CHAPTER 1

Relevance of heat transfer and heat exchangers for the development of sustainable energy systems

B. Sundén & L. Wang

1Division of Heat Transfer, Department of Energy Sciences, Lund University, Lund, Sweden.
2Siemens Industrial Turbines, Finspong, Sweden.

Abstract

There are many reasons why heat transfer and heat exchangers play a key role in the development of sustainable energy systems as well as in the reduction of emissions and pollutants. In general, all attempts to achieve processes and thermodynamic cycles with high efficiency, low emissions and low costs include heat transfer and heat exchangers to a large extent. It is known that sustainable energy development can be achieved by three parallel approaches: by reducing final energy consumption, by improving overall conversion efficiency and by making use of renewable energy sources. In these three areas, it is important to apply advanced heat transfer and heat exchanger technologies, which are explained extensively in this chapter. In addition, heat transfer and heat exchangers are important in protecting the environment by reducing emissions and pollutants. To illustrate this, several research examples from our group are used to demonstrate why heat transfer and heat exchangers are important in the development of sustainable energy systems. It can be concluded that the attempt to provide efficient, compact and cheap heat transfer methods and heat exchangers is a real challenge for research. To achieve this, both theoretical and experimental investigations must be conducted, and modern techniques must be adopted.

1 Introduction

The concept of sustainable development dates back to several decades, and it was brought to the international agenda by the World Commission on Environment and
Development in 1987 [1]. At the same time, it also provided the henceforth most commonly used definition of sustainable development, describing it as development which meets the needs of the present without compromising the ability of future generations to meet their own needs. This concept has indeed expressed people’s concern about the growing fragility of the earth’s life support systems, i.e. the use of the available resources on our planet. Among the aspects concerned, energy is certainly a very important part, and sustainable energy systems have become the worldwide concern among scientific and political communities as well as among ordinary people.

Today, the production of electricity and heat is mainly based on finite primary energy sources. Fossil fuels are combusted in such large amounts that flue gas emissions have affected the environment, e.g. green house effect and toxic pollutants. A general approach to improve the degree of sustainability of the energy supply lies in the following three aspects: reducing final energy consumption, improving overall conversion efficiency and making use of renewable sources [2]. To reduce final energy consumption is an obvious approach, which requires more energy efficient process components and systems. The energy source requirement for the same energy output can be brought down by improving overall conversion efficiency. To use renewable energy sources other than fossil fuels, such as hydropower, biomass, wind and solar energy, is an attractive approach because they are sustainable in nature.

In all three aspects, it was found that heat transfer and heat exchangers play an important role. For instance, increasing the efficiency in thermal processes for heat and power generation requires increasing the highest temperature in the process and it has to be increased further in the future. To enable the materials of the equipment, e.g. in gas turbine units, to withstand such high temperatures, cooling is needed. In this chapter, several examples will be illustrated to stress the importance of the relevant heat transfer and heat exchangers in the development of sustainable energy systems. Examples will also be given to illustrate that heat transfer and heat exchanger technologies can bring down the emissions of green house gases and other pollutants. It can be concluded that the attempt to provide efficient, compact and cheap heat transfer methods and heat exchangers is a real challenge for research, and that both theoretical and experimental investigations must be carried out and modern scientific techniques must be adopted to develop sustainable energy systems.

2 Reduction of energy consumption

The process industry remains one of the biggest sectors in consuming energy. A typical process, shown in Fig. 1, consists of three parts: chemical plant, utility plant and heat recovery network [3]. The purpose of the chemical plant is to produce products from raw materials with the supply of energy from both the utility plant and the heat recovery network. The utility plant produces power, hot utility and cold utility. The heat recovery network, which consists of many
Relevance of Heat Transfer and Heat Exchangers

Heat exchangers, aims to recover heat from hot streams to heat cold streams. Maximizing heat recovery in the heat recovery network can bring down both energy consumption and consequently flue gas emission from the utility plant. Therefore, reduction in energy consumption requires the optimization of the heat recovery network, i.e. heat exchanger networks. Advanced heat exchanger technologies can improve the efficiency of heat exchanger networks. Such technologies include compact heat exchangers, multi-stream heat exchangers, heat transfer enhancement, mini- and micro-heat exchangers, etc. [4]. Using these technologies, current processes can be improved and the final energy demands can be reduced.

Conventional heat exchangers in process industries are shell-and-tube heat exchangers. There are several disadvantages in using such units, e.g. low ratio of surface to volume, tendency of severe fouling, use of multi-pass design, low efficiency due to a relatively high pressure drop per unit of heat transfer in the shell side, etc. Most of these disadvantages are due to the relatively large hydraulic diameter. To overcome these disadvantages, compact heat exchangers have been developed. A compact heat exchanger is one which incorporates a heat transfer surface with area density (or compactness) of above 700 m²/m³ on at least one of the fluid sides [5]. The common types include plate heat exchangers (PHEs), plate-fin heat exchangers, tube-fin heat exchangers, etc. In the process industries using compact heat exchangers, energy consumption can be reduced in addition to the reduced capital cost and complexity of the plant.

Compact heat exchangers usually have a small hydraulic diameter, which results in high heat transfer coefficients. This will reduce the unit size and weight, hence the unit capital cost. In addition, the high heat transfer coefficients permit compact heat exchangers to operate under conditions with small temperature differences. This is significant in the optimization of heat exchanger networks. In the pinch

![Figure 1: Total processing system.](image-url)
Thermal Engineering in Power Systems

analysis method for the design of heat exchanger networks [6], the minimum temperature difference is the decisive parameter to construct the so-called composite curves, which are shown schematically in Fig. 2. By using compact heat exchangers, the minimum temperature difference can be reduced significantly compared to shell-and-tube heat exchangers. This makes the two lines in the composite curves approach very close to each other, which means that the heat recovery is enlarged, and at the same time, the external utility requirements are reduced. Therefore, the utility consumption in the entire plant is reduced. Due to the high heat transfer coefficients and low unit capital costs, the total capital cost for the heat recovery system can still be lower than that using shell-and-tube heat exchangers.

A multi-stream heat exchanger is a good option, when too many heat exchanger units are required. In the optimization of heat exchanger networks using the pinch technology, a large number of exchangers are often required when the network is designed in terms of two-stream exchangers. This not only increases the capital cost but also increases the complexity of the network. Therefore, it may challenge the optimal solution, and relaxation has to be made. Using multi-stream heat exchangers might be a good way to circumvent this problem, and it offers a number of potential benefits including large savings in capital and installation costs, reduction in physical weight and space, better integration of the process, etc. However, the streams connected to them should not be too far away in physical space to save piping costs. Common multi-stream heat exchangers include multi-stream plate-fin heat exchangers, multi-stream PHEs, etc. [7].

