Produced Water – Best Management Practices Conference
20-21 November 2013

Technical Paper

Compact Bulk De-watering In-Line Technology and the use of CFD in a Continual Development Programme

Dr Carl Wordsworth, Abdullah Rehman, Dr Najam Beg, Caltec Limited,
Dr Steve Howell, Abercus Limited

ABSTRACT

Conventional bulk oil-water separation is usually undertaken using gravity separators, where the phases are allowed to separate over time using gravity. One consequence of this technique is that in order to allow a sufficient residence time for the phases to separate, gravity separators tend to be large vessels. Compact separation, where high centrifugal forces are imputed on the fluids by accelerating the fluids through a cyclonic geometry, offers a step change alternative to this traditional approach. The residence time to affect phase separation is greatly reduced with the higher g-forces. This in-line compact separator technology can be used to de-bottleneck processes by removing bulk water upstream of the existing facilities. Cyclonic separators are much more compact than conventional separators and correspondingly weight is significantly reduced and pipeline design code is applicable in some cases. Also process safety is considerably enhanced due to the lower hydrocarbon inventory.

Caltec have developed a patented Water Extraction (Wx) product which is an in-line compact cyclonic separator that is designed to separate water from a water continuous oil-water mixture, the current devices are capable of recovering up to 65% of the total water at a quality of 1000ppm or better. They are designed to pipeline or process pipework code; can be installed in-line with very low pressure losses; they have no moving parts and require minimal control; have a large turn-down ratio; insensitive to inlet water cut; tolerant to gas which gives enhanced separation of fluids when present; can separate the fluids at full wellhead or process pipe line pressure and are a fraction of the size and weight of a conventional gravity separators.

Caltec's Wx technology has been extensively tested in laboratories at Cranfield, and at TUV NEL's multiphase test facilities in East Kilbride, it has also been deployed both onshore and offshore in Norway.

Computational fluid dynamics (CFD) has been extensively used in this development programme of Wx. This paper briefly describes the Wx technology and examines the continual development programme that Caltec is undertaking in order to improve its Wx oil-water separation technology. Its findings with particular reference to the CFD models that have been developed and validated are presented.

Technical Paper

1 INTRODUCTION

This paper describes the bulk-oil-water separation technology (Wx) at Caltec and the use of computational fluid dynamics (CFD) in the development and optimisation of the unit.

Most three phase separators are designed to provide sufficient disengagement space for gas/liquid phases and adequate retention time for liquid phases to establish satisfactory liquid/liquid separation. This results in designing oversized separators, thus creating issues like space, weight, pressure rating, control, hydrocarbon inventory and cost [7]. Caltec have developed an in-line compact bulk oil-water separation technology which is given the trade name of 'Wx', short for water extraction duty. The Wx unit separates the bulk of produced water from a water continuous oil-water mixture and is capable of recovering up to 65% of the total water cut at a quality of 1000ppm or better from typical North Sea crude oils.

This removal of liquids upstream of the separator debottlenecks the existing separator, thus, enhancing the performance and significantly reducing the size and weight. This approach is therefore ideally suited to offshore platforms, sub-sea applications and even land operations where the requirement to easily transport the unit is of paramount importance.

