CHEMICAL ENGINEERING TRANSACTIONS

VOL. 29, 2012

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at: www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Hon Loong Lam, Jiří Jaromír Klemeš

ISBN 978-88-95608-20-4; **ISSN** 1974-9791 DOI: 10.3303/CET1229239

CFD-based Optimisation of Spiral Wound Heat Exchanger Geometry

Julia Steube[a], Anna Lautenschleger[a], Mark Piper[a], Daniel Böe[b], Thomas Weimer[b], Eugeny Kenig*[a]

aChair of Fluid Process Engineering, University of Paderborn, Pohlweg55, D-33098 Paderborn, Germany
bMakatec GmbH, Siemensstraße 3, D-71149 Bondorf, Germany
eugeny.kenig@upb.de

Spiral wound plastic heat exchangers represent an important class of heat exchangers, with high chemical resistance, low dimensions and reasonable costs. However, their design and optimisation are mainly based on practical experience, whereas comprehensive investigations have not been accomplished yet.
In this paper, we present the results of the collaborative research performed in the context of the EU-project INTHEAT (Intensified Heat Transfer Technologies for Enhanced Heat Recovery). The focus of this work is to gain a deeper understanding of the fundamental transport phenomena occurring in spiral wound plastic heat exchangers, thus allowing their performance to be improved. The apparatus chosen for the investigation was developed by Makatec GmbH (Makatec). It consists of spiral wound plastic films kept separated by spacer filaments building grid-like arrangements. In addition to the separation function, these arrangements must ensure an efficient mixing and thus an intensified heat transfer in the exchanger, whereas the mixing process strongly depends on the specific filament grid structure.
A Computational Fluid Dynamics (CFD) based model is developed that yields a detailed description of the flow and temperature fields in the complex geometric structure of the investigated spiral wound plastic heat exchanger. The simulations are performed for a single-phase liquid flow and for volumetric flow rates used in the Makatec test rig, with the help of the commercial software STAR-CCM+ (CD-adapco).The model is validated against experimental pressure drop data obtained at Makatec. The spacer geometry has then been varied and its influence on the flow behaviour evaluated. Enhanced turbulence is supposed to have a positive impact on the efficiency of the exchanger. Based on the obtained results, new spacer geometries are suggested in order to facilitate theturbulence and thus to achieve a better exchanger performance.

1. Introduction

The efficiency of overall thermal management depends on the performance of involved heat exchangers. Polymer heat exchangers are used in applications which include the cooling or temperingof aggressive liquids, e.g., acids, caustic solutions or lubricants. Such polymer heat exchangers are developed by Makatec GmbH (Makatec) using spiral wound arrangements. Figure 1 shows a cross-section of the Makatec spiral wound plastic heat exchanger. It is operated under counter-current flow conditions, whereas the two fluids are separated by an integrated polymer membrane.
In spiral wound modules, spacers are used to provide mechanical support for the membranes. However, they also act as turbulence promoters in the channels. Several studies reveal that the

Please cite this article as: Steube J., Lautenschleger A., Piper M., Böe D., Weimer T. and Kenig E., (2012), CFD-based optimisation of spiral wound heat exchanger geometry, Chemical Engineering Transactions, 29, 1429-1434

geometric parameters of spacers have a significant influence on flow resistance and heat transfer (Cao and Wiley, 2001, Karode and Kumar, 2001). Obviously, mixing is more efficient in spacer-filled channels than in empty channels; however, the pressure drop in spacer-filled channels grows. Therefore, it is necessary to find an optimal spacer geometry in order to obtain the best performance in terms of both,heat transfer and pressure drop characteristics.

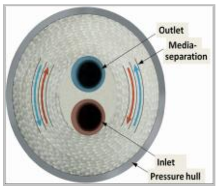

Figure1: Cross-section of a spiral

Computational Fluid Dynamics (CFD) based simulations represent a powerful tool allowing investigation of flow patterns in spacer-filled channels (Patankar, 1980).Compared to experimental studies, CFD simulations facilitate the testing of new concepts and ideas and, therefore, help to reduce development time and cost. The present study is aimed at the determination of the spacer geometry impact on the performance of plastic spiral wound heat exchangers.

2. Experimental

Schock and Miquel (1987) compared the measurement data on mass transfer and flow resistance in flat channels and spiral wound modules, both filled with various commercial spacers. They noted that spacer-filled flat channels and channels in spiral wound modules with the same spacers exhibited similar flow characteristics. This means that pressure drop and mass transfer coefficients measured in flat channels are generally applicable for spiral wound modules. We used these results and focused on the study of planar channels, the curvature was neglected. Furthermore, only a limited spacer grid section of the heat exchanger was investigated.

