features of this class of fluid dynamic codes is the ability to simulate complex fluid flows and geometric domains, both in two- and three-dimensions, also accounting for turbulence. A set of models are usually made available to the user, differing mainly in the scale of turbulence they can evaluate. The present case study has been performed by adopting a three-dimensional unstructured mesh for the pipe, an implicit method for the numerical solution of mass and momentum equations and a *k-ω* model for the turbulence. The mixture composition and phase velocities are defined at the inlet boundary. The system pressure is fixed at the outlet boundary

3.1 Geometry Preparation

In this study 4 (four) configurations or cases of bypass seats in auto-fill equipment valve will be considered for CFD analysis namely Original (baseline model), Original-taper, Larger slot and Larger slot as shown in Figure 1. There are 3 bypass channels per seat that is configured at 120° apart. These configurations have been modeled in SolidWorks 2016 and converted to IGES file format to be imported for CFD analysis in Ansys Fluent 15.0.

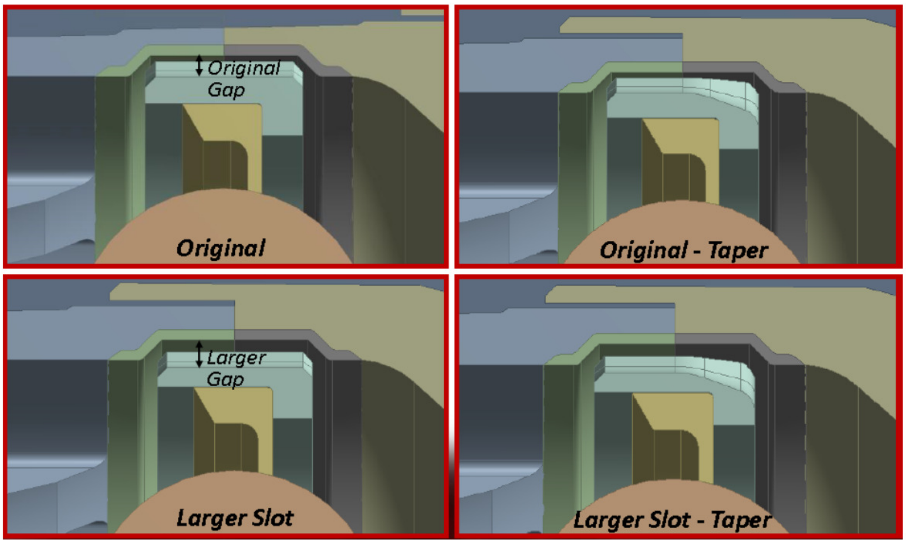


Fig. 1. Four Configurations to be considered in CFD Analysis

Table 1
Design data for all configurations

Design Feature	Configurations / Cases			
	Original	Original -Taper	Larger Slot	Larger Slot - Taper
Size of Gap (in.)	0.255	0.255	0.255	0.255
Taper Feature	No	Yes	No	Yes
Total Flow Area (in^2)	0.495	0.723	0.495	0.723
Size of Gap (in.)	0.255	0.255	0.255	0.255
Taper Feature	No	Yes	No	Yes
Total Flow Area (in^2)	0.495	0.723	0.495	0.723
Size of Gap (in.)	0.255	0.255	0.255	0.255
Taper Feature	No	Yes	No	Yes
Total Flow Area (in^2)	0.495	0.723	0.495	0.723

3.2 CFD Modelling Preparation

Due to symmetric nature of this analysis. The geometry of an auto-fill equipment valve was divided into three sections evenly at angle of 120° with orifice hole that blocked by a ball at the center of one-third pipe cross section as shown in Figure 2.

