VYSOKÉ UČENÍ TECHNICKÉ V BRNĚ

Fakulta strojního inženýrství

Ústav procesního a ekologického inženýrství

Ing. Vojtěch Turek

NEW ELEMENTS OF HEAT TRANSFER EFFICIENCY IMPROVEMENT IN SYSTEMS AND UNITS

NOVÉ PRVKY ZVÝŠENÍ EFEKTIVITY VÝMĚNY TEPLA V SYSTÉMECH A ZAŘÍZENÍCH

Zkrácená verze Ph.D. Thesis

Obor:	Konstrukční a procesní inženýrství
Školitel:	doc. Ing. Zdeněk Jegla, Ph.D.
Oponenti:	prof. Dr.-Habil. Ing. Jiří Klemeš, D.Sc., Dr.h.c.
	University of Pannonia Veszprém, Maďarsko
	Ing.Miloslav Odstrčil, CSc.
	Konzultant Brno

Datum obhajoby: 2. listopadu 2012

Keywords

flow distribution, fouling, heat transfer, shape optimization

Klíčová slova

distribuce toku, přenos tepla, tvarová optimalizace, zanášení

Místo uložení dizertační práce

Ústav procesního a ekologického inženýrství, FSI, VUT v Brně

Tato práce vznikla jako školní dílo na Vysokém učení technickém v Brně, Fakultě strojního inženýrství. Práce je chráněna autorským zákonem a její užití bez udělení oprávnění autorem je nezákonné, s výjimkou zákonem definovaných případů.

CONTENTS

1 INTRODUCTION

Many processes involve exchange of thermal energy and as such require equipment capable of transferring heat from one medium to another. These heat exchange units – commonly called heat exchangers – can therefore be found not only in various industries from the chemical and petroleum ones through pulp and paper production to the food or beverage industries (condensers, evaporators, reboilers, heat recovery steam generators, etc.), but also in households (refrigerators, boilers, hot water radiators, air conditioners, heat pumps), transportation (HVAC and engine cooling systems), electronics (heat sinks 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 operational cost, reduced emissions, and consequently also lower environmental impact.

In general, we must always start with the whole process in mind and first and foremost make sure that heat is utilized efficiently in the global sense. This involves heat transfer intensification in terms of a heat exchanger network (Klemeš and Varbanov, 2012), be it a grassroot design (Laukkanen et al., 2012) or a retrofit (Wang et al., 2012). Methods such as process integration (Zhang et al., 2012) are of paramount importance as well and many tools are being developed for this purpose (see for instance Klemeš et al., 2009). Once this is done, we can proceed to the analysis of possible improvements of heat transfer efficiency in individual heat exchange units. Here, heat transfer can be further enhanced in a number of ways from which the most common technique probably is adding fins to heat transfer surfaces. These can be of various shapes and sizes and their effect has been investigated both theoretically (see e.g. Kundu and Lee, 2012) and experimentally (Liang and Wong, 2010).

By adding fins we primarily increase heat transfer area, but increasing turbulence results in enhanced heat transfer as well. This can be achieved for example by using corrugated, dimpled, wavy, twisted, or otherwise deformed tubes as described for instance by Pethkool et al. (2011). In plate-type heat exchangers, a vast array of different plate corrugation patterns (Arsenyeva et al., 2011) and plate-fin designs (Fernández-Seara et al., 2012) are used. Turbulence can also be increased by various types of inserts – wire mesh packings, wire coils, twisted wire brushes, helical tapes with central supporting rods, or twin twisted tapes. These were investigated for example by Dyga and Płaczek (2010). Vortex generators have an analogous effect, as verified e.g. by Cheng et al. (2012). We must, however, consider the fact that enhanced surfaces, flow channel inserts, and other similar design modifications are not suitable if working fluid has high fouling propensity.

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 (see for instance Wongcharee and Eiamsa-ard, 2011). Ele-

ments such as multi-walled carbon nanotubes (Lotfi et al., 2012) can be added into the fluid as well. We could even employ oscillatory flows (Cheng et al., 2009), but this approach is not common due to the detrimental effect of vibrations on the equipment.

The above methods of local heat transfer efficiency improvement, however, may not always be suitable or feasible – especially in case of heat recovery from polluted streams (waste-to-energy plants etc.; see Stehlík, 2011). Nevertheless, efficiency depends also on the actual character of flow in the unit which, in turn, significantly affects fouling (Kukulka and Devgun, 2007) and in consequence the overall heat transfer coefficient. Additionally, since heat transfer rate is largely dependent on the available heat transfer area, massive flow parallelisation inside heat exchangers is very common. Therefore, 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 (splitting and collecting manifolds, ducts, etc.), in this thesis we will deal with shape optimization of such components.

1.1 GOALS AND OVERVIEW

The aim of this thesis is to provide mathematical models of flow systems applicable in shape optimization algorithms and also computer implementations of these models. Obviously, such models must be simple and easy to evaluate yet robust and accurate enough otherwise either the results would be useless or the optimization times necessary to obtain them would be unacceptably long.

We will discuss simplified modelling of fluid flow and analysis of flow distribution. Three different mathematical models and optimization tools based upon them will be presented. Flow instabilities, backflow, and the relationship between fouling and flow field characteristics will be briefly mentioned as well.

2 FLOW DISTRIBUTION

Heat exchanger duty depends largely on the actual heat transfer area since properties of both hot and cold streams and hence also the temperature difference are given by plant flow-sheet and are more or less fixed. Increasing heat transfer area by massive flow parallelisation is therefore a very common way of increasing heat duty while retaining compact heat exchanger design. Typical examples of exchangers with parallelised flows are the shell-and-tube heat exchanger or the plate-type heat exchanger.

One of the first papers on flow distribution in parallelised systems (Acrivos et al., 1959) described the analytical successive branch-by-branch approach decomposing an entire splitting or combining constant cross-section manifold into control volumes around the discharge ports and dealing with each of them separately. Nevertheless, only manifolds with uniformly spaced lateral pipes and

circular cross-sections distributing fluid into (or collecting fluid from) a constant-pressure environment were discussed. Another paper (Bailey, 1975) dealing with uniformly perforated manifolds having constant circular cross-sections investigated the influence of axial velocity of fluid on the actual direction of discharge. Here, friction coefficients were assumed to be calculated rather than given as constants in advance. Other existing algebraic models describe either mere division or combination of flows in manifolds with circular (Lu et al., 2008) or rectangular (Fu et al., 1994) cross-sections, or parallel flow systems with these manifolds (Ghani et al., 2012).

Considering micro-scale applications, a differential model of constant cross-section flow system was presented e.g. by Wang and Wang (2012). In spite of the fact that these models could be used in the usual process industry design problems, their complexity owing to the differential nature is quite restrictive.

More advanced models of distribution systems in common process equipment cover division or combination of flows in case of manifolds with circular (Chandraker et al., 2002) or rectangular (Habib et al., 2009) cross-sections. As for complete parallel systems, both finite-difference (Datta and Majumdar, 1980) and differential (Bajura and Jones, 1976) models have been developed. A model supporting two-phase flow was presented e.g. by Marchitto et al. (2012). Many authors also published experimental studies based of computational fluid dynamics (CFD) evaluations (see for example Gandhi et al., 2012).

Since the amounts of fluid flowing through branches of a parallel flow system (i.e., the resulting distribution) depend on pressure differences between branch ends in splitting and collecting manifolds and these can be greatly influenced by longitudinal manifold cross-section variability, an appropriate design of the manifolds can result in a much more uniform flow distribution. This is due to the fact that the differences themselves are given by pressure profiles in the manifolds and any change of a pressure profile (caused, for instance, by a locally convergent or divergent shape of the manifold) must influence the lateral flow rates. All the models mentioned so far, however, either assume manifolds of constant cross-sections or are far too complex to be employed for shape optimization. What is more, a computational tool based on one of these models would probably need significantly more time to yield a solution than a tool making use of the simpler successive branch-by-branch approach with algebraic equations. As for CFD, although it can be very useful (one obtains an extensive set of accurate data that would otherwise be unavailable), considering shape optimization such an approach is highly disadvantageous. That is, every geometry must be created, meshed, and then evaluated which, in total, can take from several hours up to several days.

