Sciforce

Journal of Artificial Intelligence and Machine Learning

Journal homepage: www.sciforce.org
ISSN : 2995-2336

Open Access Research Article

CFD Analysis of Artificial Lift Down hole tool Separator

Ramamurthy Narasimhan*

New product Engineering Manager, ChampionX Inc.USA

Executive Summary

This white paper evaluates the capability of commercially available Computational Fluid Dynamics (CFD) software to predict the performanceof an artificial lift downhole sand and gas separator. Theneed for a rigorous numerical andparametricanalysis originated fromtheproductmanagement team's objective to strengthen the design basis for a market-leading separator. Existing empirical correlations have limitations, and the theoretical investigation originally envisioned was not sufficient to represent the complex multiphase flow physics inside the tool. Therefore, the product team selected CFD as a practical method to simulate separation behavior, infer important parameter relationships, and support future optimization.

A second business objective of this CFD analysis is to provide a stronger technical basis for communicating separator performance to customers. Historically, the product team and sales organization have found it difficult to explain the behavior of complex downhole separation physics using only empirical evidence. CFD simulation provides a visual and quantitative method to demonstrate how the separator functions and to highlight the technical differentiation of the product. Once the numerical model is validated through physical testing, it can reduce dependence on repeated prototype testing, which is costly and time-consuming. A validated multiphase model can also support sensitivity studies, parametric optimization, and future product development. This white paper therefore establishes a virtual test-bench methodology for evaluating flow behavior, visualizing separation mechanisms, and estimating the performance of an artificial lift downhole separator.

Key words

ComputationalFluidDynamics(CFD),artificial lift, downholetool,separator, Hybrid-Xseparator, sandseparator,downholegasseparator,multiphasemodel,Eulerian-Eulerian model, Eulerian-granular model.

Introduction/Background

Crude oil produced from deepwells commonly contains naturally occurring gas, sand, scale, corrosive liquids, and other solid contaminants.These contaminants reduce pumping efficiency, increase maintenance requirements, and contribute to premature failuresof artificial lift equipment. To improve pumping-system reliability, downhole oil, gas, and solids separators are used upstream of the pump. The SPIRIT Downhole Gas Separator™ is a patented downhole tool installed between the packer and rod pump to help prevent gas and solids from entering the pump and reducing pump efficiency.

The SPIRIT Hybrid-X sand, gas, and solids separatorusescyclonicmotionandinternal baffling to promote separation. As the production-fluid emulsion passes through the spiral section and baffles, the mixture undergoes changes in momentum, residence time, phase distribution, and buoyancy-driven separation. Separator performance depends on several geometricparametersand well operatingconditions. Because these parameters interact with one another, physical testing across the full operating envelope is difficult, time-consuming, and expensive.

Product developers therefore require a strong analytical tool that can represent the complexmultiphase flowphysics, support visualization, and provide a basis for future validation. Thiswhitepaperdescribes the CFD modeling methodology, key assumptions, and simulation results used to evaluate downhole sand and gas separation performance.

Objective

The objective of this study is to perform a multiphase CFD analysis of the 4.5-inch Hybrid-X separator to achieve the following goals:

A. Develop and establish a CFD methodology to visualize the flow of crude oil, methane gas, and sand particles within the separator.

B. Estimate separator efficiency for removal of gas and sand from the production-fluid emulsion.

Received date: December 05, 2023 **Accepted date:** December 19 2023; **Published date:** December 23, 2023

***Corresponding Author:** Ramamurthy Narasimhan, New product Engineering Manager, ChampionX Inc.USA

Copyright: © 2023 Ramamurthy Narasimhan. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

Objective

The objective of this study is to perform a multiphase CFD analysis of the 4.5-inch Hybrid-X separator to achieve the following goals:

A. Develop and establish a CFD methodology to visualize the flow of crude oil, methane gas, and sand particles within the separator.

B. Estimate separator efficiency for removal of gas and sand from the production-fluid emulsion.

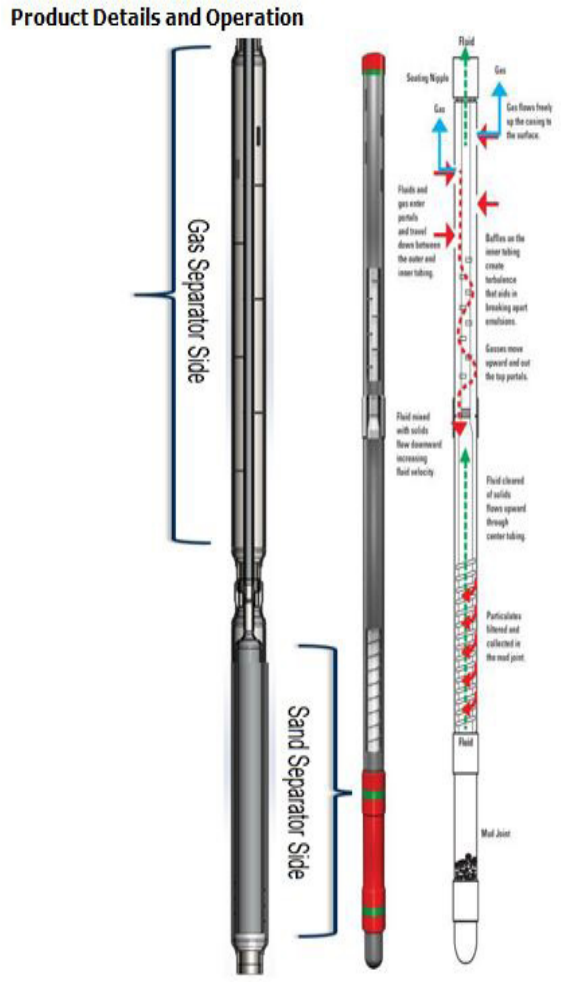


Figure1:Downhole Tool Separator

Production fluids enterthewellbore as an emulsionof crude oil,gas, and sand or other solids. The downhole tool separates desirable production fluid from unwanted gas and solid contaminants before the fluid enters the rod pump.

