THOMSON REUTERS ENDNOTE

[Kumar* *et al.*, 5(7): July, 2016] ISSN: 2277-9655
IC™ Value: 3.00 Impact Factor: 4.116

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

STUDY ON HEAT TRANSFER EFFICIENCY PARAMETRES IMPROVEMENT IN SYSTEMS or UNITS

Pankaj Kumar *

* Assistant Professor Deptt.of Mechanical Engineering, IIMT Engg. College Meerut, U.P, India

DOI: 10.5281/zenodo.57044

ABSTRACT

Heat exchangers are an important component in many industrial systems especially in process industries. So many commercial types of heat exchangers are available in market for exchange of heat as well as for recovery of waste heat. Improved heat transfer efficiency leads to decrease in energy consumption which then results in lower equipment operational and maintenance cost, lower emissions, and consequently also lower environmental impact. However, common enhancement approaches such as adding fins or tube inserts may not always be suitable or feasible – especially in case of heat recovery from streams having a high fouling propensity. Since heat transfer rate depends also on flow field characteristics, fluid distribution, and fouling which can all be greatly influenced by the actual shapes of flow system components, several simplified models for fast and accurate enough prediction of fluid distribution as well as applications for shape optimization based on these models were developed. The created applications can then be employed during the design of heat exchange units to improve their performance and reliability.

KEYWORDS: Flow distribution, Fouling, Heat transfer, Shape optimization.

INTRODUCTION

Heat exchangers are the essential engineering systems with wide variety of applications including many power sectors, refrigeration and air-conditioning systems, nuclear reactors waste heat recovery systems, chemical and food industries etc. Many processes involve exchange of thermal energy and as such require equipment capable of transferring heat from one medium to another medium. These heat exchange units commonly, called heat exchangers, can therefore be seems not only in many industries from the chemical and petroleum ones through pulp and paper production to the food or beverage industry (evaporators, condensers, heat recovery steam generators, etc.), but also in households (refrigerators, boilers, air conditioners, heat pumps, hot water converters), transportation (HVAC and engine cooling systems), electronics (heating equipments for cooling of hot chips on printed circuit boards), and in many other areas. Since by improving heat transfer efficiency we can substantially decrease energy consumption, this will result in lower equipment running cost, decrease in emissions, and hence lower environmental impact.

Another way of enhancing heat transfer is by using nanofluids, i.e., heat transfer fluids enriched with nanoparticles made of silicon dioxide, aluminium oxide, copper, or cupric oxide etc. Elements such as multi-walled carbon nanotubes can be added into the fluid as well. We could even employ oscillatory flows, but this approach is not commonly due to the determining the effect of vibrations on the equipment. The above methods of local heat transfer efficiency improvement, however, may not always be feasible especially in case of heat recovery from polluted streams (waste-to-energy plants etc.). Nevertheless, efficiency depends also on the actual character of flow in the unit which, in turn, significantly affects fouling and in consequence the overall heat transfer coefficient. Since heat transfer rate is largely dependent on the available heat transfer area, massive flow parallelization in the heat exchangers is very common. Hence, with respect to the fact that flow field characteristics, fluid distribution, and fouling can be greatly influenced by the actual shapes of flow system components like splitting and ducts, collecting manifolds, etc., in PRESENT WORK we will deal with shape optimization of such components.

COMMON PROBLEMS

Aside from maldistribution, three possible flow-related issues – backflow, instabilities, and fouling which are to be keep in mind when designing a parallel flow system or unit, be it in a heat exchanger or in any process unit. These issues can occur even in very simple systems or units and complexity of the actual system layout is rather irrelevant. More importantly, each of them can lower efficiency, cause product degradation due to insufficient heating or overheating of the fluid, or even bring about malfunction of the system.

Flow rate through an individual branch of such a system is governed by the pressure difference between inlet in the distributor and outlet in the collector, $p = p_{out}$ □p_{in}. If _p < 0, then the fluid, indeed, flows in the expected direction from the distributor into the collector. If, however, _p > 0, then the fluid flows in the opposite direction. This behavior is called "backflow" and is generally undesirable.

Any instability is caused due to a random disturbance applied by a positive feedback whereas its ultimate consequence is turbulence and random waves. This is highly undesirable – especially in high-temperature applications , since then channels are subjected to on-periodic variable due to changes in their property i.e temperatures with a common end result being mechanical failures. We should try to avoid any type of parallel flow layout which exhibits such a behavior.

By fouling we mean any accumulation of undesirable material on equipment surfaces of a process that hinders the desired operation.
It is therefore obvious that the fouling must be taken into consideration when designing any working unit which is expected to do work with a fluid having a high fouling propensity. We must eliminate as many stagnation zones with swirling character of flow as possible or at least minimize formation of eddies. Plain surfaces and suitable materials must be used for further reduction in fouling rate. Further, units should be constructed in such a manner that cleaning the surfaces of heat transfer and other essential regions is easy.