Considering the lack of models that are computationally inexpensive, easy to implement, and can be readily modified to cover even made-to-measure flow

systems, models presented in this thesis are based mainly on Bailey's paper, experimental data available in (Idelchik, 1986), and information published in monographs.

2.1 COMMON PROBLEMS

Aside from maldistribution, three possible flow-related issues – backflow, instabilities, and fouling – must be kept in mind when designing a parallel flow system, be it in a heat exchanger or in any process unit in general. These issues can occur even in very simple systems and complexity of the actual layout is therefore 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.

2.1.1 Backflow

Let us consider a simple parallel flow system consisting of a distributor, several branches, and a collector with a fluid being fed into the distributor inlet. Flow rate through an individual branch of such a system is governed by the pressure difference between its inlet in the distributor and its outlet in the collector, $\Delta p = p_{\text{out}} - p_{\text{in}}$. If $\Delta p < 0$, then the fluid, indeed, flows in the expected direction from the distributor into the collector. If, however, $\Delta p > 0$, then the fluid flows in the opposite direction. This behaviour is called "backflow" and is generally undesirable.

2.1.2 Instabilities

Any instability is caused by a random disturbance amplified by a positive feedback while its ultimate consequences are turbulence and random waves. Detailed theoretical information related to flow instabilities can be found in (Sengupta and Poinsot, 2010). Additionally, instabilities specifically related to heat exchangers were studied by Houdek (2007). In this thesis, however, we will concern ourselves only with one of the effects of instabilities, namely unsteady flow distribution. As the name suggests, it means that flow rates through individual channels are not constant in time. This is highly undesirable – especially in high-temperature applications –, since then channels are subjected to (non-periodic) variable loading due to changes in their temperatures with a common end result being mechanical failures. We should therefore try to avoid any parallel flow system layout that exhibits such a behaviour.

2.1.3 Fouling

By fouling we mean any accumulation of unwanted material on surfaces of a process equipment that hinders the desired operation. This issue is particularly

common in food industry, chemical industry, and energy industry (including waste-to-energy applications). Overall heat transfer coefficient then falls due to higher thermal resistance of the layer, which implies lower heat exchanger efficiency and, in turn, huge economic losses. For example, Hewitt (1998) provided an estimate as large as 1.4 billion USD per year for plants in the United States. It is therefore obvious that fouling must be taken into account when designing any process unit that is expected to 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 should be used to further lower fouling rate. Additionally, units should be constructed in such a way that cleaning of heat transfer surfaces and other essential regions is easy.

2.2 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. Moreover, since CFD usually denotes numerical evaluation of 3D (or less commonly 2D) geometries, we will also add numerical evaluation of pseudo-1D geometry utilizing partial differential equations to the list of available methods.

2.2.1 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.).

2.2.2 Computational Fluid Dynamics

As mentioned in the previous section, an experiment on a prototype can, under certain circumstances, yield very precise data, but what if we need to investigate some hard-to-measure quantities or simulate hard-to-duplicate conditions? Then CFD modelling is the right method to employ, since we can evaluate not only temperatures, pressures, or mass flow rates, but also flow field variables (such as turbulent intensity, vorticity magnitude, or velocity angle) anywhere within the

9

investigated geometry. There are two major drawbacks to CFD, though. First, very high computational demand necessitates usage of clusters or grids and even with these computation commonly takes many hours or days to complete. Second, accuracy of results is highly influenced by mesh quality and fineness, used models, solution methods, and other parameters. A coarse mesh results in larger numerical errors, yet using a fine mesh in the entire geometry may lead to unacceptable computational load. Thus the goal is to find a balance between accuracy and computing cost.

2.2.3 Pseudo-1D Discretization of a Flow System

Similarly as with 3D or 2D spatial discretization, we can apply the same principles to create a pseudo-1D mesh. The only difference is that now the mesh only contains nodes and edges – no faces or cells. Such an approach is advantageous for flow systems containing channels with small cross-sections compared to their lengths. Construction of a pseudo-1D mesh is then quite simple. Every channel is replaced by its axis which is discretized with a chosen spatial step as shown in Figure 1. This means that relative spatial disposition of individual channels is retained and, consequently, that changes in gravitational potential energy of fluid particles can be taken into account by any model we decide to use.

This method, however, has a serious downside to it. Due to the very limited description of geometry, we must introduce additional equations governing many phenomena (minor losses etc.) into the model. Also, accuracy tends to be impaired and any model must be fine-tuned using experimental data or at least data from CFD simulations before we can proceed to production use.

2.2.4 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. Nonetheless, once
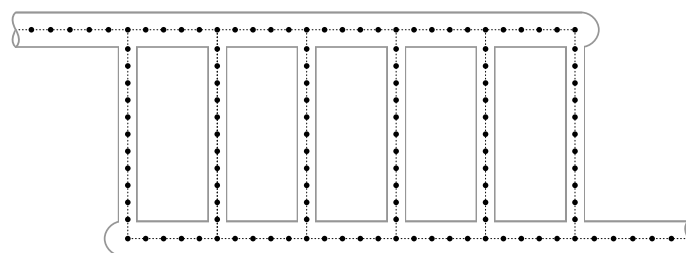

Figure 1. Pseudo-1D mesh of a simple flow system

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.

3   SIMPLIFIED MATHEMATICAL MODELS

Simplified models are well worth the additional effort that must be devoted to fine-tuning them for a specific class of flow systems because then they need substantially less time to yield accurate enough data than other, more complex models. The following section therefore lists some of the tools that have been created by the author. These are all for single-phase flow and range from applications for simulation of plain distribution into a constant-pressure environment through distributor-collector systems with complex tube coils to a shape optimization tool for a specific double U-tube heat exchanger module.

3.1   DISTRIBUTION FROM A MANIFOLD HAVING RECTANGULAR CROSS-SECTION

Since successive branch-by-branch models for distribution from manifolds with constant circular cross-sections are readily available (Acrivos et al., 1959; Bailey, 1975), we will focus on manifolds with variable rectangular cross-sections and double lateral branches (see Figure 2). Such manifolds can perform much better than those with constant cross-sections while manufacturing them is still fairly simple. Please note that any segment of a manifold between two adjacent branches will be called "manifold section" – or simply "section" – from now on.
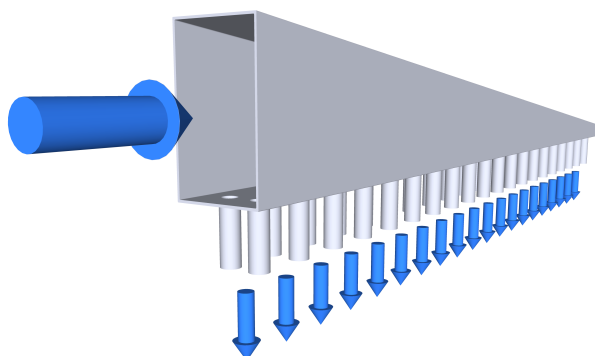

Figure 2. Distributor with variable rectangular cross-section and double lateral branches

We will assume uniform one-dimensional isothermal flow with fluid flowing as indicated by the arrows and cross-sectional areas of individual branches being small enough compared to internal areas of the nearby distributor sections. To get a clear picture of what is happening in the manifold, let us now consider one branch and one subsequent section of the distributor. A portion of fluid is discharged through the branch due to surplus static pressure in the manifold while velocity of the remaining fluid inevitably decreases and thus its momentum changes. This consequently causes increase in pressure in the downstream direction. One way to incorporate this into our model is to use axial and lateral momentum correction factors as e.g. Bajura and Jones (1976) did. This approach, however, has a notable disadvantage, since it employs integral terms depending on an actual velocity profile near the tube entrance. Hence, we will introduce the coefficient of static regain (Bailey, 1975). Moreover, although it may seem so, discharging fluid does not generally lose all its original (axial) velocity and therefore the discharge angle is greater than zero. Considering the law of conservation of mass, the amount of fluid discharging through the branch must correspond to the change in mass flow rate between the section upstream and the section downstream of the branch. However, since the joint of the distributor and the branch is usually made in such a way that it is impossible for the streamlines to suddenly bend along the edge of the entrance, the stream is contracted due to axial momentum of fluid particles. This means that we need to introduce one more correction factor – the discharge coefficient. As for manifold sections, static pressure varies not only due to friction, but also because of changes in gravitational potential energy of fluid particles and changes in cross-sectional area.

