Numerical and experimental investigation of a wire-net compact heat exchanger performance for high-temperature applications

Jojomon Joseph, Michel Delanaye, Rabia Nacereddine, Andres Giraldo, Mehdi Rouabah, Jan G Korvink, Juergen Brandner

► To cite this version:

Jojomon Joseph, Michel Delanaye, Rabia Nacereddine, Andres Giraldo, Mehdi Rouabah, et al.. Numerical and experimental investigation of a wire-net compact heat exchanger performance for high-temperature applications. Applied Thermal Engineering, 2019. hal-02446520

HAL Id: hal-02446520

<https://hal.science/hal-02446520>

Submitted on 20 Jan 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Numerical and experimental investigation of a wire-net compact heat exchanger performance for high-temperature applications.

JOJOMON JOSEPH¹, MICHEL DELANAYE¹, RABIA NACEREDDINE¹,
ANDRES GIRALDO¹, MEHDI ROUABAH¹, JAN G. KORVINK² AND
JUERGEN J. BRANDNER²

¹*Mitis SA, Liege, Belgium*

²*Karlsruhe Institute for Technology, Institute of Microstructure Technology,
Hermann-von-Helmholtz-Platz 1, 76344 Eggenstein-Leopoldshafen, Germany
Email: jojomon.joseph@partner.kit.edu*

The objective of this paper is to have a detailed investigation of the microchannel performance of a complex wire-net compact heat exchanger and investigate its collector performance experimentally for a micro combined heat and power system. Localised turbulence can enhance the heat exchanger performance. Besides, this will increase the pressure losses also. Shifting the Reynolds critical to smaller Reynolds number by using perturbators will control the pressure losses and enhance the microchannel thermal efficiency. In this paper, we consider microchannels with wire-net perturbators. The wire-net micro heat exchanger is assembled as a stack of counterflow flow passages with optimised thickness separated by thin foils. A metallic wire-net mesh is inserted in the flow passages to provide the required stiffness and enhance the microchannel efficiency.

A parametric study was conducted on various heat exchanger parameters to optimise the heat exchanger size, thermal effectiveness and pressure losses for a micro-CHP system. Besides, a detailed investigation of the wire-net flow physics was made using a higher order Reynolds stress turbulence model to obtain the full velocity gradient tensor. This could detail the effect of anisotropic flow physics in the isotropic wire-net microchannels. Lambda 2 criteria was implemented to investigate the flow mixing of the centrally convected non-disturbed mass flow. Furthermore, the analysis of the turbulence production terms provided a deeper insight into flow attachment and detachment near the wire-net intersections. The heat exchanger was experimentally tested, and it was found that the collector pressure losses are not sufficiently low as compared to the microchannel pressure losses. The microchannel conjugate heat transfer thermal effectiveness is in good agreement with the overall experimental efficiency.

Keywords:

Received 01 December 2018

1. INTRODUCTION

Combined heat and power (CHP) production is a process of producing both electricity and heat with high efficiency. According to the International Energy Agency (IEA), CHP shows the potential to reduce the primary energy consumption in the residential sector. Besides, they help to reduce greenhouse emissions, promotes decentralisation of electricity production and has lower initial investment and distribution losses compared to conventional grids [1, 2, 3]. However, miniaturisation and cost reduction remain a challenge for the industry to reach broad adoption [2]. The MITIS Micro Gas Turbine (MGT) system consists of a compressor, combustor, turbine, recuperator and generator. The compressor and turbine are radial-flow designs, and similar to conventional engine turbochargers. The combustion chamber works in flameless regime, providing high effectiveness with low emissions [4, 5]. The air preheating is carried out by a wire-net micro heat exchanger. Preheating typically doubles the electrical effectiveness of the unit while reducing the amount of heat recoverable [2, 5]. The objective of this paper is to investigate the performance of a wire-net micro heat exchanger which preheats the combustion air, therefore reducing the amount of injected fuel [5] for the micro-CHP system.

A compact heat exchanger (CHE) regularly consists of microchannels with collectors optimised for the distribution of the flow. Here we use a plate gas to gas (air) counterflow CHE, with wire-net microchannel configuration (patent number: WO 20161124472, Figure 1). The wire-net placed in between the partition foils shifts the critical Reynolds number to lower values and enhances heat transfer mechanism by localised laminar/ turbulent mixing. The wire-net structures influence the CHE performance. Balaras [6] reviewed and classified heat transfer enhancement techniques as surface, fluid and combined and compound methods. The surface method includes extended geometries such as wire-net structures (fins/protrusions), structural vibrations etc. High convective heat transfer is associated with larger protrusions which creates greater channel obstruction with an increased acceleration of the flow [7]. Conversely, smaller protrusions yield more intense 3D transport with high turbulence intensity, resulting in high heat transfer. Tao et al. [8] suggested heat transfer enhancement by decreasing the thermal boundary layer, introducing protrusions inside the microchannels and by increasing the velocity gradient near the heated surface. Conjugate heat transfer (CHT) analysis carried out by Paolo [9] on a cube, pointed that the heat transfer was minimum when the boundary layer was undisturbed.

Transverse vortex generators and longitudinal vortex generators play a major role in local and global heat transfer enhancements [10]. However, longitudinal vortex generators perturb the viscous sublayer along

the flow direction and enhance the heat transfer [11] as compared to transverse vortex generators. The study made by [12] M.Fiebig revealed that counter-rotating vortex generators arrangements enhanced the heat transfer by more than twice that of the co-rotating configuration. In addition, he inferred that longitudinal vortices are remarkably persistent in both laminar and turbulent boundary layers and enhance the heat transfer locally and globally. Johnson and Joubert in 1969 made a study on the influence of longitudinal vortex on heat transfer and found that cylindrical drag was reduced due to separation delay caused by the longitudinal vortex enhanced the heat transfer [13].