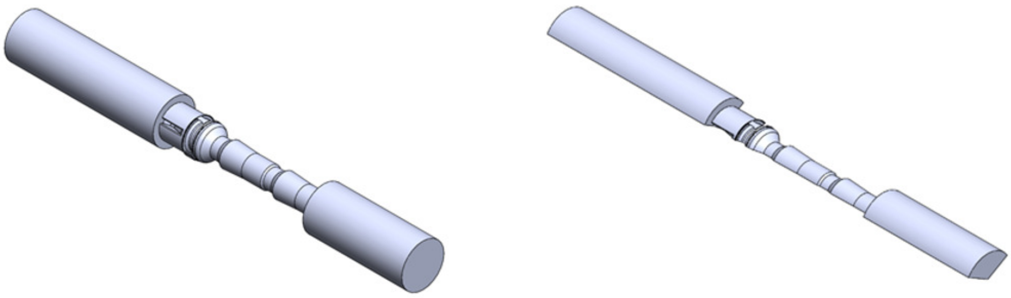


Fig. 2. Geometry of full and one-third of the auto-fill equipment valve

3.3 Meshing

Two areas of refinement were performed using body of influence function. Figure 3 shows solid body for Area 1 that was used for mesh refinement at the bypass channel area with element size of 0.75 mm. Figure 4 shows Area 2 solid body for mesh refinement in order to capture wake flow after bypass channel area with element size of 1.25 mm. Figure 5 shows the mesh generated, it has combination of three element types, tetrahedral, hexahedral and prism. Table 2 shows number of mesh for every configuration.

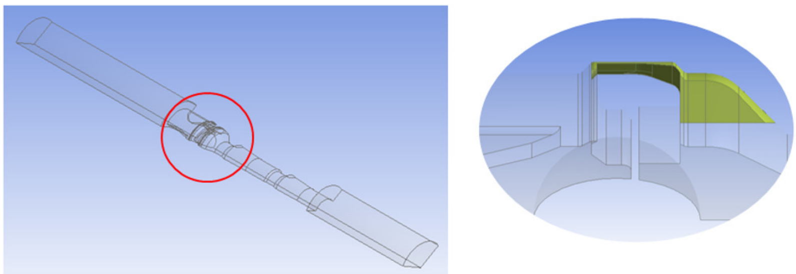


Fig. 3. Mesh refinement for area 1

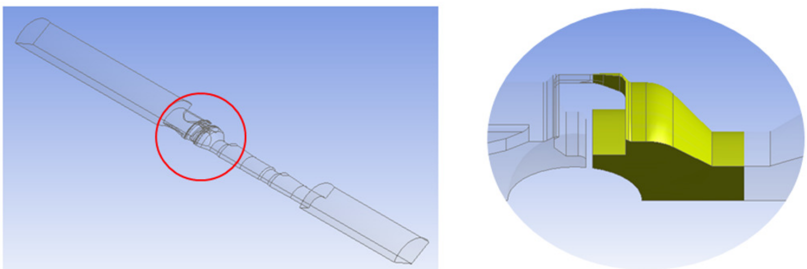


Fig. 4. Mesh refinement for area 2

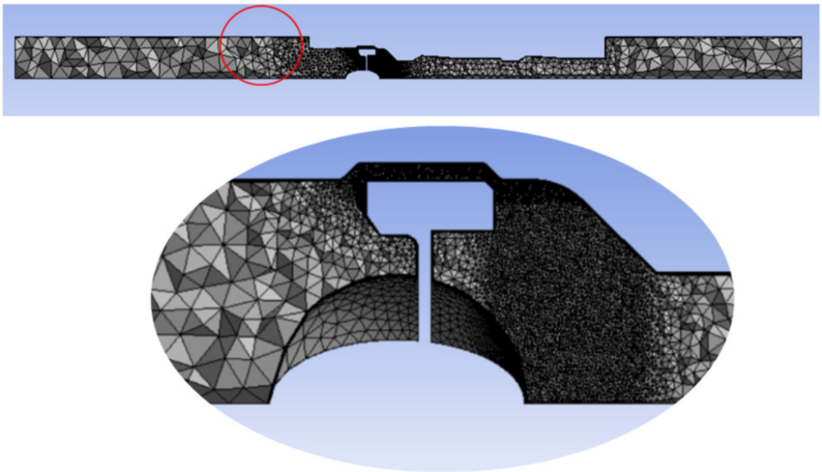


Fig. 5. Mesh generated for original-taper configuration

Table 2
Total elements generated for every configurations

Cases	Number of Elements (million)
Original	1.82
Original-Taper	2.96
Larger Slot	2.86
Larger Slot-Taper	3.35

