VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL INVESTIGATION ON THE EFFECT OF THE INLETS ON THE PERFORMANCE OF OIL/WATER SEPARATION IN HYDROCYCLONE

Khor Y. Yin, Hussain H. Al-Kayiem and William P. K. Son
Mechanical Engineering Department, Universiti Teknologi Petronas, Bandar Seri Iskandar, Perak Malaysia
E-Mail: hussain_kayiem@petronas.com.my

ABSTRACT

Hydrocyclone are widely used in oil and gas industry particularly in this study similar to de-oiling process or improve purification of discharged water. As offshore oil production started to decline over time with high water cut in the production stream, this usually cause unprofitable production for huge expenditures in water handling. The periodically collapse in oil price further heightened the awareness on production technology to reduce unit costs. This paper offers a comprehensive suite of numerical simulation to predict the performance of oil-water hydrocyclone in an effort to reduce amount of water production at surface. Separation performance impact from single, dual and quad inlets configuration is simulated based on a tested turbulence and multiphase model. This investigation was carried out using Computational Fluid Dynamics in ANSYS-FLUENT 14 environment. The hydrocyclone swirling flow was simulated using *RNG* swirl dominated k-ε Turbulence model while the interface between crude oil and water was achieved using the Discrete Phase model. Based on the simulation results, it is concluded that hydrocyclone separation efficiency is affected by inlet velocities at tangential inlet and number of tangential inlets.

Keywords: hydrocyclone, separation efficiency, DOWS, turbulence, tangential inlets.

INTRODUCTION

As time passing by, water production becomes unavoidable due to oil is naturally accompanied by an underlying aquifer and nature of water is intrinsically more mobile than light oil. Factors lead to high water production include water coning during high production drawdown, water contact rises after primary depletion, water breakthrough due to water injection in secondary recovery project or reservoir in strong water aquifer environment. The increased water production will consequently translate into higher operating expenditure. The operational expenditures include cost of lifting oil together with water from the ground, produced water disposal costs, facilities debottleneck and corrosion management associated with large volumes of produced water.

To rectify the poor performance of the wells concerned due to simultaneous production of excessive unwanted water, this research effort focus on the development of Downhole Oil-Water Separation (*DOWS*) technology. *DOWS* is a hydrocyclone-based system for downhole separation of water and produced oil inside a well. The separated produced water will be re-inject to the other water bearing zone. Globally, it has been successfully implemented in a number of onshore wells, for example Alberta Canada, France, North Sea and China [1]. The longest recorded run life is about 590 days in onshore. Nevertheless, based on the feedback and feasibility study on onshore and offshore installations, the utilization of *DOWS* in offshore platform is limited due to its complicated apparatus set up, unfavorable well trajectory, space constraint for power generation and difficulty of well re-entry for troubleshooting. The concept of an ideal offshore installation is to have no moving parts and ability to maintain energy for production, injection and separation. This is essential to reduce installation complexity without any artificial lift suction as suction/discharge pump and motor. Such concept demands a relative positioning of the *DOWS* in between a high pressure production layers on top and a low pressure water zone below to accomplish hydrocyclone separation. Preliminary assessment for deployment in Malaysian oil fields is deemed favorable due to availability of multi-stack reservoir with various depletion rates.

In this study, numerical modelling using computational fluid dynamics (*CFD*) is used to quantitatively predict hydrocyclone performance for a range of design and operating parameter with the assumption that model parameters were determined from the available experimental data and field data from offshore Sarawak, Malaysia. The adopted modelling strategy enables model validation work with experiment data and model optimization at the same time to suit offshore wells in Sarawak. In addition, hydrocyclone design amendments can be done numerically before the employment of pricey experimental investigation. The flow behavior in hydrocyclone is modeled and simulated using ANSYS-Fluent 14.5 commercial software to examine and study the outcome of separation efficiency versus inlet velocities and evaluate the impact from different number of tangential inlets.