The effect of flow pattern influences the global heat transfer and pressure losses. High heat transfer and low-pressure losses are formed along the flow reattachment and separation lines of leading-edge vortices [14]. However, areas of minimum convective heat transfer were found in regions of maximum turbulence production areas (coincided with maximum Reynolds normal stresses) due to high-velocity shear gradients. Thus, the maximum heat transfer coefficient occurred in the proximity of the reattachment points, where the wall shear stress is small, but the velocity fluctuations normal to the wall are high [15]. Valencia [16] showed that at Reynolds number 2000 heat transfer was enhanced by 30% due to turbulence together with a five-fold increase in friction factor. Increasing the blockage ratio and reducing the aspect ratio increased the global Nusselt number [17]. The heat transfer coefficient was very low on the upstream side in dimple like protrusions while downstream due to flow reattachment, heat transfer coefficient was increased [18]. The surface roughness also plays an essential role in the micro-channel performance [19] since at micro level it is nearly impossible to obtain a smooth surface. Morini and Spiga [20] investigated the role of viscous dissipation in the liquid flowing through heated microchannels. Heat transfer enhancement in the microdevices cannot be solved by indefinitely reducing the microchannel dimensions due to viscous dissipation effect.

The partition foil that separates the hot and cold fluid plays a vital role in heat transfer enhancement. Brandner et al. [21, 22] compared the performance of heat exchange devices with various microstructures manufactured on thin foils to enhance heat transfer. The smaller the foil thickness, the lower the axial conduction thereby leading to high heat transfer. Low heat conductivity material could improve the CHE effectiveness rather than very low or high heat conductivity materials. Among the flow arrangements, countercurrent flow arrangement will provide high effectiveness [23]. However, the cross-flow configuration tends to give similar results to the counter-current arrangement for higher Reynolds number and partition foils having low thermal conduction.

2. WIRE-NET HEAT EXCHANGER

The counterflow gas-gas (air) wire-net compact CHE arrangement (Figure 1, top left) is similar to the conventional plate-type CHE arrangement. Smooth partition foils replace the conventional corrugated partition foils. A wire-net is inserted in between the partition foils to bring the corrugation effects and thereby induce localised laminar/ turbulent mixing. In addition, the wire-net also prevent buckling of partition foils and therefore enhances the mechanical stability.

The CHE consists of collectors, two hot and two

FIGURE 1. Arrangement of wire-net brazed CHE and simplification scheme to reduce the computational domain

cold collector inlets with several microchannels made from a pile of counterflow passages separated by thin foils. The flow channels are delimited by frames with integrated collectors. The geometries of both frames

are identical, and they are positioned on top of each other by mirroring, as shown in Figure 1, top left. The collectors distribute the flow to the microchannels, and their inlets are placed perpendicular to the microchannel inlets. In this paper, computational fluid dynamics (CFD) computations were carried out only for microchannels while experimental tests were performed on a complete CHE model with collectors. CFD computations were adopted due to the complexity in experimentally investigating the microchannel flow physics. The primary aim of the CFD simulations is to examine the microchannel design features and use the experimental results to analyse the collector performance.

The inverted Brayton cycle (IBC) has a subatmospheric hot inlet side/ hot side, and Brayton cycle (BC) has a high-pressure cold inlet side/ cold side. The pressure at the outlet was increased and decreased (based on IBC/BC operating conditions) step by step to achieve the desired pressure inside the microchannels. The two different operating conditions, IBC and BC are mentioned in Table 1.

2.1. Numerical microchannel model

2.1.1. Boundary conditions

The CHE assembly is shown in Figure 1, top left. Two types of geometrical symmetry exist in a plane that is parallel to the inlet flow direction. The first symmetry is along half of the collector diameter of hot and cold collector inlets (see Figure 1, bottom), and second symmetry is at the centre of the wire-net CHE (Figure 1, top right). It is computationally expensive to carry out a CHT analysis by holding the first symmetry due to complex channels with a large computational domain. In conclusion, the rectangular first symmetry, (see Figure 1, bottom) and second symmetry (see Figure 1, top right) is shown in Figure 1.

Similarly, periodic boundary conditions were implemented in the plane which is perpendicular to the inlet flow direction. In other words, a periodicity exists in half of the microchannel foil thickness (Figure 1, top) on every hot and cold side alternatively. Thus, all hot and cold microchannels were simplified into a single hot and cold microchannels with partition foils above it, as shown in Figure 1, top right. In conclusion,

- Periodicity between the partition foils of hot and cold side
- Symmetric boundary conditions on the sides.
- No-slip conditions at wall (fluid and solid) boundaries

CHE performance depends on a wide range of parameters, e.g. wire-net cell size, microchannel dimensions and thickness of the partition foil. In this paper, square type wire-nets were used in which the opening size, longitudinal and diagonal size of the wire-net remains constant and all together it is mentioned as wire-net sizing. The structural radius of the wire-net is kept constant to decrease complexity.