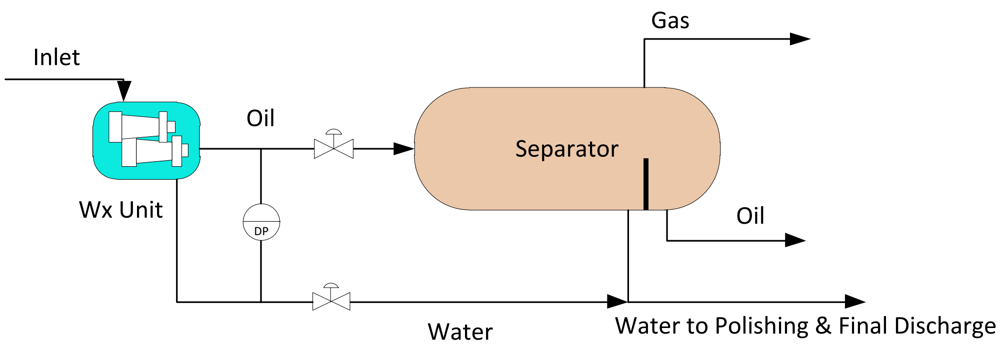

Figure 1: Wx debottlenecking Separator

Wx is designed to pipeline or process pipework code depending on location; they can be installed in-line, have a large turn-down ratio, can handle free gas, can separate the fluids at full pipe line pressure and are a fraction of the size and weight of the conventional gravity separators. The development of Wx follows the development of Caltec compact separator known as "I-SEP" which has enjoyed numerous field applications and has a number of applications including gas-liquid separation, sand separation and multiphase metering, see Figure 2.

In recent years there has been significant advancement in the field of CFD and it is now capable of simulating most complex multiphase flows to some degree, though, this CFD work was simulated using water/oil multiphase mixture only. This is because the state-of-art is still not capable of modelling all features of multiphase flow such as the gas floatation mechanism which showed enhanced separation performance during the trials data collection. The setting-up of correct physical models along with preparing flow aligned mesh is also critical for any meaningful results.

This paper also demonstrates the sensitivity of particle droplet size on the CFD predictions, the agreement of CFD pressure drop predictions with experimental data and discusses the two most commonly used CFD turbulence models:
1) Standard K-Epsilon (k-ε) 2) Reynolds Stress Turbulence Model (RSTM)

2 Wx TECHNOLOGY

Generation of high "g" forces to separate fluids of different densities is not new and dates back to the late 19th century, although the main part of the development work has been carried out since the 1950s.

The Wx technology makes use of Caltec's patented 'I-SEP' to develop a bulk oil-water separation system. The Wx technology uses a novel cyclonic separation system in order to take fluids with inlet water cuts of 65% or more (so long as the fluid is in the water continuous phase) and produce separated water containing approximately 500-1000ppm or less of oil in water for typical North Sea hydrocarbons.

The patented I-SEP [1] consists primarily of a compact dual involute and a specially designed separation chamber between the two involutes, Figure 2.

Figure 2: I-SEP Cyclonic Separator

The function of the first involute is to generate a spin and high "g" forces as the fluids enter the separator. The fluid then travels uni-axially and the high "g" forces help the fluid droplets with different densities to coalesce and be separated. The denser phase is pushed towards the wall of the separation chamber and the lighter phase gathers within the centre core. The fluids maintain their spin and tangential velocities along their path, which helps to continue the separation of phases with different densities.

Key design features, which becomes part of the know-how and the confidential information held by the designers and suppliers of these units contribute to the performance and efficiency of each unit in different applications.

A significant amount of development work has been carried out on reverse flow cyclones; particularly in applications where the flow fluctuations are minimal; the flow rate of one phase is very small compared to the main fluid phase; and the operating pressures are low [2, 3].The unique design of I-SEP makes is significantly different from conventional hydrocyclones due its tolerance to flow perturbations and minimal pressure drop, typically less than 1bar.

2.1 Benefits of Compact Separation Technology

Compact separators, such as I-SEP or Wx, offer the following benefits:

- Compactness with a small footprint
- Easy to control and no active level control is required
- No moving parts
- Pipeline design code
- Suitable for subsea application
- Can be rated to high pressures and temperatures
- Very low fluid inventory, with increased safety

- Relatively insensitive to the motion of floating platforms or vessels
- Tolerant to flow fluctuations and turn-down
- Comparatively low pressure drop across the units
- Ability to handle free gas
- Could help the existing gravity separators to improve their performance
- Key components can be easily replaced if required
- A wide range of applications, may act as flow regime conditioner

2.2 Wx Previous Testing

Figure 3 shows the Wx designed for 4000bpd that was designed and tested in Norway, both onshore and offshore, using real crude oil.

The oil-water separator was manufactured to pipeline code B31.3, ANSI Class 600, using 316L stainless steel, the design and mechanical calculations were reviewed and approved by Bureau VERITAS, (B.V.), and the unit was CE marked and had PED approval undertaken by B.V.

