

Optimization Of Autonomous Inflow Control Device Performance Through Evaluation Study On Pressure Distribution On Levitating Disc

Muhammad Aiman bin Baharin, PM Ir Dr. Nadiyahnor Md Yusof

Faculty of Chemical Engineering, Universiti Teknologi Mara

Abstract— Oil and gas industry is facing critical challenge in maintaining production performance. High water production due to strong aquifer and water injection decrease the performance of a well. Implementation of Autonomous Inflow Control Device inside a completion system shall help improve the total performance of a well. The device work autonomously where a levitating disc control the inflow based on the composition and velocity of the fluid produced. Each field require specific design of the device to be implemented. The purpose of this paper is to determine the optimum flow area for the device to work effectively in a well condition specified. This will verify the effects of a few design parameter of the device focused such as inlet area, disc size and outlet size. Simulations using ANSYS Fluent were done by adjusting the design parameters to obtain favourable results. The results were analyzed based on the differential pressure exerted on the disc. These optimum parameters shall indicate that the levitating disc will work effectively and autonomously inside the completion string during production. It will also verify that the design parameters are critical to ensure that the production performance was enhanced.

I. INTRODUCTION

PROBLEM STATEMENT

An Autonomous Inflow Control Device uses the principle of differential pressure to autonomously control the flow of oil entering the completion tubing[6]. The different pressure created by different types of fluids will cause the device to close, stopping the unwanted materials or fluids from entering the production stream. This application is commonly coupled with horizontal drilling to increase production by extend the reach into the reservoir. The result of this horizontal drilling causes the water inside the formation to flow out of the reservoir to the wellbore. This unwanted flow of water, if not control, will reduce the reservoir pressure and reduce the necessary drive for the oil to be push out of the reservoir to the wellbore. It will also prevent uniform sweep of the oil inside the formation[7].

The implications stated above make it very important for the AICD to reduce the water production. Otherwise, this technique will only make the production performance become worse. As stated in the research background, these problems will results to uneconomical condition of the well. Thus, it became unprofitable causing the well to be shut off and the oil reserved to stay underground.

In order to make it effective and efficient, the flow of the fluid flowing into the tubing string and through the device must be

optimized. Optimizing the flow by determining the optimum flow area for the device is the main purpose of this research. By ensuring the optimum flow velocity across the device, it will increase the effectiveness of the AICD in enhancing oil production.

OBJECTIVE

The purpose of this research is to determine the optimum flow velocity by making flow area as the main variable of this research. Theoretically, the smaller the diameter, the higher the flow velocity thus creating higher differential pressure across the AICDs. This research will determine how to make the AICDs implemented in the completion string restrict the flow of water in the most efficient way. The differential pressure created by water across AICDs will autonomously adjust the flow restriction[8]. This way, the part which causing high water production will decrease the production rate or completely closed to stop flow from that region and avoid unnecessary reservoir pressure reduction from loss of reservoir fluid.

The mechanism of AICD to restrict inflow of fluid into the completion string is based on a levitating disc inside the device that will autonomously restrict or completely close the flow area based on the differential pressure created by the flow at the inlet[9]. The material of the plate used must be taken into consideration in determining the optimum condition for the device to effectively work. By adjusting the composition of fluid flow through the AICD, the results at which the compositions that contain high water content and low water content will compared across all flow area. This way, the optimum flow required for the device to work effectively can be determined.

Other than that, in the process of optimizing the flow, flow resistance will be taken into account. The flow resistance is also important in an analysis especially in conducting mathematical approach. However, the flow resistance will be kept constant along all the flow parameter and flow optimization.

At the end of these simulations, the optimum flow should be determined based on the highest differential pressure exerted on the plate during the high water saturation fluid flow. However, the parameter of the flow design must have the opposite results during the low water saturation fluid across the device which open the flow path so that the production can flow if it does not contains high amount of water. These two optimum results will determine that the flow condition is optimum for the device to effectively work. If the fluid passing through plate has high water content, the plate will close the device and restrict the flow. If the fluid passing through the plate has low water content, the device will be left open.