Figure 2: MAKATEC test rig

The configuration of the Makatec test rig displayed in Figure 2 allows the analysis of different spacer geometries. It is used to investigate the influence of the geometry on the pressure drop at varying flow rates. The test rig consists of PVC (polyvinylchloride) plates with adapters for the liquid inlet and outlet. Furthermore, there exist three measurement points thatcan be used for the determination of pressure and liquid volume flow rates. The investigated grid samples have a length of 200 mm and a width of 40 mm. In Figure3, the experimentally investigated spacer geometries are represented with relevant geometric parameters. Filaments have circular cross-sections and an equal diameter. The geometric parameters used to characterise a spacer grid are L, the distance between spacer filaments, α, the angle between crossing spacer filaments and d, the spacer diameter.

Makatec investigated two different spacers that differ in material and geometry. The PP (polypropylene) spacers have a diameter of d=1.3 mm. Within the grid, they have a crossing angle of α=45° in flow direction. Overlapping spacers have a mergence of 0.6 mm. The distance between parallel spacers is L=4.0 mm. The advantage of this mesh is a cheap price combined with a good chemical resistance. It can be used for applications with a maximum

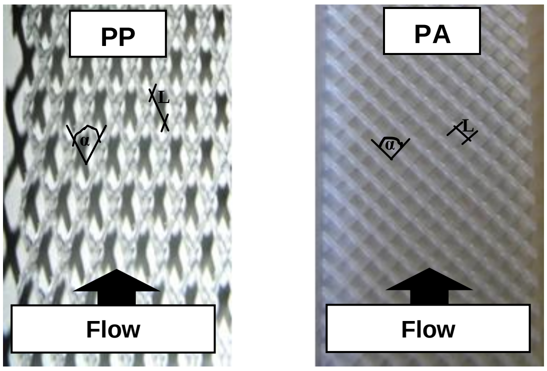

Figure3: Experimentally investigated spacer geometries.

temperature of 90°C. In this study, the pressure drop of the PP spacer has been measured for water volume flow rates of 40– 500 l/h at a temperature of 20 °C.
The PA (polyamide) spacer grid has a higher temperature resistance, but also a significant higher price and a lower chemical resistance compared to the PP spacers. The PA spacers have a diameter d=1.0 mm. Spacers cross at an angle of α=90° in flow direction and have a mergence of 0.6 mm. The distance between parallel spacers isL=2.55 mm (cf. Figure 3).The pressure drop of the PA spacer was investigated for water flow rates of 20 – 350 l/h at a temperature of 20 °C.

3. Simulations

The basis for a comprehensive description of fluid flow and heat transfer in CFD are the principles of conservation of mass, momentum and energy. These principles are expressed mathematically, in terms of partial differential equations (Bird*et al*., 2007).Regarding their non-linear nature, these differential equations cannot be solved analytically. Instead an approximate numerical solution is sought out, which is achieved by discretising the domain and equations, for instance, by using the finite volume method, and by subsequently solving the discretised system (Patankar, 1980).
In this work, the simulation studies were carried out using STAR-CCM+, a commercial CFD tool by CD-adapco, with implemented finite volume method. The simulations in STAR-CCM+ are performed, based on a CAD model of the spacer geometry (see Figure 4).

Figure 4: CAD draw of the periodic element

The required computer memory and the numerical errors increase when the number of numerical cells grows.On the other hand, the grid needs to be fine enough for yielding stable and adequateresults.To reduce the computational effort keeping a fine grid, a periodic element was identified which consists of only one spacer inter-section. Figure 5illustratesthe selection of a periodic element.
The simulations of such a periodic element were carried out using periodic boundary conditions that realise the computational domain reduction. These conditions can be applied to systems with inherent periodicity (like the studied flow in the periodic geometry) and just transfer relevant information (e.g., on velocities, on fluxes) from one edge to the opposite one. In particular, Periodic (1) condition describes a fully-developed hydrodynamic flow, whereas the flow rate is specified at the inlet interface. The Periodic (2) condition ensures the mapping of velocity profiles.
The pressure drop was calculated for the spacer geometries experimentally investigated by Makatec (see above).The calculations were performed with both a laminar and a turbulent (k-ε) flow model. The latter model includes two extra transport equations to evaluate the turbulent properties of the flow, namely k, the turbulent energy, and ε, the isotropic dissipation rate. The simulations with both liquid flow models were convergent and showed equal results. This means that the spacer geometries and flows studied by Makatec bring about just laminar flow conditions.