3.1.1 Governing Equations

Amounts of fluid discharging through individual branches are given by the variation of static pressure along the distributor and thus finding equations governing static pressure means we can predict the actual discharge flow rates. To do so, we will need four basic equations governing pressure changes due to outflow through branches, friction, changing distributor cross-section, and minor losses. Scheme of the investigated distributor is shown in Figure 3 in which superscript "U" denotes variables just upstream of a branch, "D" variables just downstream of a branch, and "M" variables related to the middle of a section.

We will first deal with pressure variations near branch entrances. Let us assume we already know pressure, velocity, and density just upstream of the ith branch – p_i^{U}, v_i^{U}, and ρ_i^{U}. Pressure change due to change in momentum of fluid particles

Figure 3. Scheme of the investigated distributor

near the branch is governed by the Bernoulli equation for adiabatic compressible flow (Clancy, 1975) modified as shown by Bailey (1975),

$$\left(\frac{\gamma}{\gamma-1}\right)\frac{p_i^U}{\rho_i^U}+C_{r,i}\frac{\left(v_i^U\right)^2}{2}=\left(\frac{\gamma}{\gamma-1}\right)\frac{p_i^D}{\rho_i^D}+C_{r,i}\frac{\left(v_i^D\right)^2}{2},\tag{1}$$

in which $C_{r,i}$ denotes the coefficient of static regain and $\gamma = c_p/c_v$ heat capacity ratio. Value of γ is constant throughout the entire system, because specific heats are temperature-dependent (Coulson and Richardson, 1999, vol. 1, p. 8) and the flow is – per our assumption – isothermal. Since Bailey (1975) successfully validated his model against a series of experiments and the geometries he investigated were relatively similar to our flow system, in this model we use the same formula for coefficient of static regain as he did. Discharge through the ith branch, given by the excess static pressure, can subsequently be calculated using

$$\dot{m}_i^B=b_i h_i\left(\rho_i^U v_i^U-\rho_i^D v_i^D\right)=\frac{\pi d_i^2}{2}C_{d,i}\sqrt{\frac{\left(p_i^U+p_i^D\right)\left(\rho_i^U+\rho_i^D\right)}{2}}.\tag{2}$$

Here, b_i denotes width of the cross-section at the ith branch and $C_{d,i}$ discharge coefficient which, again, is taken from (Bailey, 1975). To calculate mean fluid velocities in the branches, equal static pressures are assumed at tube exits:

$$p_i^B=\frac{p_i^U+p_i^D}{2}-\zeta_i\frac{\rho_i^U+\rho_i^D}{2}\frac{w_i^2}{2}=p^B=\text{const.}\qquad\forall i.\tag{3}$$

Coefficients of hydraulic resistance of branch entrances, ζ_i, depend not only on entrance geometry, but also on fluid velocities just upstream and in the branch (see Idelchik, 1986, Chapter 7).

Now, let us focus on the subsequent distributor section. Pressure loss caused by friction can be modelled using the Darcy-Weisbach equation (White, 1998, p. 340). Considering the fact that changes of cross-section width and height per one section are relatively small, we can approximate hydraulic diameter as well as fluid density and velocity without any significant loss of accuracy by values in the middle of the section and thus obtain

$$\Delta p_i^{\text{fr}} = -\frac{1}{4} f_i l_i \frac{b_i^{\text{M}} + h_i^{\text{M}}}{b_i^{\text{M}} h_i^{\text{M}}} \rho_i^{\text{M}} \left(v_i^{\text{M}} \right)^2 \tag{4}$$

with f_i being Darcy friction factor for the ith section and b_i^{M} width of the respective distributor cross-section. Pressure change caused by change in distributor cross-section is calculated using the original Bernoulli equation for adiabatic compressible flow and the continuity equation. Hence, for ith distributor section we have

$$\left(\frac{\gamma}{\gamma - 1} \right) \frac{p_i^{\text{D}}}{\rho_i^{\text{D}}} + g z_i + \frac{\left(v_i^{\text{D}} \right)^2}{2} = \left(\frac{\gamma}{\gamma - 1} \right) \frac{p_{i+1}^{\text{U}}}{\rho_{i+1}^{\text{U}}} + g z_{i+1} + \frac{\left(v_{i+1}^{\text{U}} \right)^2}{2} \tag{5}$$

and

$$\rho_i^{\text{D}} b_i h_i v_i^{\text{D}} = \rho_{i+1}^{\text{U}} b_{i+1} h_{i+1} v_{i+1}^{\text{U}}. \tag{6}$$

Should the system contain any elements causing minor losses (similarly as in case of branch entrances), we can calculate them in a manner analogous to Equation 3 with coefficients of hydraulic resistance taken for instance from (Idelchik, 1986).

Having obtained the mass flow rates through individual branches, their non-uniformity is used to assess suitability of a given distributor geometry. With pure distribution we do not expect backflow in any of the branches and therefore the percentage

$$\delta = 100 \cdot \left(1 - \min_i \left\{ \dot{m}_i^{\text{B}} \right\} \Big/ \max_i \left\{ \dot{m}_i^{\text{B}} \right\} \right) \tag{7}$$

can be employed. The closer the value of δ to zero, the more uniform the distribution and hence the more suitable the geometry.

As for the model of incompressible flow, the equations it uses are a bit simpler because here the density is constant. Since for low Mach number flows the incompressible approximation is valid even for fluids like air (Acheson, 1990, p. 58), such a simplification may bring substantial shortening of evaluation time. In any case, it is clear that both models require us to either solve several relatively complex

implicit equations or create custom iterative mechanisms. With respect to the fact that solving implicit equations numerically can be quite troublesome and we might want to be able to use different formulae for physical properties of the modelled fluid, it is far better to implement the latter approach.

3.1.2   Simulation Tool: Multi-Platform Java Application

Both the compressible and the incompressible models have been implemented in Java so that one can easily evaluate distributors of the described class (see screenshot of the application in Figure 4). The tool can also be used for distributor shape optimization. Even though the brute-force approach is employed to search given optimization spaces, optimum is usually obtained within a couple of seconds due to the simplicity of the model. As can be seen, every parameter is fully customizable.

Pure distribution, however, is not as common as complete distributor-collector flow systems present in virtually any heat exchanger. This is why we will deal only with such configurations from now on.