Heat transfer enhancement for shell-and-tube heat exchangers should be also considered in the optimization of heat exchanger networks. It reduces the capital cost because of the small size needed for a given duty. It also reduces the temperature driving force, which reduces the entropy generation and increases the second law efficiency. In addition, heat transfer enhancement enables heat exchangers to

Figure 2: Composite curves.
operate at a smaller velocity but still achieve the same or even higher heat transfer coefficient. This means that a reduction in pressure drop, corresponding to less power utilization, may be achieved. All these advantages have made heat transfer enhancement technology attractive in heat exchanger applications. For the tube side, different geometries (e.g. low-finned tubes, twisted tubes, grooved tubes) and tube inserts (e.g. twist taped inserts, wire coil inserts, extended surface inserts) have been developed [8]. For the shell side, improvements have been also made, e.g. helical baffles and twisted tube heat exchangers [4].

More heat transfer and heat exchanger technologies are available to improve the process, and consequently to reduce the final energy consumption. These may include micro- and mini-heat exchangers, integrated chemical reactor heat exchangers, etc. Due to the space constraints in this chapter, these technologies are not explored in detail. However, the possibilities of their application in process industries should not be underestimated.

3 Improved efficiency of energy conversion

There are many ways to improve the efficiency of thermal power plants, but heat transfer and heat exchangers play a significant role in all means. This can be highlighted by considering as an example a power plant that uses gas turbines. The original Brayton cycle for the power plant only needs a compressor, a combustion chamber and a power turbine; this concept can be found in any textbook on thermodynamics, e.g. Cengel and Boles [9]. However, the thermal efficiency is usually very low in such systems, and improvements can be made by employing the concept of intercooling, recuperation (regeneration) and reheating. Such a flow sheet is illustrated in Fig. 3, and the corresponding thermodynamic cycle is shown in Fig. 4. Two stages of gas compression are provided to reduce the power consumption for compression due to the lower inlet temperature of the gas in the second compression stage by using an intercooler. Because the compression power required is reduced, the net power output is certainly increased. The concept of recuperation is the utilization of energy in the turbine exhaust gases to heat the air entering the combustion chamber, thus saving a certain amount of fuel in the combustion process. This will certainly increase the overall thermal efficiency as well. In addition, the turbine output can be increased by dividing the expansion into two or more stages, and reheating the gas to the maximum permissible temperature between the stages. Although the power output is improved, the cost of additional fuel will be heavy unless a heat exchanger is also employed. These concepts can be also seen in the thermodynamic cycle in Fig. 4. The cycle 1-2-3-4-1 corresponds to the simple Brayton cycle. The cycle 9-11-12-2 represents the intercooling and the cycle 15-14-13-4 represents the reheating. The cycles 4-7-12-5 and 4-6-2-5 represent recuperation in the case of intercooling and no intercooling, respectively. This concept has already been incorporated in some real gas turbines, e.g. LMS100 from GE makes use of an intercooler, Mercury 50 from Solar Turbine makes use of a recuperator and GT24/26 from ASL TOM uses sequential combustion. These features significantly increase the efficiency of gas turbines, and a great deal of work has been done for the design of reliable heat exchangers that are operated at higher temperatures.
Thermal Engineering in Power Systems

The thermal efficiency and power output of gas turbines will increase with increasing turbine rotor inlet temperature, which corresponds to the temperature at point 3 in Fig. 4. This is the reason why modern advanced gas turbine engines operate at high temperatures (ISO turbine inlet temperature in the range of 1200–1400°C), and the trend is to operate at even higher temperatures. To enable this, in addition to material innovation cooling technologies must be developed for the combustion chamber, turbine blade, guide vane, etc. Over the years, film cooling, convection cooling and impingement cooling have been developed for both combustion chamber (see Fig. 5) and turbine blade (see Fig. 6), and the technique of transpiration cooling is still under development due to engineering difficulties. In addition, more advanced high temperature materials such as superalloys of single crystals and ceramic coating significantly contribute to the high turbine inlet temperature operation. With these advanced cooling technologies, reliable and high-efficiency power plants can be sustained.

The above blade cooling technologies are for air-cooled gas turbines, but a new turbine cooling concept is available, i.e. steam cooled gas turbines. Steam provides several benefits over air as a cooling medium. First, steam provides higher heat...
Relevance of Heat Transfer and Heat Exchangers

Transfer characteristics because its heat capacity is higher than that of air. Second, the use of steam as a cooling medium reduces the use of cooling air, which means that more cooling air is available for the combustion process, which contributes to improving emissions. Third, reduction in cooling air results in less temperature dilution of the hot gas caused, while mixing with the cooling air. This increases the turbine inlet temperature, which results in more power availability. Finally, no ejection of cooling air to the main gas flow means aerodynamic loss is minimized. With this technology, the efficiency of the gas turbine is greatly enhanced; the best example of this is GE’s H class gas turbine, which is the first gas turbine to achieve 60% efficiency in the combined cycle power plants. However, to design such turbines, the heat transfer characteristics of steam as a cooling medium must be thoroughly understood, which requires extensive research. For the gas side, the use of different fuels can lead to a significant change in properties for the gas in the turbine part. The Integrated Gasification Combined Cycle application operates on hydrogen, and consequently the syngas will increase the amount of heat transferred to rotating and stationary airfoils due to increased moisture content and

Figure 5: Cooling concepts of combustion chamber: (a) Film cooling; (b) Transpiration cooling; (c) Enhanced convective cooling; (d) Impingement cooling.

Figure 6: Cooling concepts of gas turbine blade: (a) convection cooling; (b) impingement cooling; (c) film cooling; (d) transpiration cooling.
mass flow. Thus, research will provide a better understanding of heat transfer mechanisms in a syngas environment.

Another way to improve energy conversion efficiency is to use combined cycles incorporating steam turbines, fuel cells, etc. A combined cycle with steam turbines is a relatively old but still very effective approach, and heat transfer and heat exchangers play a significant role in this approach without any doubt. Here, a brief discussion is given for the heat transfer issues associated with fuel cells. Figure 7 shows a typical configuration for a combined cycle using both a gas turbine and a fuel cell. As is well known, fuel cells can convert the chemical energy stored in the fuel into electrical and thermal energy through electrochemical processes. Because these processes are not subject to the Carnot cycle limitation, high electrical efficiencies can be obtained. Typical fuel cell types include phosphoric acid fuel cells, proton exchange membrane, solid oxide fuel cell and molten carbonate fuel cell, etc. [10].

The operation principle indicates that heat and mass transfer play an important role in fuel cells [10]. One typical fuel cell construction is the flat plate design for solid oxide fuel cells, shown in Fig. 8. As can be seen, the fuel in fuel ducts has both heat and mass transfer on the top wall with the anode, and the air in air ducts has both heat and mass transfer on the bottom wall with the cathode. In addition, two-phase flows exist in fuel ducts after a part of the fuel is consumed. Therefore, the conditions of fluid flow and heat transfer in air and fuel ducts have great effects on the performance of fuel cells and consequently the entire power cycle. Most of the current designs are based on constant values of Nusselt number and friction factor. Such rough estimations cannot meet future developments, and considerable research efforts must be given to this complex heat and mass transfer problem.