Figure2: Gas Separators Section and Baffles

The separator consists of a gas-separation section at the top and a sand-separation section at the bottom, as shown in Figure 1. The emulsion enters the gas separator through the upper section, as shown in Figure 2. Internal baffles agitate the emulsion, reduce incoming-flow momentum, and help break gas-liquid emulsions. Separated gas escapes through the annulus and rises toward the surface. The current separator model includes six baffles that promote gas separation and provide an upward path for gas to exit through the slots.

The remaining liquid and entrained sand move downward toward the spiral separator. In this region, the flow passes through a spiral passage formed by a thick wire wound around the internal pipe, as shown in Figure 3.

The spiral passage imparts rotational motion to the mixture and promotes separation between oil and sand. The separated oil enters the central pipe through the lower opening shown in Figure 3 and is then lifted by the rod pump. Separated sand is driven downward and collected in the mud joint.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

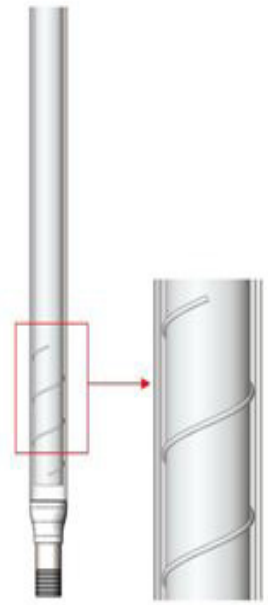


Figure3: S and Separator Section and Spiral

CFD analysis details

The CFD study was performed on the 4.5-inch Hybrid-X separator to establish a repeatable multiphase analysis approach and to estimate gas and sand separation efficiency.

a) Develop and Establish a Methodology to visualize the flow of crude oil, methane gas & sand particles (emulsion)

b) Estimate the separator efficiency of Down hole Separator to separate the emulsion.

Figure4: Design Modeller Geometry

This section summarizes theCFDanalysis workflow. ANSYSCFX was used for flow visualization and multiphase-flow analysis. The major analysis steps included geometry preparation, fluid-domain extraction, mesh generation, boundary-condition definition, solver setup, and post-processing of phase volume fractions and flow fields.

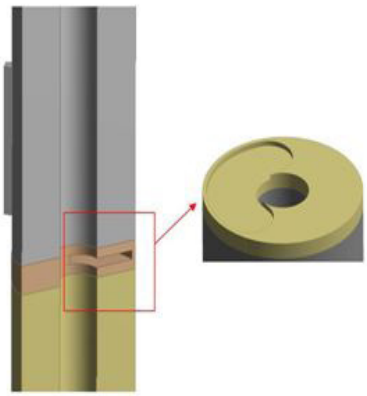


Figure5: Baffles Modelling

a) ANSYS Design Modeler: ANSYS Design Modeler was used to import the neutral CAD geometry into the ANSYS environment and extract the fluid domain. Boolean operations were used to isolate the flow volume from the solid geometry. Figure 5 shows the resulting fluid domain after extraction.

b) ANSYS Workbench Mesher: ANSYS Workbench Mesher was used for grid generation and geometry discretization. Mesh density was refined in the annular flow passages between the inner and outer tubes to capture local flow currents and particle movement. Inflation layers were added near walls to represent boundary-layer effects and improve viscous-force approximation. Figure 6 shows the meshed geometry.

c)ANSYS CFX: ANSYS CFX was used as the solver to compute the multiphase flow field in the discretized fluid domain.

Methodology and Analysis-S and / Solid Separator

Oil is the primary product of the production process, while sand and other contaminants reduce downhole pump efficiency and reliability when not separated effectively. The SPIRIT sand/solids separator uses patented centrifugal motion to capture sand and solids before theyinterferewithrod-pump operation. Inthesand-separatorsimulation, the primaryphase was represented by mass-averaged crude oil and methane gas, while the secondary phase was represented by sand particles. An Eulerian-granular multiphase model was used to simulate the liquid-solid separation behavior.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

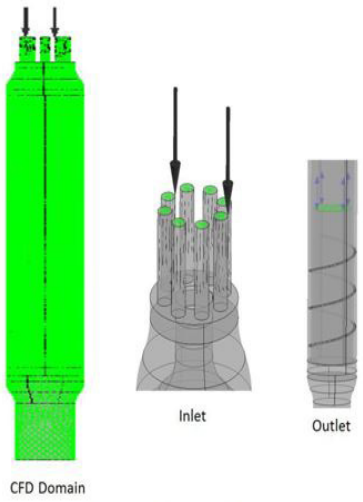


Figure7: Boundary Conditions

5.1. Boundary Conditions: Appropriate initial and boundary conditions are required for the CFD solver to solve the continuity and Navier-Stokes equations and achieve convergence. Boundary locations are shown in Figure 7. The operating conditions used for the sand-separator analysis are summarized below.

a) The type of fluid material used–Crudeoil, Methanegas, generics and particles

b) Volume fraction of fluid satin let–Crudeoil(70to90%)and s and particles (up to 10%).

c) Mass flow rate of emulsion–100bbl/day

d) Density of the fluids.Crudeoil–862Kg/m^3ands and particles–2500Kg/m^3.

e) Mean diameter of thes and particles– 1mm

f) Viscosity of crude oil–7.5Centi Poise.