3.4 Analysis Setup

Fluent 15.0 with steady state RANS Turbulence Model was used as processor in this analysis. Turbulence model selected for this analysis is *k-ω* SST due its suitability to solve flow at boundary layer in pipe and economical computational time. Fluid for this analysis is API 12 ppg mud with density and viscosity of 1438 kg/m^3 and 0.025 kg/m.s respectively with flowrate of 8 bpm (0.02 m^3/s).

One face inlet, one face outlet, two symmetry faces and walls boundary condition for other faces as shown in Figure 6. Inlet for all configurations was defined with pressure-inlet with initial gauge pressure as listed in Table 3. Outlet for all configurations was defined with pressure-outlet with gauge pressure of 0 Pa. All wall faces was defined as no-slip condition wall. Table 4 shows solution methods defined for this analysis with residual of 0.001.

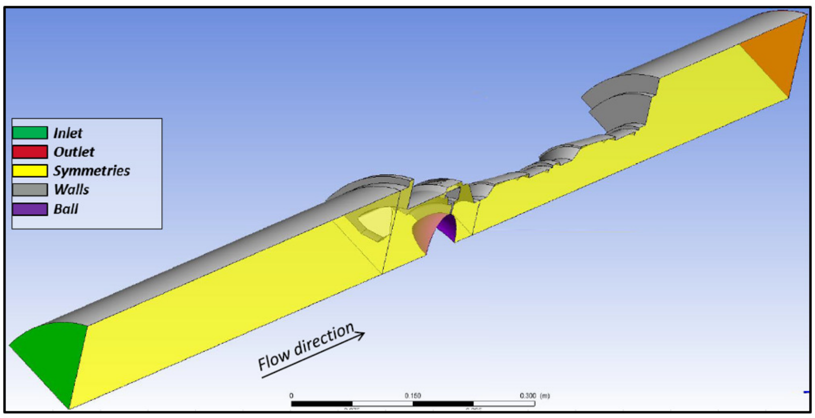


Fig. 6. Boundary conditions

Table 3
Pressure defined for inlet boundary condition

Cases	Initial Gauge Pressure (kPa)	Gauge Total Pressure (kPa)
Original	2720	2720.327
Original-Taper	1660	1660.327
Larger Slot	700	700.27
Larger Slot-Taper	445	445.327

Table 4
Solution methods for CFD analysis

Pressure-Velocity Coupling	
Scheme	SIMPLEC
Skewness Correction	0
Spatial Discretization	
Gradient	Green-Gauss Node Based
Pressure	PRESTO!
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind

4. Result Analysis

CFD simulations that have been carried out for all bypass seat cases. Due to complexity nature of erosion modelling for multiphase flow in FLUENT, modeler license could not be obtained and also lacking of competent resources to perform erosion modelling simulation, writer has decided to proceed the project for single phase flow simulation only and dropped the requirement for erosion modelling. Thus, writer will only discussed on velocity field and contours, relative total pressure and there will be no discussion for erosion rate prediction.

4.1 Validation Analysis

For validation process, result from initial analysis had been compared to CFD analysis performed by Felten [9]. Validation analysis was performed using the same mesh and setting and at flowrate of 3 bpm and for configurations Original and Original-taper only. Tables 5 present results of validation analysis compared to Felten [9]. It shows that current CFD analysis produce results with difference less than 3% compared to Felten [9]. Hence, it has been decided that the current CFD model was good to be used for CFD analysis with 8 bpm.