Figure 7: Reference fuel cell and gas turbine system layout [11].
The above analysis demonstrates that high efficiency of power conversion can be reached with the help of relevant heat transfer and heat exchanger technologies. Therefore, attempts to provide compact, efficient heat transfer methods and heat exchangers and at the same time allowing a cheap and relatively simple manufacturing technique are real challenges for research.

4 Use of renewable energy

Hydropower, biomass, wind and solar energy are regarded the most important renewable and sustainable energy sources. Hydropower is, of course, dependent on the earth’s contour, and it is not substantial for those countries with flat earth surface. Biomass appears to be an attractive option for many countries, and technologies for the conversion of biomass into electricity and heat are highly similar to the technologies for other solid fossil fuels. Wind and solar energies are strongly fluctuating sources, but they are very clean, with no pollutant emissions and have received great attention. In these renewable energy systems, heat transfer and heat exchangers play an important role as in those systems described earlier.

Consider now a simple solar energy system as an example. Figure 9 is a schematic view of a typical domestic hot water heating system designed for residential applications. When there is sun, the photovoltaic (PV) module produces power, which runs a small circulating pump. Antifreeze is pumped through the solar collectors and is heated. The fluid then returns to a reservoir in the heat exchanger module. Water coils in the reservoir absorb the heat from the solar fluid. The domestic water flows through these heat exchanger coils by natural thermosiphon action. As the water is heated, it rises and returns to the top of the tank, drawing cold water from the bottom of the tank into the heat exchanger.

It should be pointed out that no external heat exchanger as shown in Fig. 9 was used historically. Instead, the heat exchangers were coils of copper pipes located at the bottom of the solar storage tank. The current design shown in Fig. 9 has a number of advantages. First, the system performance is enhanced. External heat
exchangers can be configured so that the potable water circulates by natural convection (i.e. it thermostombs), which means that excellent temperature stratification can be achieved in the storage tank. With the hot water remaining at the top of the tank, usable hot water is available more rapidly with an external heat exchanger. Second, the thermodynamic efficiency is improved with the external heat exchanger configuration. The rate of heat transfer is directly proportional to the difference in temperature between the water being heated and the antifreeze from the solar collectors. With the external heat exchanger configuration, the heat exchanger coil is always surrounded by the very cold water, which means that thermal efficiency is greatly improved. Third, low cost can be achieved due to the long lifetime of the external heat exchanger compared to the solar tank. The external heat exchanger can be saved when the solar tank develops a leak, and thus cost saving is achieved. However, the heat transfer mechanism involved in the external heat exchanger is highly complex. Both forced convection and natural convection have important impacts. Shell-and-tube heat exchangers may serve well in this condition, but compact heat exchangers (such as PHEs) also claim superior operating condition. Because this practical application is still in its infancy, more research is expected in the future.

In addition, the solar collector using the PV module is a special heat exchanger. On one side of the surface, solar energy (radiant energy) is absorbed. This energy is transferred to the second side of the coolant. This is a quite complex heat transfer problem, not only because it involves both the radiant and the convection heat transfer but also because it is a time-dependent issue. The solar energy varies with time and location, and this must be taken into account in the use of this renewable energy.

The importance of heat transfer and heat exchangers has been illustrated for the solar energy system. Similar conclusions can be reached when dealing with the other types of renewable energy systems. However, they are not fully exploited here due to space constraints.
5 Reduction of emission and pollutant

Heat transfer and heat exchangers are also important in reducing emissions and pollutants. As illustrated earlier, they play an important role in the development of sustainable energy systems. The reduction of final energy consumption means less prime energy (e.g. fossil fuels) consumption, which results in overall reduction in emissions and pollutants. Improved efficiency of power plants certainly also reduces the primary energy consumption as well as the consequent emissions. Alternative fuels like biofuels (including biomass and waste utilization) are said to be neutral in terms of CO$_2$. The other renewable energy sources – solar, hydropower and wind – simply are clean enough and no emissions exist at all. In addition, by considering the pressure drop and associated pressure losses (work loss) in the heat transfer processes and attempting to reduce it, the consumption of electricity will be decreased, which is also beneficial. Therefore, heat transfer and heat exchangers are important for the protection of the environment, with regard to their role in the development of sustainable energy system.

The above influences on emissions and pollutants are obviously the indirect effect. However, heat transfer and heat exchangers can also have a direct effect on reducing emissions and pollutants in many situations. One example is their presence in internal combustion engines. In diesel combustion engines, exhaust gas recycling (EGR) was used for a while because this has been found to be an efficient method to reduce NO$_x$. However, particle emissions are increased and the engine performance is reduced. It has been recognized that if the exhaust gas is cooled in a heat exchanger, the above-mentioned problems can be overcome or at least partially avoided. In addition, the NO$_x$ emission will be further reduced as shown in Fig. 10. In this situation, several factors must be considered. First, due to the limited space in automobiles, an EGR cooler must be compact and lightweight. Second, because the cooling water is taken from the total engine cooling water, the amount of cooling water for the EGR cooler is limited and must be kept as small as possible. This means that the EGR cooler must have high thermal efficiency. Third, the EGR cooler is always subject to unsteady or oscillatory operation and is also severely affected by fouling, which means that the operating reliability and lifetime are extremely important in selecting the heat exchanger type. Therefore, a compact heat exchanger (e.g. a brazed plate heat exchanger) may be a better option, although shell-and-tube heat exchangers are currently often used in automobiles. To design an EGR cooler giving very reliable performance and durability, further research must be carried out.

Another example is the combustion chamber in gas turbine systems. It is well known that the production of NO$_x$ is related to high flame temperatures. One way to reduce the flame temperature is to use high air to fuel ratios [13]. This means that much more compressor air is needed for combustion and consequently less air is available for the cooling of combustion chambers and turbine blades. However, low temperature zones lead to unburned hydrocarbons. Thus, the emission control and the cooling system are coupled and need careful attention. This is another evidence that heat transfer design has a direct effect on reducing emissions and pollutants.
6 Some examples of recent research

6.1 Case study of a heat exchanger network design using the pinch technology

A heat recovery system at a Swedish pulp mill has been investigated. At the mill, there is a big amount of hot water and thin liquor coming from the washing and bleaching process. These hot streams exchange heat with some cold streams, which will be used in the digesting plant. Since the hot streams labelled 2 and 3 contain a small content of fibres and some other substances, fouling may occur quite easily. Therefore, the process is very appropriate for PHEs because of the characteristic of easy cleaning. Specially designed PHEs, called wide gap PHEs, are used for the streams 2 and 3.