Flow Modeling in CFX

The fluid domain was generated using DesignModeler. The Eulerian-granular multiphase model was used for the sand-separator simulation. The setup included an inhomogeneous mixture model, a fluid-dependent turbulence treatment appropriate for the viscous laminar regime, buoyancy based on density differences, the Gidaspow momentum-transfer model, and coupled volume-fraction control.

These selections enabled the model to represent relative motion between the crude-oil mixture and dispersed sand particles while accounting for density-driven separation and interphase momentum transfer.

Methodology and Analysis-Gas Separator

In the gas-separator simulation, crude oil was modeled as the primary phase and methane gas as the secondary phase. An Eulerian-Eulerian multiphase model was used to represent gas-liquid separation. The mean gas-bubble diameter is an important input to the model because it influences interphase drag, buoyancy response, and separation behavior.

Boundary conditions:

a) The type of fluid material used–Crude oil, Methanegas.

b)Volume fraction of fluid satin let–Crudeoil(70to90%) Methanegas(upto20%).

c) Massflowrateofemulsion–100bbl/day

d) Densityofthefluids.Crudeoil–862Kg/m^3,methanegas–0.656Kg/m^3.

e) Mean diameter of thes and particles– 1mm

f) Viscosity of crude oil–7.5Centipoise, methanegas–0.0103 Centipoise.

Flow Modeling in CFX

Thegas-separator fluiddomainwas preparedusingtheEulerian-Eulerian multiphasemodeland the boundary conditions listed above.

The gas-separator setup included an inhomogeneous mixture model, a fluid-dependent turbulence model appropriate for viscous laminar flow, buoyancy based on density differences, the Ishii-Zuber momentum-transfer model, and coupled volume-fraction control.

Secondary Phase Diameter Estimation

The gas-phase diameter is required as an input to the CFD model. A practical guideline for estimating this value is the critical Weber number. The Weber number is a dimensionlessratioofdeforming inertialforcesto stabilizingcohesive forces. It is commonly used to characterize droplet or bubble breakup behavior in multiphase flows and emulsions.

The critical Weber number represents the condition atwhichadropletor bubble beginstobreakup.Forthis separator analysis, the critical Weber number for the gas-oil interface was identified as 13.

Reference	We_c	Exp. Fac.	Notes
Lenard (1904)	5.8	FF	Calc. for water – u_c = 8 m/s; d_c = 5.5 mm
Merrington and Richardson (1947)	15.4–29.8	FF	Calc. by Hinze (1948)
Lane (1951)	10.8	VWT	Calc. from $u_c^2 d$ = const. for water
Volynskii (1948)	11–15.8	HWT	Average value 14
Buhman (1954)	2.2–3.6	HWT	For water 2.6–3.5; (3.9–10.4)
Hinze (1955)	13	HWT	For gas oil
Krzeczkowski (1980)	11–38	HWT	For water 11
Nichiporenko et al. (1982)	10.9–17.7		For liquid metal; 17.7 for bag breakup
Wierzba (1985)	14		For water
Ishiki (1959)	9.26–29		For water 11.1–14.6 = $f(d)$
Haas (1964)	11.2	SO	For mercury
Naida et al. (1973)	8.4–12.1	SO	Average value for tin 10.9
Yoshida (1985)	10–48	SO	For water 10–31 = $f(d)$
Hanson et al. (1963)	7.2–47.6	HST	For water 7.2–14.3 = $f(d)$
Simpkins (1971)	13	HST	For water
Gelfand et al. (1972)	12–16	HST	For liquid nitrogen
Simpkins and Bales (1972)	14	HST	For water
Gelfand et al. (1973)	10–50	HST	For water 10
Gelfand et al. (1974)	10	HST	For water and kerosene
Reichman and Temkin (1974)	7	HST	Calc. from $u_c^2 d$ = const. for water
Korsunov and Tishin (1971)	15–32	CN	For transformer oil
Lopatiev (1975)	14.6–99.6	VS	For low viscosity liquids 14.6–21
Cavery and Grany (1979)	20–30	CN	For Al/Al_2O_3 agglomerates
Borisov et al. (1986)	40–60	HST	For water and kerosene

$We_{critical} = 13$

Table1: Weber Number Reference

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

CALCULATION SHEET		
Jet Diameter, d	0.01270	m
Column diametr, D	0.073152	m
Jet / column area ratio,b	0.030141	
Crude oil volume fraction	0.80	
Methane gas volume fraction	0.20	
Gas/liquid volumetric flow ratio, α	0.250	
Mass flow rate of emulsion	0.662400	m^3/h
Velocity of jet, u	1.453	m/s
Mixing zone length, L	0.1016	m
Density of crude oil, ρ	862.0	kg/m^3
Average energy dissipation rate, Ɛ	369.11	
Critical weber number	13	
Surface tension of crude oil, σ	30.18	Dynes
Surface tension of crude oil, σ	0.000308	1dyn = 10^-5 N
Maximum stable diameter of completely circular bubble	0.5847	mm

$$\epsilon = \frac{\rho u^{*3}}{2L}[b - 2b^2 - b^3(1+\alpha)^2 + 2b^3(1+\alpha)]$$

$$d_m = \left[\frac{Wecri * \sigma}{2}\right]^{3/5} * \rho^{-1/5} * \epsilon^{-2/5}$$

Figure8: Mean Diameter Calculation

The Weber-number approach described above was used to estimate the mean gas-bubble diameter. The calculation is shown in Figure 8.