II. METHODOLOGY

This research is about determining the optimum flow through an Autonomous Inflow Control Device for the device to effectively work. The simulation does not involve irregular condition of actual reservoir which requires further configuration setting of the whole completion string. The simulations are simply about fluid dynamic through an AICD where the velocity and pressure of the flow will be observed.

The flow parameters are based on the simple generic model reservoir parameters from the simulation in a journal of Offshore Technology Conference in Houston, Texas. The wellbore diameter is 0.156m but the completion tubing that is used in the simulation will be 5-1/2 inch. The initial reservoir pressure is 157.5 bars. The oil density is 805 kg/m³, oil viscosity is 1.7cP, oil formation volume factor is 1.17m³, gas oil ratio (GOR) is 60m³/m³. Water density is 1045 kg/m³, water viscosity is 0.45cP and water formation volume factor is 1.07m³/m³. Gas density is 0.8 kg/m³, gas viscosity is 0.017cP and gas formation volume factor is 0.0068m³/m³. As used in the simulation of the journal, the gas rate is kept constant at 144000m³/day[5].

The device that will be used in the simulation is based on the Autonomous Inflow Control Device by Tendeka which named FloSure. There are no actual production data to be used as reference because of the confidentiality of the design used. Every field has different flow parameters which require different design or in this case different flow area specific to be used by the field[10].

III. COMPUTATION MODELING AND SIMULATION

The flow simulation across the Autonomous Inflow Control Device will be done using ANSYS Fluent software.

First, before designing the simulation, the scope, objective and parameter to be analyzed must be specified. This study is regarding the flow optimization through the Autonomous Inflow Control Device. It is to analyze the optimum flow areas in the design of the device used by Tendeka FloSure Autonomous ICD. There will be a few points inside the device that concern this study in term of flow area optimization.

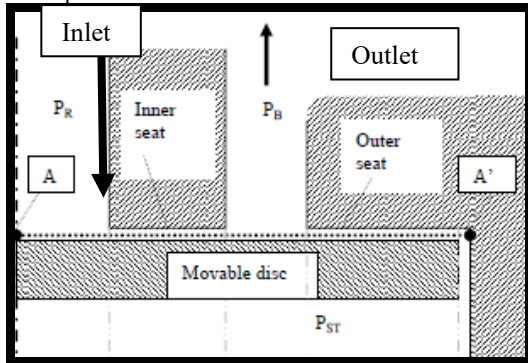


Figure 1: AICD cross section

BASIS OF SIMULATIONS

The fluid specified for the first simulation will have the total density equal to fluid composed of zero water content and 30% gas content. Thus, the oil composition is 70% with the density of 805 kg/m³ and viscosity of 1.75cP. This set of parameters shall represent the fluid produced in the early stage of the well's life, or the reservoir life. Density of the mixture will be calculated using the formula given in the Tendeka FloSure Autonomous ICD Datasheet as figure 2. Other parameters set in the simulations will be based on the data in Table 1 on Appendices.

$$\rho_{mix} = \alpha_{oil} \rho_{oil} + \alpha_{water} \rho_{water} + \alpha_{gas} \rho_{gas}$$

$$\mu_{mix} = \alpha_{oil} \mu_{oil} + \alpha_{water} \mu_{water} + \alpha_{gas} \mu_{gas}$$

Figure 2: Density and viscosity equation

The pressure at the inlet of the device as labelled in figure 1 will be set to 157.5bar in respond with the reservoir pressure stated in the research scope. The suitable pressure at the outlet will be calculated in respond to the targeted flow rate of 100m³/day. The flow rate of the fluid will be adjusted based on the result of the simulation. Note that 100m³/day of flow rate is relatively high because the simulation of the fluid flow is only through half of the cross section of a single device. The flow rate will later be adjusted most probably lower than the preliminary simulation.