Figure 3: Wx-4 Tested Onshore/Offshore Norway

Figure 4 below shows a larger Oil-Water separator unit that was tested onshore in Glasgow.

Figure 4: Wx-12 Unit Tested at TUV NEL

The following conclusions were drawn from the various test programmes [4, 5]:

1. The separator was capable of recovering up to 55% of the total water with a water quality of 1000ppm or better oil in water
2. By allowing a small amount of gas, it is possible to increase the water recovery to ~ 65% whilst still maintaining 1000ppm or less oil in water
3. The separator was not affected by the inlet liquid droplet size of the fluids entering the system, over the range of conditions tested
4. The effect of inlet water cut on the separator performance is small. This is true as long as the fluids are in a water continuous regime
5. The turndown ratio of the separator has been shown to be large, with the unit able to operate with high separation efficiency at flow rates between 20% - 120% of the design flow rate
6. Gas can be injected into the system with up to 70% GVF having little effect on the separator performance. However at very high levels of gas the pressure drop across the equipment will become high, therefore it is usually recommended to include a gas-liquid separator such as I-SEP upstream of the oil-water separation to reduce the overall pressure loss through the system
7. The pressure loss experienced by the separator under the design flow conditions was identified as being close to1.6bar

3 CFD INVESTIGATIONS

Computational Fluid Dynamics (CFD) has been used on several occasions to optimise separator designs and several papers have been written to that effect. CFD operates on the basis of fundamental laws of fluid dynamics such as thermodynamics, conservation, momentum, energy and Euler equations. The Navier-Stokes equation, a set of non-linear partial differential equations, describes the mass and momentum conservation of a fluid. Solving the Navier-Stokes equations involves the use of sophisticated solution algorithms on a discretised domain of the geometry of interest with the appropriate set of physics and flow boundary conditions. These work by processing initial fluid flow conditions which form the basis for the prediction of the next flow conditions. As such it offers a means of testing theoretical advances for conditions unavailable experimentally and eliminates several physical experimentation cycles of the design process. A typical CFD program will have the following steps [8]:

a) Geometry preparation (CAD)
b) Mesh Generation (importing CAD, cleaning surfaces & meshing)
c) Pre-processing (Defining physics, setting-up boundary conditions & solvers)
d) Solver calculations (Run)
e) Post processing (Reports, plots and graphs, scalar & vector scenes, streamlines)

Like many other oil/water systems, Wx requires a secondary polishing system to achieve over board discharge limits. Therefore, CFD work was undertaken in order to understand the performance of Wx in detail; to study how the system operated and how changes to its design could help move towards larger cleaner volumes of produced water. To achieve this Abercus Limited were engaged to develop a CFD model of the Wx separator.

There were three parts to the CFD study:

1) Build a CFD model and compare results with experimental data
2) Fine tune the CFD model to best match with experimental data
3) Use the best CFD model to predict Wx performance for modified designs

By examining the actual pressure losses, separation efficiencies, flow velocities, convergence monitors and residuals it was possible to validate the CFD model that could be used during future design phases to analyse how changes in the Wx system could possibly affect the separation efficiency etc. This paper only discusses part 1 of the CFD study, part 2 and 3 will be discussed in a second paper due to be published early 2014.

3.1 CFD Package

The CFD package used throughout this work was STAR-CCM+ v7.02. Previously, the default meshing tools available in STAR-CCM+ were tailored towards programmed meshing whereby the CFD mesh is largely automated. Whilst this can bring significant benefits in terms of minimising the effort required generating a mesh, this approach generally results in a mesh with more cells. Therefore, the subsequent CFD simulations may take longer to run when compared to traditional meshing methods which are more interactive and can allow more efficient meshes to be constructed. However, the recent versions of STAR-CCM+, i.e. v7.04 onwards, has improved on this and have provided with new meshing feature called 'Directed meshing' to allow more control when generating a flow-aligned mesh.