2.1.2. CHT solver specifications

A total mass flow (m_{to}) of air passes through the counter-flow collector of CHE. A single collector inlet mass flow is calculated from the total mass flow based on the number of collector inlets. The microchannel inlet flow is calculated from the overall mass flow based on the number of collector inlets, the number of microchannels and MSR coefficient. The latter is obtained from simplifying the computational domain with the help of symmetric and periodic boundary conditions. The number of microchannels along with the microchannel size reduction coefficient is utilized to calculate the microchannel inlet mass flow from the collector inlet mass flow.

$$m_\mu = \frac{m_{to}}{n_{pl} * MSR} \quad (1)$$

The microchannel inlet mass flow is calculated using Eq.(1). Moreover, several microchannel inlet mass flows were calculated for different microchannel and collector arrangements, to investigate the dependency of CHE performance on mass flow.

The temperature difference between hot and cold inlets is at its minimum 650 K. Thus, compressibility and radiation effects may also have a significant impact on the performance. However, the effect of radiation to the outside was not taken into consideration because of the excellent insulation (5 cm thick special incombustible woollen layer) to the ambient. CHT simulations were carried not only based on the total massflow (m_{to}) (using Eq. (1)), but also for several mass flows to investigate the dependency of microchannel performance on the particular range of massflows. The Reynolds number (based on inlet hydraulic diameter) in the microchannel and collectors are in the range of 30-2000 and 2500-20000 respectively.

The governing differential equations were discretised using a finite volume method on a tetrahedral mesh with 12 to 14 million cells. The continuity and momentum equation is expressed as,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = S \quad (2)$$

$$\frac{\partial(\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla \cdot P + \rho \vec{g} + \vec{F} \quad (3)$$

S- is the mass added to the continuous phase from the dispersed second phase which is zero

for the numerical model. Where $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces which is zero for the CHT model. The energy conservation equation is expressed as,

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (\vec{v}(\rho E + P)) = -\nabla \cdot \left(\sum_j h_j J_j \right) + S_h \quad (4)$$

Where S_h is the heat source and $E = h - \frac{P}{\rho} + 0.5v^2$. h is the enthalpy for ideal gas and J_j is the diffusion flux of different species. The energy equation for solid regions are,

$$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot (\vec{v} \rho h) = -\nabla \cdot (K \nabla \cdot T) + S_h \quad (5)$$

Where K is the conductivity and the part of equation containing this term represents convective energy transfer due to rotational or translational motion of the solids.

At low mach numbers, the density does not depend on pressure but only on temperature. For a perfect gas we have,

$$P = \rho RT \quad (6)$$

Such that,

$$\partial \rho = \left(\frac{\partial \rho}{\partial P} \right)_T \partial P + \left(\frac{\partial \rho}{\partial T} \right)_P \partial T = \frac{\rho}{T} \partial T \quad (7)$$

since, $\frac{\partial \rho}{\partial P} = 0$ and the density simplifies into,

$$\rho = \rho_0 \frac{T_0}{T} \quad (8)$$

The flow is thus considered incompressible, but density varies as a function of the inverse of the temperature as well as the fluid is dilatible. Thus a segregated solver was introduced instead of a coupled solver. Some of the general assumptions made for the CHT analysis are:

- 3D, Steady-state, laminar for a microchannel without wire-net and turbulent model for a microchannel with wire-net.
- Ideal gas, steady-state, incompressible flow (segregated solver)
- Radiation losses and body forces were neglected.
- Smooth partition foils relative to microchannel hydraulic diameter to neglect surface roughness effects.

The porosity of the wire-net microchannels is approximately 83%, which will induce the transition to localised turbulence at low Reynolds number [24]. The CHT simulations were performed using the CFD solver FLUENT. For the velocity-pressure coupling, the SIMPLE algorithm has been applied. The thermal efficiency, ε , % is calculated using the relation

$$\varepsilon, \% = \frac{C_h(T_{in}^h - T_{out}^h)}{\min(C_c, C_h)(T_{in}^h - T_{in}^c)} * 100 \quad (9)$$

$$C_c = m_{\mu c} C_{p_c}; C_h = m_{\mu h} C_{p_h} \quad (10)$$

with $C_{p_{h,c}}$ being the hot and cold fluid capacity rates. $T_{in,out}^h$ and $T_{in,out}^c$ are the inlet and outlet temperatures of the counter-flow channels. The total pressure losses, $\Delta P, \%$ is calculated using,

$$\Delta P, \% = \frac{P_{in} - P_{out}}{P_{ab}} * 100 \quad (11)$$

P_{in}, P_{out} are the inlet and outlet total pressure losses. A parametric study was conducted using the lower order 3 equation SST $k-\omega$ turbulence model. SST $k-\omega$ model was implemented by Kanaris et al. [25] and Pelletier et al. [26], for Re in the range of respectively 500–6000 and 3500. The SST $k-\omega$ model switches between wall function and Low Re-implementation. Since its computationally cheap as compared to the RSM model, all the parametric studies were carried out using the SST $k-\omega$ model. Besides, a detailed investigation of the wire-net flow physics is made using a higher order Reynolds stress turbulence model to obtain the full velocity gradient tensor. This could detail the effect of anisotropic flow physics in the isotropic wire-net microchannels. The mesh for the RSM model was highly refined (relative to the computational mesh of the $k-\omega$ model) at proximities and curvatures to capture any sharp gradients.

2.1.3. Mesh sensitivity analysis

Mesh resolution along with the mesh quality plays a vital role in the accuracy and convergence of the CFD results. There is no exact methodology to investigate the error due to spatial discretisation. The only method is to conduct an independent mesh study. The standard way to characterise the mesh refinement is by checking the global and local convergence of flow characteristics. Here in this study, the convergence (local) of the temperature distribution along the central line is examined and plotted in Figure 2, top.

