National Conference On Recent Trends And Developments In Sustainable Green Technologies
Journal of Chemical and Pharmaceutical Sciences www.jchps.com ISSN: 0974-2115

# DESIGN OPTIMIZATION OF A HEAT EXCHANGER HEADER WITH INLET MODIFIER

**S.Prakash*, S.Prabhakar, M.Saravana Kumar, K.Annamalai**
Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Missions University, Chennai, India
***Corresponding author: E.Mail:prakash.mech94@gmail.com**

**ABSTRACT**

One of the common problems observed in the shell and tube heat exchanger is erosion of shell of surface due to the force of fluid which is entering through header of the shell. This problem of erosion could be minimized by placing the modifiers at near inlet of shell which will divert the flow direction. The objective of the project is to analyse the flow characteristics at the inlet header of shell and tube heat exchanger with modifiers using the Computational Fluid Dynamics tool. A heat exchanger is a device that is used to transfer heat energy between two fluids. Shell and tube heat exchanger is built up of a bundle of round tubes mounted in a cylindrical shell with the tube axis parallel to that of the shell. One fluid flows inside the tubes, the other flows across and along the tubes. The heat exchanger would be modelled with modifiers of opening angle 30degree, 45 degree and 60 deg and curved profile. The parameters such as wall shear, intensity of turbulence, pressure variation would be considered for analysis. A comprehensive comparative analysis would be carried out for optimizing the position of modifier.

**KEYWORDS -** Heat exchanger, Transfer heat energy

## INTRODUCTION

A heat exchanger is a device that is used to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at deferent temperatures and in thermal contact. In heat exchangers, there are usually no external heat and work interactions. Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single- or multi component fluid streams. In other applications, the objective may be to recover or reject heat, or sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control a process fluid. In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak. Such exchangers are referred to as direct transfer type, or simply recuperators. In contrast, exchangers in which there is intermittent heat exchange between the hot and cold fluids via thermal energy storage and release through the exchanger surface or matrix are referred to as indirect transfer type, or simply regenerators. Such exchangers usually have fluid leakage from one fluid stream to the other, due to pressure differences and matrix rotation/valve switching. Common examples of heat exchangers are shell-and tube exchangers, automobile radiators, condensers, evaporators, air pre heaters, and cooling towers.

## SHELL AND TUBE EXCHANGERS

This exchanger, shown in Fig is generally built of a bundle of round tubes mounted in a cylindrical shell with the tube axis parallel to that of the shell. One fluid flows inside the tubes, the other flows across and along the tubes. The major components of this exchanger are tubes (or tube bundle), shell, frontend head, rear-end head, baffles, and tube sheets, and are described briefly later in this subsection. A variety of different internal constructions are used in shell-and-tube exchangers, depending on the desired heat transfer and pressure drop performance and the methods employed to reduce thermal stresses, to prevent leakages, to provide for ease of cleaning, to contain operating pressures and temperatures, to control corrosion, to accommodate highly asymmetric flows, and so on. Shell-and-tube exchangers are classified and constructed in accordance with the widely used TEMA (Tubular Exchanger Manufacturers Association) standards (TEMA, 1999), DIN and other standards in Europe and elsewhere, and ASME (American Society of Mechanical Engineers) boiler and pressure vessel codes. TEMA has developed a notation system to designate major types of shell-and-tube exchangers. In this system, each exchanger is designated by a three-letter combination, the first letter indicating the front-end head type, the second the shell type, and the third the rear-end head type.

The three most common types of shell-and-tube exchangers are (1) fixed tube sheet design, (2) U-tube design, and (3) floating-head type. In all three types, the front-end head is stationary while the rear-end head can be either stationary or floating depending on the thermal stresses in the shell, tube, or tube sheet, due to temperature differences as a result of heat transfer. The exchangers are built in accordance with three mechanical standards that specify design, fabrication, and materials of unfired shell-and-tube heat exchangers.