The Weber-number approach described above was used to estimate the mean gas-bubble diameter. The calculation is shown in Figure 8.

Inferenceof CFD results:

Results and Discussions and separator

a) Sand volume fraction plot along the length of the sand separator is shown for full domain & for center cut section. As shown in the Figure 9.

b) Figure 9 shows the sand volume fraction across the full domain. The highest sand accumulation occurs in the mud-joint region. Traces of sand are also visible along the sand separator, indicating a gradual change in phase concentration along the tool length.

c) Totals and volume fraction considered at the in let was10%of total mass flowrate.

The corresponding sand mass-flow input was 0.0141 kg/s, based on prior test data.

d) Morethan 80% ofthetotal sand VF lies at the bottom ofmud joint. As shown in the Figure 9.

e) The remaining sand volume fraction is distributed through the spiral region and, to a smaller extent, toward the outlet.

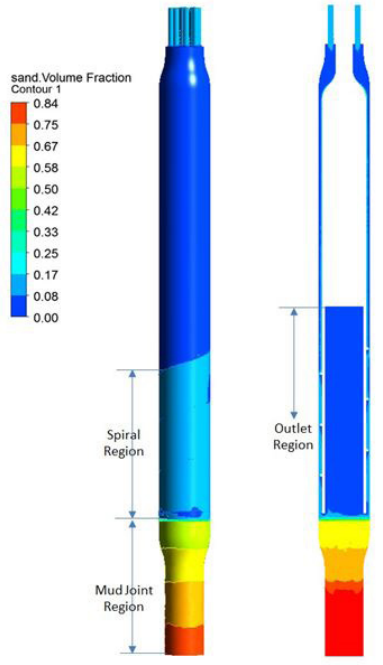


Figure 9:CFD Results of S and Separation

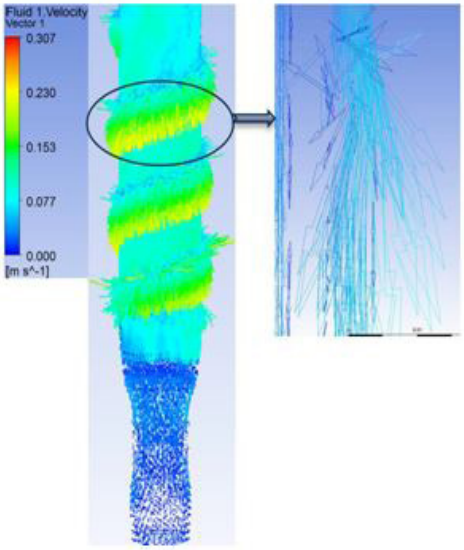


Figure10: Vector Plot S and Separation

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

Fluid flow Vector plots

The spiral section forces the fluid to circulate around the pipe instead of moving directly downward. Figure 10 shows the direction of fluid-particle motion. As the fluid passes through the spiral passage, velocity decreases and residence time increases, which supports phase separation.

Figure 10 indicates that fluid velocity reaches approximately 0.3 m/s at the beginning of each spiral and decreases to approximately 0.07 m/s after passing over the spiral. This reduction in velocity, combined with the rotational flow path, promotes bulk sand separation.

Substantial local backflow and recirculation are observed near the spiral region. These recirculation zones help lighter crude oil move upward while sand particles migrate downward toward the mud joint. Additional study is recommended to determine the optimum level of turbulence because excessive agitation may generate froth and reduce separation effectiveness.

Sand Separator Efficiency:

Sand-separation performance can be evaluatedby the amount of sandc arried over to the liquid outlet. Key factors affecting sand-separation efficiency include:

a) Amount of sand entrainment in liqui doutlet

b) Length of the separator

c) Sand-Liquidratio

d) Angle of the separat orde sign and sizing

e) Location of Baffles.

Separator efficiency was calculated from the amount of sandc arried over to the liquid outlet. The sand flow rate at the liquid outlet was evaluated in CFD-Post using the function calculator. The calculated results are summarized below.

a) Mass flow average values for sand volume fraction atliqui doutlet is calculated to be of 9.96%

b) 90.04% which is aremaining portionis filled with oil at liqui doutlet

c) Thus separator efficiency will be 90.04%.

Result sand Discussion Gas Separator

Gas separation can be visualizedusing flowtraces and gas-volume-fraction plots. The CFD solver distinguishes between oil and gas phases by solving the appropriate governing equations for each phase.