Fundamentals of the hydrocyclone & modelling

The hydrocyclone separation is fundamentally straight forward which is explained by schematic diagram of hydrocyclone exhibited in Figure-1 below. The pressurized oil-water mixture is transmitted horizontally

VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

and tangentially through an inlet into a vertical cylinder, resulting in an intense swirling motion within the cyclone. Basically, small inlet will promote droplet break-up while big inlet will result in inadequate swirl intensity as stated by Vazquez *et al*. 2004 [2]. The conical shape of the hydrocyclone speeds up the mixture to stream in a helical manner, generating a free vortex and producing large centrifugal forces. The cylindrical section facilitates the particles to experience the centrifugal force, hence prolonging the residence time so that denser fluid can travel towards the wall. The cone further restricts the downward flow, generating a central spiraling upward flow. The narrower the cone angle, the smaller percentage of secondary phase leaks from underflow. Nevertheless, the cone angle is proportionate to the pressure disparity between the overflow and underflow. The oil-rich stream discharges through the overflow outlet and a water-rich stream emits the system via the underflow.

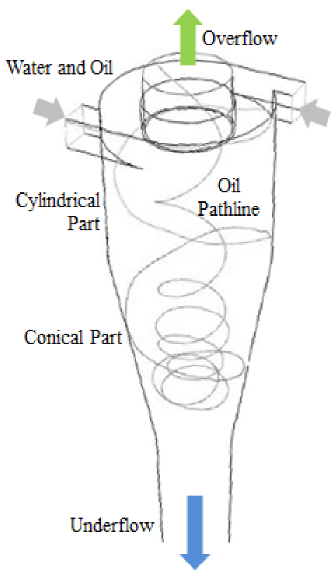

Figure-1. Basic design of a hydrocyclone.

Three key factors for an effective separation are density difference, surface tension between phases and water phase salinity. Separation efficiency improves with higher density differential; hence, to attain maximum density difference in the system, the most positive crude oil condition is at higher temperature and at bubble point pressure when oil is at its minimum density. Whereas for the formation water is best to be more saline, containing higher salt content. This will enhance the separation process as a result of the higher surface tension as well as the coalescence between droplets is augmented together with higher density nature of the salt water [3]. This paper examined the feasibility of *DOWS* technologies that is intended to reduce water cut from an oil producing well.

The hydrocyclone exhibited in Figure-2 which was experimentally studied in 1980 has been a significant reference for liquid-liquid hydrocyclone model [4].

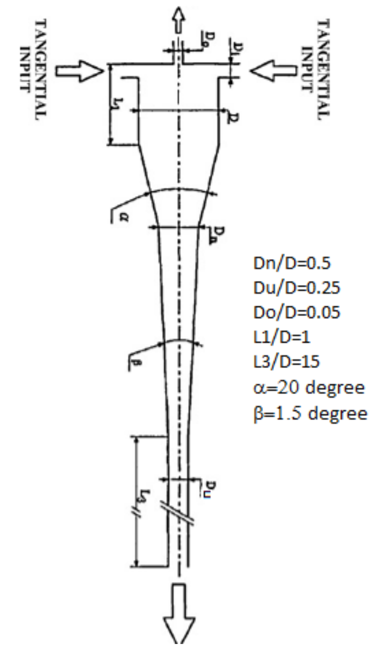

Figure-2. Hydrocyclone geometry, D = 60 mm.

This design has an inlet chamber and a reducing section to accomplish enhanced tangential acceleration of the fluid, lessening the pressure plunge and the shear stress to a tolerable level. Most of the separation is accomplished at the tapered segment where incremental lowering of conical segment lessens the rate of swirl intensity thus facilitating prolonged residence time for the denser phase. Ultimately, a long tail pipe cylindrical segment in which the smallest and lighter droplets migrate to the reversed flow core at the axis are being isolated into the overflow exit.