The network investigated contains three hot streams and five cold streams. The existing network is presented in grid form in Fig. 11. All the existing heat exchangers are PHEs and the total heat transfer area is 1436.5 m². The heat capacity flow rate, supply and target temperatures, physical properties and allowable pressure drop of each stream are given in Table 1. It should be pointed out that the allowable pressure drops are treated as the pressure drops to promote
Relevance of Heat Transfer and Heat Exchangers

6.1.1 Grassroots design

The composite curves are plotted in Fig. 12 for DTMIN 6°C.

The optimal hot and cold utilities as well as the estimated total heat transfer area are calculated. The optimal hot and cold utility requirements are 1788 and 6800 kW, respectively. By comparing these figures with those in Fig. 11, it is obvious that the hot and cold utility consumption in the existing network could be reduced by 44.5% and 17.4%, respectively.

For the total heat transfer area, the estimation is carried out based on the proposed method. The total heat transfer area is a function of both DTMIN and the
Thermal Engineering in Power Systems

It is obvious that there is an optimal corrugation angle corresponding to the minimum area after DTMIN is specified. In addition, it is easy to understand that the heat transfer area becomes larger for lower DTMIN for the same corrugation angle. For DTMIN = 6°C, the minimum total heat transfer area is 1095 m², and the optimal corrugation angle is about 60°. This value is much lower than the existing exchanger area. This is probably due to the fact that the existing exchangers do not fully use the allowable pressure drop. In addition, a small part of the exchangers have lower corrugation angles, which also increases the total heat transfer area. In the calculation, the fouling resistances for stream 1, streams 2 and 3, and the rest are taken as 0.0001, 0.0003 and 0.00008 m²·K/W, respectively. The hydraulic diameters for the wide gap PHEs and the normal PHEs are 0.022 and 0.008 m, respectively.
The variation in DTMIN causes variations in utility consumption, heat transfer area and most likely in the structure of the network. The variation in the corrugation angle causes variation in heat transfer area. Therefore, the optimal DTMIN and corrugation angle should be determined before any network generation. The annual costs for energy and exchanger area can be estimated by the following relationship:

\[
\text{Capital cost} = 2700 \cdot \text{Area}^{0.85}
\]

\[
\text{Energy cost} = 1400 \cdot \text{Hot utility} + 400 \cdot \text{Cold utility}
\]

The estimation for the capital cost is based on some experience of a PHE manufacturer and the estimation for the energy cost is provided by the staff at the mill. The hot utility is a live steam, and the cold utility is the normal cold water. The units for cost, area and utility are Swedish Crown (SEK), m² and kW. Now, it is possible to plot a graph of the total annualized cost versus DTMIN and the corrugation angle. The plot is given in Fig. 14, and the optimal DTMIN and corrugation angle are close to 1°C and 62°, respectively. The optimal DTMIN is quite small because the energy cost is the dominant part in the total cost.

The Pinch design suggested by Linnhoff et al. [6] is employed to design the network. The minimum DTMIN is taken as 6°C after considering the minimum temperature difference for PHEs. The optimal corrugation angle for this value of DTMIN is close to 60°. The final design is shown in Fig. 15. After the detailed calculation is carried out, the total heat transfer area is 1247 m². The deviation between predicted and calculated values is about 12%. Considering the fact that the vertical alignment is assumed in the prediction while it is actually not in the network synthesis, this deviation is acceptable for the pre-optimization. Hence, it demonstrates that the suggested method is suitable for the optimization of heat exchanger networks using PHEs.

As for the potential use of multi-stream PHEs, the heat exchangers 2 and 4, 3 and 5, and 6 and 7 are likely to be combined as three-stream PHEs. By doing so,
the capital cost and installation cost are greatly reduced. The process is also made more integrated. The operability concern can be solved by several ways suggested above. Although the detailed calculation is not carried out here, the potential for use of multi-stream PHEs is obvious.

6.1.2 Retrofit design

The reason for the excessive consumption of both hot and cold utility is that there is heat transfer (exchange 7 in Fig. 11) across the pinch point. To reduce the utility cost to the optimal level, this cross-pinch heat transfer must be eliminated. The suggested retrofit design is shown in Fig. 16.

As can be seen, the exchanger 7 has been moved to another place and some of the plates are removed. In addition, two new exchangers 8 and 9 are added. By doing so, both the hot and cold utility consumption are reduced. The two new heat exchangers 8 and 9 have heat transfer areas of 122 and 82 m$^2$, respectively. The investment for the new exchangers is 0.248 MSEK, and the payback period is only about 12.5 months. The payback period is very short because the running cost is much higher than the capital cost in this case. It also demonstrates why process integration is so important in industry where energy cost is high.

However, the utility consumption is still not the minimum value because there is still a small amount of heat transferred across the pinch in exchanger 9. This can be eliminated if the low-temperature end of stream 8 is heated by stream 2. However, the energy reduction is small, whereas the cost of the exchanger, connecting pipe and others is quite high. The structure of the network also becomes more complicated, which is not good for operability. Hence, the payback period will be rather long.
6.2 High temperature heat exchangers

High temperature heat exchanger technology has become important for improving the performance of power generation. There is a need to develop various types of high temperature heat exchangers in different applications such as hydrogen production, reforming process of solid oxide fuel cells, generation of high temperature gas, low emission power plants. In this section, monolithic heat exchangers are considered and some specific problems are addressed.

6.2.1 Monolithic exchangers

6.2.1.1 Ceramic monolith structures

Ceramic monolith structures are used in the industry today and they are produced in large numbers by using the extrusion technique. They are unibody structures composed of interconnected repeating cells or channels (Figs. 17 and 18). They are increasingly under development and evaluation for many new reactor applications [14, 15], e.g. chemical process and refining industries, catalytic combustion, low emission power plants. However, monoliths are mainly used where only one fluid flows through all the channels. An example is the monolithic exhaust structure in automotive applications. In endothermic and slow reactions such as steam reforming of hydrocarbons, large amounts of heat are needed to maintain reaction rates. If the catalysts were deposited on tubes, usage of monoliths would be more efficient, leading to greater reaction rates and a smaller reactor [16]. Additionally, there would be a great improvement in mechanical integrity. Especially, it would be advantageous if two fluids in monolithic channels can exchange heat and/or mass. The reason why monoliths are not widely used in these applications is because of complex technique for feeding and distributing the two fluids in and out of the channels.