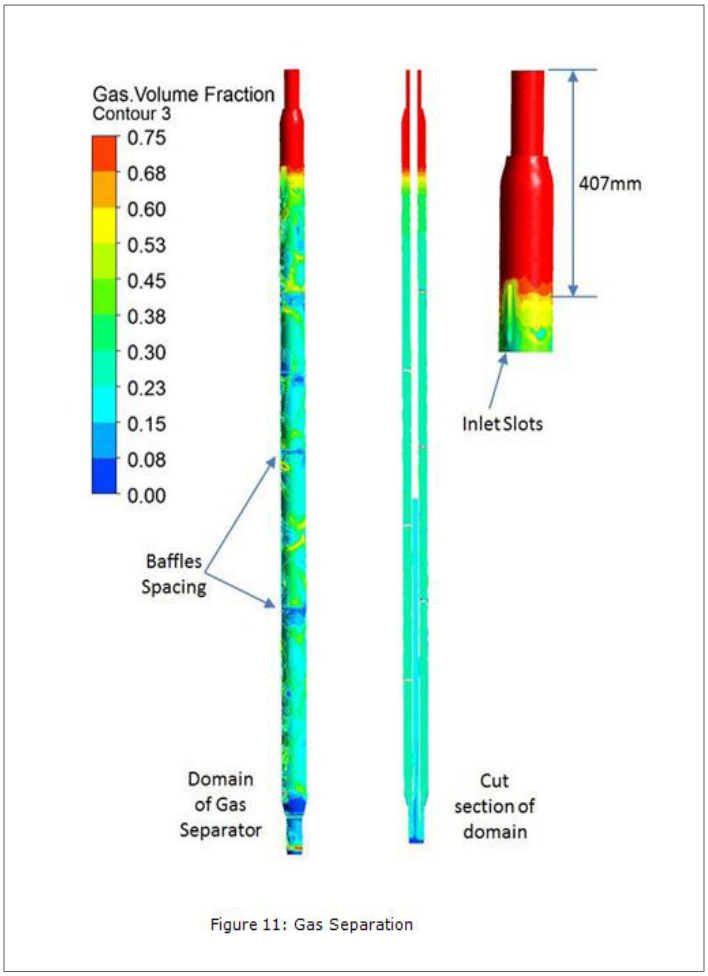


Figure 11: Gas Separation

Gas Separator Results: Gas-separation results are shown in Figure 11. The following observations can be inferred from the simulation results.

a) Gas volume fraction plot along the length of the Gas separator isshown for full domain & for a cut section

b) Methane gas begins to flow along with fluid from inlet slots shown in the Figure11

c) Figure 11 shows the gas volume fraction across the fluid domain. The highest gas concentration occurs near the top region because buoyancy and gravity effects are represented in the model.

d) The simulation confirms that buoyancy is a primary mechanism for gas separation. Traces of gas remain along the separator length, indicating that separation occurs progressively throughout the domain.

e) Up to 75% of the total VF of Gas lies at the top of the separator, a length of 407 mm from top.

f) Remaining gas volumelies up tothe length spiral region and somepercentage is traced in the liquid outlet.

g) The percentage of gas entrained in the oil determines the efficiency of the gas separator.

Flow visualization at baffles: The simulation shows that gas separation occurs near the baffles. Baffle count, shape, size, and placement have a significant influence on separation performance. The following results describe the role of the baffles in promoting gas separation.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

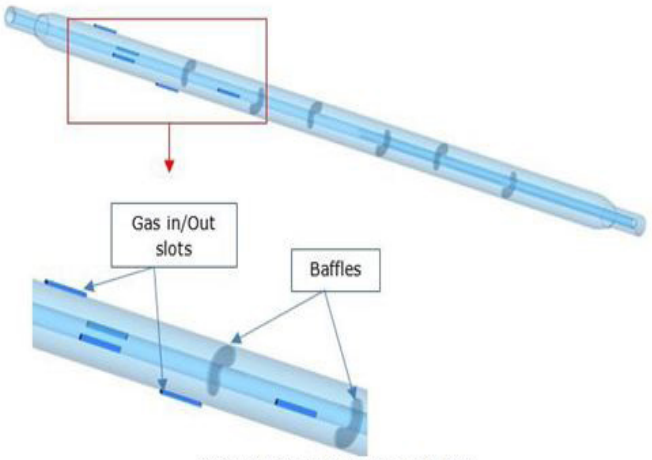


Figure12: Gas Separation at Baffles

h)Baffles are arranged in alternative angles as shown in Figure13.

i) The alternate baffle arrangement reduces fluid momentum, redirects the incoming stream, and increases flow agitation.

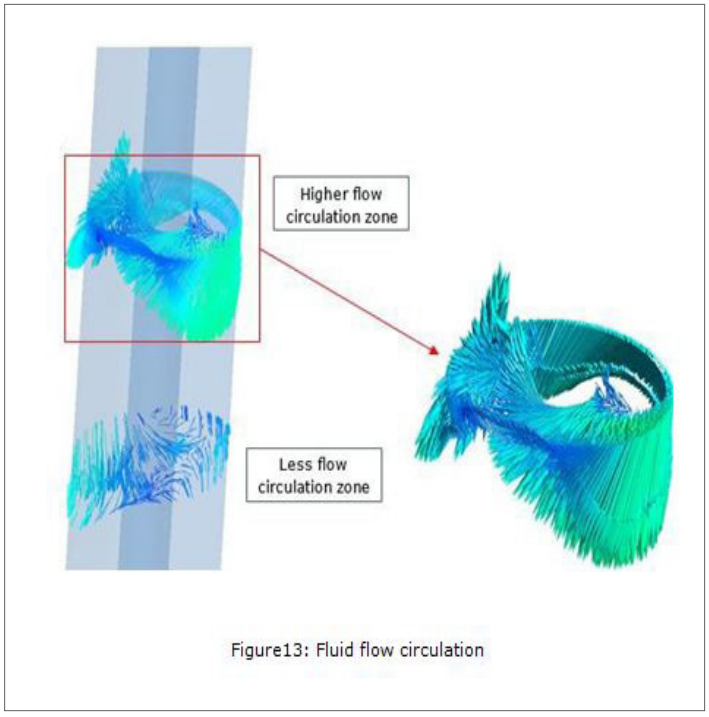


Figure13: Fluid flow circulation

j) The baffles change the direction of the inlet stream and assist in separating bulk liquid from gas.

k) The recirculation visible in Figure 12 adds controlled agitation and supports bulk gas-liquid separation.