Moving on to the geometry of the simulation, the design of the body of the flow will be based on figure 1 with initial inlet flow area of 2.5mm representing only the radius of the device since the simulation is only half the device. The distance between the levitating disc and the right wall will be initially specified to 1mm. The levitating disc will be set fully open, leaving the flow area above the disc of 2mm as illustrated in figure 3.

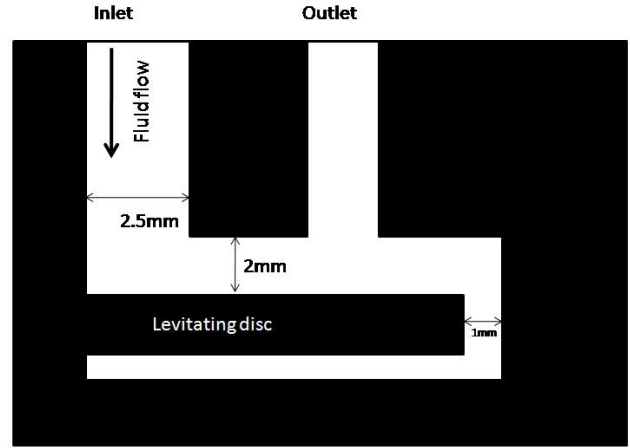


Figure 3: Illustration for the first simulation

After setting up the geometry of the flow area, that geometry will be meshed so that the analysis was done in smaller fraction of the body. Higher mesh number will results in greater precision of simulation and analysis of the flow through the device.

Result of the simulation should provide us with pressure profile across the cross section of the Autonomous Inflow Control Device. The main concern of the simulation is the velocity distribution along the flow area. If there are so little different in velocity across all the concerning flow area specified in the geometry of the flow, the flow rate of the fluid flowing through will be adjusted. This situation is related with the practice in the field. The production flow rate will be reduced in order to optimize the functionality of the device.

Later, pressure and velocity concerning area along the device will be recorded and compared. The velocity above the disc is the most important as stated in the theory; low velocity of oil will leave the flow area open. The area below the disk will be set as the minimum distance between the disk and the sitting.

The simulation will then be repeated with different density and viscosity. The geometry and meshing of the body will be kept the same. The density and viscosity of the fluid used will contain 10% water of density 1045kg/m³ and viscosity of 0.45cP. the gas composition will be kept constant at 30%. This will left the oil content in the same flow rate with 60%. The total density and viscosity will be calculated using the same formula as in figure 2. The analysis and simulation will be repeated with 20%, 30%, 40%, 50%, 60% and 70% water saturation. This will cause the total density of the fluid to be increased as the water content increased. Meanwhile, at the same time, the viscosity will reduced enabling the fluid to flow easier and create high velocity flow inside the device. In real situation where the levitating disc can move, the high velocity fluid flowing above the disc will suck the disk move it closer to the upper sitting increasing the fluid restriction due to smaller flow area.

INLET SIZE MODIFICATION

All those steps will represent the performance of the device and its sensitivity towards the flowing fluid's properties with flow area of 2.5mm radius. The simulation will be repeated with decreasing flow area at the inlet. 2.25mm inlet flow area will be used for the next simulation representing decrease of 0.5mm of diameter of the device. The simulations will be repeated with the same flow rate used before. The same set of variables will also be used as before. The different fluid density and viscosity caused by setting variety of fluid composition ranging from zero water content to 70% water content with the increment of 10% for every set of simulation will be repeated. The disc placement closer and full open will also be used. This is to show how the fluid flows in both of the condition. The results between this two flow areas will show whether the disc will remain closer as the fluid flow is low density and high viscosity or not. The decrease of the inlet flow area will be repeated until it reaches 0.5mm representing 1mm in diameter of the inlet. If the first two decrements of inlet flow area are unfavourable, where they produce worse results, the inlet flow area should be increased until it reach optimum area.

DISC SIZE MODIFICATION

After the simulations based on the different of inlet flow area were done. Further changes will be done to the device. The next modification of the device is distance of the disc to the right wall based on the diagram. In other words, the size of the disc will be reduced. This modification will leave the space between the disk and the body of the device became bigger. The decrease of size will be 0.25mm at a time. The same rule applies when decreasing the size of the levitating disc. If the performance becomes worse after a few simulations, the size of the levitating disc will be increased instead of decreased. The increment will be done until the side of the disc touches the body of the device leaving no flow area to the bottom of the disc.

