

Petroleum Engineering, Oil and Gas

December 06 -07, 2018 | Dubai, UAE

CFD modelling of the fluid behaviour in oil wells

Abdualmagid Algadi

Energy, Emissions and the Environment Research Group, United Kingdom

Computational Fluid Dynamics (CFD) can be used to analyse the fluid behaviour in oil wells, where the CFD allows conducting three-dimensional (3D) simulations of interpenetrating or immiscible fluids that comprise impacts of pressure, temperature and fluid mass transfer in detail. Recently, the CFD is used to more varied challenges such as sand transport, coupled mechanical modelling of failing wellbores and gas well clean-up. CFD has been utilized to model the crushed area around perforation tunnels and as well to predict good performance based on laboratory testing. Where the reservoir is involved in these models they are fully coupled and flow has enabled deep in the reservoir and in to each point along the well.

Capturing the simultaneous flow of fluids in the reservoir and in the well is fundamental, to predict performance in wells and reservoirs with complicated geometry. Further advances that assist sand failure and sand transport to be modelled are also illustrated. The case is employed for use of CFD or similar modelling processes where complicated wells or complicated reservoirs have considered. CFD modelling for the oil and gas sector providing informed analysis of the data that can assist designers and operators solve flow issues, extend life of flow lines and ensure the efficient delivery of the product.

e: Abdualmagid.Algadi@hud.ac.uk*Notes:*