Dimension of this standard hydrocyclone with dual tangential inlet from Colman and Thew *et al.,* 1980; as exhibited in Figure-2 is utilized as this has been the widely reported design as per numerous researchers such as Young *et al*., 1990; and Gomez *et al*., 1990. With the model efficiency is highly dependent on the turbulent flow velocities, inlet configuration is compared to evaluate tangential velocity impact. Figure-3 illustrates the design configuration of single, dual and quad inlets.

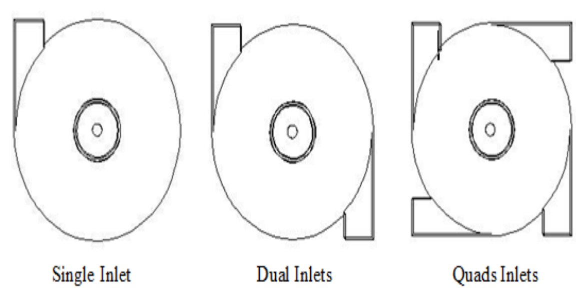

Figure-3. Hydrocyclone inlet congifuration.

VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Numerical modelling

Numerical analysis is widely used to predict and maneuver the hydrocyclone to attain the desired performance. The *DOWS* model can be divided into analytical solutions and numerical simulations. Boundary specifications are based on analytical solution and actual field data where flow rate is a function of differential pressure. With the potential productivity feeding from upper zone and underflow injecting at lower disposal zone, *DOWS* separation concept is described as exhibited in Figure-4. This figure illustrates the operating conditions in the hydrocyclone which are established by the Material Balance and injectivity formula. The flowing Material Balance utilizes the concept of "Pseudo-Steady-State" or stabilized flow to assess well productivity at longer production time in a bounded reservoir. Flow enters the pseudo-steady-state regime when the pressure momentarily reaches all boundaries after sufficient drawdown duration. Throughout this period, the decrease in pressure rate is nearly identical at all points in the wellbore and reservoir. Thus, the difference between the average reservoir pressure and wellbore pressure reaches a constant with respect to time. Pseudo-steady-state productivity index is defined as the production rate divided by the differences of average reservoir pressure and wellbore pressure. Thus, the productivity index is basically constant as in (1).

$$Q = PI\ (P - \overline{P}wf) \tag{1}$$

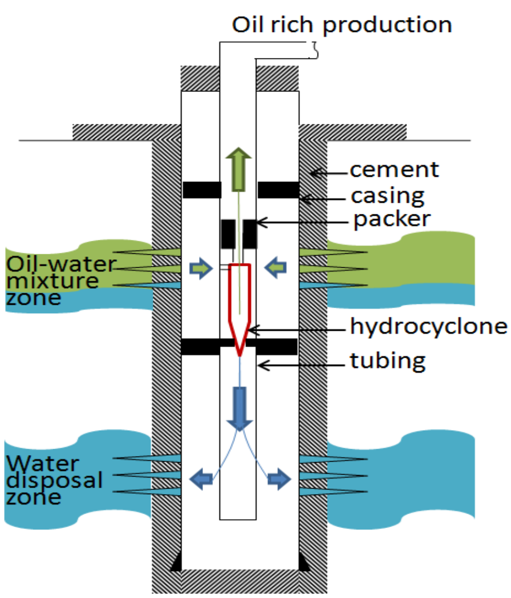

Figure-4. Illustrates that the oil-water mixture with high pressure enters cyclone to complete separation, causing the oil rich stream to exit the system via overflow to production tubing and heavier water stream being dispose to depleted water zone via underflow outlet.