Table 5
Results for validation analysis for flowrate 3BPM

Cases	Parameters	Felten [9]	CFD	% Diff.
Original	Velocity (ft/s)	107.7	102.4	0.66
	Total Pressure (psi)	79	80	1.36
Original-Taper	Velocity (ft/s)	83.1	82.9	0.13
	Total Pressure (psi)	48	47	3.01

4.2 Result Analysis

Tables 6 shows maximum velocity and total pressure along the valve at fluid flowrate of 8 bpm for all configurations/cases. Original case required highest total pressure to maintain flowrate of 8 bpm and generate highest velocity in slot/bypass channel compared to other configurations/cases as shows in Figure 7 and Figure 8 respectively.

Table 6
Maximum velocity and total pressure for all configurations

Cases	Parameters	CFD
Original	Velocity (ft/s)	272.2
	Total Pressure (psi)	395
Original-Taper	Velocity (ft/s)	198.6
	Total Pressure (psi)	240
Larger Slot	Velocity (ft/s)	122.7
	Total Pressure (psi)	100
Larger Slot-Taper	Velocity (ft/s)	89.1
	Total Pressure (psi)	65

Table 7
Total pressure at point A and point B and the total pressure difference

Cases	Total Pressure (psi) (Point A)	Total Pressure (psi) (Point B)	Total Pressure Difference (psi)
Original	395	5.5	389.5
Original-Taper	241	3.8	237.2
Larger Slot	101	8.4	95.6
Larger Slot-Taper	65	2.7	62.3

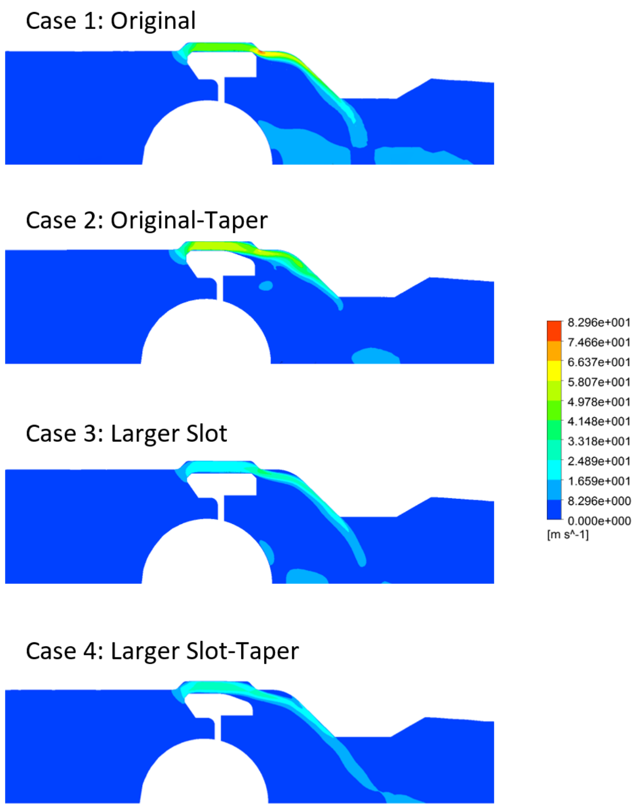


Fig. 7. Velocity Contour at symmetry plane for all configurations

Overall, it can be observed that the Total Pressure differentials are less that extrusion ring extrusion pressure (400 – 600 psi) even though with high velocity fluid flowing through the valve, we cannot have a solid conclusion stating that it is unlikely the auto-fill feature will be deactivated at 8 bpm. The absence of another phase of flow (solid particle), circulation time and also erosion rate, we could not predict the rate of eroded surface that likely to happen. However, due to the fact that velocity magnitude is high, it can be assumed that internal wear on the valve could occur, resulting in higher than expected deactivation rates. Based on the Velocity and Total Pressure results, the Original case will likely deactivate the valve as the Total Pressure is consider at the threshold of extrusion ring pressure. The highest Total pressure for Original case was contributed by smallest flow area at bypass channel that it had and there is no taper feature to diffuse the flow and the highest velocity occurred at the exit of bypass channel is due to orifice effect.