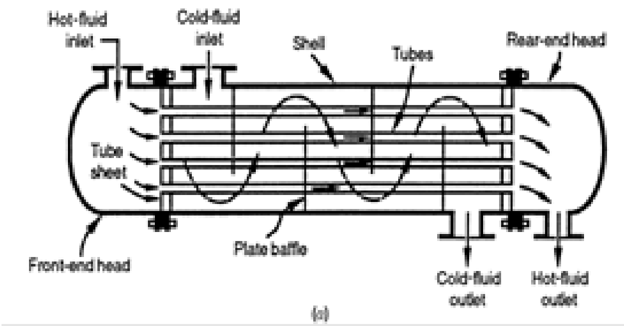



**Fig1.1 Shell and tube heat exchanger**

Class R is for the generally severe requirements of petroleum and related processing applications. Class C is for generally moderate requirements for commercial and general process applications. Class B is for chemical process service. The exchangers are built to comply with the applicable ASME Boiler and Pressure Vessel Code, Section VIII (1998), and other pertinent codes and/or standards. The TEMA standards supplement and define the ASME code for heat exchanger applications. In addition, state and local codes applicable to the plant location must also be met. The TEMA standards specify the manufacturing tolerances for various mechanical classes, the range of tube sizes and pitches, baffling and support plates, pressure classification, tube sheet thickness formulas, and so on, and must be consulted for all these details. They are custom designed for virtually any capacity and operating conditions, such as from high vacuum to ultrahigh pressure [over 100 MPa], from cryogenics to high temperatures [about 11008 C] and any temperature and pressure differences between the fluids, limited only by the materials of construction.

**COMPUTATIONAL FLUID DYNAMICS**

Prior to the era of computers, the availability of analytical solutions to the various industrially relevant Fluid dynamics problems were rendered dormant due to the unavailability of the requisite computing power to solve the millions of mathematical equations that led to the results. However, with the arrival of personal computing, the immense number-crunching power of the digital devices were put to use by programmers to develop codes that infused the equations of fluid flow into the memory of the computers by means of algorithms and programming methods. This marked the arrival and extensive use of Computational Fluid Dynamics or CFD in the solution of industrial problems. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows.

**ANSYS CFX**

ANSYS CFX is a commercial Computational Fluid Dynamics (CFD) program used to simulate fluid flow in a variety of applications. CFX allows engineers to test systems in a virtual environment. It has been applied to the simulation of water flowing past ship hulls, gas turbine engines (including the compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, and more. It is scalable. CFX software delivers powerful computational fluid dynamics (CFD) technology for simulations of all levels of complexity. ANSYS CFX takes advantage of data and information common to many simulations. This begins with common geometry: Users can link to existing native computer-aided design (CAD) packages as well as create and/or modify CAD models in an intuitive solid modeling environment. The ANSYS CFX solver uses the most modern solution technology with a coupled algebraic multi-grid solver and extremely efficient parallelization to help ensure that solutions are ready for analysis quickly and reliably. Animations of flow simulations are easily generated, and 3-D images can be directly created using the freely-distributable 3-D viewer from **ANSYS CFX.**

**MODELING OF GEOMETRY**

In the core area of the project, the flow field of air through a carburetor venturi is simulated in three dimensions. 3-D CAD modeling tool Pro/Engineer wildfire 2.0 was used to model the heat exchanger flow domain and the model was subsequently exported to the ANSYS CFX meshing environment for further processing.