Benefits of momentum reduction:

a) The velocity contours in Figure 14 show momentum reduction across the baffle region, as reflected by lower velocity values in the contour legend.

b) Momentum change is one of the key drivers of separation, and the CFD results provide a clear visualization of the flow variation.

c) CFD results can also be used to evaluate the optimum number of baffles needed to improve separation efficiency.

d) The kinetic energy of the gas-liquid stream dissipates across the baffle, and the flow is redistributed downstream. Figure 15 summarizes the velocity regions after the baffle.

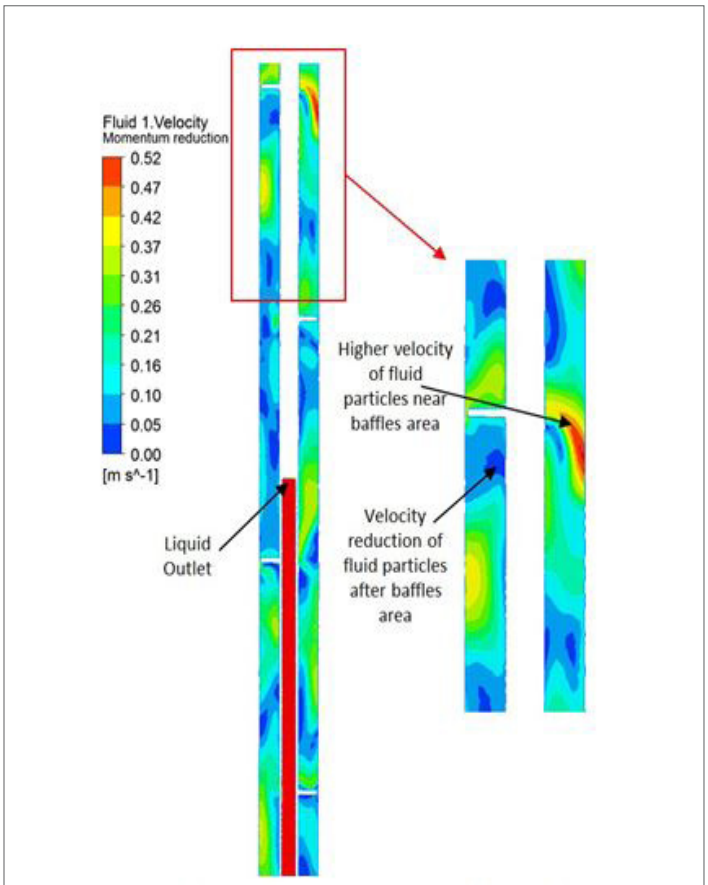


Figure14:Velocity Contour Gas Separation

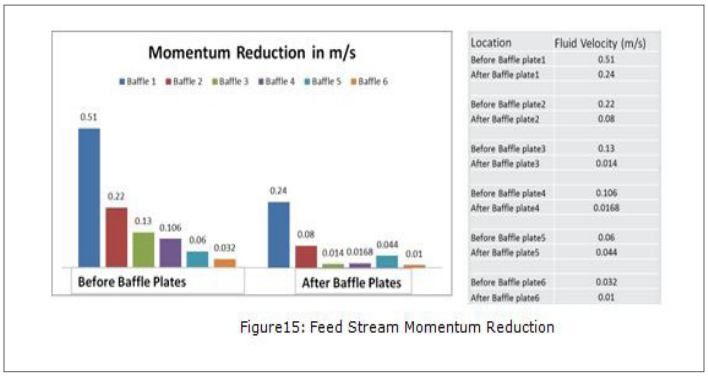


Figure15: Feed Stream Momentum Reduction

Gas Separator Efficiency

Gas-separation performance is affectedby several factors, including:

a) Amount of gas entrainment in liquid outlet

b) Length of the separator

c) Gas-Liquidratio

d) Angle of the separat ordesign and sizing

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

e) Location of Baffles.

Gas-separator efficiency was evaluated from the amount of gas carried over to the liquid outlet. The gas flow rate at the liquid outlet was calculated in CFD-Post using the function calculator. The results are summarized below.

a) Mass flow average values for gas volume fractionat liquid outle tis calculated to be of17.31%

b) 82.69% which is are maining portion is filled with oil at liqui doutlet

c) Thus separator efficiency will be 82.69%.

Conclusions

Amultiphase CFD simulationwas completedforboth thes and separator and gass eparator of the 4.5-inch Hybrid-X separator. The study established a numerical methodology for analyzing the separation of crude oil from me thane gas andsand particles. Based on the CFD results, the following conclusions can be drawn:

The multiphase CFD methodology can interpret the volume fraction of individual phases, including sand accumulation in the mud joint and gas venting through outlet slots. Gas and sand quantities can be estimated virtually from the simulation results.

The centrifugal motion generated in the spiral region of the sand separator supports sand separation and can be quantified for future design optimization.

The number of baffles and the ratio between the inner and outer tubes play important roles in gas separation. These parameters can be further optimized using CFD results.

Baffle-plate arrangements help break froth and promote bulk gas-liquid separation. CFD analysis provides a practical method to visualize and understand these mechanisms.