3.2 Mesh Generation

This study was carried out using the previous STAR-CCM+ version, therefore, the traditional interactive meshing procedures were adopted to allow the body fitted mesh to be constructed, in line with QNET best practice. The body fitted mesh has a base mesh edge length of 5mm and a cell count of around one-million. Whilst in order to achieve a similar accuracy (with respect to numerical diffusion) using the automated meshes it is anticipated that the edge length may need to be halved, which could lead to a further eight-fold increase in the cell count. A comparison of the cell count is shown in Table 1 below:

Table 1 – Comparison of cell count

Mesher	No. of Cells
Gambit (Manually prepared Hexahedral mesh)	950,000
STAR-CCM+ (Automated Trimmer mesh)	2,160,000
STAR-CCM+ (Automated Polyhedral mesh)	2,890,000

A view of the body-fitted mesh within the I-SEP inlet is shown below.

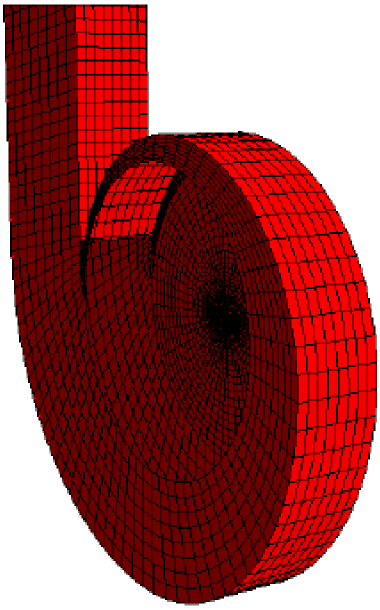

Figure 5: Gambit Mesh of I-SEP Inlet

This type of hexahedral mesh does not only have a lower cell count but is also aligned with the flow within the I-SEP and within each cyclone, therefore, enhancing the accuracy of CFD prediction for swirling flows.

3.3 Physics

As all the boundary conditions considered were steady, so the steady-state CFD solver was used by default. The Eulerian multiphase model (EMP) was used throughout this study, which is the most general and established CFD formulation and is appropriate for modelling multiphase flows involving gas-liquid or liquid-liquid separation. For example, droplets or bubbles of the secondary phase dispersed randomly within the primary or continuous phase, i.e. oil bubbles dispersed in water. The Eulerian multiphase model solves momentum, enthalpy and continuity equations and tracks the volume fractions for each phase but uses a single pressure field for all phases [10].

The multiphase segregated flow method was used along with Eulerian, as it is suitable for incompressible flows. In segregated flow approach the governing equations are solved sequentially for each phase, i.e. segregated from one another. The pressure is assumed to be the same in all phases. The volume fraction gives the share of the flow domain that each phase occupies. Each phase has its own velocity and physical properties [10].

Standard k-ε model was used throughout the study as at the time of the work, the RSTM was not yet implemented with the EMP framework in STAR-CCM+. A standard K-Epsilon turbulence model is a two-equation model for which only initial and/or boundary conditions need to be supplied and the transport equations are solved for the turbulent kinetic energy (k) and its dissipations rate (ε) [9]. The QNET best practice guidance recommends the use of Reynolds Stress Turbulence Model (RSTM) to correctly capture the anisotropic transition of free to forced vortex in cyclonic flows such as in Wx. Standard k-ε models and other models are based on assumptions of isotropic turbulence and are not suitable for strongly swirling/cyclonic flows as they tend to over predict the turbulent viscosity and exaggerate the forced vortex.

A second study was performed at a later date that compares the results of various models along with the standard k- ε turbulence model vs. RSTM. However, this work will be included in a second paper to be published next year.