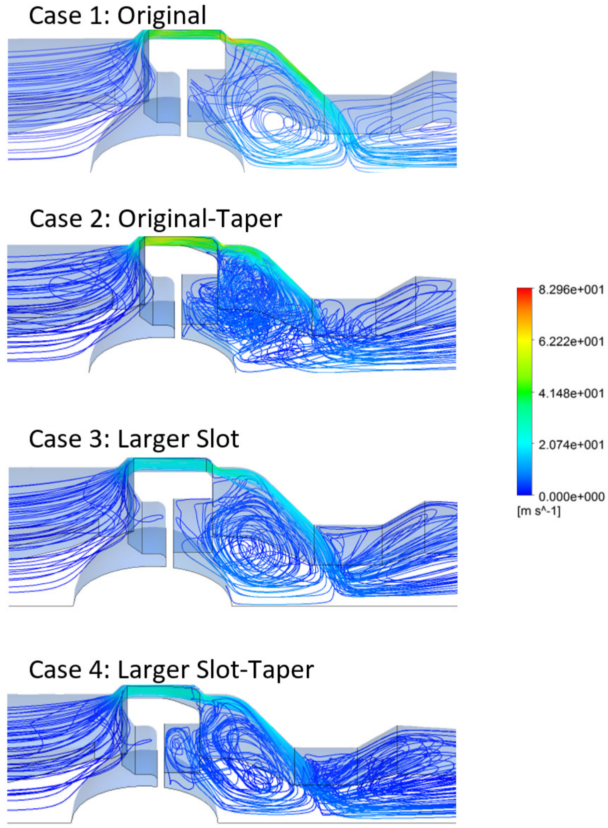


Fig. 8. Velocity streamline at symmetry plane for all configurations

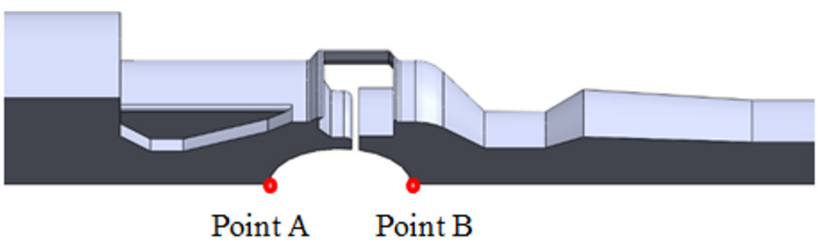


Fig. 9. Locations of total pressure on the auto-fill valve setting ball

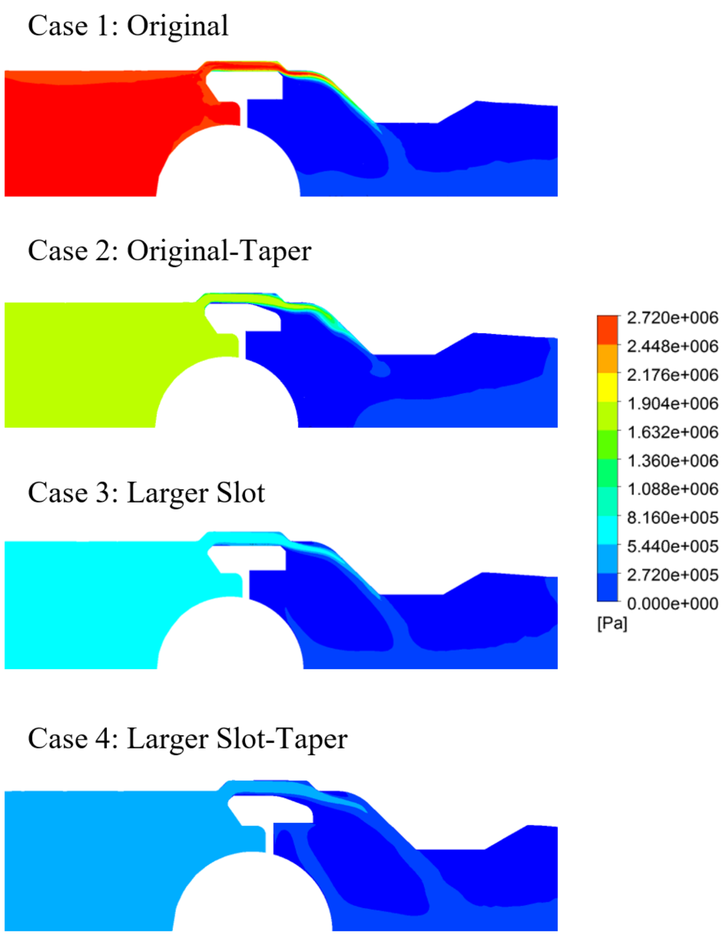


Fig. 10. Velocity contour at symmetry plane for all configurations

Figure 9 shows two locations of total pressure taken on the center of auto-fill valve setting ball to calculate total pressure difference. Original configuration has the highest total pressure difference compared to other configurations as shown in Tables 7. High total pressure difference for Original configuration is due to high total pressure at inlet to maintain 8 bpm flow rate through small slot cross section area. From the results, it was found that geometry with taper configurations create lower total pressure at Point B. Lower total pressure happen because taper geometry moves some of fluid closer to Point B which subsequently reduce pressure at area around Point B due to Bernoulli Effect.

5. Conclusion

The objectives of this study were to develop and propose 4 bypass seat designs for CFD Simulation at 8 bpm flowrate in order to assess flow behavior and erosion characteristic so that from the results, seat designs can ranked. Subsequently, potential suggestions on how to reduce the erosion rate can be proposed. Due to complexity nature of erosion modelling for multi-phase flow in FLUENT, modeler license could not be obtained and also lacking of competent resources to perform erosion modelling simulation, writer has decided to proceed the paper for single phase flow simulation only and dropped the requirement for erosion modelling. Thus, erosion rate predication and its characteristic have not been discussed, only velocity field and contours, relative total pressure has been discussed.

Acknowledgement