OUTLET SIZE MODIFICATION

The next modification to be done is the flow area of the outlet. Same method will be applied to the outlet area. If there is no significant effect or if the increases of the outlet flow area made the performance worse, the area should be increase on order to produce more optimized results. After all the parameter and changes had been done and went through simulations, analysis will be made to select the best parameter for the device considering both high and low water content of fluid flow in the simulations. For example, the pressure profile across the above surface of the levitating disc will look like figure 5 below;

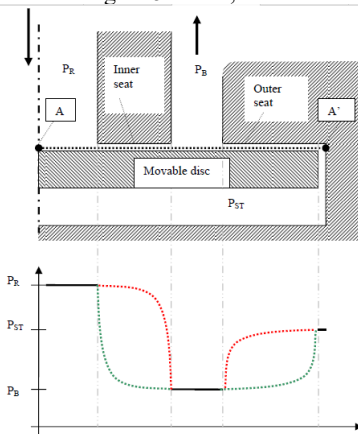


Figure 4: Pressure profile at levitating disc in closed position [9]

After choosing the best flow areas for the device to work effectively, run another set of simulations with different flow rate but using the optimized design obtained in the previous sets of simulations. This is to show the effects of flow rate to the specific design of Autonomous Inflow Control Device.

IV. RESULTS AND DISCUSSION

BASIC SIMULATION AND INDUSTRIAL CONCERN

The industrial concern about high water production needs the implementation of AICD which restrict the flow of fluid containing high water content to reduce water production and delay water breakthrough. For the device to work autonomous feature of the device should be reliable to distinguish the fluid flowing through. The basic simulation ran without any modification showed that pressure distribution acted on the disc varies according to water content of the fluid flowing.

This basic simulation is the base simulation use in this research that run on the parameter of 2.5mm inlet radius, 5mm outlet opening and 29mm radius levitating disc. The effects can be seen on the top surface of the disc based on pressure distribution along the 29mm surface of figure 6. However the working principle of the device should allow the disc to be lifted due to the change of water content in the fluid flowing through the device[9]. These results drive the analysis to extend its observation to bottom surface of the surface of the disc in figure 7. As the trend of pressure exerted along the disc depicted in the graph followed the trend showed in the article[9], the simulation can be considered reliable.

The results in figure 7 showed that the pressure exerted below the bottom surface of the disc increase as the fluid flowing through the device contained higher water content. This allows the disc to be lifted up as the fluid content change. Thus, the device was proven to be capable of controlling the flow autonomously as stated by Tendeka[10]. This is due to higher density lower viscosity of water increased the density and reduced the viscosity of the mixture. According to Bernoulli, kinetic energy which is in the function of density and viscosity increased as more water flow below the surface of the levitating disc and no way out. That causing the pressure exerted on bottom of the levitating disc to increase as more water contain in the flowing fluid.

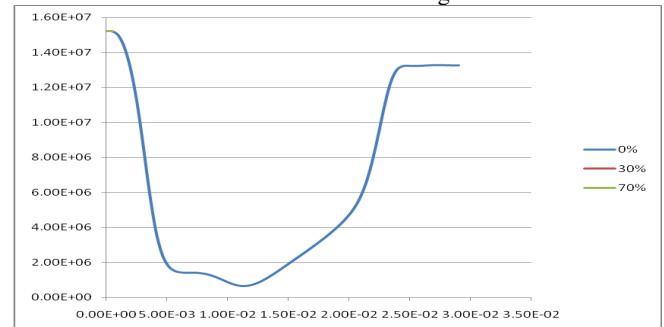


Figure 5: Pressure on the topside of the disc with different of mix fluid with water content