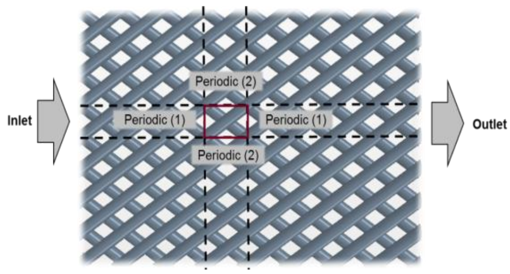

Figure 5: Selection of a periodic element with the respective boundary conditions.

4. Results

The developed CFD model was validated using the experimental pressure drop data obtained by Makatec. A comparison of experimental and calculated pressure drop values at different flow rates V is shown in Figure 6. The average relative deviation between these values is 17.75 % for the PA spacer grid and 23.08 % for the PP spacer grid. The deviations can be attributed to the very small length of the experimentally investigated spacer section and related non-established flow effects. These effects are not considered in the simulation studies, because the overall spiralwound heat exchanger would not be influenced by them. Under such conditions, the agreement between the simulated and experimental values can be considered satisfactory and the CFD model validated.

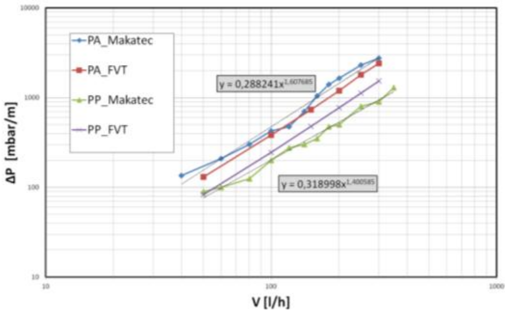

Figure 6: Comparison between experimental and numerical pressure drop values.

A visualisation of the liquid flow streamlines in Figure 7demonstrates that, in the laminar flow regime, the liquid flows parallel to the spacer filaments. This means that, under laminar flow conditions, the spacers can hardly act as turbulence promoters.

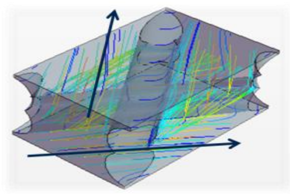

Figure 7:Visualisation of streamlines.

Geometry optimisation should be aimed at fulfilment of two criteria, namely pressure drop reduction and intensified heat transfer via enhanced turbulence. These requirements are contradictory, and hence, a balance between them must be found. Therefore, we investigated some changes in the geometric parameters α, L and d (see above) which appeared promising for the pressure drop reduction.

4.1 Variation of α

The impact of the angle between the spacer filaments is investigated by comparing the pressure drop values for the PA and PP spacer geometries (see Figure 3). The angles between the spacer filaments were „exchanged" and the simulations were carried out for the same volume flow rates.The distance between parallel spacer filaments was kept at the same value. Figure 8 explains the change in spacer angle for the PA grid geometry. The results of these simulation studies shown in Figure 9 demonstrate, that pressure drop decreases with a decreasing filament angleα.This can be explained by a smaller number of disturbance nodes and by less intensive re-direction at lower α-values.

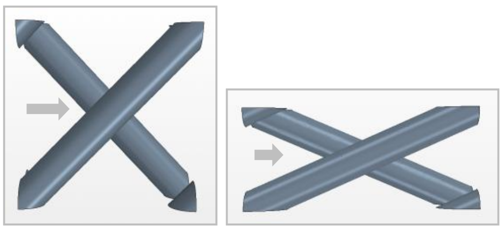

Figure 8: Change in spacer angle for PA grid geometry.

Based on Figure 9, it can be concluded that the PA spacer grid causes a lower pressure drop than the PP spacer grid. This can be attributed to the specific structure of these spacer grids, namely, to the fact that the PP spacers, with their thicker filaments at the same channel heights, leave less room for the flow than the PA spacers. To evaluate the influence of these variations on the overall heat exchanger performance, further investigations are necessary.

4.2 Double arrangement of Geometry I

As discussed above, pressure drop strongly depends on the size of the channel. Therefore, we investigated a double arrangement of the spacer geometry with a spacer angle of α=45° which is shown in Figure 10as a possible modification reducing pressure drop in a channel. Figure 11 illustrates the results of these investigations.

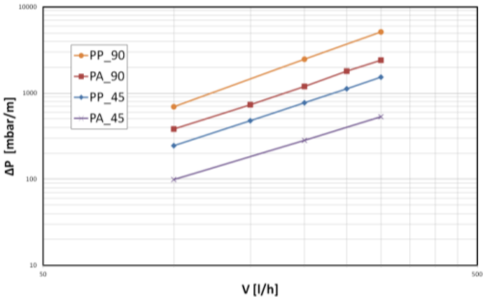

Figure 9: Pressure drop as a function of flow rate at different spacer angles.