The QNET guidance also indicated that it was not necessary to resolve the boundary layer conditions since the turbulence was generated within the bulk of the main flow. In line with this, the boundary layers were not resolved as part of this study, instead, the standard wall function approach was used to describe the turbulent behaviour in the near wall regions. Some additional modelling assumptions made are listed below:

- Acceleration due to gravity was included
- Each case was isothermal
- All internal wall surfaces were assumed to be smooth
- Oil droplets were 100 micron in diameter (this is an estimate – droplet diameter and was not measured during the experimental studies)
- Water was specified as the continuous (primary) phase and oil as dispersed (secondary) phase
- The Schiller-Naumann law was used exclusively to define the particle drag coefficient
- Then density and viscosity for water, oil and gas were defined as in table below:

Table 2: Fluid properties

Property	**Water**	**Oil**	**Gas**
Density (kg/m^3)	1035	797	1.18
Viscosity (Pa.s)	1.0×10^{-3}	$1.5 \text{x} 10^{-3}$	$1.0 \text{x} 10^{-5}$

3.4 Cases Considered

During this study three cases were considered, Table 3 shows the relevant data for each of the cases examined.

Table 3: Cases considered for CFD Study

CASE	**Inlet Flow Rates (m^3/hr)**			**Water Recovery (%)**	**Pressure (barg)**		**Oil in Water (ppm)**
	Water	**Oil**	**Gas**		**Inlet**	**Water Outlet**	
A	20	4.9	-	60.2	59.9	58.8	590
B	22.5	2.5	-	64.4	62.4	61.1	977
C	20.5	5.0	10.9	57.8	60.8	58.2	804

Figure 6 shows a sketch of the Wx CFD model developed:

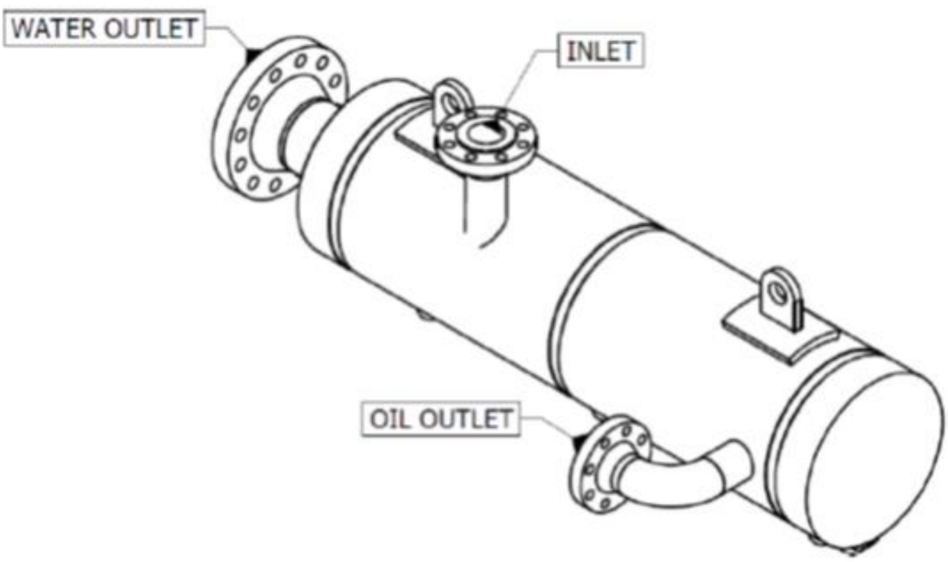

Figure 6: Model of Wx-4

4 CFD RESULTS/DISCUSSION

Figure 7 and Figure 8 show examples of the CFD plots that were obtained from this study, the two plots show the oil contraction at a range of locations with the Wx-4 model. Similar plots were also obtained for pressure contours, turbulence viscosity and velocity vectors. The total mass balance across the device, as reported at the inlet and the two outlets, was within a few hundredths of one-percent, so mass is converged.