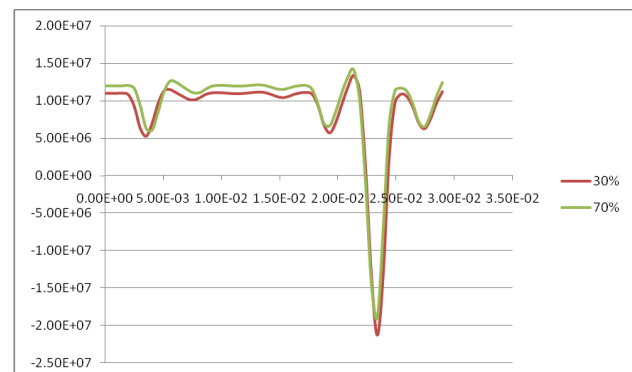


Figure 6: Pressure on the bottom side of the disc with different of mix fluid with water content

INLET SIZE MODIFICATION AND IMPLEMENTATION OF INFLOW CONTROL DEVICE

The implementation of AICD in production system can improve flux equalization as high as 45% according to a study on balancing flux equalization and production[2]. However this improvement

came with the cost of reducing the well productivity up to 55%. For the improvement of the flux equalization to worth its cost, the device have to work efficiently. Thus, this lead to modification intended in this paper to further distinguish the flow of different water content

The sizes of the inlet were increased to 5mm radius with other parameters follow the basic simulation. The graph in figure 8 shows that by increasing inlet radius show better pressure difference between two fluid compositions, 30% and 70% compared to figure 6. However the different between the simulations done are too small. The difference only distinguishes those two flows by 0.4%. However, the modification still bring about changes to the effectiveness of the device if compared to the basic simulation where there are almost no significant changes can be seen on the top surface of the disc as the fluid content changes. This is due to the kinetic energy reduced and conserved in the form of pressure. Referring to Bernoulli's theorem, the total energy of a flowing fluid is the same through any point of the flow. When inlet size increased, the velocity at the inlet decreased. These low velocities reduce the disruption cause to the velocity when the momentum of the fluid flowing changes as it hit the levitating disc. When the velocity is conserved, the higher difference in kinetic energy can be obtained as the fluid flowing has different density and viscosity.

The implementation of this device which as stated in the journal can improve flux equalization and delay water breakthrough[2] with the cost of reducing well productivity should taken the device specification into critical consideration as it well proven this research that the changes made to device will affect its effectiveness. Even though for this part of simulation, the significant of the modification is only 0.4%, it still changes the outcome of the implementation of the device if it used in the field.

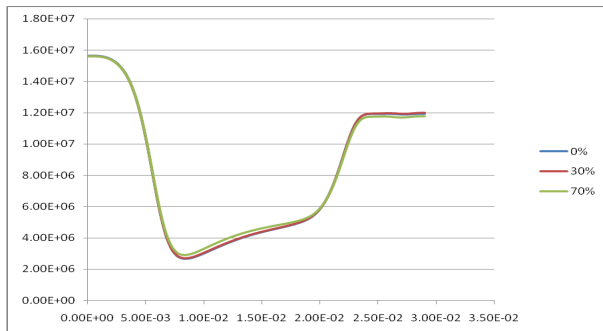


Figure 7: Pressure distribution on the topside of the disc with 5mm inlet radius

OUTLET MODIFICATION

The outlet opening were reduced to 2.5mm with the used of 5mm inlet radius. This is because 5mm inlet radius show better efficiency compared to 2.5mm inlet radius. The results in figure 9 show better trend in term of pressure exerted on the surface of the levitating disc compared to figure 8. The differences in pressure distribution improve as the outlet opening was increased. The trends distinguish the flow by 3.3%. This results show that the outlet opening is more sensitive to changes if compared to the inlet radius of the device.

The reduction of the outlet area cause higher outflow velocity. This results to higher kinetic energy that will further increase by difference in density and viscosity. Thus, the pressure distributions will further vary as different composition flows through it. The results show better trend in term of pressure exerted on the surface of the levitating disc. The differences in pressure distribution improve as compared to the previous parameter modified. The trends distinguish the flow by 3.3%. This results show that the outlet opening is more sensitive to changes if compared to the inlet radius of the device.