Three different mesh models (highly refined, average and coarse mesh) were selected based on the mesh sensitivity analysis. The cell number was regulated by managing the base size. In order to decrease the solver complexity and computational size, all three mesh models were converted into polyhedral cells using the Ansys Fluent solver. CHT simulations were conducted for three different mesh models by holding the same inlet boundary conditions. The temperature profile (along with the centre of the microchannel) was plotted from inlet to the outlet of a single hot inlet side for the counterflow CHT model. From the analysis, its evident that the uncertainty in the temperature

distribution is relatively low ($\pm 1\%$) between the highly refined model and the average mesh model (see Figure 2, top).

FIGURE 2. Mesh sensitivity analysis (top), CHE test bench (centre), Uncertainty analysis (bottom)

3. EXPERIMENTAL TESTING

The primary objective of the experimental testing is to characterise the collector pressure losses and compare the CHT thermal effectiveness relative to the overall CHE effectiveness. Figure 2, centre shows the test bench configuration for the CHE. The temperature is measured with type K thermocouples, with a standard limit of uncertainty equal to 2.2°C . Additionally, measurement of volumetric flow

BC	Temperature	Pressure
Hot inlet side	1054(K)	1 (Bar)
Cold inlet side	407 (K)	3 (Bar)
IBC	Temperature	Pressure
Hot inlet side	1054 (K)	0.4 (Bar)
Cold inlet side	298 (K)	1 (Bar)

TABLE 1. CHE operating conditions for BC and IBC.

Instrument	Range (0-FS)	Accuracy
Vol. Flowmeter	167 L/min - 850 L/min	$\pm 3\%FS$
Pressure sensor	0-2 BAR	$\pm 0.25\%FS$
Thermocouples	$-200^{\circ}C$ to $1250^{\circ}C$	$2.2^{\circ}C$

TABLE 2. Characteristics and accuracy of instrumentation

is taken from a GAS-VIEW device, which can measure volumetric flow in a range of 167 L/min to 850 L/min, with an accuracy of $\pm 3\%FS$. Moreover, effectiveness is calculated using Eq. (9). To reduce heat losses to the environment, there was a 5 cm layer of insulating material all around the CHE and the inlet and outlet collectors. Furthermore, pressure sensors were installed in the primary collectors that feed the mass flow to the secondary collectors and then to the microchannels. The temperature was analysed for 10 minutes and the uncertainty in temperature measurements for experimental hot inlet temperatures (based on CFD inlet conditions) is depicted in Figure 2, centre. The instrumental uncertainties is shown in Table 2. Table 3 shows the thermal effectiveness and pressure losses obtained at different temperatures. The thermal effectiveness differences between the CFD and experimental result is in the range of 0.3 to 4.4 % have been obtained. The differences mainly arise due to conductivity and radiation losses to the environment as well as the effect of collector maldistribution which is not included in the CFD model. In conclusion, the microchannel CHT model predicts the thermal efficiency with reasonable accuracy ($< 4.4\%$).

Apart from that, the collector pressure loss is calculated from the pressure loss difference between CFD and experiments at all temperatures, which is around 40-50 % of the microchannel pressure losses. Based on the heat exchanger arrangement, the contribution of collector towards thermal effectiveness is inadequate. Thus, it is not worth losing high-pressure/power losses in the collectors. Besides, the integrated secondary collectors will also contribute to the CHE

$T_{in}^{h,c}$ (K)	η_{exp} (%)	η_{CFD} (%)	ΔP_{exp} (%)	ΔP_{CFD} (%)
574	85.1	85.5	0.67	0.33
774	83.5	86.1	0.74	0.44
974	82.3	86.8	1.02	0.58

TABLE 3. Experimental and CFD results

pressure losses.

FIGURE 3. Thermal effectiveness (top) and pressure drop characteristics (bottom) of free microchannels and wire-net microchannels.

4. CHT RESULTS AND DISCUSSION

CHT Simulations were carried out for several mass flows to investigate the dependency of microchannel performance on mass flow. The temperature drop/temperature potential (at each cross-section) between the counter-flow channels is a major factor that influences the heat exchanger performance. The pressure losses and thermal effectiveness were calculated using Eq. (9 & 11). From the CHT analysis, we found an optimal Reynolds number/mass flow where thermal effectiveness reaches a maximum, see Figure 3, top & Figure 5. Studies made by Y. C. Fu et al. [27] on wire-net structures of compact heat exchangers showed a similar trend for thermal efficiency. The Reynolds number based on microchannel inlet is calculated using Eq. (12),

$$Re = \frac{\rho D_h V_{in}}{\mu} \quad (12)$$

In general we consider three mass flow regimes from the

thermal efficiency curve,

- Smallest mass flow/Reynolds number
- Optimum mass flow /Reynolds number
- Highest mass flow /Reynolds number

The temperature distribution along the centre of the free microchannel is depicted in Figure 4, top. The non-dimensionalised velocity distribution along the centre of a single cold-inlet channel is illustrated in Figure 4, bottom. Along the microchannel length, the velocity increases from cold-inlet to cold-outlet due to high-temperature gain from the hot-inlet side. In order to balance the momentum equation, the density decrease due to the temperature increase is equilibrated by the velocity increase [28].