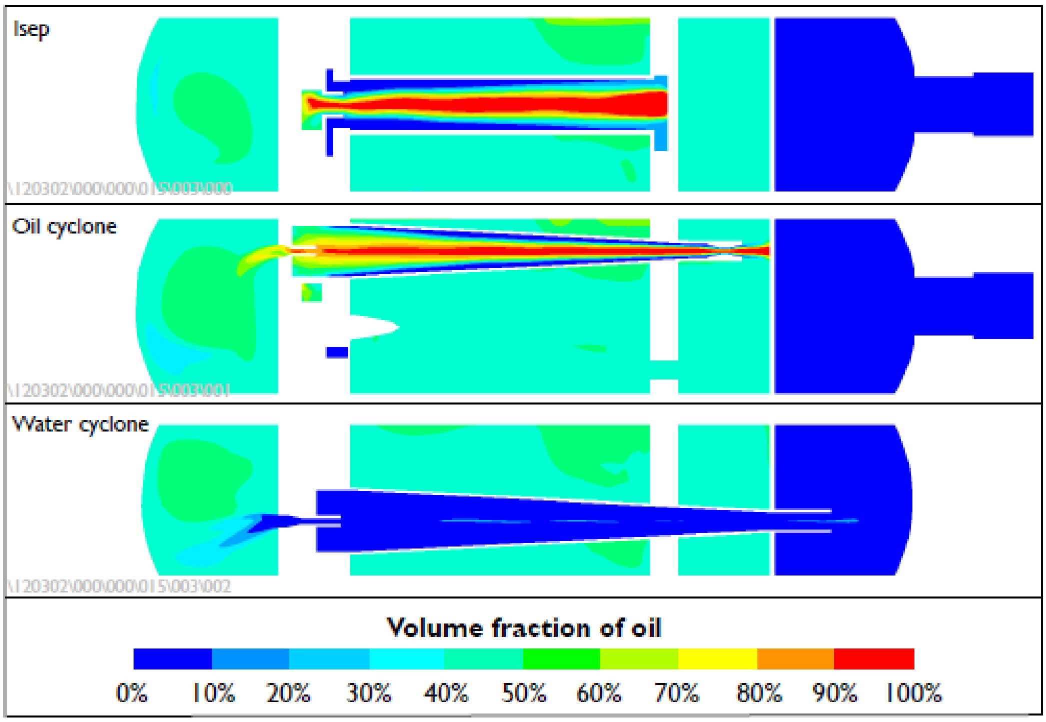

Figure 7: Contour Plots along Centre plane for Wx

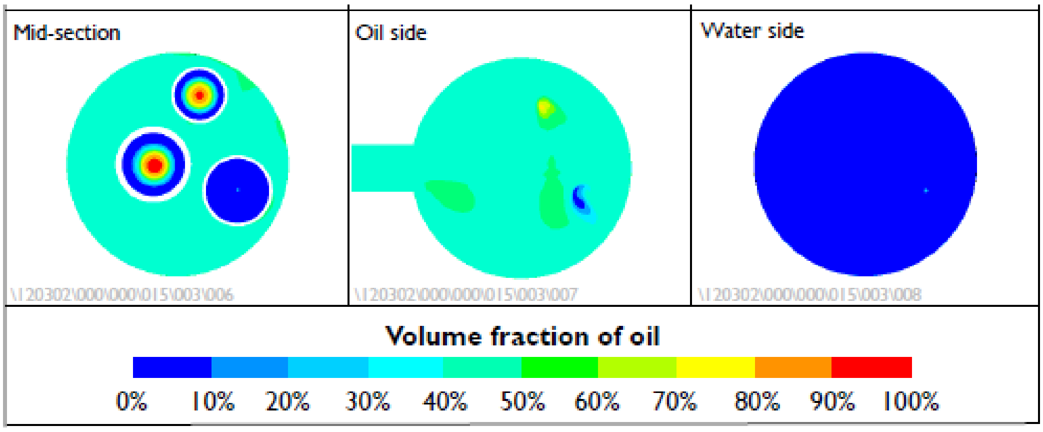

Figure 8: Contour Plot at Selected Sections through Wx-4

Table 4 shows the summary results obtained from the CFD studies of Cases A, B and C.