Figure 16: Grid structure of the retrofit design.
Selimovic and Sunden [17] focused on the compact ceramic heat exchanger where two fluids are fed and distributed into individual channels in a multi-channels structure. Their study shows three different approaches of modelling: analytical, experimental and numerical modelling. The exchanger is of monolithic shape where heat and mass is transferred in rectangular channels. Usually, for the pressure drop calculations of standard channel shapes, different available correlations can be applied. However, when these channels are attached to a manifolded and connected to other components, complex geometries are involved and then modelling with correlation parameters may be unsuccessful. Similar to PHEs, the pressure drop, as well as thermal performance, depends on distribution of fluid. Therefore, it is important to investigate how good the flow distribution is from the main port pipe into the channels. The analytical investigation made here includes both U- and Z-type configurations.

Monolithic ‘honeycomb’ structure has been manifolded by two stage manifolds where either U-type or Z-type manifolds can be used to distribute the flow rate uniformly through each branch. This stage manifold can be compared to the manifolding of PHEs. The main difference compared to PHEs is that each branch will further divide the flow to the monolithic structure with specified channel arrangement. This stage manifolding is called I-type manifold here. More detailed picture of I-type manifold can be observed in Fig. 19. Concerning the monolithic
channels, two different gas distributions (channel arrangements) are investigated: the checkerboard and linear (Figs. 18 and 19). The important physical characteristics are then the size of the channel through which the gaseous reactants and products traverse wall thickness, and the total monolith’s compactness.

Rafidi and Blasiak [18] developed a two-dimensional simulation model to find out the temperature distribution of the solid storing material and flowing gases and other thermal and flow parameters for this regenerator and compared the computed results with experiments. Because of geometric symmetry of the honeycomb

Figure 19: I-type manifold assembly: 1 and 2 – manifold top, 3 – dividing plates, 4 – monolithic channels, 5 – collecting plate, 6 and 7 – manifold bottom, 8 – checkerboard channel arrangement.
structure, mathematical analysis was made on one honeycomb cell, or matrix, that formed a small part of the regenerator cross-section along the flow path. The regenerator is composed of two different materials along the heat exchangers, one is 0.2 m long alumina and the other is 0.1 m long cordierite.

Figure 19 shows the dimensions of a heat regenerator used in a twin-type 100 kW HiTAC (high temperature air combustion) regenerative burner. The regenerator dimensions are 150 × 150 × 300 mm$^3$. The cell size is attributed to 100 cells/in$^2$ and hence, the specific heat transfer area is 4200 m$^2$/m$^3$. All flue gases generated by combustion are sucked again by the burners and pass through the regenerators. The honeycomb compact heat regenerator has relatively high effectiveness of about 88% and recovers 72% of energy contained in combustion flue gases at nominal operating conditions. Consequently, the energy storage and the pressure drop are calculated and the thermal performance of the honeycomb heat regenerator is evaluated at different switching times and loading. The model takes into account the thermal conductivity parallel and perpendicular to flow direction of solid and flowing gases. It considers the variation in all thermal properties of solid materials and gases with temperature. Moreover, the radiation from combustion flue gases to the storage materials was considered in the analysis.

6.3 Heat load prediction in combustors

Different phenomena such as complex flow field and heat release by combustion are involved in the heat transfer process in combustion chambers. This section concerns prediction of heat load and wall temperature in a gas turbine combustor by taking different phenomena into account. Two-dimensional axisymmetric models were used to model the flow field and combustion in a premixed combustor with two different cooling schemes. The $k$–$\epsilon$ turbulence model and Eddy Dissipation Concept were used for modelling turbulent flow and combustion, respectively. In the modelling of heat transfer through the walls, a conjugate heat transfer formulation was applied. The temperatures calculated by the models were compared with experimental data. The results showed that in the mid part of the liner, the prediction of the wall temperature is good, although worse agreement is found in other parts. In addition, radiative heat transfer has been studied. The results showed that radiative heat transfer in simple and ribbed duct cooling schemes can increase the average inner wall temperature up to 33 and 40 K, respectively.

Here computational fluid dynamics (CFD) simulations are used to, first, predict the temperature and heat transfer rate to the combustor wall (called liner wall hereafter) by using a conjugate heat transfer method and, second, study quantities of convective and radiative heat transfer in this type of combustor. The analysis is carried out on a VT4400 LPP combustor developed by Volvo Aero Corporation. A slightly simplified geometry is used to simulate this combustor and some experimental data of inner and outer liner wall temperatures were provided to validate the simulation results.

6.3.1 Combustor description and its modelling

The VT4400 LPP is a lean premixed combustor, which is fuelled by natural gas. In the case of measured data, the equivalence ratio has been set to 0.59. The supplied
Relevance of Heat Transfer and Heat Exchangers

Air from the compressor is divided into two parts. The primary air, after passing through a cooling duct, enters the swirl system and mixes with the natural gas and is then burnt. The height of the cooling duct is 8 mm. The primary air and swirl number are about 1.57 kg/s and 0.6, respectively. By using the geometrical data and definition of swirl number (see eqn (1)), the axial and tangential velocities at the inlet of the combustor can be set.

\[
S_n = \frac{R_2^2}{R_2^2} \int_0^{R_1} \frac{U_z U_0 r^2 dr}{R(a)} \int_0^{R_1} \frac{U_z^2 r dr}{R_2^2} \tag{1}
\]

At the second inlet, the secondary air is mixed with the burnt gases before the entrance to the turbine. In the experiments, the combustor was equipped with two different cooling schemes; a simple duct and a ribbed duct with thermal barrier coating (TBC) on the inner side of the liner wall.

The thickness of the liner wall is 1.5 mm and its thermal conductivity is about 25 W/m K. In the second scheme, a TBC layer with thermal conductivity 1.3 W/m K has been used. The inlet temperature from the compressor is 662 K and according to the experiment this temperature is increased by 48 K at the outlet of the channel. The described combustor was modelled by a two-dimensional geometry (see Fig. 20). The model was meshed by two-dimensional (three-dimensional with one cell thickness) multi-block axisymmetric grids. A grid dependence study was carried out in the simple cooling duct case and 42,580 cells showed satisfactory accuracy. Then this meshed model was improved for the ribbed duct and TBC case and the number of cells reached 70,090. To capture the temperature distribution, the liner has been divided into 10 cell thickness. For boundary conditions, inlet and pressure boundaries were used for inlet and outlet, periodic and symmetry boundaries were used for the \( r-z \) faces in the liner and cooling duct, respectively.

6.3.2 Governing equations and solution methods

To model the flow field, continuity and Navier–Stokes equations were solved. The turbulence was modelled by solving the transport equations for the turbulent kinetic energy and turbulent dissipation, which are implemented in the standard \( k-\varepsilon \) model. The summarized governing equations are listed in Table 2.