At lower Reynolds, the potential/ temperature difference between the counter-flow channels (perpendicular to the flow direction) is nearly zero (see Figure 4, top) due to high residence time inside the counter-flow channels [29]. However, the slope of the temperature drop curve at smaller Reynolds number is less steep (relative to other cases, see Figure 4) due to low enthalpy input in the microchannel inlets. As the Reynolds number increases (from smaller to optimum Reynolds number), the thermal efficiency also increases, due to an increased input in enthalpy. Thereby, the slope of the temperature gets steeper at an optimum Reynolds number (Figure 3, top). The enthalpy input is higher at higher Reynolds number; still, the thermal efficiency drops as it advances from optimum to higher Reynolds number (see Figure 3). This is because, the boundary layer is thin at higher mass flows and in a relatively thin boundary layer, friction Shear Stress (viscous shearing force, Eq. 14) is very large [30]. This is evident from the shear velocity profile (Eq. 14) which is shown in Figure 3(b). The section of centrally advected non-disturbed mass flow increases and does not enhance the heat transfer.

$$\tau_w = \mu \left[\frac{\partial u}{\partial y} \right]_w \quad (13)$$

$$U^* = \mu \sqrt{\frac{\tau}{\rho}} \quad (14)$$

It is evident from the Figure 4, top that there is a thermal enhancement at higher Reynolds number due to the wire-net localised mixing. This could increase the working range and thereby reduces the size and cost of the CHE. The wire-net configuration shifts the critical Reynolds number to smaller values due to the wire-net structures localised turbulent mixing. From the design point of view, size and in turn costs will be reduced by,

- Shifting the optimum working range to higher mass flows
- Flattening the effectiveness curve at higher mass flow regimes

4.1. Parametric study

A parametric study was performed for various microchannel lengths and heights. However, the study was not carried for various microchannel widths, because an increase in width will introduce strong recirculation zones thereby increasing the pressure losses together with a reduction in thermal efficiency. The strong recirculation will change the pure counter-flow to cross-flow configuration thereby decreasing the efficiency.

FIGURE 4. Temperature distribution and velocity (non-dimensional) distribution along the microchannel length

4.1.1. Micro channel length

Increasing the microchannel length enhances the thermal effectiveness for both IBC and BC operating conditions. An increase in length gives more surface area improving heat recovery. Variation of thermal

effectiveness is saturated for BC operating condition compared to IBC configuration (see Figure 5, top). The optimum velocity (thermal efficiency) is shifted to higher inlet velocities for IBC operating conditions (compared to BC) while the optimum mass flow (thermal effectiveness) remains same for IBC and BC operating conditions. The lower pressure of IBC with respect to the BC influences the inlet velocities and thereby shifted the optimum to higher velocity (see Figure 5, top).

4.1.2. Micro channel wire net cell size (WNs)

Figure 5(centre) shows the thermal effectiveness for varying wire-net cell size. An increase in wire-net cell size (2.5 mm to 3 mm) did not enhance the thermal effectiveness but decreased the pressure losses (see Figure 4, bottom). Increase in wire-net cell size results in less number of perturbations/ square wire-net structures (for the same microchannel length), thereby lowers pressure losses. A further increase in wire-net cell size will induce buckling of the partition foils due to the pressure difference between the counter-flow microchannels.

4.1.3. Micro channel foil thickness (Fth)

Figure 5(centre) and Figure 4(bottom) shows the microchannel thermal effectiveness and pressure losses for two different foil thicknesses. The pressure losses did not change remarkably based on foil thickness. However, foil thickness strongly influenced the microchannel thermal efficiency. As the foil thickness decreases, axial conduction also decreases improving heat transfer between the channels. On the other hand, thinner foils may reduce the life of the CHE because of hot corrosion and erosion. As the thickness decreases, oxidation becomes a major limiting factor for its service life [31]. The operating conditions (BC and IBC) could introduce buckling of the thin partition foils.

4.1.4. Micro-CHP cycle efficiency on overall effectiveness

We, MITIS developed a computer code to estimate the effectiveness of the MGT cycle at IBC and BC conditions. This code integrates all of the MGT components for both IBC and BC. Figure 5(bottom) shows the cycle effectiveness of the MGT system for various microchannel inlet mass flows. Microchannel inlet mass flow corresponds to the MGT mass flow/ total CHE mass flow based on Eq. (1). The wire-net cell size of 2.5 mm shows higher pressure losses and thermal effectiveness compared to 3 mm wire-net cell size as shown in Figure 4(bottom) & Figure 5(bottom). At optimum mass flow, cycle effectiveness is affected by the thermal effectiveness pattern of the CHE. At higher mass flows, pressure losses have a more

significant influence on cycle performance irrespective of thermal effectiveness.

FIGURE 5. Thermal effectiveness for different length (top), Thermal effectiveness on various Fth & WNs sizing (centre), Combined effect of CHE pressure & thermal effectiveness on MGT

4.2. Flow physics

RSM turbulence model was implemented to infer the physics since it solves the turbulent stress transport equation and obtains the full velocity gradient tensor rather than the highly simplified eddy viscosity models. Thus, it could capture the anisotropic wire-net flow physics. The

K-omega SST model is an eddy-viscosity model, which calculates only the diagonal part of the velocity gradient tensor. RSM solves for the main physical mechanisms that drive turbulence: production, redistribution, turbulent transport, viscous diffusion, dissipation. RSM solves for 27 terms instead of 9 which makes it computationally expensive. In order to reduce the computational complexity, 1 Bar pressure operating condition was maintained in both cold and hot microchannels simultaneously.