Pseudo-Steady-State is derived from Darcy law for radial flow into a vertical well located at the center of a closed isotropic circular reservoir as in (2)

$$\overline{p} - p_{wf} = \frac{141.2 B_o \mu q}{kh}\left(\ln\frac{r_e}{r_w} - \frac{3}{4}\right) \tag{2}$$

The above equation is combined with productivity index equation (3).

$$PI = \left(\frac{0.00708\ h\ k}{\ln\frac{r_e}{r_w} - 0.75 + s}\right)\frac{kro}{B_o \mu} \tag{3}$$

As for injectivity aspect, it is a crucial feature as the flood life is depending on the rate at which water can be injected and disposed. The magnitude of injection depends on the rock and fluid properties. Well injectivity is a function of skin, the distance between the injector and producer, accompanied by the pressure drop, and also a function of the formation thickness, oil viscidness and effective permeability to the displaced fluid. The calculation can be undertaken analytically which we assume the displacing and displaced fluids are compressible, have mobility ratio of one and the reservoir has uniform properties. For direct line drive, it can be estimated as stated in (4):

$$I = \frac{1.538x10\ \ k\ kro\ h\ \Delta p}{\mu \log\ (a/r_w) + 0.682\ d/a - 0.798} \tag{4}$$

Since fluid flow occurs from top to bottom, the above equation is combined with a productivity index equation

$$Q = PI\ (P - P_{wf}) = I\ (P - P_{wf}) \tag{5}$$

Nevertheless, as this region regularly suffers formation damage due to fine migration, clay swelling, paraffin and asphaltene precipitation, deposition of scales from injected water or filtrations of particulates from injected water, these damages can be characterized as skin factor [5]. Figure-5 illustrates the ranges of sensitivity run on three highly uncertain variables, namely delta pressure (Δ P), Darcy factor (k permeability) and non-Darcy factor (S skin). From this run, it is determined that flow rate of the field is ranging from 0.4m/s to 2.2m/s, with base velocity at 2m/s.

VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

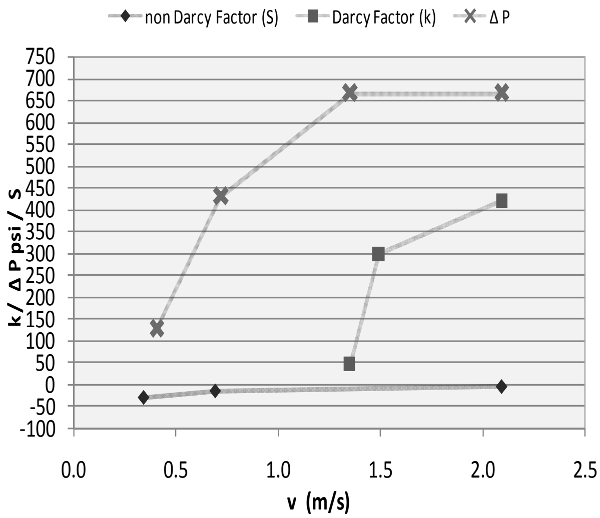

Figure-5. Sensitivity plot illustrates the cross flow rate between upper and lower zone are highly sensitive to non-Darcy skin factor, follow by delta pressure and Darcy factory like permeability. Flow rates at field condition are ranges from 0.5 m/s to 2.2 m/s.

Numerical modelling options for hydrocyclone complex flow behaviour has been evaluated based on previous literature reviews. RNG swirl dominated k-ε Turbulence Model was selected for radius < 44mm according to few authors recommendation such as Dyakowski *et al*.,1993; Fraser *et al*. ,1997; He *et al*. ,1999; Narasimha *et al*., 2003[6]. The RNG-based k-ε turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called renormalization group methods. Strong swirling flow and rapid strained flow accuracy refinements has been appropriately accounted for the hydrocyclone modeling.

The stated model was solved using three-dimensional unstructured hexahedral. In order to capture the reversed flow at the hydrocyclone, block structured mesh is generated using ANSYS ICEM workbench meshing. This model comprised of 492186 cells as exhibited in Figure-5 for $Do = Dn$ = 15mm.