METHODS FOR FLOW DISTRIBUTION PREDICTION

There are three main methods one can employ to predict flow rates and pressure profiles in individual branches of a flow system. Each of these methods provides a different level of accuracy and has different requirements considering time necessary for flow evaluation, cost and computing power. These methods are, in no particular order, experiment on a prototype, computational fluid dynamics, and successive branch-by-branch approach. Among of them we are focusing on successive branch-by-branch approach.

Experiment on a Prototype

Building a flow system prototype and measuring flow rates and pressures (or any other quantity for that matter) as necessary will, obviously, provide high-quality data. However, there is a shortcoming we must consider, i.e., prototypes sometimes do not allow us to fully imitate operating conditions of real equipment. In such a case the obtained data may be inaccurate to a certain degree depending on properties of the working fluid (density and viscosity variation with temperature etc.)

Successive Branch-by-Branch Approach

Branch-by-branch approach is a special case of pseudo-1D discretization. It simplifies the problem even further by en bloc evaluation of each segment of a channel between two points where fluid is split or merged. In other words, it examines a flow system sequentially using a very coarse pseudo-1D mesh and therefore suffers from similar problems as pseudo-1D discretization.

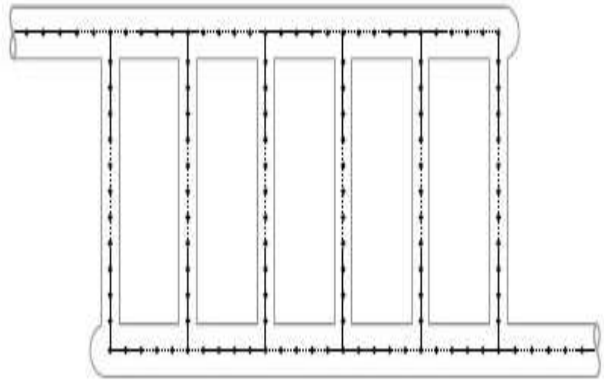

Figure . Pseudo-1D mesh of a simple flow system

Nonetheless, once a model is fine-tuned for a certain class of flow system geometries, its production use does not pose any significant risk. This method is implemented in the majority of the models mentioned further due to the extremely fast evaluation and effortless modification of geometry characteristics. Section 3.3 demonstrates that it can be used even for relatively complex flow systems.

Overall heat transfer coefficient then falls due to higher resistance (thermal) of the layers, that is implies lower efficiency of heat exchanger and, in turn, huge economic losses.

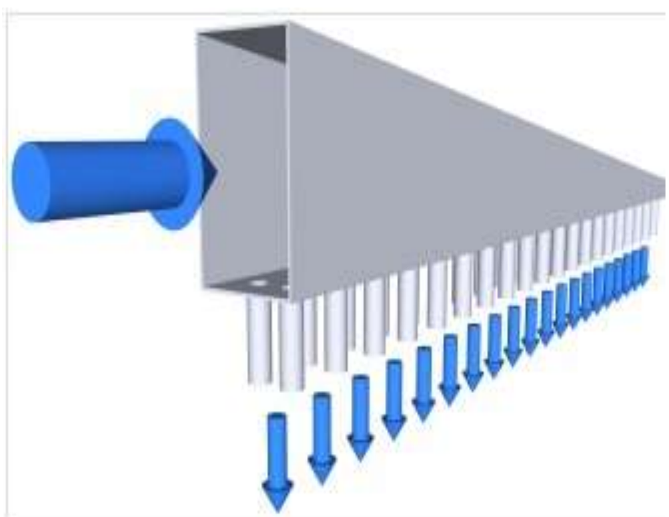

Figure. Distributor with variable rectangular cross-section and double lateral branches

It is therefore obvious that the fouling must be taken into account consideration when designing any working unit that is expected to do work with a fluid having a high fouling propensity. We must eliminate many zones with swirling character of flow if possible or at least minimize formation of eddies. Plain surfaces and suitable materials should be used to further lower fouling rate. In general, the units should be constructed in such a way that cleaning of heat transfer surfaces and other essential regions is easy.