4.2.1. Surface Nusselt number and wall shear stress

In a locally turbulent flow, heat transfer outside laminar sublayer occurs through conduction and convection. But the heat transfer mechanism inside the viscous sublayer is dominated by conduction [32]. As a result, thermal resistance appears in the viscous sublayer. In the wire-net structure, minimum and maximum local Nusselt number occur at locations where the cross section is at its minimum and maximum. The local Nusselt number is higher in the converging sector of the wire-net rather than in the diverging section. This is because the converging section has a higher average velocity and velocity gradient that increases heat transfer ratio. At higher mass flow, Nusselt number is higher, and heat transfer is enhanced compared to the smaller mass flow. This substantially proves the validation of Figure 6, top. The wall shear stress is proportional to Nusselt number [33] as seen in Figure 6 (top & bottom) and increases locally near the wire-net intersections thereby enhances the heat transfer together with higher pressure losses.

4.2.2. Wire-net induced vortices

Figure 7, a shows the velocity, λ_2 plotted in the midplane of the microchannel geometry. Higher order RSM analysis was performed to have the symmetric part $S_{ij} = \frac{1}{2}(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})$; and the skew-symmetric part $\Omega_{ij} = \frac{1}{2}(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i})$; of the velocity gradient tensor. λ_2 criterion looks for local pressure minimum but removes the effects from unsteady straining and viscosity by cancelling those terms [34]. A vortex is defined as a connected region with two negative eigenvalues of $S^2 + \Omega^2$. The vorticity mixing is maximum at a higher mass flow (1.57e-5 kg/s), which is expected due to the localised turbulence. The strong counter-rotating vortices near the wire-net intersections shown in Figure 7, a mix the centrally convected non-disturbed mass flow. The counter-rotating vortices that arise from the near wall increased the wall-normal velocity fluctuations and thereby enhanced the heat transfer.

4.2.3. Turbulence production on heat transfer enhancement

Higher order RSM directly calculates the three normal turbulent stresses ($-\rho u^2, -\rho v^2, -\rho w^2$) and three tan-

gential turbulent shear stresses ($-\rho \overline{uv}, -\rho \overline{vw}, -\rho \overline{wu}$) by solving the stress transport equation [35]. The RSM model captures the anisotropy of the flow pattern [36]. Turbulent kinetic energy production (mean of turbulent normal stresses, see Figure 7, b is dominant after the cylindrical cross-section which is due to the localised mixing of wire-net cross-sectional structures. Among the three normal stresses, wall normal stress appears to be dominant ($> 75\%$) as seen in Figure 7, c which shows an enhanced mixing of centrally convected fluids and also an increased pressure loss. As a result, the major share of the localised turbulence induced by the wire-net enhanced the heat transfer rather than just increasing the pressure losses. The strong wall-normal velocity fluctuations are produced by the wire-net cross-sectional structures together with the near wall regions of partition foils, this takes energy from the streamwise velocity and damps the energy to the walls thereby enhancing heat transfer. In particular, turbulent production terms do not require modelling in RSM since they only involve moments of order less or equal to two [37]:

$$P_{ij} = \overline{u_i u_k} \frac{\partial U_i}{\partial x_k} - \overline{u_j u_k} \frac{\partial U_j}{\partial x_k} \quad (15)$$

In the RSM turbulence production terms (see Eq. 14) are not modelled. Influence of the wire-net intersections is evident from the turbulence production term, $\rho \overline{uv} d\overline{U}/dx$. The stream wise mean velocity gradient, $d\overline{U}/dx$ is supposed to be positive and negative before and after the circular cross-section due to strong favorable and adverse pressure gradient zones [38]. The $\rho \overline{uv}$, vertical flux of stream wise momentum produced by the wire-net walls suppress these strong flow detachment/ separation zones behind the cylindrical cross-sections. This separation is suppressed by the nearby wire-net cross-sectional walls as seen in Figure 7, d thereby, enhancing the mixing.

5. CONCLUSION

When the Reynolds number increases, the pressure loss also increases in line with a heat transfer enhancement. Thus, the best way to obtain higher thermal effectiveness and heat transfer is to shift the critical Reynolds number to lower values by using perturbators. The metallic wire-net inserted in the flow passages provided the required stiffness and enhanced the microchannel effectiveness. Thermal effectiveness reaches its maximum at a particular optimum mass flow due to the boundary layer development.

A parametric study was carried on various CHE parameters for MGT optimisation. The strong counter-rotating vortices near the wire-net intersections mix the centrally convected non-disturbed mass flow. The flow separation is alternatively suppressed by the adverse and favourable pressure gradients (created by the wire-net intersections). This could reduce the size of flow detachment zones lowering the pressure

FIGURE 6. Surface Nusselt number (top), Wall shear stress (bottom)

found that the collector pressure losses is around 40-50 % of the microchannel pressure losses. Furthermore, thermal effectiveness is in good agreement with the CHT model, and it validates the microchannel CHT model assumptions. Additionally, it is possible to increase the predicted effectiveness by changing the insulating material or with more compact insulation, and thereby, reducing heat losses to the environment. Thus, in turn, this will increase the calculated effectiveness and reduce the differences between CFD and experiments in terms of microchannel thermal effectiveness.

6. ACKNOWLEDGEMENT

This ITN Research Project MIGRATE (www.migrate2015.eu) is supported by European Community H2020 Framework under the Grant Agreement No. 643095. The research of MITIS has been funded by grants Nanocogen+ and Nanocogen+2 from DGO4 and DGO6 directorates of Wallonia.

FIGURE 7. $Velocity.\lambda_2$ criteria for vorticity study(top), Turbulent Kinetic Energy(bottom, b), Reynolds stress, uv(bottom, c), Turbulence Production-DU/DX.uv(bottom, d) at higher mass flow

losses with an enhanced heat transfer. Among the turbulent Reynolds stresses, wall-normal velocity stress is found to be dominant. As a result, the significant share of the localised turbulence induced by the wire-net enhanced the heat transfer rather than increasing only the pressure losses.