The flow in hydrocyclone is a multiphase flow consisting of dispersed oil particles throughout the produced water. Multiphase flow can be solved by a number of techniques including Eulerian Mixture or Volume of Fluid models and Lagrangian approach. In this study, Discrete Phase Modelling (*DPM*) technique was selected to reflect the particle separation behaviour using Lagrangian approach. This approach is the best option since each and individual particle path can be traced based on force balance on individual particle hence allowing quantify separation efficiency at overflow outlet to be done.

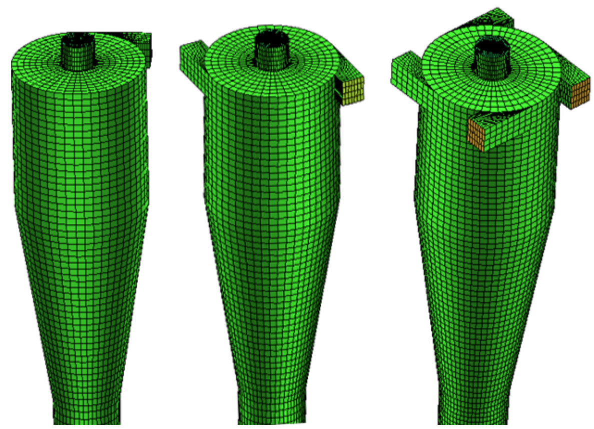

Figure-6. Meshed hydrocyclone geometry used in CFD simulation.

Each particle is balanced between centrifugal force, gravity, drag, and buoyancy lift. The advantages of *DPM* also include detail analysis of oil particle size distribution impact. For Pressure-Velocity coupling, SIMPLE algorithm is employed while First Order Upwind scheme is applied for the momentum. As for pressure interpolation scheme, PRESTO! (PREssure STaggering Option) scheme is adopted. These selections were the recommended scheme with the aim of improving the estimation for highly swirling flow simulations.

To validate this numerical model, experimental study from the previous literature with the closest operating data is compared. Assuming isothermal and steady state condition, the inlet boundary specifications are assumed as an inflow flow rate ranges from 40 lpm to 70 lpm with pressure at inlet set at 90psi as per literature [5]. The primary phase is assumed to be water with a density of 1000 kg/m³ and a viscosity value of 0.97 cP. Inert liquid spherical particles, where oil is assumed with density and viscosity of 840 kg/m³ and 0.45 cP respectively. The oil droplet sizes are ranged from 2.3µm to 200µm with mean drop size of 55µm. In most cases, larger droplets mobilize faster and increase cyclone efficiency [3].

RESULTS AND DISCUSSION

The separation performance of hydrocyclone is closely related to tangential velocity where it describes the value of centrifugal acceleration force. This paper will study the impact of inlet configuration on 50% flow split for single inlet, dual inlets and quad inlets. Average peak tangential velocity is at 2m/s same for all configuration for comparison purpose. For the experimental data, Colman's 1980 work [8] is used to validate the D =60mm hydrocyclone performance at flow rate as shown in Figure-7. On the other note, velocity vector at for each inlet configuration in shown in Figure-8 to Figure-10.

VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

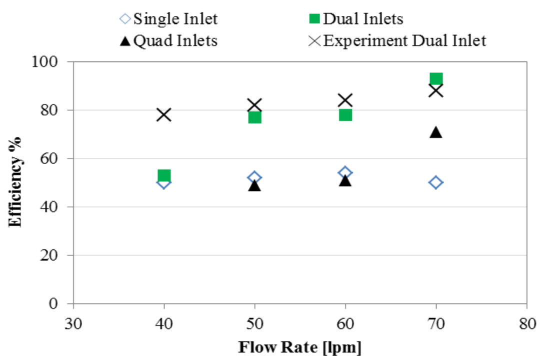

Figure-7. Comparison of hydrocyclone model efficiency with experimental data.

