DESIGN & ANALYSIS OF SHELL & TUBE HEAT EXCHANGER FOR DIFFERENT TYPES OF BAFFLES USING CFD

M Tech Dissertation

Submitted in

partial fulfillment of the requirements

for the degree of

MASTERS OF TECHNOLOGY

In

Thermal Engineering

By

Gajipara Kaushal Manojbhai

(160303210004)

Under the supervision of

Asst. Prof. Kruten Patel

May 2018

DEPARTMENT OF MECHANICAL ENGINEERING

PARUL INSTITUTE OF ENGINEERING &TECHNOLOGY

FACULTY OF ENGINEERING &TECHNOLOGY

PARUL UNIVERSITY

P.O. Limda – 391 760, GUJARAT, INDIA

ABSTRACT

In present day shell and tube heat exchanger is the most common type heat exchanger widely used in oil refinery and other large chemical process, because it suits high pressure application. So it is very important to study the behavior of such heat exchangers under different conditions. In present study single pass segmental baffle shell and tube heat exchanger is designed with conventional correlation available along with the reference literature available. After that computational fluid dynamics(CFD) package, Fluent is utilized for modelling single pass shell and tube heat exchanger with segmental baffles. Then change in baffle geometry made and different baffles like perforated baffle, hexafoil hole baffle are tested in CFD Fluent for performance analysis. Also experimental setup is prepared for segmental baffle case, using PVC as shell material and copper as tube material. CFD results for same are compared with the experimental one for segmental baffles. Further investigation is carried out in Fluent. The purpose of this investigation is to improve thermo-hydraulic performance of the heat exchanger with the use of different types of orifice baffles. It is investigated that as the mass flow rate increases, the heat transfer coefficient increases and pressure drop also increases. For same flow rate perforated baffle is having almost same heat transfer coefficient as hexafoil hole baffle but the pressure drop for perforated one is quite higher than the other one. A large computational effort is involved for the memory access of the computers and computing time for the simulation of the complex geometries associated with the dense grids. The available computational fluid dynamics software package FLUENT is applied to determine the related problems. Standard k - ε turbulence model is allowed to predict the three-dimensional flow and the heat transfer characteristics.