Journal of Chemical and Pharmaceutical Sciences www.jchps.com ISSN: 0974-2115

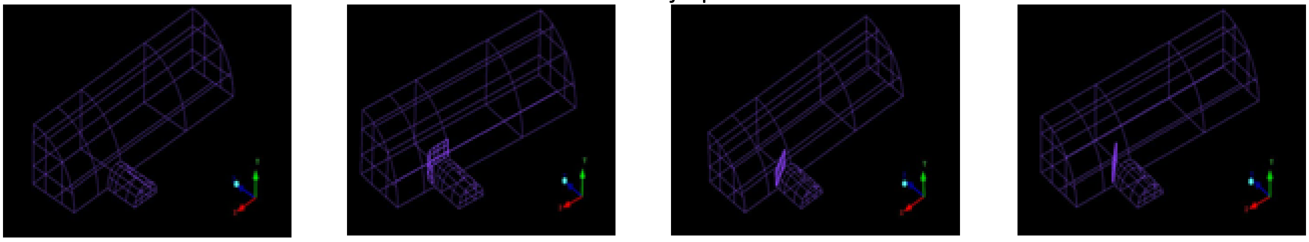


**Fig 3.1 (a) Without Modifier Fig 3.1 (b) Curved Modifier Fig 3.1(c) 30 deg opening Fig 3.1 (d) 45 deg opening**

In this paper, for the purpose of CFD analysis, the commercially available CFD software package – ANSYS CFX 11.0 has been utilized taking into account its robust prowess and capacity at modeling the flows through complex geometry flow domains. The geometry was modeled according to the dimensions specified in the thesis published by Michael Paul Schott from the University of Queensland, Australia.

## CREATION OF REGION AND MESHING

This step defines creation of regions and geometry. 2D region is created for defining inlet and outlet. Creation of regions facilitates to assign boundary condition for inlet, outlet and other defined regions. The model is exported in IGES format and is used in ICEM- CFD tool.

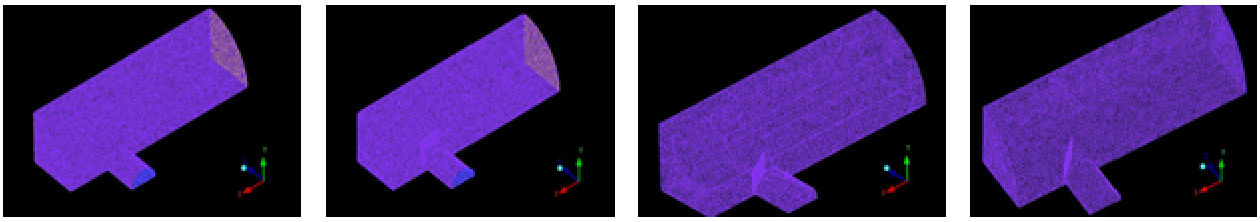


**Fig 3.2(a) Without Modifier Fig 3.2(b) Curved Modifier Fig 3.2(c) 30 deg opening Fig 3.2(d) 45 deg opening**

Surface and volume meshes were generated using this tool by defining the type of meshing element and mesh element size.The advantage of applying regions to the geometry is that they are directly associated to the model; if we remesh the geometry, they remain associated with the model. Once a new mesh is created, the regions are automatically reassigned. Mesh generation is the process by which spatial discretization of CFD model is accomplished. Meshing is based on tetrahedron element discretization.

## RESULT AND DISCUSSION

Figure shows the distribution of wall shear for the forward opening of modifier. The model without modifier shows that wall shear concentration on wall surface straight to the inlet. For the 30 deg opening position of modifier, wall shear has been reduced on side wall of shell and increased concentration is observed flow surface of the shell. The 45 deg opening shows that increased wall shear concentration on side wall of the shell straight to the flow entry.