It is observed that the separation efficiency of dual inlets configuration is at highest performance, follow by quad inlets and lastly single inlet. The findings are consistent with Colman *et al*., 1984 and Thew *et al*., 1984 [4].

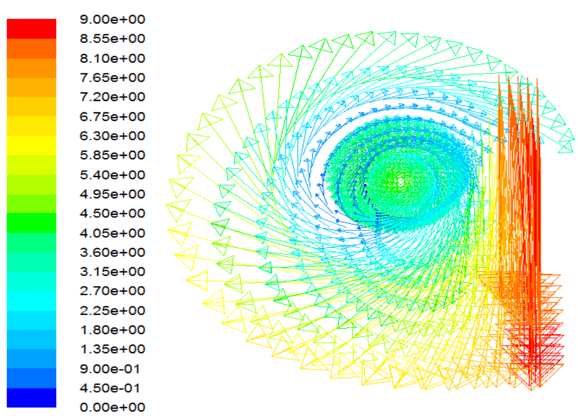

Figure-8. Single inlet model with velocity vectors colored by velocity (m/s).

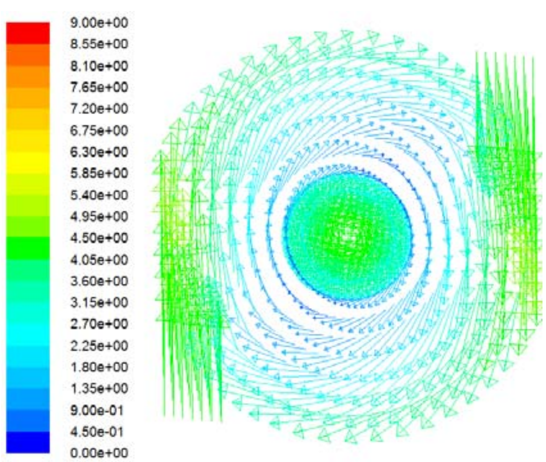

Figure-9. Dual inlets model with velocity vectors colored by velocity (m/s).

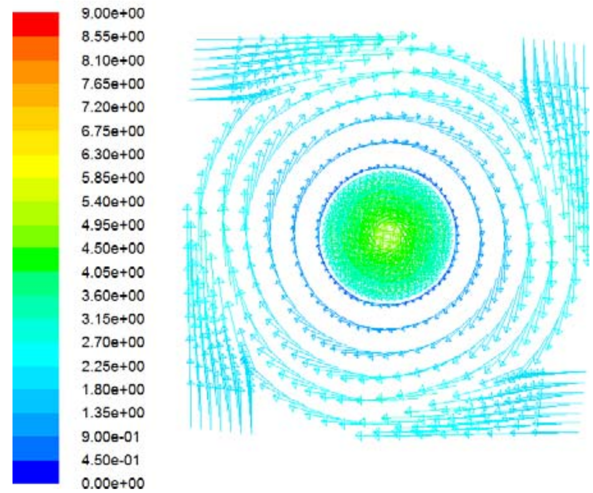

Figure-10. Quad inlets model with velocity vectors colored by velocity (m/s).

Comparing the velocity vector for all inlets configuration, the dual inlets velocity vector in Figure-9 is observed to have a better performance due to more symmetrical swirl as compared to other configurations. It is shown that more stable reversed flow of oil particle can be maintained at the centre core by having twin inlets. Single inlets in Figure-8 on the other hand illustrates a downward swirling effect due to high velocity for the same mass flow rate which causing more light particles are leaving through underflow outlet. As for Quad inlets in Figure-10, the mixture flow is disrupted at short circular section, leading to emulsification phenomena where droplets breakdown happened.

