عنوان مقاله:

Computational Fluid Dynamics and Experimental Study of Two-Fluid-Spill-Return (TFSR) Atomiser(A new mixed (model of Two-Fluid and Spill-Return Atomisers) (Patent's Introduction

محل انتشار:

چهارمین همایش بین المللی توسعه فناوری در نفت، گاز، پالایش و پتروشیمی (سال: 1400)

تعداد صفحات اصل مقاله: 18

نویسندگان:

Amin Nakhaei - *MSc. Petroleum & Gas Eng., University of Salford, Manchester, UK*

Faeze Soltani, - *BSc. Chemical Eng., University of Kerman, Kerman, IRI*

خلاصه مقاله:

Spray atomisation is a common process found in different industries, where sprays under high injection pressure are widely recorded in the literature. Nonetheless, research on spray breakup mechanism under low pressure in any industry is still relatively rare. The process of liquid atomisation is largely used in the petroleum and gas industry and fields, descaling pipeline and spray and atomisation systems, Health and Safety Systems (HSE), Fire Suppression, Manufacturing, Agriculture and Steel Cutting Systems and etc. I this context, the computational fluid dynamics CFD is one of the best solutions because it can give high quality results with low cost, observing that to achieve this purpose it is necessary refinements in the models and mainly perform an extended study of the boundary conditions. Thus, in this work it was used the software CFX ۱۴ to study the atomisation process with different models and boundary conditions. Numerical model in the morphology of air chose as "Continuous Fluid" and the morphology of water chose as "Dispersed Particle Transport Fluid" for simulating the spray formation from a TFSR Atomiser device in the current study.This paper introduce the novel patent**: Two-Fluid-Spill-Return (TSFR) Atomiser is a new mixed system model of Two-Fluid and Spill-Return atomisers and discuss and recommend the utilisation of the this new atomiser model in design and analysing of varies parameters related in oil and gas industries. This is to achieve the optimum particle diameters (SMD, best performance of impact forces, particles distributions, varies range of pressure drops and etc. to provide optimum and useful ranges on the differ utilisations. The results form CFD analysis show the fluid particle diameter (SMD) about ۱۶ microns, average final normal speed ۱۷۰ [m s^-۱] and average pressure in range of ۲۰ -۲۵ Pascal based on the loading properties of CFX Simulations (Table ۱). Generally results show the increasing of spill outlet by decreasing in range of air pressure and rise in the distance from water-air spray point. In addition the stage ۴ (Tangent point) compare to other stages shows the increasing of circulation of liquid and then rising of spill outlet flow rate.

کلمات کلیدی:

TFSR, CFD, HSE, SMD, CFX