![Figure 20: The combustor model for the case of ribbed duct.](image-url)
Thermal Engineering in Power Systems

The premixed turbulent combustion was modelled by a one-step reaction for burning methane. The reaction rate was approximated by using the Eddy Dissipation Concept [19] and implemented in the source term of the species transport equation. According to this model, the reaction rate is given by:

$$a_{re} = \frac{-\dot{\nu}}{\text{min}}$$

(8)

where $a_{re}$ is the reaction rate, $\dot{\nu}$ is the reaction time, and min is the minimum of the reaction rates. The solution domain was discretized by the finite volume method and the STAR-CD CFD code was used for all computational processes. The convective terms of the transport equations were handled by a second order scheme (MARS) [20] and the SIMPLE algorithm [21] was used for the pressure-velocity coupling. Convergence criteria for solution of all equations were set on $1.0 \times 10^{-4}$ and besides that the temperature data at some boundaries were controlled.

6.3.2.1 Heat transfer

Heat transfer through the liner wall was modelled by a conjugate heat transfer formulation, whereas other walls were modelled by their thermal resistance and environment temperature. Convective heat transfer through the liner wall was modelled by a conjugate heat transfer formulation, whereas other walls were modelled by their thermal resistance and environment temperature.
on both hot and cold sides of the liner was modelled by the standard wall function relations [22], which are valid in the log-law region of the turbulent boundary layer. For this reason, the values of \( y^+ \) in near the wall regions were kept in the range of 30–40. Also, to mitigate the effect of circulation zones in the case of the ribbed duct channel, a non-equilibrium wall function has been imposed, which can take the effect of pressure gradient into account. In the radiative heat transfer part, the radiative transfer equation (RTE) [22] in participating media has been solved by the S4 discrete ordinates method with 24 directions. This selection gives satisfactory accuracy [23] for absorption and emission and keeps the computational efforts as low as possible. Also, because of the low temperature in the cooling duct, the radiative heat transfer was only considered on the inside of the liner. Combustion of methane generates some \( \text{CO}_2 \) and \( \text{H}_2\text{O} \) and these are the most important participating gases in the absorption and emission of thermal radiation. To take their effects into account, the spectral line weighted sum of grey gases method (SLW) [24] has been used, which is an accurate model. For calculation by the SLW method, five grey gases, optimised by a conjugate gradient method were selected. For each grey gas, the blackbody weight is calculated and then the grey gas absorption coefficient and its blackbody weight are used in solving the RTE. For the radiative boundary conditions, the liner wall emissivities have been set to 0.7 and updated wall temperatures are used during the solution process.

6.3.3 Results and discussion

6.3.3.1 Flow and temperature fields

The axial velocity and temperature fields inside the liner and before mixing with the secondary air are shown in Fig. 21. As can be seen, the velocity changes sharply close to the swirler with negative values near the liner wall. The sharp variation is mitigated along the length of the liner. At \( z/R_0 = 1.5 \) (about 74 mm along the liner), the velocity profile starts to change direction near the wall and therefore a stagnation point is formed. By further increase in \( z/R_0 \), the velocity is stabilized in the new direction and its profile in the radial direction will be flattened.

Similar to the velocity field, the temperature field varies sharply close to the swirler. The main reason for the variation is the combustion of fuel in the region.
and negative velocities at the centre and wall regions of the liner. By increasing \( z/R_0 \) up to 1, the temperature does not change considerably near the wall regions; however, by further increase, the temperature is decreased. This can be due to the fact that the combustion process has been completed and convective heat transfer.

**6.3.3.2 Simple cooling duct** Temperature distributions inside and outside of the liner wall are shown in Fig. 22. As can be seen, the peak of the wall temperature is predicted at a distance 74 mm from the entrance. This is also the position for zero axial velocity (see Fig. 21). Radiation has increased the wall temperature both on the inner and on the outer sides and its effect is stronger in the low temperature zones. Because of radiation, the average temperature has increased about 33 K on the inner wall. In addition, comparison of predictions and experimental data shows that at the beginning of the liner, the agreement is better, whereas with increasing liner length, the difference between predictions and experimental data becomes larger.

Total heat loads to the wall with and without radiation are shown in Fig. 23. It can be clearly seen that the increase in radiative heat flux near the entrance of the
cooling duct is large and low radiative heat flux occurs at the hot region of the wall. In the mid part, the heat load with radiation is almost constant, 450 kW, which means that the temperature difference between the two sides of the wall for a short distance is almost constant. The average heat load without radiation is about 388 kW and radiation increases this value by 8%. Because of the small wall thickness, the heat load is very sensitive to the temperature difference between the two sides of the liner wall and it is noted that for 1 K difference, the heat load changes about 17 kW/m$^2$.

6.3.3.3 Ribbed cooling duct and TBC In Fig. 24, the temperature distributions on the inner and outer liner walls for the case of a ribbed duct with TBC are shown. In this case, the influence of radiation is higher than that for the simple duct case. The average temperature on the inner wall has increased by 40 K. The agreement between predictions and experimental data at the entrance of the liner is somewhat poor, but it is obvious that radiation is important. At the middle and end parts, the predictions for the outer face of the wall are very good, but the values of the inner face have been overpredicted.

This might be due to error in the experimental data, because the outer wall at the same position is well predicted. In that part of the liner, the flow and temperature near the wall are stabilized, so the sharp decrease in the inner wall temperature is doubtful. The predicted heat load is shown in Fig. 25. Similar to the simple cooling duct case, the radiative heat load is stronger near the entrance of the cooling duct. In this case, the average convective heat load is about 393 kW/m$^2$ and the radiative heat transfer increases the heat load by about 7%.

In summary, the wall temperatures and heat loads in a premixed combustor have been predicted. The results showed that in the mid part of the liner, the prediction of the wall temperature is good, but poorer agreement exists in other parts. In addition, radiative heat transfer has been included in the study. The results showed that radiative heat transfer for simple and ribbed duct cooling schemes can increase the average inner wall temperature by 33 and 40 K, respectively. As an extension of this study, the accuracy of the model in prediction of wall temperature and heat loads to the walls can be investigated by using different wall treatments such as a two-layer wall function approach or applying a low Reynolds model.
6.4 CFD methods in analysis of thermal problems

CFD can be applied to heat exchangers in quite different ways. In the first way, the entire heat exchanger or the heat transferring surface is modelled. This can be done by using large scale or relatively coarse computational meshes or by applying a local averaging or porous medium approach. For the latter case, volume porosities, surface permeabilities and flow and thermal resistances have to be introduced. The porous medium approach was first introduced by Patankar and Spalding [25] for shell-and-tube heat exchangers and was later followed by many others.

Another way is to identify modules or group of modules, which repeat themselves in a periodic or cyclic manner in the main flow direction. This will enable accurate calculations for the modules, but the entire heat exchanger including manifolds and distribution areas are not included. The idea of streamwise periodic flow and heat transfer was introduced by Patankar et al. [26].