WAY OF INCREASE IN ACCURACY

As before, fluid enters the system through the distributor where it is split into individual tubes of the tube bundle. Then it is heated or cooled as required and subsequently it enters the collector to be merged again into a single stream. The model described below is based on (Ngoma and Godard, 2005), but features the following modifications:

• The quantities which are evaluated along the manifolds instead of considering those to be mass points.
• Mixing of fluid streams having a temperature different is supported at tube entrances and exits in both manifolds due to the possibility of backflow.
• Geometry of each tube in a bundle can be defined arbitrarily as a function instead of being specified only by a number of equidistant passes.
• Heat flux into each tube can, again, be defined as a function instead of being constant throughout the entire tube bundle; and
• Three types of tube ends can be simulated – exserted, conical, and circular bellmouth – instead of the tubes being just flush with manifold walls.

The former two improvements should provide a noticeable increase in accuracy while the latter three improvements make the new model easily applicable to a wider range of process units.

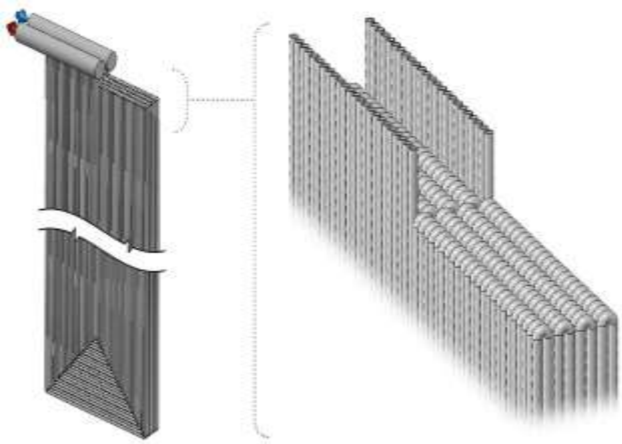

Figure: Sample parallel flow system

Figure shows a sample parallel flow system with moderately complex tube bundle that is easily evaluable using the discussed pseudo-1D model. It was always made sure that the used regression sub-models were statistically sound, i.e., that residuals were consistent with a normal distribution having zero mean and approximately constant variance. Coefficients of determination of the obtained sub-models were in all cases greater or equal to 99.5 %.

CONCLUSION

In present work we focused on shape optimization of flow systems in heat exchange units since we can significantly increase heat transfer efficiency via improvement of flow distribution and abatement of fouling. Additionally, we discussed the effect of flow field characteristics upon fouling rate. Two variants of each of them exist so that both compressible and incompressible flows can be analyzed. The model is based on the simplified branch-by-branch approach and describes pure distribution from a manifold with variable rectangular cross-section into a constant-pressure environment. It simplifies the problem even further by en bloc evaluation of each segment of a channel between two points where fluid is split or merged. The concluded summary represented that by improving heat transfer efficiency we can substantially reduce energy consumption, hence will result in lower equipment operational cost, reduced emissions, and consequently also lower environmental impact.

ACKNOWLEDGEMENT

I have been able to complete this study with the help of a few but very important people who had trust in my abilities. I would like to thank **Kapil Dev**, who guided me in this paper and **Mr. Satish Maurya** who helped me in completing this paper successfully.

REFERENCES

[1] Garcia-Cascales, J.R., Vera-Garcia, F., Corberan-Salvador, J.M., Gonzalvez-Macia, J., Assessment of Boiling and Condensation *Heat Transfer Correlations in the Modelling of Plate Heat Exchangers,* International Journal of Refrigeration, 30, 2007, 1029-1041.

[2] Vera-Garcia, F., Garcia-Cascales, J.R., Corberan-Salvador, J.M., Gonzalvez-Macia, J., Assessment of Condensation *Heat Transfer Correlations in the Modelling of Fin and Tube Heat Exchangers,* International Journal of Refrigeration, 30, 2007, 1018-1028.

[3] Kumar, K. and Venkatarathnam, G., *Optimization of matrix heat exchanger geometry*. J. Heat Transfer, 122, **3**, 2000, 579–586.

[4] Davidson, J. and S. Mantell, "Use of polymers in liquid-to-liquid heat exchangers; Applied to low-cost solar water heating systems", Task III Report, University of Minnesota, 1998.

[5] Brunger, A.P., Hollands, K.G.T., Van Decker, G.W.E., *Heat-Exchange Relations for Unglazed Transpired Solar Collectors with Circular Holes on a Square or Triangular Pitch*, Solar Energy, 71, **1**, 2001, 33-45.

[6] Andrew, M.H., Shaaban, A.H., Jamil, A.K., and Ian, G.S., Salavani, R., *An Experimental, Numerical, and CFD Investigation into the Heat Transfer Characteristics in Porous Media Using a Thermal Non-Equilibrium Model*, Proc. Of IHTC2006, the 13th IHTC, Sydney, Australia, Aug. 21-25, 2006.

[7] Thome, J.R., *Boiling In Microchannel: a review of experiment and theory,* International Journal of Heat and Fluid Flow, 25, 2004, 128-139.