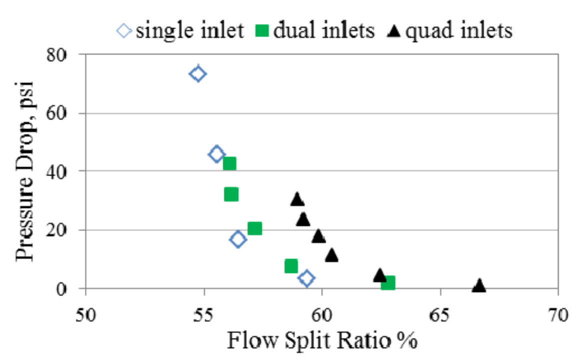

Figure-11. Pressure drop at underflow vs. flow split ratio for volumetric flow rate from 40lpm -250lpm.

Since the conceptual development of this *DOWS* is to re-inject produced water into the aquifer, it is necessary to study its effect on separation process on pressure drop. Figure-11 shows for single, dual and quad inlets configuration model, the resultant pressure drop is proportionate to the total volume discharged downward. This can be explained by the fact that the pressure loss is a function of flow rate. Flow split is the ratio of overflow rate to the inlet flow rate, hence, the lower the flow split, the higher fluid exit through underflow outlet. This parameter can either be controlled by overflow diameter or exposed backpressure at underflow outlet. For same volumetric flow rate, velocity across quad inlets is lesser compared to single inlet. This give lesser friction loss in quad inlets follow by dual inlets as per comparison illustrated in Figure-11.

CONCLUSIONS

The numerical study on flow and particle motion in hydrocyclone complex flow phenomena were able to predict reasonably with different inlet configuration using *RNG* swirl dominated k-ε Turbulence Model and *DPM*. The simulation results proved that separation efficiency with dual inlets are at better performance. Through this work, it is also concluded that pressure drop can be managed with different flow split for desired operating condition. However, investigations should be continued on hydrocyclone with higher feed oil concentration and various particle size distributions.

ACKNOWLEDGEMENTS

The authors acknowledge Universiti Teknologi PETRONAS for sponsoring the research under the YUTP-FRG 015-3AA-A34 research grants. Also, the support from PETRONAS-Sarawak Resource Management is highly appreciated.

NOMENCLATURE

q = flow rate
$\overline{P}$ = reservoir pressure
P_{wf} = wellbore flowing pressure
PI = productivity Index
B_o = formation volume factor (RB/STB)
h = net thickness (ft)
re = drainage radius (ft)
r_w = wellbore radius (ft)
μ = viscosity (cp)
I = Injectivity rate (bbl/day)
k = reservoir permeability (md)
k_{ro} = relative permeability to oil
ΔP = pressure difference between injector and producer (psi)
d = distance between lines of injectors and producers (ft)
a = distance between producers (ft)

REFERENCES

[1] Y. K. Bangash, Downhole Oil Water Separation (DOWS) Systems In High-Volume/High HP Application. SPE Wood Group ESP, Inc. SPE 81123, 2003.

[2] C. Oropeza-Vazquez, E. Afanador, L. Gomez, S. Wang, R. Mohan, O. Shoham, G. Kouba, Oil-Water Separation in a Novel Liquid-Liquid Cylindrical Cyclone(LLCC©) Compact Separator Experiments and Modeling". University of Tulsa, Tulsa,2004.

[3] K.B. Yasser, R.B Michael, E.C. Stephen, Downhole Rotary Water Separation System, Wood Group ESP Inc., Oklahoma City, US. US 6547003 B1,2000.

[4] C.H. Gomez, Oil-Water Separation In Liquid Liquid Hydrocyclones (LLHC) Experiment and Modeling, University of Tulsa, Tulsa, 2001.

[5] S. Abdus, M. I.Ghulam, L. B.James. Practical Enhanced Reservoir Engineering: Assisted with Simulation Software.PennWell Corporation, Tulsa,Oklahoma,USA,2007

[6] M. Narashimha, M. Brennan, P.N. Holtham, A Review of CFD Modelling For Performance Prediction Of Hydrocyclone, The University of Queensland, Australia, 2007.