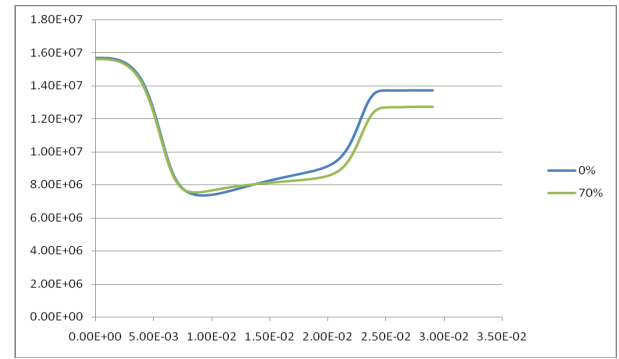


Figure 8: Pressure distribution of the top surface with the outlet opening of 2.5mm

DISC SIZE MODIFICATION AND WORKING MECHANISM

This set of simulations was done by decreasing the radius of the levitating disc from 29mm which leaves 1mm space between the disc and inner wall of the device to 25mm radius which increases the space to 5mm. The inlet and outlet sizing is 5mm and 2.5mm respectively.

These parts of simulations show that the decrease of the disc radius affects the device performance in a negative way as shown in figure 9. The pressure distribution with no zero water content is lower than high water content. The difference between pressure exerted by 30% water content and 70% water content is closer than it did when the disc is bigger. This show that flow became less distinguish as the disc size was reduced. It happened because the kinetic energy that supposed to vary the pressure exerted on the surface flow below the disc as the area to the bottom of the disc was increased. These simulations were done with at least 2 cells for every space in the device during meshing. With the size of 0.5mm per cell, the disc size was not increased bigger than 29mm so that it leaves at least 1mm of space between the disc and inner wall or 2 cells.

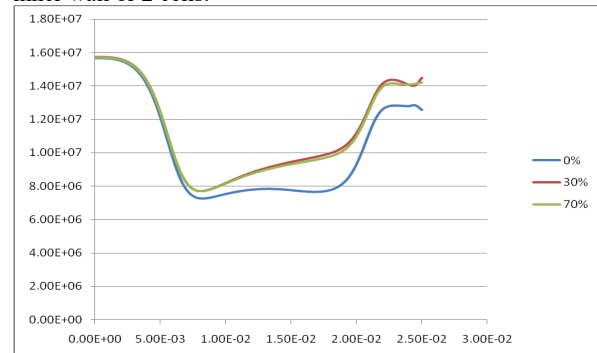


Figure 9: Pressure distribution on top surface of the disc

V. CONCLUSION

This research was done to study the effects of modification done to the device to the ability of the device to differentiate different fluid flowing. Taking account the basic working mechanism of the Autonomous Inflow Control Device where the device are able to control the inflow of fluid through it autonomously, previous study on the performance of its implementation was analyze. This highlighted the importance for the device to work efficiently. This was also because its implementation came with the cost of reducing the well productivity to ensure flux equalization, uniform sweep efficiency and higher production efficiency.

Optimization of the device specification was proven in this research to be a critical element for the device's ability to work autonomously and effectively. The modification of the outlet especially showed the most sensitive effects compared to others. However the size of disc also needs to be designed meticulously as

it may affects functionality. All this parameters need to be designed base on the specific condition of the reservoir and the well where the device will be implemented.

Other than that, high solid content can also affect the efficiency of the device in field. It can clog the device and reduced its effectiveness. Thus, it is important to use sand filter at the production string to filter out the solid from entering the device.

ACKNOWLEDGMENT

Thank you to my supervisor PM Ir. Dr. Nadiyahnor Md Yusof for supervising and helping this research. Universiti Teknologi Mara for the opportunity to do this research.