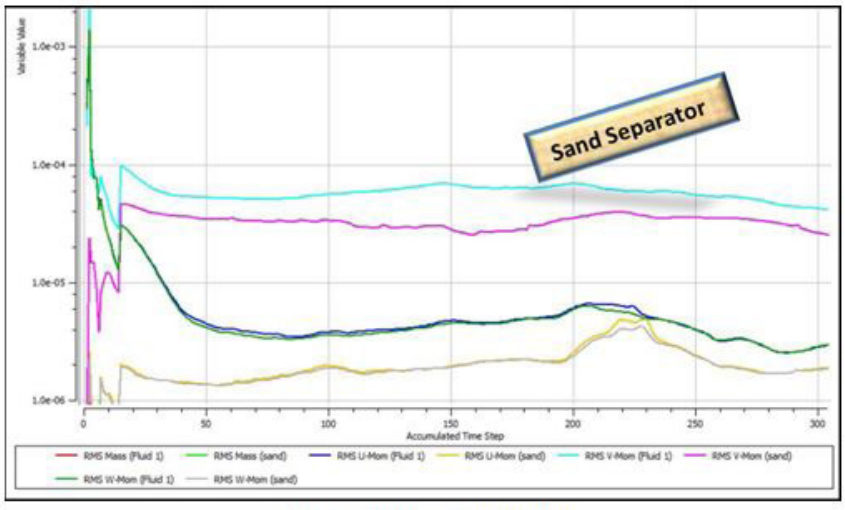
Separator performance is influenced by fluid composition, well conditions, tool orientation, and operating angle. CFD provides a strong analytical method to evaluate these effects and predict performance trends.

From a business perspective, CFD results provide engineering evidence that can help explain separator performance to customers, especially because downhole behavior is difficult to observe directly during operation.

Model validation remains a required next step because CFD is a numerical approximation. However, the qualitative simulation results provide useful insight into the separator's internal flow behavior and support future physical validation and design optimization.

APPENDIX A

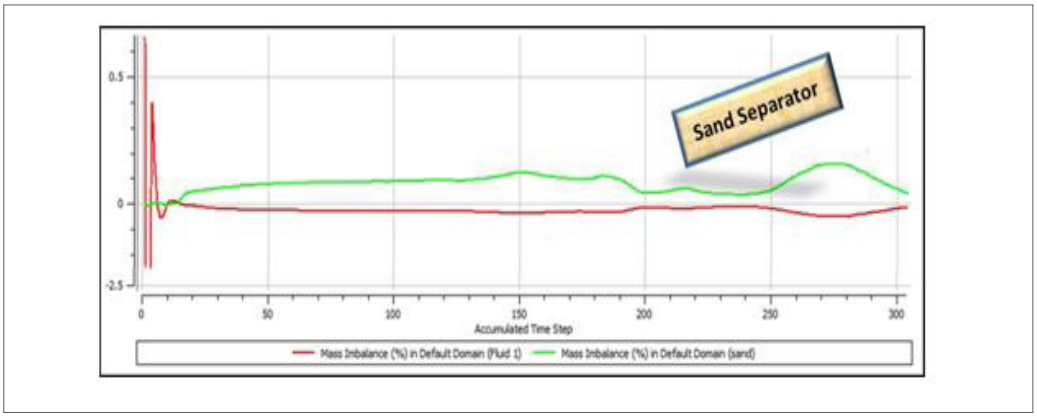
Convergence plots show whether the numerical solution has reached the specified convergence criteria. Residuals measurehowaccurately the governing equations have been solved. Lower residuals indicate a more accurate and stable numerical solution.



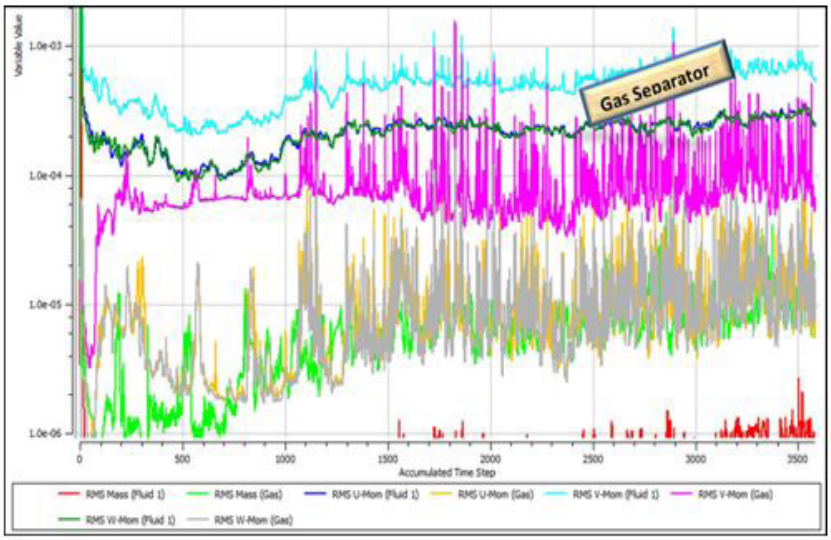
Plots for mass and momentum

Mass imbalance for the liquid and sand phases was monitored using a residual monitor. The acceptable mass imbalance criterion was less than 1%.