The finite volume method is a popular method particularly for convective flow and heat transfer. It is also applied in several commercial CFD codes. Further details can be found in [21, 27]. In heat transfer equipment like heat exchangers, both laminar and turbulent flows are of interest. While laminar convective flow and heat transfer can be simulated, turbulent flow and heat transfer normally require modelling approaches in addition. By turbulence modelling, the goal is to account for the relevant physics by using as simple a mathematical model as possible. This section gives a brief introduction to the modelling of turbulent flows.

The instantaneous mass conservation, momentum and energy equations form a closed set of five unknowns \( u, v, w, p \) and \( T \). However, the computing requirements, in terms of resolution in space and time for direct solution of the time dependent equations of fully turbulent flows at high Reynolds numbers (so-called direct numerical simulation (DNS) calculations), are enormous and major developments in computer hardware are needed. Thus, DNS is more viewed as a research tool for relatively simple flows at moderate Reynolds number. In the meanwhile, practicing thermal engineers need computational procedures supply-
ing information about the turbulent processes, but avoiding the need to predict effects of every eddy in the flow. This calls for information about the time-averaged properties of the flow and temperature fields (e.g. mean velocities, mean stresses, mean temperature). Commonly, a time-averaging operation, called Reynolds decomposition is carried out. Every variable is then written as a sum of a time-averaged value and a superimposed fluctuating value. In the governing equations, additional unknowns appear, six for the momentum equations and three for the temperature field equation. The additional terms in the differential equations are called turbulent stresses and turbulent heat fluxes, respectively. The task of turbulence modelling is to provide the procedures to predict the additional unknowns, i.e. the turbulent stresses and turbulent heat fluxes with sufficient generality and accuracy. Methods based on the Reynolds-averaged equations are commonly referred to as Reynolds-averaged Navier–Stokes (RANS) methods.

6.4.1 Types of models
The most common turbulence models for industrial applications are classified as

- zero-equation models,
- one-equation models,
- two-equation models,
- Reynolds stress models,
- algebraic stress models and
- large eddy simulations (LES).

The first three models in this list account for the turbulent stresses and heat fluxes by introducing a turbulent viscosity (eddy viscosity) and a turbulent diffusivity (eddy diffusivity). Linear and non-linear models exist [28–30]. The eddy viscosity is usually obtained from certain parameters representing the fluctuating motion. In two-equation models, these parameters are determined by solving two additional differential equations. However, one should remember that these equations are not exact, but approximate and involves several adjustable constants. Models using the eddy viscosity and eddy diffusivity approach are isotropic in nature and cannot evaluate non-isotropic effects. Various modifications and alternative modelling concepts have been proposed. Examples of models of this category are the $k–\varepsilon$, and $k–\omega$ models in high or low Reynolds number versions as well as in linear and non-linear versions. A lately popular model is the so-called V2F model introduced by Durbin [31]. It extends the use of the $k–\varepsilon$ model by incorporating near-wall turbulence anisotropy and non-local pressure–strain effects, while retaining a linear eddy viscosity assumption. Two additional transport equations are solved, namely one for the velocity fluctuation normal to walls and another for a global relaxation factor.

In Reynolds stress equation models, differential equations for the turbulent stresses (Reynolds stresses) are solved and directional effects are naturally accounted for. Six modelled equations (i.e. not exact equations) for the turbulent stress transport are solved together with a model equation for the turbulent scalar dissipation rate $\varepsilon$. Reynolds stress equation models are quite complex and require
large computing efforts and for this reason, are not widely used for industrial flow and heat transfer applications.

Algebraic stress models (ASM) and explicit ones such as EASM present an economical way to account for the anisotropy of the turbulent stresses without solving the Reynolds stress transport equations. One idea is that the convective and diffusive terms are modelled or even neglected and then the Reynolds stress equations reduce to a set of algebraic equations.

For calculation of the turbulent heat fluxes, most commonly a simple eddy diffusivity concept is applied. The turbulent diffusivity for heat transport is then obtained by dividing the turbulent viscosity by a turbulent Prandtl number. Such a model cannot account for non-isotropic effects in the thermal field but still this model is frequently used in engineering applications. There are some models presented in the literature to account for non-isotropic heat transport, e.g. the generalized gradient diffusion hypothesis and the WET (wealth = earnings × time) method. These higher order models require that the Reynolds stresses are calculated accurately by taking non-isotropic effects into account. If not, the performance may not be improved. In addition, partial differential equations can be formulated for the three turbulent heat fluxes, but numerical solutions of these modelled equations are rarely found. Further details can be found in [32].

The LES is a model where the time-dependent flow equations are solved for the mean flow and the largest eddies, while the effects of the smaller eddies are modelled. The LES model has been expected to emerge as the future model for industrial applications, but it still limited to relatively low Reynolds number and simple geometries. Handling wall-bounded flows with focus on the near wall phenomena like heat and mass transfer and shear at high Reynolds number present a problem due to the near-wall resolution requirements. Complex wall topologies also present problem for LES.

Nowadays, approaches to combine LES and RANS based methods have been suggested.

6.4.2 Wall effects
There are two standard procedures to account for wall effects in numerical calculations of turbulent flow and heat transfer. One is to employ low Reynolds number modelling procedures, and the other is to apply the wall function method. The wall functions approach includes empirical formulas and functions linking the dependent variables at the near-wall cells to the corresponding parameters on the wall. The functions are composed of laws of the wall for the mean velocity and temperature, and formulae for the near-wall turbulent quantities. The accuracy of the wall function approach is increasing with increasing Reynolds numbers. In general, the wall function approach is efficient and requires less CPU time and memory size, but it becomes inaccurate at low Reynolds numbers. When low Reynolds number effects are important in the flow domain, the wall function approach ceases to be valid. The so-called low Reynolds number versions of the turbulence models are introduced and the molecular viscosity appears in the diffusion terms. In addition, damping functions are introduced. Also, the so-called two-layer models have been suggested where the transport
equation for the turbulent kinetic energy is solved, while an algebraic equation is used for, e.g. the turbulent dissipation rate.

6.4.3 CFD codes
Several industries and companies worldwide are nowadays using commercially available, general-purpose, so-called CFD codes for simulation of flow and heat transfer topics in heat exchangers, investigations on enhanced heat transfer, electronics cooling, gas turbine heat transfer and other application areas, e.g. fuel cells. Among these codes are: FLUENT, CFX, STAR-CD, FIDAP, ADINA, CFD2000, PHOENICS and others. Also many universities and research institutes worldwide apply commercial codes besides in-house developed codes. However, to apply such codes successfully and to interpret the computed results, it is necessary to understand the fundamental concepts of computational methods.