Author would like to express his sincere appreciation to thesis supervisor, Assoc. Prof. Dr. Nor Azwadi bin Che Sidik, for encouragements, guidance, critics, motivations and advices towards continually support and interest of this study. Special dedication for my wife, Siti Nur Farida Indera Putra for her patience, sacrifices for all these years and endless supports given to author during completing this paper. Last but not least, I would like to extend my sincere appreciation to Mohd Hasrizam, families and entire course mate for assistances that have been given whether direct or indirectly throughout the process to complete this project.

References

[1] Shaughnessy, John Martin, William Thomas Daugherty, Rick L. Graff, and Todd Durkee. "More ultra-deepwater drilling problems." In *SPE/IADC Drilling Conference*. Society of Petroleum Engineers, 2007.

[2] Chen, Xianghui, Brenton S. McLaury, and Siamack A. Shirazi. "Application and experimental validation of a computational fluid dynamics (CFD)-based erosion prediction model in elbows and plugged tees." *Computers & Fluids*33, no. 10 (2004): 1251-1272.

[3] Finnie, Iain. "Erosion of surfaces by solid particles." *wear* 3, no. 2 (1960): 87-103.

[4] Finnie, Iain, and Y. H. Kabil. "On the formation of surface ripples during erosion." *Wear* 8, no. 1 (1965): 60-69.

[5] Wong, Chong Y., Christopher Solnordal, Anthony Swallow, Steven Wang, Lachlan Graham, and Jie Wu. "Predicting the material loss around a hole due to sand erosion." *Wear* 276 (2012): 1-15.

[6] Lester, D. R., L. A. Graham, and Jie Wu. "High precision suspension erosion modeling." *Wear* 269, no. 5 (2010): 449-457.

[7] Fluent, Ansys. "12.0 User's guide." *Ansys Inc* (2009).

[8] Parslow, G. I., D. J. Stephenson, J. E. Strutt, and S. Tetlow. "Investigation of solid particle erosion in components of complex geometry." *Wear* 233 (1999): 737-745.

[9] Chen, Xianghui, Brenton S. McLaury, and Siamack A. Shirazi. "Application and experimental validation of a computational fluid dynamics (CFD)-based erosion prediction model in elbows and plugged tees." *Computers & Fluids*33, no. 10 (2004): 1251-1272.