3.2   DISTRIBUTOR-COLLECTOR SYSTEM WITH CIRCULAR MANIFOLDS

Now we will describe a pseudo-1D model of a relatively simple distributor-collector system. Such parallel flow systems, as they are sometimes called, often contain manifolds with constant circular cross-sections and are commonly used in heat exchange units. Employing pseudo-1D discretization is beneficial here, because the model remains uncomplicated due to cross-section invariability.
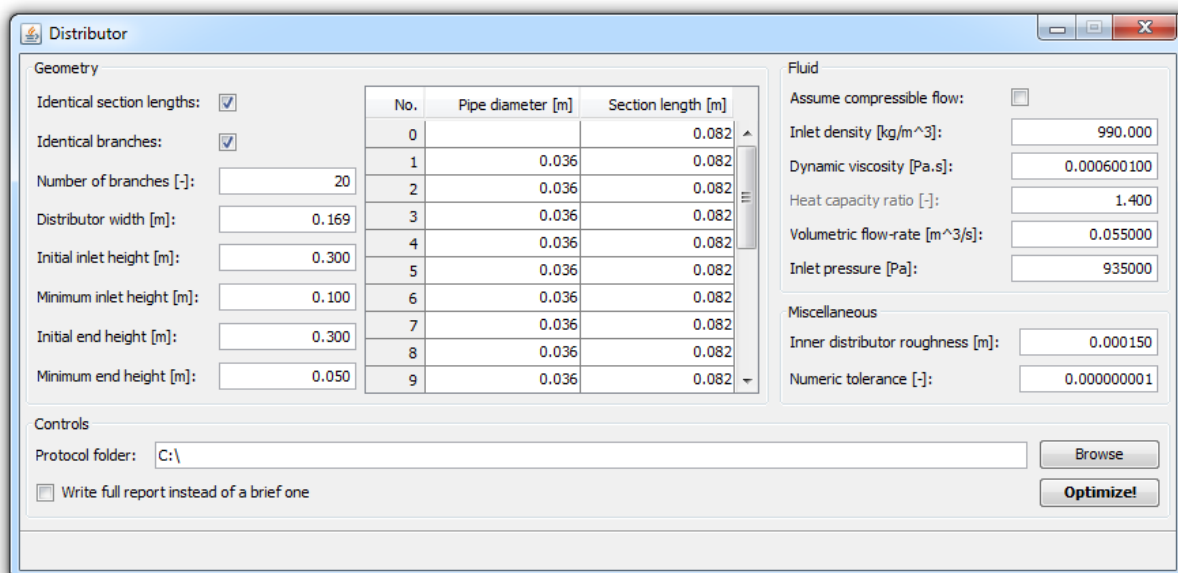

Figure 4. Screenshot of the multi-platform Java application

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:

- quantities are evaluated along the manifolds instead of considering those to be mass points;
- mixing of fluid streams of different temperatures 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 5 shows a sample parallel flow system with moderately complex tube bundle that is easily evaluable using the discussed pseudo-1D model.
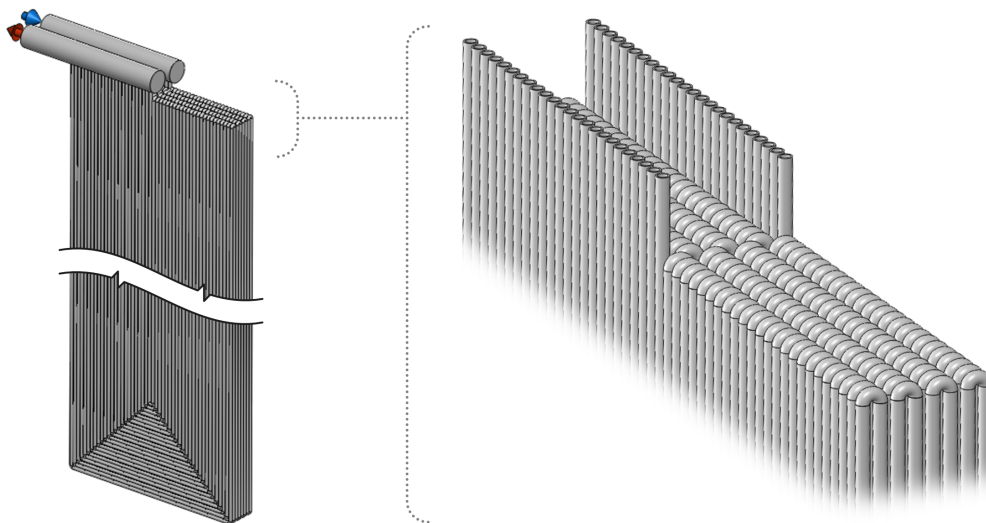

Figure 5. Sample parallel flow system

3.2.1 Governing Equations

For a one-dimensional steady single-phase flow in a channel having constant cross-section, conservation laws can be written as follows (Ngoma and Godard, 2005):

$$\text{mass:} \quad \frac{\partial (\rho v)}{\partial x} = 0, \tag{8}$$

$$\text{momentum:} \quad \frac{\partial (\rho v^2)}{\partial x} + \rho g \sin \beta + \frac{\partial p}{\partial x} + \frac{\partial p}{\partial x}\bigg|_{\text{friction}} = 0, \quad \text{and} \tag{9}$$

$$\text{energy:} \quad \frac{\partial}{\partial x}\left[\rho v \left(H + \frac{v^2}{2} + g x \sin \beta \right) \right] = \dot{q} \frac{U}{S}, \tag{10}$$

where x denotes position along a manifold or a tube, β angle of inclination, H enthalpy, $\dot{q}$ heat flux density, U channel circumference, and S cross-sectional area. The friction term in Equation 9 can be calculated analogously to Equation 4 and minor losses between nodes due to flow through an entrance, exit, or a bend, can be incorporated similarly as in Equation 3. Implementation-wise, this can be easily done by employing an automatically generated indicator function on the set of pseudo-1D mesh nodes for each minor loss type.

Considering control volumes enclosing entrances and exits of lateral tubes through which fluid flows in or out of the manifolds, the equation system cannot be the same, because inflow or outflow causes momentum changes. To factor in the subsequent pressure changes, the present model uses the simplified approach involving the coefficient of static regain (see Section 3.1). The other thing we must take into account is that now streams of different temperatures can mix near each tube end.

3.2.2 Spatial Discretization of the Flow System

Although splitting manifold, combining manifold, and tubes in the tube bundle can be discretized separately, it is beneficial to do it in such a way that key nodes are shared between these subsystems. This means that the inlet node of each tube is identical to a corresponding node in the splitting manifold and, similarly, the outlet node of each tube is identical to a corresponding node in the combining manifold. For numerical reasons, it may also be advantageous to keep spatial step constant if possible, although sharing of key nodes usually does not permit equidistant spacing around tube entrances and exits.

3.2.3 Approximation of Partial Derivatives and Non-Linear Terms

Partial derivatives can be approximated for example using the forward finite difference method (Ames, 1992, p. 16). Should any equation contain non-linear terms, we can either completely rely on an internal solver incorporated in the technical

computing environment of our choice or approximate such non-linearities e.g. by their first order Taylor expansions (Patankar, 1980, p. 49).

3.2.4 Comparison with Other Models

The only models of single-phase flow validated with experimental data that the author was able to find were those assuming adiabatic flow systems. Figure 6 compares data obtained using the present model with predictions of models by Wang and Yu (1989) and Ablanque et al. (2010). It can be seen that agreement among the models is good, especially between the present model and the one by Ablanque et al. (2010).

3.2.5 Simulation Tool: Maple Worksheet

Maplesoft Maple (Maplesoft, 2012) was chosen for computer implementation of the pseudo-1D model for its symbolic computation capabilities. The worksheet is built with maximum automation in mind and thus only the input data, i.e.,

