Proceedings of the Canadian Society for Mechanical Engineering International Congress 2021
CSME Congress 2021
June 27-30, 2021, Charlottetown, PE, Canada

Validated CFD Simulation for Flashing Flow in Inflow Control Devices in SAGD

Tarek H. Nigim[1], Da Zhu[2], Carlos F. Lange[1*]
[1]Department of Mechanical Engineering, University of Alberta, Edmonton, Alberta, CANADA
[2]RGL Reservoir Management INC, Calgary, Alberta, CANADA
*e-mail address: carlos.lange@ualberta.ca

ABSTRACT

Using inflow control devices (ICDs) in the thermal oil recovery wells can increase the oil production rate by more than 50% with more uniform production along the well. The main objective of this paper is to describe an industrial design tool for ICDs by means of advanced, predictive computational models of ICDs that reveal the potential occurrence of flashing through the device and accurately predict the effect on the pressure distribution. In the present work, we provide a computational model that has the ability to predict the flashing flow inside ICDs. Our validated computational fluid dynamics (CFD) model is developed on the basis of state-of-the-art commercial software (Ansys-Fluent). Our multiphase numerical model is applied to time-averaged, two-phase, adiabatic, turbulent flows to solve and predict flashing flow. The numerical model can predict the phase change between liquid and vapour phases based on mechanical effects. This can be reached by comparing the local pressure in each cell in the numerical field with the local vapour pressure. The vapour bubbles nucleate and grow when the local pressure (p) of the phase is lower than the local saturation pressure (p_v). Our current numerical methods assume monodisperse bubbles and solve the continuity and momentum equations for the mixture (liquid + vapour phase), and the volume fraction equation is solved for the secondary phase (vapour). Furthermore, the effects of different turbulent models on the computational field have been carried out. The model has been applied to a typical converging-diverging nozzle to verify its validity. Following validation, the model has applied to an existing ICD. The results of the developed model showed the multiphase flow field of a converging-diverging nozzle under flashing conditions and allowed us to map the flow patterns and behaviour inside the nozzle. The predicted flashing behaviour agrees well with data from published experimental work. Results produced by the model can be used to estimate relevant design factors, such as pressure drop and erosion potential. The advanced use of CFD allows computational flow modelling to be applied as an industrial design tool, offering significant design cost savings, accelerated production, reduced steam-to-oil ratios, and reductions in water use and emissions.

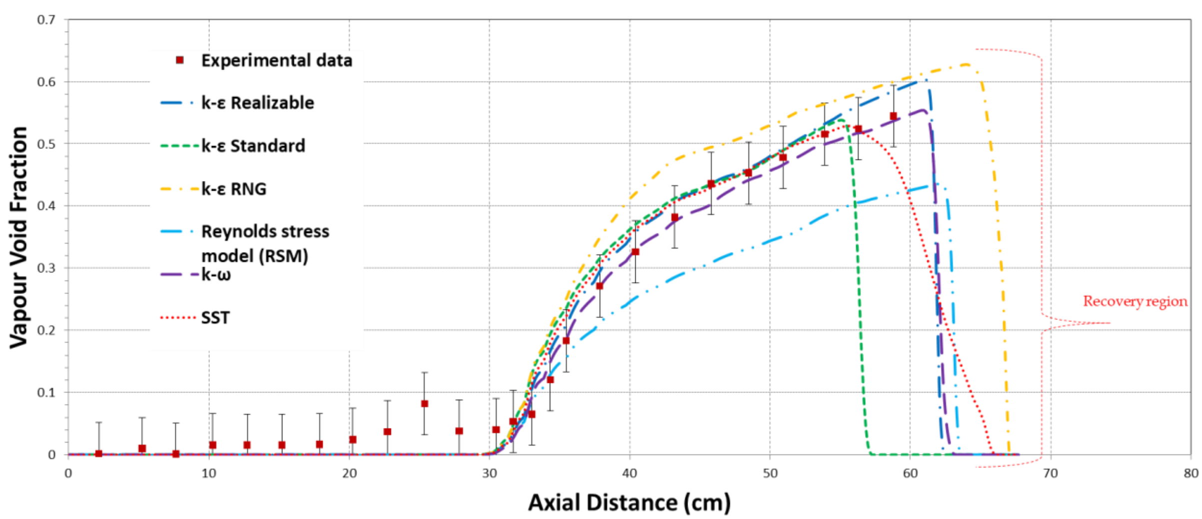

Figure 1 Comparing results of area-averaged void fraction of turbulent models on multiphase flashing flow of experimental and CFD under flashing condition