From the experimental testing of the CHE, it was

Nomenclature

3D	Three dimensional
BC	Brayton cycle
CFD	Computational fluid dynamics
CHT	Conjugate heat transfer
E	Total enthalpy (Joules)
micro-CHP	micro-Combined Heat & Power
IEA	International energy agency
IBC	Inverted brayton cycle
J	Diffusion flux of different species
K	Conductivity(W/(m.K))
S, S_h	Mass source, heat source
MSR	Microchannel size reduction
MGT	Micro gas turbine
RSM	Reynolds stress model
$P, P_{in}/P_{out}, P_{ab}$	Inlet and outlet total pressure (Pa), Inlet absolute pressure (Pa)
D_h	Hydraulic diameter
R	Universal gas constant (J/mol. K)
Re	Reynolds number
U	Mean velocity along flow direction(m/s)
τ_w	Wall shear stress (Pa)
U^*	Friction/shear velocity (m/s)
V	Mean velocity perpendicular to flow direction (m/s)
V_{in}	Inlet velocity (m/s)
u	Velocity fluctuations along flow direction (m/s)
v	Wall normal velocity fluctuations (m/s)
w	Velocity fluctuations in the plane parellel to partition foils (m/s)
ρ	Density (kg/m3)
$T, T_{in,c}^{h,c}$	Inlet and outlet total temperature for hot and cold inlet sides (K)
m, m_{co}	Collector mass flow (kg/s)
Cp_c, Cp_h	Specific heat capacity for cold and hot (J/kg.K)
m_{to}	Total inlet mass flow (kg/s)
$m_{\mu}, m_{\mu h}, m_{\mu c}$	Inlet and outlet mass flow for hot and cold inlet sides (kg/s)
n_{pl}	Number of plates/micro channels
Fth	Partition foil thickness
WNs	Wire-net cell sizing
μ	Viscosity (kg/m.s)
η_{exp}	Experimental thermal effectiveness (%)
η_{CFD}	CFD thermal effectiveness (%)
ΔP_{exp}	Experimental pressure losses (%)
ΔP_{CFD}	CFD pressure losses (%)
ε	Thermal effectiveness
λ_2	Lambda 2 vorticity criteria
P_{ij}	Turbulence production term