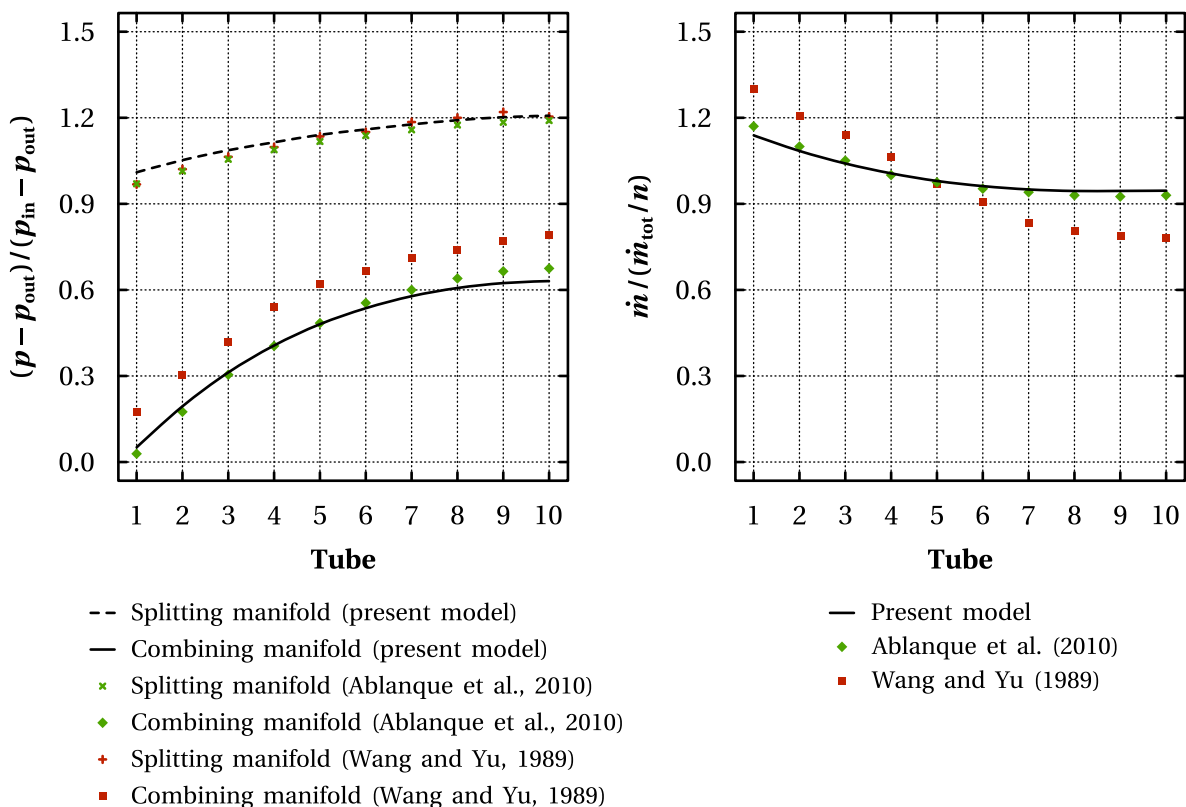- characteristic dimensions of the flow system and function describing geometry of tubes in the tube bundle,

Figure 6. Comparison of results yielded by three different models. Flow system inlet and outlet pressures are denoted p_{in} and p_{out}.

- pressure and temperature at inlet and functions for fluid properties calculation,
- function describing heat flux into individual tubes, and
- general data such as spatial step or numerical tolerance,

must be entered by the user. Spatial discretization of the flow system is performed by an internal algorithm. Due to the fact that no elements other than wyes and bends causing minor losses are assumed to be present, such losses are, too, calculated without user interaction.

The worksheet can be used to optimize distributor and collector diameters. The brute-force approach is employed, but the model is computationally simple enough to swiftly search the optimization space and find the optimum manifold diameter. Suitability of individual geometries must now be assessed via the relative standard deviation from uniform flow distribution,

$$\delta = \frac{100}{\dot{m}_{\mathrm{id}}} \sqrt{\frac{1}{n} \sum_{i=1}^{n} (\dot{m}_i - \dot{m}_{\mathrm{id}})^2}, \tag{11}$$

because backflow can occur in the present model. Here, $\dot{m}_{\mathrm{id}}$ denotes mass flow rate through one tube corresponding to a uniform flow distribution, n number of tubes in the tube bundle, and $\dot{m}_i$ mass flow rate thought the ith tube.

3.3 DOUBLE U-TUBE HEAT EXCHANGER MODULE

The double U-tube heat exchanger module, shown in Figure 7, is a part of a high-temperature heat exchanger used for preheating of fluidizing and combustion air. The exchanger contains several such modules stacked in a vertical hexahedral shell and each of them consists of two stepless manifolds and two sets of U-tubes. These manifolds are, in principle, identical to the distributor discussed in Section 3.1 and the mathematical model is very similar as well. This time, however, we must consider the fact that pressure drops in individual U-tubes are different from each other and that the U-tube outlet pressures must correspond to pressures at branching points in the collector. To increase the accuracy of predictions, the formula for coefficient of static regain was derived by means of approximating data obtained by evaluation of many configurations of such distribution systems using the fluid flow modelling software ANSYS Fluent (Fluent, Inc., 2006). Fine meshes were always generated to ensure data from CFD were accurate enough.

3.3.1 Coefficient of Static Regain

One can predict flow distribution without ever using estimates of coefficients of static regain (see Bajura and Jones, 1976) or, alternatively, calculate their values exactly (see Wang et al., 2001). Nevertheless, the former scenario requires us to
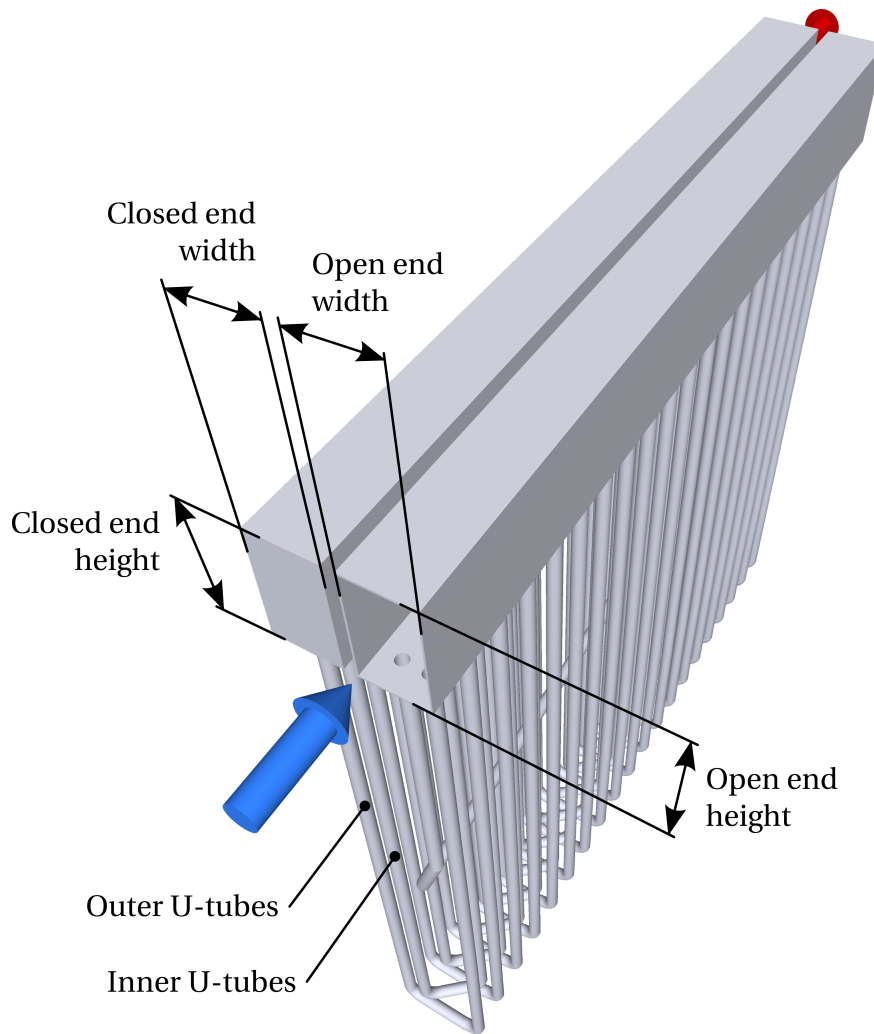
Figure 7. Double U-tube heat exchanger module

solve non-linear partial differential equations with a detailed knowledge of the actual velocity profiles in the ducts being necessary. Similarly, the latter scenario demands solving non-linear ordinary differential equations and then using the calculated coefficients in the usual manner. In both cases it is quite difficult a task even for a geometrically very simple system. On the other hand, Bailey (1975) demonstrated that it is possible to estimate values of the coefficients with sufficient accuracy using a function of hole-to-duct area ratio and velocity ratio. However, this formula might not perform optimally in the studied family of parallel distribution systems and thus a comprehensive set of 282 various geometries was evaluated in ANSYS Fluent to acquire relevant data.

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 %.

3.3.2 Simulation Tool: Multi-Platform Java Application

Computer implementation of the above model for both compressible and incompressible fluids was, again, done in Java to ensure the resulting application can be run on a wide variety of operating systems (see screenshot of the main window in Figure 8). The implemented model does not take gravity into account as its effect is negligible no matter how dense a working fluid flows through the module or how the module is oriented with respect to the Earth's gravitational field. The graph in Figure 9 obtained using the new coefficient of static regain shows mass flow rates through individual U-tubes in a module containing manifolds with linear changes of cross-section widths and heights, open end dimensions 140×110 mm, closed end dimensions 185×500 mm, and all U-tube ends being exserted 12 mm into the manifolds (the total mass flow rate of air was 1 kg/s).