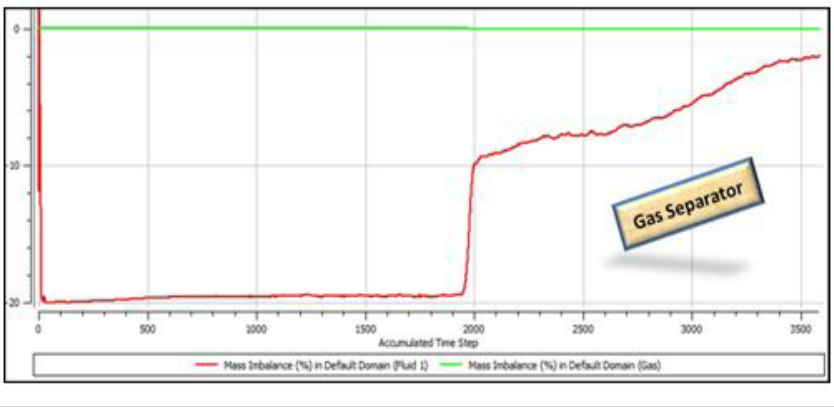
Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305



Mass and momentum residuals were reviewed to verify numerical stability and convergence of the sand-separator model.



Plots for mass and momentum



Mass imbalance for the liquid and gas phases was monitored using a residual monitor. The acceptable mass imbalance criterion was less than 1%.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305

APPENDIX B

Modeling the Flow Physic and Operation of the Separator

Sand separation was modeled as a multifluid, immiscible-flow problem. The numerical methodologyforthiscomplexcase includes severalstages thatareassigned within the commercial CFD code, with key user-defined inputs based on research literature and fluid-mechanics calculations.

The following sections describe the primary stages and fluid-dynamic considerations used in the analysis.

Selection of multi phase model

Multi phase flowrefers to the simultaneous flow of materials in different phases, such asgas, liquid, andsolid. It may also involve materials with different properties that flow together while remaining macroscopically distinct.

The primary phase is treated as the continuous fluid,while the secondary solid phase is modeled as dispersed particles within the continuous phase. A representative particle diameter must be assigned to the secondary phase to calculate interphase interaction, including drag.

If the secondaryphase has aparticle-size distribution, each representative particle diameter can be modeled as aseparatephase.

Flow Regimes

Flow observations and Reynolds-number estimates indicate that the separator operates in a viscous flow regime. The selected multiphase model is therefore appropriate for representing the dominant separation mechanisms.

Elastic Regime	Plastic Regime	Viscous Regime.
Stagnant	Slow flow	Rapid flow
Stress is strain dependent	Strain rate independent	Strain rate dependent
Elasticity	Soil mechanics	Kinetic theory

Governing Equations

The momentum equation represents the balance of forces acting on a fluid or particle phase. Although the equation may be described using a Lagrangian approach, it can be transformed into an Eulerian frame for multiphase CFD modeling.

$$\frac{\partial}{\partial t}(\alpha_s \rho_s) + \nabla \cdot (\alpha_s \rho_s \vec{u}_s) = \dot{m}_{fs}$$

Interfacial forces

Define primary fluid and secondary fluid

The momentum forces acting on each phase include gravitational, buoyancy, drag, and interphase interaction forces. These forces create relative motion between the primary and secondary phases. Gravity is a dominant contributor to separation because heavier sand particles tend to settle downward, while lighter oil and gas phases tend to move up ward.

Force sacting on a solid particle in a multi phase flow:

Particle Particle Interaction Force (Fp) : The particle-particle interaction force represents momentum exchange caused by collisions or contact between particles in a gas-liquid-solid flow.

$$F_P = \frac{\pi}{4} d_P^2 \frac{m_s}{A} (v_M - v_P)$$

Particle-Liquid Momentum Transfer Force (FM): This force represents the linear momentum transferred between the liquid phase and the sand-particle phase.

$$F_M = \frac{F^{\wedge}\left(\frac{qu}{2a^2\rho_P}\right)}{\left[1 + \left(\frac{\bar{\rho}_L}{2\bar{\rho}_P}\right)\right]}$$

Viscous Force (FV): The viscousforcecan be represented using Stokes' law for a sphericals and particle moving through a viscous fluid.

$$F_v = 3\pi\mu d_P (v_M - v_P)$$

Buoyancy Force (FB): Buoyancy force represents the upward force acting on a phase due to density differences in a gas-liquid-solid multiphase flow.

$$F_B = \frac{1}{6}\pi d_P^3 \rho_M g$$

Gravity Force (FG): Gravity force acts on the particle mass and promotes downward movement of the higher-density solid phase.

$$F_G = \frac{1}{6}\pi d_P^3 \rho_P g$$

Frictional Drag Force (FD): Drag forcerepresents the resistance experienced bya suspended particle moving relative to the surrounding liquid phase.

$$F_D = \alpha(v_M - v_P)$$

References

1. Computational of Multi phase Flow by Volume of Fluid and High Order Front Tracking Methods – Stephane Zaleski.

2. The study of Multi phase Flow for PetroleumProductionusing Computational Fluid Dynamics – Norhafizuddin

3. De formation and breakup of liquid drops in a gass treamat nearly critical We ber numbers by A. Wierzba, January 1990, Volume 9, Issue 1, pp 59-64

4. Multiphase flow models by www.Ansys.com

5. Performance of Internals in Three Phase Tank Separators – Dominic George Akpan.

Citation: Ramamurthy Narasimhan. (2023) CFD Analysis of Artificial Lift Down hole tool Separator. International Journal of Artificial intelligence and Machine Learning, 1(4), 1-10. https://doi.org/10.55124/jaim.v1i4.305