Table 4: Comparison of calculated and measured Data

CASE	**Experimental Pressures (barg)**		**Measured Conc. of oil (ppm)**	***CFD Predicted Pressures (barg)***		***CFD Predicted Conc. of Oil (ppm)***
	Inlet	**Water Offtake**		**Inlet**	**Water Offtake**	
A	59.9	58.8	590	58.9	58	5307
B	62.4	59.9	977	61.0	59.9	3155
C	60.8	58.2	804	58.7	57.8	85762

The pressures predicted at the inlet and water offtake were under-predicted by STAR-CCM+, as can be seen in Table 4. It was thought the discrepancy may partly be due to uncertainties relating to the neighbouring pipework of the experimental work such as:

- The probe locations were unknown so averaged pressures were reported across the entire cross-section of the pipe at each opening
- The incoming pipeline incorporated a choke valve upstream of the pressure probe, which was not included in the CFD model, but which may have had an effect
- The disagreement in predicting pressures may also be as a result of using standard k-ε turbulence model for this study. It is thought more accurate prediction could be achieved using the anisotropic Reynolds Stress Turbulence modelling approach

Later a second study, was carried out using Eulerian multiphase model along with Reynolds stress turbulence model in Fluents and all the predicted pressures and pressure drops were in very good agreement with the experimental data in all cases, demonstrating the limitations of using the standard k-ε model compared to the recommended best practice guidelines in QNET. A detail analysis and findings of this work will be presented in a second paper, to be published next year.

The oil carry over at the water offtake was over predicted by STAR-CCM+ v7.02:

- Case A 5307ppm predicted against 590ppm measured
- Case B 3155ppm predicted against 977ppm measured
- Case C 85672ppm predicted against 804ppm measured

The initial results being far off, the obvious thing to check was the specified oil droplet size as a large part of the problem was the unknown droplet size because this was never measured during the experiments. After carrying out a few runs with different oil droplets size the following graphs was produced.

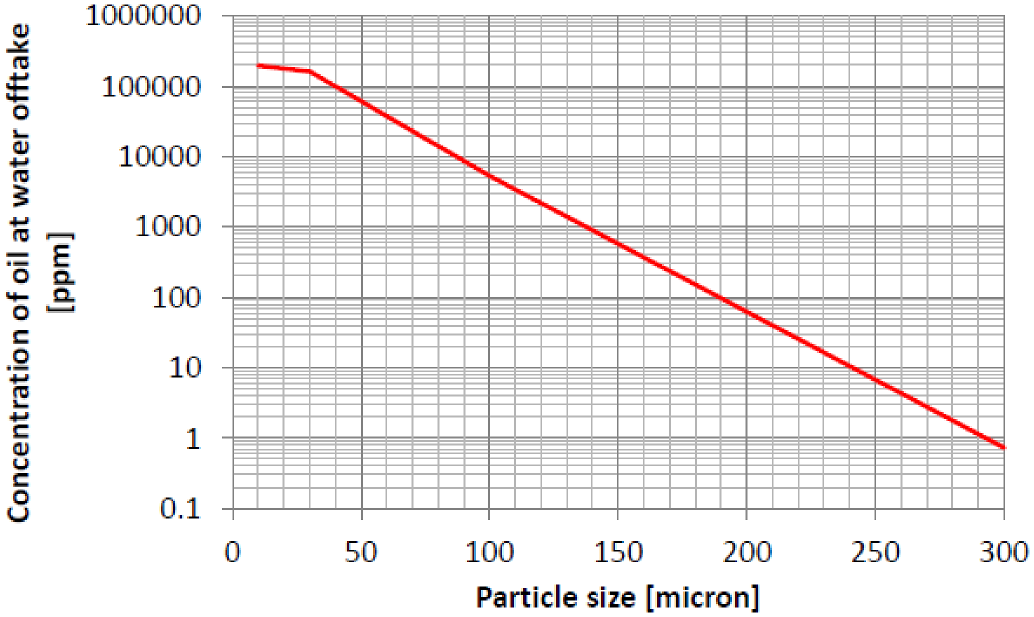

Figure 9: Variation of Oil Carry Over at Water Offtake with Particle Size

It is interesting to note that the quantity of oil carried over to the water side of the separator was very sensitive to the size of the oil droplets specified, see Figure 9.

For example in Case A:

- As the droplet size falls from 10 to 30 microns, the concentration of oil at the water offtake approaches 197,000ppm.
- As the droplet size increases to 300microns, the concentration falls to around 1ppm.
- Therefore, a droplet size of around 150microns would correlate to an oil carry-over of 500ppm, as measured. This also highlighted the importance of measuring the oil droplet size entering the separator to aid the performance prediction using CFD.