Optimization algorithm employed in this application is a bit more complex compared to the brute force approach implemented in case of pure distribution where only one optimization variable was present. Now we have eight optimization variables – open and closed end widths and heights for both manifolds. Direct
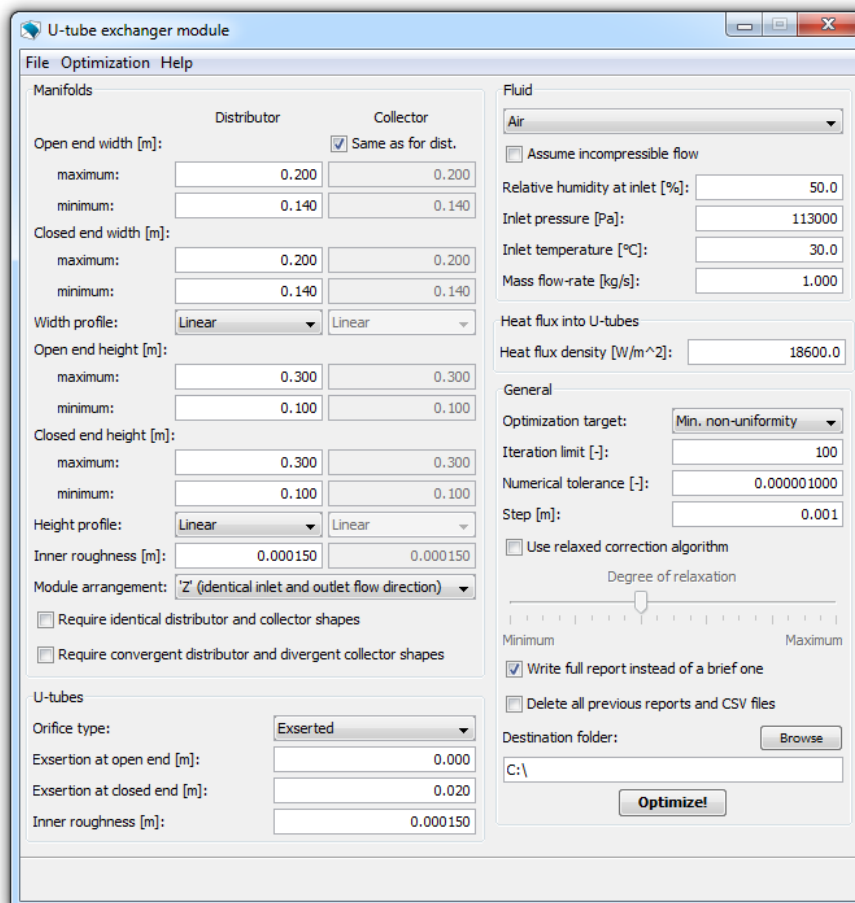

Figure 8. Main window of the application for analysis of flow in the double U-tube heat exchanger module
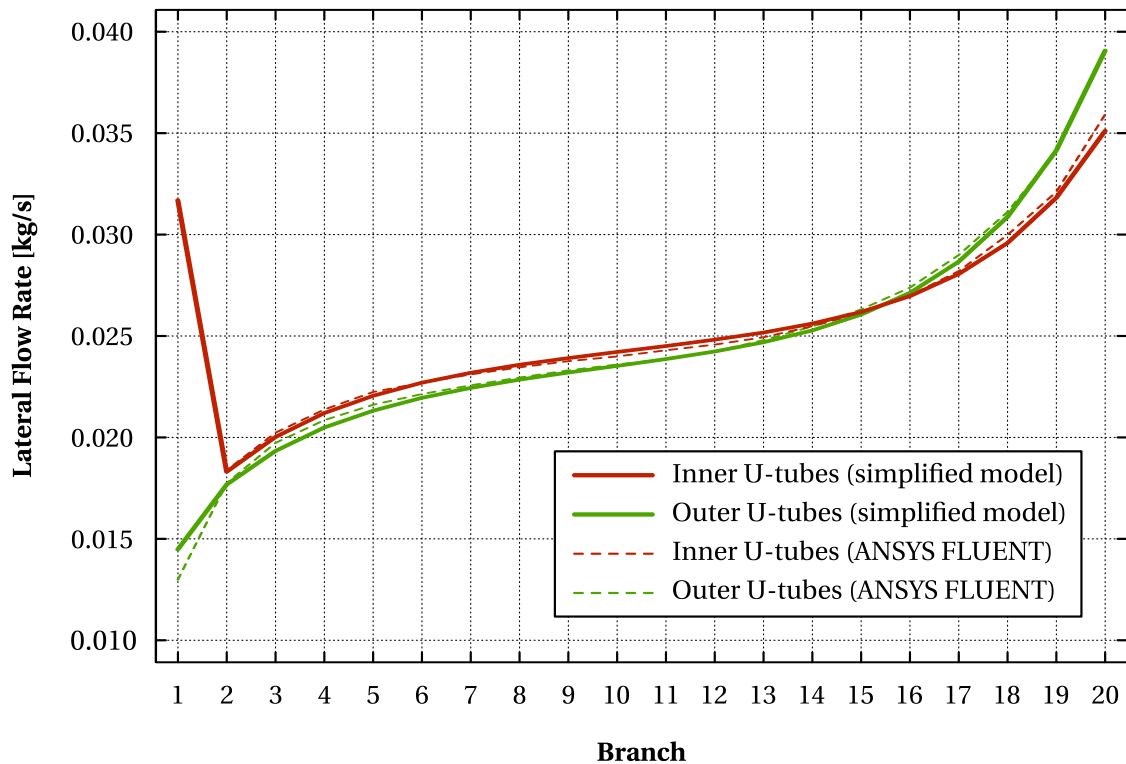
Figure 9. Sample results obtained using the new coefficient of static regain

search methods must be used because calculating or approximating gradients and derivatives would be virtually impossible. Hence, the Hooke and Jeeves method (Ravindran et al., 2006, pp. 92–97) with two additional modifications and the Golden section method (Ravindran et al., 2006, pp. 51–53) are implemented. Although a more robust 2D optimization method could be used, the modified Hooke and Jeeves algorithm was chosen due to its ease of implementation and generally shorter evaluation times (Wetter and Wright, 2004). A higher-dimensional implementation was avoided because then local optima may possibly exist which apparently would be undesirable. This problem, however, must be researched further in order to determine whether the objective function is smooth and monotone in higher-dimensional spaces. If so, then such an approach would bring a substantial decrease in optimization time.

4 APPLICATION OF COMPUTATIONAL FLUID DYNAMICS

Simplified models described in the previous sections can provide excellent results in very short time frames. Yet, when geometry of a flow system is too complex or when we need detailed information regarding flow field variables, these models are inadequate. Then computational fluid dynamics is the right tool for the purpose. In addition, CFD can be used for analysis of fouling. As an example, let us consider an inlet tube sheet in a preheater which is a part of a liquid and gaseous wastes incineration unit. This exchanger preheats process waste gas (PWG) by

high-temperature flue gas, however, there are two major problems. The PWG stream contains a relatively large amount of sticky liquid droplets that we are not able to extract because there is no room for an additional droplet separator in the unit. As a consequence, the inlet tube sheet gets clogged up very rapidly by jelly-like deposits and therefore the preheater's efficiency drops significantly. This then leads to non-uniform thermal expansion of individual tubes in the tube bank and, eventually, it results in mechanical failures of many of the tubes. Vortical character of flow in inlet regions of individual tubes aggravates the issue even more. Figure 10 shows vorticity magnitude above the inlet tube sheet as well as a photograph of the actual fouled tube sheet. The fouling pattern matches vorticity pattern just above the tube sheet quite nicely. Hence, increasing distribution uniformity is crucial here because by this we can in many cases eliminate stagnation zones and thus also considerably lower fouling rate. Nonetheless, this approach to fouling analysis must be researched further before it is ready for production use.