The pressure drop decreases compared to the pressure drop for a single layer of Geometry I, whereas the flow pattern remainslaminar. Thus, the advantage of a double layer arrangement of the spacer grids is the opportunity to reduce the pressure drop by enlarging the channels; this can be promising for applications in which it is not possible to change the spacer geometry due to process requirements (see above). However, any conclusion on the efficiency improvement of a spiral wound heat exchanger with a double layer arrangement can be done only on the basis of a thorough heat transfer analysis.

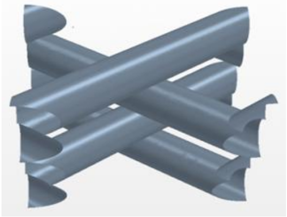

Figure 10: Double arrangement of Geometry I.

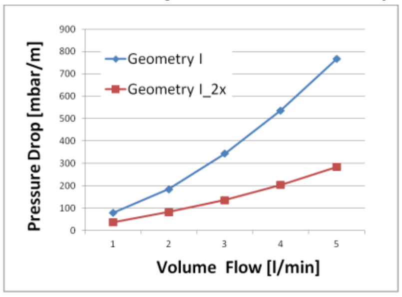

Figure 11: Pressure drop as a function of flow rate for different geometries.

4.3 Enhancement of turbulence

As we see, straight filaments appear to promote laminar flow in the heat exchanger. To enhance turbulence, one may try to change the filament shape. Therefore, we suggested a spacer grid with interwoven filaments and simulated this arrangement to analyse the impact on the development of turbulence and its influence on the pressure drop. As can be seen in Figure 12, the liquid flow pattern becomes turbulent, which should be advantageous for heat transfer. However, the pressure drop within this spacer arrangement is about three times higher, compared to the same spacer geometry with straight filaments. Thus, only in case of very significant positive changes in the heat transfer performance can this variant be considered as a relevant optimisation measure.

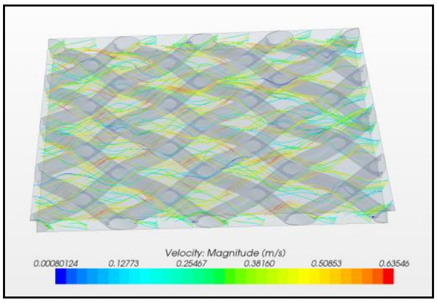

Figure 12:Visualisation of turbulent flow pattern within an interwoven spacer grid

5. Concluding remarks

A CFD model describing the fluid dynamics in the spiral wound heat exchanger developed by Makatec was elaborated and implemented in commercial software tool STAR-CCM+. This model was validated using the experimental pressure drop data of Makatec, and a satisfactory agreement between simulated and measured values was found.

Using this model, the spacer geometry was varied to identify its impact on the pressure drop characteristics. In particular, the variation of spacer angle α and the channel size were tested.

The investigations clearly show that the pressure drop reduction can be achieved by enlargement of the channels within the spacer grid, for instance, applying double layers of spacer grids or grids with large filament diameters or bigger distances between the filaments.

Another way to geometry optimisation of the spiral wound heat exchangers is to enhance turbulence and hence heat transfer. In this regard, a grid geometry with interwoven spacers was investigated. The results showed indeed a turbulent flow pattern, but with significantly increased pressure drop characteristics. It is thus necessary to investigate temperature fields and heat transfer parameters in all studied geometries and compare them. This work will be done in the near future in the context of the the EU project INTHEAT.Moreover, further investigations are going to be performed with new materials which are appropriate for use in sulphuric or hydrochloric acidapplications.

Acknowledgements

The financial support of the European Commission in the context of the 7th Framework Programme of the European Union (Activity 2.1, Research for SMEs, FP7-SME-2010-1-262205-INTHEAT) is greatly acknowledged.

References

Cao, Z., Wiley, D.E., Fane, A.G.,2001, CFD simulation of net-type turbulence promoters in a narrow channel, J. Membr. Sci. 185, 157–176.

Karode, S.K.,Kumar, A., 2001, Flow visualization through spacer filled channels by computational fluid dynamics. Part I. Pressure drop and shear rate calculations for flat sheet geometry, J. Membr. Sci. 193, 69–84.

Patankar, S. V., 1980, Numerical Heat Transfer and Fluid Flow, Hemisphere Publ. Corp., McGraw-Hill, New York, USA.

Schock, G., Miquel, A., 1987, Mass transfer and pressure loss in spiral wound modules, Desalination 64, 339–352.

Bird, R. B., Stewart, W.E., Lightfoot, E. N.,2007, Transport Phenomena, 2nd Edition, John Wiley & Sons, Inc., New York, USA.