References

- [1] 1. Youngs, B., K.J. Neylon, and J.A. Holmes, *Recent Advances In Modeling Well Inflow Control Devices In Reservoir Simulation*. International Petroleum Technology Conference.
- [2] 2. Lee, B., S.A. Faizal, and A. Galimzyanov, *An Innovative ICD Design Workflow to Balance Flux Equalization and Well Productivity in Horizontal Wells*. Society of Petroleum Engineers.
- [3] 3. Gualdrón, M.B.G., et al., *Autonomous Inflow Control Devices (AICD) Application in Horizontal Wells Completions in Rubiales Area, Heavy Oil Reservoir*. Society of Petroleum Engineers.
- [4] 4. Al Hasani, M.A., et al., *Diagnosis of Excessive Water Production in Horizontal Wells Using WOR Plots*. 2008, International Petroleum Technology Conference.
- [5] 5. Mojaddam Zadeh, A., et al., *Optimal Inflow Control Devices Configurations for Oil Rim Reservoirs*. Offshore Technology Conference.
- [6] 6. Aadnoy, B.S. and G. Hareland, *Analysis of Inflow Control Devices*. 2009, Society of Petroleum Engineers.
- [7] 7. Bybee, K., *Inflow-Control Devices for Reducing Water Production*.
- [8] 8. Zeng, Q., et al., *A novel autonomous inflow control device design and its performance prediction*. Journal of Petroleum Science and Engineering, 2015. **126**(Supplement C): p. 35-47.
- [9] 9. Halvorsen, M., G. Elseth, and O.M. Naevdal, *Increased oil production at Troll by autonomous inflow control with RCP valves*. Society of Petroleum Engineers.
- [10] 10. Tendeka, *Data Sheet - Inflow Control*, in *FloSure Autonomous ICD*, Tendeka, Editor. 2015.
- [11] 11. Vela, I., et al., *Well Production Enhancement Results with Inflow Control Device (ICD) Completions in Horizontal Wells in Ecuador*. Society of Petroleum Engineers.
- [12] 12. Fernandes, P., Z. Li, and D. Zhu, *Understanding the Roles of Inflow-Control Devices in Optimizing Horizontal-Well Performance*. Society of Petroleum Engineers.
- [13] 13. Barragán-Hernández, V., et al., *A strategy for simulation and optimization of gas and oil production*. Computers & Chemical Engineering, 2005. **30**(2): p. 215-227.
- [14] 14. Morooka, C.K., I.R. Guilherme, and J.R.P. Mendes, *Development of intelligent systems for well drilling and petroleum production*. Journal of Petroleum Science and Engineering, 2001. **32**(2): p. 191-199.
- [15] 15. Fayal, Z.F., B. Lakhdar, and N. Zoubir, *Horizontal Well Performance Flow Simulation CFD-Application*. Society of Petroleum Engineers.
- [16] 16. Ismael, M., et al., *Well Completion Data Adjustment Workflow for Reservoir Simulation*. International Petroleum Technology Conference.
- [17] 17. Frantz, J.H., Jr., J.P. Spivey, and C.W. Hopkins, *Practical Production Data Analysis*. Society of Petroleum Engineers.
- [18] 18. Henriksen, K.H., E.I. Gule, and J.R. Augustine, *Case Study: The Application of Inflow Control Devices in the Troll Field*. Society of Petroleum Engineers.
- [19] 19. Todman, S., G. Wood, and M.D. Jackson, *Modelling and Optimizing Inflow Control Devices*. Society of Petroleum Engineers.
- [20] 20. Marir, B., A. Allouti, and D.O. Cobb, *Simulation Modeling of ICDs and MLTBS in a Green Field Offshore Abu Dhabi*. Society of Petroleum Engineers.
- [21] 21. Porturas, F., *Enhanced Production with ICD and AICD Completions in Oil Wells: Case Studies From Latin America*. 2016, Society of Petroleum Engineers.
- [22] 22. Rushd, S., et al., *CFD Simulation of Pressure Losses in Eccentric Horizontal Wells*. Society of Petroleum Engineers.
- [23] 23. Cai, J.J., et al., *Hydrate Prevention in Subsea Oil Production Dead-Legs*. 2013, Offshore Technology Conference.