5 SUMMARY

In this thesis 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. As flow analysis is necessary in the process, three simplified mathematical models were presented. Two variants of
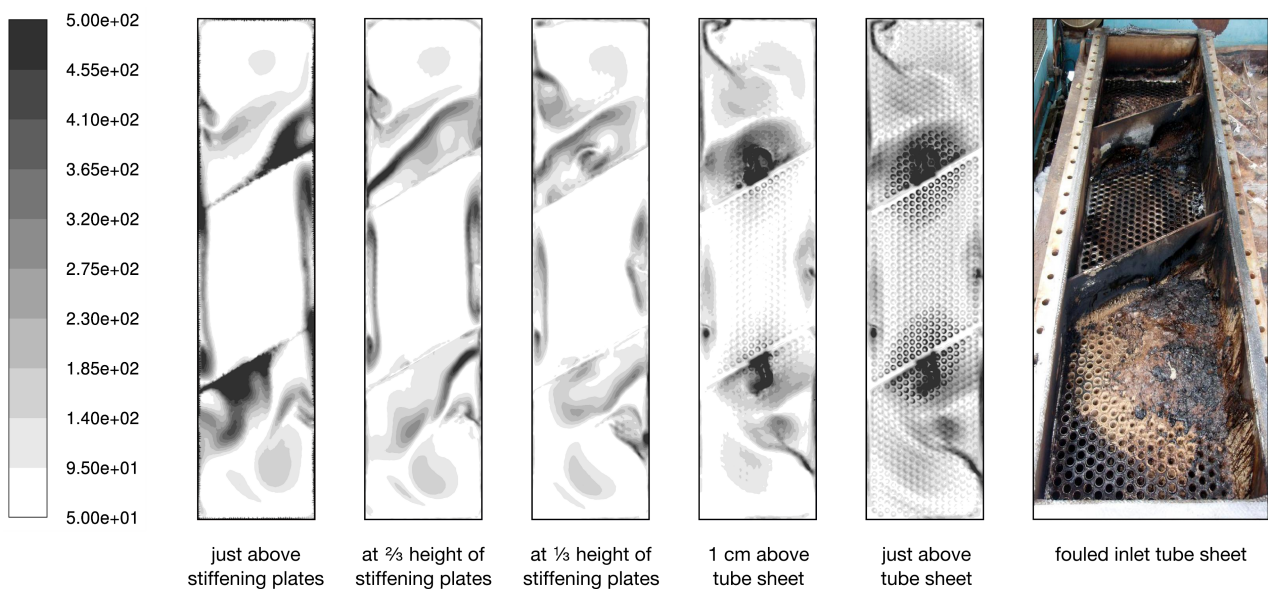

Figure 10. Vorticity magnitude (1/s) in several layers above the inlet tube sheet (left) and a photograph of the actual tube sheet (far right)

each of them exist so that both compressible and incompressible flows can be analysed. The models were built with their use in optimization algorithms in mind, that is, they were made as simple and easy to evaluate as possible while retaining reasonable accuracy of the provided data and applicability to a wide range of flow system geometries.

The first 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. The respective application software can then be run in virtually any modern operating system due to its implementation in Java. Although the brute-force optimization algorithm is employed, results are provided within seconds given the simplicity of the mathematical model.

The second model was built using partial differential equations and works with a pseudo-1D mesh of a parallel flow system. Only circular manifolds with constant cross-sections are supported with respect to the differential nature of the model, since otherwise its complexity would prohibit utilizing it as a core of an optimization algorithm. The advantages, however, are fully automated generation of the mesh and possibility to specify shapes and thermal loads of each of the tubes in the bundle as functions thus rendering the model to be capable of evaluating even relatively complex flow systems. This model has been implemented in Maplesoft Maple and, again, employs the brute-force optimization algorithm. In spite of this, no significant increase in optimization time should be noticeable as only one optimization variable is present.

The last mathematical model describes a parallel flow system consisting of manifolds with variable rectangular cross-sections and a double U-tube bundle. Similarly as in case of pure distribution, the simplified branch-by-branch approach is applied, but here a relatively complex hybrid optimization algorithm is used to shorten optimization times as much as possible because now we search for the optimum in an eight-dimensional space. In order to increase accuracy of the results that the model provides, formula for coefficient of static regain was derived using data obtained by evaluation of 282 geometries, which belonged to the respective class of flow systems, in ANSYS Fluent. As for computer implementation, the corresponding optimization package has been written in Java so that users can benefit from its functionalities without being limited in their choice of operating system or hardware platform.

REFERENCES

Ablanque N., Oliet C., Rigola J., Pérez-Segarra C. and Oliva A. (2010) Two-phase flow distribution in multiple parallel tubes, *International Journal of Thermal Sciences* **49**(6), 909–921.

Acheson D. (1990) *Elementary Fluid Dynamics*, Oxford University Press USA, New York, NY, USA, 3rd edn.

Acrivos A., Babcock B. and Pigford R. (1959) Flow distribution in manifolds, *Chemical Engineering Science* **10**(1), 112–124.

Ames W. (1992) *Numerical methods for partial differential equations*, Academic Press, Inc., San Diego, CA, USA, 3rd edn.

Arsenyeva O., Tovazhnyansky L., Kapustenko P., Perevertaylenko O. and Khavin G. (2011) Investigation of the new corrugation pattern for low pressure plate condensers, *Applied Thermal Engineering* **31**(13), 2146–2152.

Bailey B. (1975) Fluid flow in perforated pipes, *Journal of Mechanical Engineering Science* **17**(6), 338–347.

Bajura R. and Jones E. (1976) Flow distribution manifolds, *Journal of Fluids Engineering* **98**(4), 654–666.

Chandraker D., Maheshwari N., Saha D. and Venkat Raj V. (2002) Experimental and analytical investigations on core flow distribution and pressure distribution in the outlet header of a PHWR, *Experimental Thermal and Fluid Science* **27**(1), 11–24.

Cheng L., Luan T., Du W. and Xu M. (2009) Heat transfer enhancement by flow-induced vibration in heat exchangers, *International Journal of Heat and Mass Transfer* **52**(3–4), 1053–1057.

Cheng Z., He Y. and Cui F. (2012) Numerical study of heat transfer enhancement by unilateral longitudinal vortex generators inside parabolic trough solar receivers, *International Journal of Heat and Mass Transfer* **55**(21–22), 5631–5641.

Clancy L. (1975) *Aerodynamics*, John Wiley & Sons, Inc., Hoboken, NJ, USA.

Coulson J. and Richardson J. (1999) *Chemical Engineering*, Butterworth-Heinemann Ltd., Oxford, UK, 6th edn.

Datta A. and Majumdar A. (1980) Flow distribution in parallel and reverse flow manifolds, *International Journal of Heat and Fluid Flow* **2**(4), 253–262.

Dyga R. and Płaczek M. (2010) Efficiency of heat transfer in heat exchangers with wire mesh packing, *International Journal of Heat and Mass Transfer* **53**(23–24), 5499–5508.

Fernández-Seara J., Diz R. and Uhía F. (2012) Pressure drop and heat transfer characteristics of a titanium brazed plate-fin heat exchanger with offset strip fins, *Applied Thermal Engineering*, **in press**, DOI: 10.1016/j.applthermaleng.2012.08.066.

Fluent, Inc. (2006) *Fluent 6.3 User's Guide*, Fluent, Inc., Lebanon, NH, USA.

Fu H., Watkins A. and Yianneskis M. (1994) The effects of flow split ratio and flow rate in manifolds, *International Journal for Numerical Methods in Fluids* **18**(9), 871–886.

Gandhi M., Ganguli A., Joshi J. and Vijayan P. (2012) CFD simulation for steam distribution in header and tube assemblies, *Chemical Engineering Research and Design* **90**(4), 487–506.