6.4.4 Ducts with bumps
As a duct with bumps is considered, this type of duct appears in some rotary regenerative heat exchangers. The basic idea with introduction of bumps is to design corrugated ducts as indicated in Fig. 26 for ducts with triangular cross section. The intention is that this corrugation should affect the flow field and introduce low Reynolds number turbulence and a swirling motion as sketched in Fig. 26. At a certain distance downstream the corrugation element, the turbulence and the swirling motion will be attenuated and gradually the intensity of the fluctuations will be reduced. Therefore, at a position upstream where the complex flow pattern (strong secondary cross-sectional flow and separated flow) has been significantly weakened or has disappeared, a new corrugation element is introduced to re-establish the violent and swirling like motion. CFD calculations have been performed and a non-orthogonal structured grid was employed. Periodic conditions were imposed in the main flow direction. About 40,000 control volumes (CVs) were used, 30 × 60 CVs in the cross-sectional plane. The existence of a secondary flow was revealed and a result is shown in Fig. 27. It is obvious that a swirling motion is created by the bumps and the triangular cross section. The Reynolds number corresponding to the flow in Fig. 27 is about 2000. In the simulations, a low Reynolds number $k–\varepsilon$ model was used. The secondary motion exists also for laminar cases, as it is partly geometry-driven. It is found that the heat transfer is enhanced compared to a smooth duct, but the pressure drop increase is high.

Further details of this investigation can be found in [33].

Figure 26: Conjectured flow pattern in a duct with bumps.
6.5 Flow structures in ribbed ducts

Ribbed duct flows are encountered in numerous engineering applications, e.g. turbine blades and combustor walls cooling. The flow behind a rib is typically characterized by flow separation and subsequent reattachment. Flow in a separated shear layer is complicated by the presence of reverse flow and a high level of the turbulence intensity. Despite the substantial progress in experimental and numerical studies on turbulent flows with separation, our understanding of this phenomenon is far from complete. The first review of the experimental data for separated flow was provided by Bradshaw and Wong [34] for flow over a backward-facing step. On the basis of single point measurement, they concluded that the shear layer split in two parts at the reattachment point and the bifurcation caused a rapid decrease in turbulence shear stress. Troutt et al. [35] showed that the separated shear layer was dominated by the large-scale vortices, which retained their organization far downstream of the reattachment region. Ruderich and Fernholz [36] indicated a self-similar behaviour for the mean and fluctuating quantities in a short region upstream of the reattachment point. The data of Castro and Haque [37] showed that the turbulent structure of the separated shear layer differed from that of a plane.

Figure 27: Secondary flow velocity vectors in a cross sectional plane midway over a corrugation element.
mixing layer between two streams. On the other hand, they argued that the flow close to the wall within the recirculation region had some features reminiscent of a laminar boundary layer in a favourable pressure gradient. Thereafter, Hasan [38] confirmed that the reattaching shear layer did split into two and a low-frequency flapping motion of the shear layer is observed. In the numerical simulation of turbulent flow over a backward-facing step, Le et al. [39] pointed out that the turbulent kinetic energy budget in the recirculation region is similar to that of a turbulent mixing layer.

Previous research mostly focused on the flow with separation induced by a backward-facing step, which is considered to be the benchmark to study this phenomenon. The flow past a rib is more complicated because it involves an additional separation in the upstream region of the obstacle. On the other hand, the geometry of rib has also an essential influence on the flow separation and reattachment. According to Fröhlich et al. [40], separation from continuous and curved surfaces displays a strong spatial and temporal fluctuation of the separation line; meanwhile, the mean location of reattachment is sensitively dependent on that of separation. These characteristics imply that the separation from contoured protrusions is more elusive than that from obstacles with sharp edge.

In our recent work, square-shaped, transversely placed ribs were employed to investigate the separated flow in a square channel. To highlight the physical mechanism of flow separation, only one wall of the channel is fitted with periodic ribs. The ribs obstruct the channel by 15% of its height and are arranged 12 rib heights apart. The inter-rib spacing is set such that the reattachment is allowed to take place on the portion between consecutive ribs and a distinct redevelopment region is introduced prior to a re-separation over the next rib.

Many numerical and analytical studies [41–44] were carried out to investigate the characteristics of flow separation in a ribbed channel based on the DNS or LES techniques. Corresponding to the numerous simulation works, very few experimental works, however, were executed to give high-resolution velocity measurements and turbulent properties. On the basis of literature review, Islam et al. [45] conducted an experimental study on the turbulent water flow in a rib-roughened rectangular channel by particle image velocimetry (PIV).

Given the limited body of experimental data, experiments are performed to study the unsteady turbulent flow inside a square, ribbed channel. In this study, two-dimensional PIV technique is implemented to measure the instantaneous velocity fields and turbulent statistical quantities. The research reported here was undertaken to fulfill two objectives, i.e. to gain sight into the physical process of separation and to provide experimental data of ribbed channel flows for validation of CFD models.

Wang et al. [46] examined experimentally the flow structures and turbulent properties associated with the flow separation in a square, ribbed channel by using PIV. The Reynolds number, based on the bulk-mean velocity and the channel hydraulic diameter, is fixed at 22,000. The ribs obstruct the channel by 15% of its height and are arranged 12 rib heights apart. Due to the flow periodicity, the investigated domain ranges from the sixth to seventh ribs. Two-dimensional velocity measurements are made in both the vertical symmetry plane and the horizontal planes.
The instantaneous velocity gives evidence that the separated shear layer is dominated by the coherent vortices, which are generated by the Kelvin–Helmholtz instability. Similar to the plane mixing layers, the growth rate of the separated shear layer is linear with respect to the streamwise direction. Moreover, it is noticed that the turbulence production, for both turbulent kinetic energy and shear stress, has a remarkable peak at \( y/e = 1 \), which approximately coincides with the inflexion point \( \frac{d^2\langle u \rangle}{dy^2} \). Two distinct features near reattachment have been identified. First, the maximum shear stresses decrease rapidly just downstream of the reattachment. Second, the anisotropy parameters deviate to a small extent from unity near reattachment. Further downstream of the reattachment, the acceleration in the inner part coupled with deceleration in the outer part makes the redevelopment of the boundary layer different from the behaviour of an equilibrium boundary layer.

7 Conclusions

To develop sustainable energy systems, one must minimize the final use of energy, improve the efficiency of energy conversion and use renewable energy sources. In all these aspects, heat transfer and heat exchangers play a significant role, and this fact has been reviewed and illustrated throughout this article. In addition, heat transfer and heat exchangers also have great influence on the reduction of emissions and pollutants directly and indirectly. Therefore, the attempt to provide efficient, compact and cheap heat transfer methods and heat exchangers is a real challenge for research. Both theoretical and experimental investigations must be conducted, and modern scientific techniques must be adopted, such as CFD, laser techniques, liquid crystal thermography. By doing so, sustainable energy systems can be established, and this will contribute to global sustainable development.

Acknowledgement

Financial support has been received from Swedish Energy Agency, Swedish Scientific Council as well as EU. Several graduate students have been involved in performing the various projects.

References