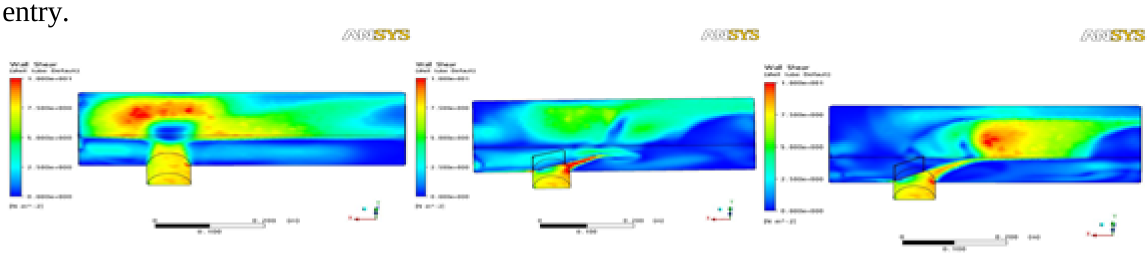


**Fig 4.1 (a) Without modifier Fig 4.1 (b) 30 deg opening Fig 4.1 (c) 45 deg opening**

4.2 Pressure Distribution

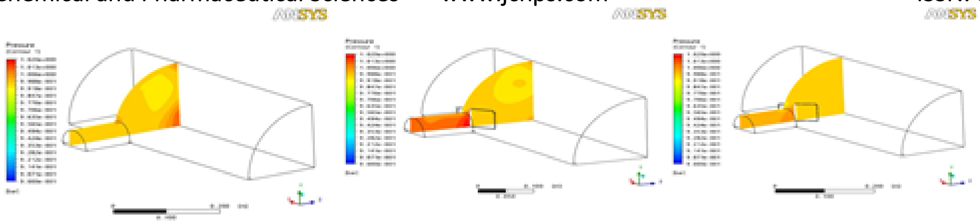
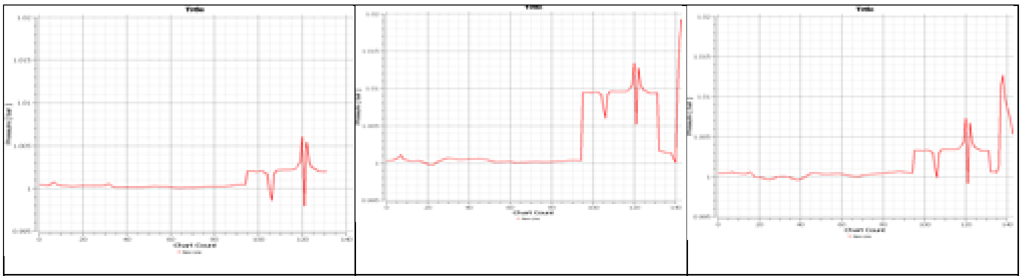


**Fig 4.2 (a) Without modifier Fig 4.2 (b) 30 deg opening Fig 4.2 (c) 45deg opening**

Fig shows the pressure distribution for the forward opening modifier. Significant pressure rise has been observed at the near wall of 30 deg opening position and slight increase in pressure has been observed in the case of 45 deg.



| Fig 4.3 (a) without modifier | Fig 4.3 (b) 30 deg opening | Fig 4.3 (c) 45 deg opening |
|---|---|---|

Figure shows the pressure distribution for the backward opening modifier. 30 deg opening modifier shows the increase in pressure at fluid entry and also near the wall of modifier and shell. In the case of 45 deg opening position pressure rise is observed at near modifier wall.

**CONCLUSION**

Thus computational analysis of shell and tube heat exchanger with inlet modifier has been carried out and the following conclusion has been drawn. The curved modifier shows comparatively reduced concentration of wall shear and shear strain rate on the side wall of shell as well as flow surface than other modifier positions. This will consequently reduce failure of the surface by erosion. The wall shear on the surface of curved modifier is also considerably less. By placing modifiers at the inlet the rise in pressure observed comparatively less in the case of curved modifier the intensity of turbulence as well as turbulence eddy dissipation is significantly less in the case of curved modifier.

**REFERENCE**

Bremhorst, K. & Flint, P. J. 1991, 'Effect of flow patterns on the erosion-corrosion of shell and tube heat exchangers', Journal of Wear, vol. 145, no. 1, pp. 123-135.
Huber, P.M. 1999, CFD Study of Erosion-Corrosion in a Shell and Tube Heat Exchanger Header, Undergraduate thesis, University of Queensland, Brisbane
Purchase, A. 2000, CFD Investigation of Erosion-Corrosion in the First Pass of a Shell and Tube Heat Exchanger, Undergraduate thesis, University of Queensland, Brisbane.
Salameh, T. 2001, Computational Fluid Dynamics Investigation of Erosion- Corrosion in a Heat Exchanger Header, Undergraduate thesis, University of Queensland, Brisbane.
Patankar, S.V. 1980, Numerical Heat Transfer and Fluid Flow, McGraw Hill Book Company, New York.
John David Anderson (1995), Computational Fluid Dynamics: The Basics with Applications, , McGraw Hill
ANSYS Europe Ltd, ANSYS CFX solver modeling guide, V11, Technical report, ANSYS Europe Ltd, 2007.