Ghani F., Duke M. and Carson J. (2012) Effect of flow distribution on the photovoltaic performance of a building integrated photovoltaic/thermal (BIPV/T) collector, *Solar Energy* **86**(5), 1518–1530.

Habib M., Ben-Mansour R., Said S., Al-Qahtani M., Al-Bagawi J. and Al-Mansour K. (2009) Evaluation of flow maldistribution in air-cooled heat exchangers, *Computers & Fluids* **38**(3), 677–690.

Hewitt G. (Ed.) (1998) *Heat Exchanger Design Handbook*, Begell House, Inc., Redding, CT, USA.

Houdek P. (2007) Flow instabilities in heat exchangers, Ph.D. thesis, Czech Technical University in Prague.

Idelchik I. (1986) *Handbook of Hydraulic Resistance*, Hemisphere Publishing Corporation, Washington, DC, USA, 2nd edn.

Klemeš J., Bulatov I. and Koppejan J. (2009) Novel energy saving technologies evaluation tool, *Computers & Chemical Engineering* **33**(3), 751–758.

Klemeš J. and Varbanov P. (2012) Heat integration including heat exchangers, combined heat and power, heat pumps, separation processes and process control, *Applied Thermal Engineering* **43**, 1–6.

Kukulka D. and Devgun M. (2007) Fluid temperature and velocity effect on fouling, *Applied Thermal Engineering* **27**(16), 2732–2744.

Kundu B. and Lee K. (2012) Analytic solution for heat transfer of wet fins on account of all nonlinearity effects, *Energy* **41**(1), 354–367.

Laukkanen T., Tveit T. and Fogelholm C. (2012) Simultaneous heat exchanger network synthesis for direct and indirect heat transfer inside and between processes, *Chemical Engineering Research and Design* **90**(9), 1129–1140.

Liang S. and Wong T. (2010) Experimental validation of model predictions on evaporator coils with an emphasis on fin efficiency, *International Journal of Thermal Sciences* **49**(1), 187–195.

Lotfi R., Rashidi A. and Amrollahi A. (2012) Experimental study on the heat transfer enhancement of MWNT-water nanofluid in a shell and tube heat exchanger, *International Communications in Heat and Mass Transfer* **39**(1), 108–111.

Lu F., Luo Y. and Yang S. (2008) Analytical and experimental investigation of flow distribution in manifolds for heat exchangers, *Journal of Hydrodynamics, Ser. B* **20**(2), 179–185.

MAPLESOFT (2012) *Maple User Manual*, Waterloo Maple, Inc., Waterloo, ON, Canada.

MARCHITTO A., FOSSA M. and GUGLIELMINI G. (2012) The effect of the flow direction inside the header on two-phase flow distribution in parallel vertical channels, *Applied Thermal Engineering* **36**, 245–251.

NGOMA G. and GODARD F. (2005) Flow distribution in an eight level channel system, *Applied Thermal Engineering* **25**(5–6), 831–849.

PATANKAR S. (1980) *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation, Washington, DC, USA.

PETHKOOL S., EIAMSA-ARD S., KWANKAOMENG S. and PROMVONGE P. (2011) Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube, *International Communications in Heat and Mass Transfer* **38**(3), 340–347.

RAVINDRAN A., RAGSDELL K. and REKLAITIS G. (2006) *Engineering Optimization: Methods and Applications*, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2nd edn.

SENGUPTA T. and POINSOT T. (Eds.) (2010) *Instabilities of Flows: With and Without Heat Transfer and Chemical Reaction, CISM Courses and Lectures*, vol. 517, Springer-Verlag, Inc., New York, NY, USA.

STEHLÍK P. (2011) Conventional versus specific types of heat exchangers in the case of polluted flue gas as the process fluid – A review, *Applied Thermal Engineering* **31**(1), 1–13.

WANG J., GAO Z., GAN G. and WU D. (2001) Analytical solution of flow coefficients for a uniformly distributed porous channel, *Chemical Engineering Journal* **84**(1), 1–6.

WANG J. and WANG H. (2012) Discrete approach for flow field designs of parallel channel configurations in fuel cells, *International Journal of Hydrogen Energy* **37**(14), 10881–10897.

WANG X. and YU P. (1989) Isothermal flow distribution in header systems, *International Journal of Solar Energy* **7**(3), 159–169.

WANG Y., SMITH R. and KIM J. (2012) Heat exchanger network retrofit optimization involving heat transfer enhancement, *Applied Thermal Engineering* **43**, 7–13.

WETTER M. and WRIGHT J. (2004) A comparison of deterministic and probabilistic optimization algorithms for nonsmooth simulation-based optimization, *Building and Environment* **39**(8), 989–999.

WHITE F. (1998) *Fluid Mechanics*, McGraw-Hill, Inc., New York, NY, USA, 4th edn.

WONGCHAREE K. and EIAMSA-ARD S. (2011) Enhancement of heat transfer using CuO/water nanofluid and twisted tape with alternate axis, *International Communications in Heat and Mass Transfer* **38**(6), 742–748.

ZHANG N., SMITH R., BULATOV I. and KLEMEŠ J. (2012) Sustaining high energy efficiency in existing processes with advanced process integration technology, *Applied Energy*, **in press**, DOI: 10.1016/j.apenergy.2012.02.037.

Curriculum Vitae

Osobní informace

Jméno a příjmení	Vojtěch Turek
Datum narození	1. května 1984
Státní příslušnost	Česká republika

Vzdělání

Období	září 2008 →
Instituce	Vysoké učení technické v Brně, FSI
Studijní obor	doktorský, Konstrukční a procesní inženýrství
Období	září 2007 – červenec 2008
Instituce	Università degli Studi dell'Aquila, Itálie (studium v rámci programu Double Diploma)
Studijní obor	magisterský, Ingegneria dei Modelli Matematici
Období	září 2003 – srpen 2008
Instituce	Vysoké učení technické v Brně, FSI
Studijní obor	magisterský, Matematické inženýrství
Období	září 1999 – květen 2003
Instituce	VOŠ, SPŠ a OA Kopřivnice
Studijní obor	Konstrukce a diagnostika motorových vozidel

Pracovní zkušenosti

Období	duben 2010 →
Zaměstnavatel	Vysoké učení technické v Brně, FSI
Vykonávaná funkce	technický pracovník

Znalosti a dovednosti

Mateřský jazyk	čeština
Další jazyky	angličtina: pokročilý italština: začátečník
Počítačová gramotnost	programování v jazycích C#, Java, Borland Pascal; uživatelská znalost aplikací MathWorks MATLAB, Maplesoft Maple, GAMS, ANSYS Fluent, Autodesk AutoCAD a Microsoft Office

Abstract

Improved heat transfer efficiency leads to decrease in energy consumption which then results in lower equipment operational cost, reduced 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. In addition, accuracy of one of the models was further increased by fine-tuning it using data obtained by evaluation of 282 flow systems in the fluid flow modelling software ANSYS Fluent. The created applications can then be employed during the design of heat exchange units to improve their performance and reliability.

Abstrakt

Zvýšení efektivity výměny tepla vede k poklesu spotřeby energie, což se následně projeví sníženými provozními náklady, poklesem produkce emisí a potažmo také snížením dopadu na životní prostředí. Běžné způsoby zefektivňování přenosu tepla jako např. přidání žeber či vestaveb do trubek ovšem nemusí být vždy vhodné nebo proveditelné – zvláště při rekuperaci tepla z proudů s vysokou zanášivostí. Jelikož intenzita přestupu tepla závisí i na charakteru proudění, distribuci toku a zanášení, které lze všechny výrazně ovlivnit tvarem jednotlivých součástí distribučního systému, bylo sestaveno několik zjednodušených modelů pro rychlou a dostatečně přesnou predikci distribuce a také aplikace pro tvarovou optimalizaci distribučních systémů využívající právě tyto modely. Přesnost jednoho z modelů byla dále zvýšena pomocí dat získaných analýzou 282 distribučních systémů v softwaru ANSYS Fluent. Vytvořené aplikace pak lze využít během návrhu zařízení na výměnu tepla ke zvýšení jejich výkonu a spolehlivosti.