For Case C the oil carry over predicted was particularly high. This is because for three-phase scenarios it is known that oil droplets adhere to gas bubbles which promote subsequent separation producing cleaner water phase – this mechanism is known as gas floatation. Modelling this phenomenon remains beyond the current state-of-the-art for CFD and an area requiring further development.

5 CONCLUSION

Although, CFD can capture important aspects of the flow, with respect to the bulk flow and related pressure drops, the use of correct models is critical together with flow aligned mesh. STAR-CCM+ has now implemented a new feature that allows a body-fitted hexahedral mesh to be constructed entirely within the STAR-CCM+ environment.

This study highlighted that the standard k-ε does not accurately capture the anisotropy of swirling flows and thus, under-predicts the pressure drops. Therefore, the use of Reynolds Stress Turbulence model is necessary for strong swirling flows. The trade-off, however, is the significant computation overhead required by RSTM to reach convergence as it involves solving seven additional equations as compared to standard k-ε.

With regards to the system performance prediction for the specified bubble size of 100microns, CFD over-predicted the carryover of oil but there was uncertainty regarding the bubble size as it was not measured. The sensitivity on performance due to bubble size was only identified as a result of this study. Having said that, the accurate prediction of bubble size still remains a challenge, both experimentally and for CFD, and it's not ideal to prescribe droplet size. The tuning of the droplets break-up and coalescence models in CFD to predict the droplet size or distribution at the inlet is an area for further work.

This work also demonstrated that gas flotation can still not be modelled using CFD and more research and development is required in this area by the CFD companies and experts.

The above concludes part one of the conducted CFD study. Some further CFD work on Wx technology will be presented in a second paper due to be published early 2014. As a result of the above work and that carried out in second part, Caltec Limited and Abercus Limited have been able to contribute in the development of CD-Adapco's CFD software – STAR-CCM+ by suggesting the implementation of Reynolds Stress Turbulence model with the Eulerian multiphase framework, which is due to be made available in STAR-CCM +v8.06.

7 REFERENCES

[1] MM (Sacha) Sarshar, Dr N A Beg, The Applications and Performance of a Novel Compact Separator in the Oil and Gas Industry, UK 2nd GCC – EU Advanced Oil and Gas Technology Conference, Abu Dhabi, UAE, May 21 - 22 2001

[2] L Svarsky, M T Thew, Kluwear, Hydrocyclones Analysis and Applications, Academic Publishers.

[3] P. Wood, BHRA, 3rd International Conference on Hydrocyclones – Oxford, The Fluid Engineering Centre, 30th Sep – 2nd Oct 1987.

[4] Dr Carl Wordsworth and Dr Najam Beg, Caltec Limited, In-Line Water Extraction from Crude Oil using Compact Separation Technology, SPE 147163-PP, SPE ATCE Denver 2011

[5] Dr Carl Wordsworth (Caltec), Dr Najam Beg (Caltec), The Use of Compact Separation Technology for Debottlenecking Separation Systems, Tekna Separation Technology 2012, Stavenger 2012

[6] ERCOFTAC QNET Application Challenge AC£-03 http://qnet-ercoftac.cfms.org.uk/w/index.php/AC_3-03

[7] Ali Pourahmadi Laleh and William Y.Svrcek, University of Calgary, and Wayne D. Monnery, Chem-Pet Process Tech Limited, Computation Fluid Dynamics-Based Study of an Oilfield Separator – Part ii: An Optimum Design, Oil and Gas Facilities, Vol. 2 No. 1, February 2013

[8] M. Abdulkadir and V. Hernandez-Perez, The Effect of Mixture Velocity and Droplet Diameter on Oil-water Separator using Computation Fluid Dynamics (CFD), World Academy of Science, Engineering and Technology 37, 2010

[9] Versteeg, H.K. and Malalasekera, W., An Introduction to Computation Fluid Dynamics: the Finite Volume Method. 2nd ed. Pearson Educational Limited, 2007.

[10] STAR-CCM+ help manual.