REFERENCES

- [1] Pehnt, M., Cames, M., Fischer, C., Praetorius, B., Schneider, L., Schumacher, K., Vob, J.-P., Micro cogeneration towards decentralized energy systems, DOI: 10.1007/3-540-30821-0, Springer-Verlag Berlin Heidelberg, 2006.
- [2] Hawkes AD, Leach MA. Cost-effective operating strategy for residential micro-combined heat and power. *Energy*, 32(5):711–23, 2007.
- [3] IEA Energy Efficiency Policy Recommendations; 2011.
- [4] M. Delanaye, A. Giraldo, R. Nacereddine, M. Rouabah, V. Fortunato, A. Parente: Development of a recuperated flameless combustor for an inverted Bryton cycle microturbine used in residential micro-CHP, Proceedings of the ASME 2017 Turbomachinery Technical Conference and Exposition, TurboExpo 2017.
- [5] Beith R. Beith, Small and micro combined heat and power (CHP) systems, advanced design, performance, materials and applications, Woodhead Publishing, ISBN: 978-1-84569-795-2, 2011.
- [6] Balaras, C. A., A Review of Augmentation Techniques for Heat Transfer Surfaces in Single-Phase Heat Exchangers, *Energy*, 15 10 , pp. 899-906, 1990.
- [7] P. T. Roeller, J. Stevens and B. W. Webb, Heat Transfer and Turbulent Flow Characteristics of Isolated Three-Dimensional Protrusions in Channels, DOI:10.1115/1.2910607, *J. Heat Transfer* 113(3), 597-603, 1991.
- [8] Tao, W. Q., He, Y. L., Wang, Q. W., Qu, Z. G., and Song, F. Q., A Unified Analysis on Enhancing Single Phase Convective Heat Transfer With Field Synergy Principle, *Int. J. Heat Mass Transfer*, 45 24 , pp. 4871-4879, 2002.
- [9] Paolo C., Maria V., De B., Gianpaolo R., Conjugate heat and mass transfer in drying: A modeling review, ISBN:0260-8774, *Journal of Food Engineering*, 2016.
- [10] Fiebig, M., Vortex generators for compact heat transfer exchangers. *J. Enhanced Heat Transfer* 2 (1), 43-61, 1995.
- [11] Incropera, F.P., DeWitt, D.P., Introduction to Heat Transfer. John Wiley and Sons, USA, pp. 249-251, 1996.
- [12] M.Fiebig, Vortices, Generators and Heat Transfer, ISBN: 0263-8762, Chemical Engineering Research and Design, 1998
- [13] Johnson, T. R. and Joubert, P. N., The influence of vortex generators on the drag and heat transfer from a circular cylinder normal to an airstream, *J Heat Transfer*, 91: 91-99, 1969.
- [14] Hajime N., Tamotsu I., Takayuki T., Local heat transfer around a wall-mounted cube in the turbulent boundary layer, *International J. of Heat and Mass Transfer*, ISBN: 0017-9310, 2001.
- [15] E. R. Meinders, K. Hanjalic and R. J. Martinuzzi, Experimental Study of the Local Convection Heat Transfer From a Wall-Mounted Cube in Turbulent Channel Flow, *J. Heat Transfer* 121(3), 564-573, DOI:10.1115/1.2826017, 1999.
- [16] Alvaro Valencia, Turbulent flow and heat transfer in a channel with a square bar detached from the wall, *Numerical Heat Transfer: Part A: Applications*, vol. 37, issue 3, pp. 289-306, DOI: 10.1080/104077800274307, 2000.
- [17] M. Meis , F. Varas , A. Velázquez , J.M. Vega, Heat transfer enhancement in micro-channels caused by vortex promoters, *International Journal of Heat and Mass Transfer*, ISBN: 0017-9310, 2010.
- [18] Johann Turnow, Nikolai Kornev, Valery Zhdanov, Egon Hassel, “Flow structures and heat transfer on dimples in a staggered arrangement”, *International Journal of Heat and Fluid Flow*, 35, pp. 168–175, 2012.
- [19] Giulio Croce , Paola D’agaro, Carlo Nonino: Three-Dimensional Roughness Effect On Micro-channel Heat Transfer And Pressure Drop, *International Journal of Heat and Mass Transfer* 50 (2007) 5249–5259.
- [20] Morini, G., and Spiga, M., 2007: The Role of the Viscous Dissipation in Heated Microchannels, *ASME J. Heat Transfer*, 129, pp. 308-318.
- [21] Y. Yang, I. Gerken, J. J. Brandner, G. L. Morini. (2014): Design and Experimental Investigation of a Gas-to-Gas Counter-Flow Micro Heat Exchanger. *Experimental Heat Transfer* 27:4, pages 340-359.
- [22] Yang, Y., Morini, G.L., Brandner, J.J., 2014: Experimental analysis of the influence of wall axial conduction on gas-to-gas micro heat exchanger effectiveness. *Int. J. Heat Mass Transf.* 69, 17-25.
- [23] W. M. Kays, A. L. London, Compact heat exchangers, McGraw-Hill, 1964.
- [24] A. Kanaris, A. Mouza, S. Paras, Optimal design of a plate heat exchanger with un- dulated surfaces, *Int. J. Therm. Sci.* 48 (2009) 1184–1195.
- [25] O. Pelletier, F. Stromer, A. Carlson, CFD simulation of heat transfer in compact brazed plate heat exchangers, *ASHRAE Trans.* 111, 2005, 846–854.
- [26] Yang, Y.T. and Hwang, C.Z.: Calculation of turbulent flow and heat transfer in a porous-bed channel”, *Int. J. of Heat and Mass Transfer*, 46, pp. 771-780 (2003).
- [27] Yanchen Fu, Jie Wen, Cuizhen Zhang, An experimental investigation on heat transfer enhancement of sprayed wire-mesh heat exchangers, *International Journal of Heat and Mass Transfer*, Volume 112, 2017, Pages 699-708, ISSN 0017-9310.
- [28] Mushtaq I. Hasan, A.A. Rageb, M. Yaghoubi, Homayon Homayoni, Influence of channel geometry on the performance of a counter flow microchannel heat exchanger, *International Journal of Thermal Sciences*, Vol. 48, Iss. 8, 2009, pp. 1607-1618, ISSN 1290-0729.
- [29] Hermann Schlichting, Klaus Gersten, Boundary-Layer Theory, Springer-Verlag Berlin Heidelberg 2017, ISBN: 978-3-662-52917-1.
- [30] Chungpyo Hong, Yutaka Asako, Gian Luca Morini, Danish Rehman, Data reduction of average friction factor of gas flow through adiabatic micro-channels, *International Journal of Heat and Mass Transfer*, Vol. 129, 2019, pp. 427-431, ISSN 0017-9310.
- [31] George Y. Lai, High-temperature corrosion and material applications.
- [32] Qiuwang W., Qiuyang C., Ling W., Min Z., Yanping H., Zejun X., Experimental study of heat transfer enhancement in narrow rectangular channel with longitudinal vortex generators, ISBN: 0029-5493, *Nuclear Engineering and Design*, 2007.

-
- [33] Mahmud, S., Sadrul Islam, A.K.M. & Das, P.K., Numerical Prediction of Fluid Flow and Heat Transfer in a Wavy Pipe, J. of Therm. Sci. (2001) 10: 133, <https://doi.org/10.1007/s11630-001-0054-1>
 - [34] Vivianne H., Methods for Vortex Identification, Master's Thesis, ISSN: 1404-6342, Mathematics, Lund University, 2012.
 - [35] Pope, S. (2000). Turbulent Flows. Cambridge: Cambridge University Press. DOI:10.1017/CBO9780511840531
 - [36] Andrew E., and Jie C., Comparison of turbulence models in simulating swirling pipe flows, Applied Mathematical Modelling, Vol 34, p. 2840–2849, ISBN:0307-904X, 2010.
 - [37] Remi Manceau: Industrial codes for CFD.
 - [38] Farhana A., Muhammad A. R. S., Numerical analysis of the flow separation and adverse pressure gradient in laminar boundary layer over a flat plate due to a rotating cylinder in the vicinity, The Japan Society of Fluid Mechanics and IOP Publishing Ltd, Fluid Dynamics Research, Volume 50, Number 2, 2018.
 - [39] E. Agelidou, M. Henke, T. Monz, M. Aigner, Numerical investigation of an Inverted brayton cycle micro gas turbine based on experimental data, Proceedings of ASME Turbo Expo 2018, June 11-15, 2018, Oslo, Norway.
-