

Computational Fluid Dynamics Simulation of Reservoir for Dwell Time Prediction

Authors : Nitin Dewangan, Nitin Kattula, Megha Anawat

Abstract : Hydraulic reservoir is the key component in the mobile construction vehicles; most of the off-road earth moving construction machinery requires bigger side hydraulic reservoirs. Their reservoir construction is very much non-uniform and designers used such design to utilize the space available under the vehicle. There is no way to find out the space utilization of the reservoir by oil and validity of design except virtual simulation. Computational fluid dynamics (CFD) helps to predict the reservoir space utilization by vortex mapping, path line plots and dwell time prediction to make sure the design is valid and efficient for the vehicle. The dwell time acceptance criteria for effective reservoir design is 15 seconds. The paper will describe the hydraulic reservoir simulation which is carried out using CFD tool acuSolve using automated mesh strategy. The free surface flow and moving reference mesh is used to define the oil flow level inside the reservoir. The first baseline design is not able to meet the acceptance criteria, i.e., dwell time below 15 seconds because the oil entry and exit ports were very close. CFD is used to redefine the port locations for the reservoir so that oil dwell time increases in the reservoir. CFD also proposed baffle design the effective space utilization. The final design proposed through CFD analysis is used for physical validation on the machine.

Keywords : reservoir, turbulence model, transient model, level set, free-surface flow, moving frame of reference

Conference Title : ICCFD 2020 : International Conference on Computational Fluid Dynamics

Conference Location : London, United Kingdom

Conference Dates : December 10-11, 2020