Performance estimation on micro gas turbine plant recuperator

Laura Alina STIKA[1], Jeni Alina POPESCU*[,1], Sorin Gabriel TOMESCU[1], Valeriu-Alexandru VILAG[1]

*Corresponding author
[1]National Research and Development Institute for Gas Turbines COMOTI,
220D Iuliu Maniu, Bd., sector 6, cod 061126, OP76, CP174, Bucharest, Romania,
laura.stika@comoti.ro , jeni.popescu@comoti.ro*, gabriel.tomescu@comoti.ro,
valeriu.vilag@comoti.ro

DOI: 10.13111/2066-8201.2016.8.4.10

Received: 05 September 2016/ Accepted: 10 October 2016/ Published: December 2016

International Conference of Aerospace Sciences "AEROSPATIAL 2016"
26 - 27 October 2016, Bucharest, Romania, (held at INCAS, B-dul Iuliu Maniu 220, sector 6)
Section 5 – Systems, Subsystems and Control in Aeronautics

Abstract: *In order to accomplish a thermodynamic analysis of a micro gas turbine plant, while satisfying conditions related to different configurations, different input data sets and a series of imposed thermodynamic parameters, assumptions are necessary regarding the individual performance of the components, for enabling the correlation of overall performance with the performance of the components. The use of a recuperator, even if it can improve a micro gas turbine cycle efficiency, by reducing the fuel consumption for an imposed turbine inlet temperature, will induce a pressure loss on the hot and cold sections, with possible negative influence on the cycle efficiency. In order to investigate this pressure loss, a numerical study was conducted on several cases, by modifying the geometry of a simplified recuperator, using as input a reference micro gas turbine thermodynamic cycle. The comparison between the different results led to the quantification of recuperator's performance as a function of its geometry.*

Key Words: *micro gas turbine, recuperator, CFD, effectiveness*

1. INTRODUCTION

Given that The EU depends on imports for over half of its energy needs and given the forecast of mitigation and depletion of traditional energy resources [1], without sufficient development of renewable resources, which led to issuing directives on energy saving, the focus is on using high efficiency cogeneration plants for combined production of electric and thermal energy. This is thought a solution for producing energy under standard costs, which achieves a significant saving of energy resources, a high level of energy efficiency up to 90% [2], with low pollutant emissions.

From a technical standpoint, the current trend is to increase the efficiency by optimizing the gas-dynamics and the combustion; to increase the global efficiency of the micro gas turbines by introducing waste heat from flue gases; to increase the mechanical efficiency by reducing the mechanical losses of transmissions; to develop high speeds bearings; to simplify the constructive solutions by eliminating the reducing gear box and directly driving the electric generator by the micro gas turbine; to reduce the emission level by optimizing the combustion process; to improve the heat transfer in recuperators through gas dynamics and geometry optimization; to use the latest technologies and materials; and to reduce the need for maintenance [3]. Although the generating capacity of the micro gas turbines is beyond the limits defined for the regime of micro-cogeneration, they are well suited to generate combined (thermal / electrical) power for locations such as apartment complexes, groups of commercial buildings, small businesses [4]. Another advantage of the micro gas turbines is their small size relative to the amount of energy they produce.

The main components of a cogeneration group are: the air inlet, also with noise reduction role; the micro gas turbine, which is a reduced scale single-shaft turbomachine, usually equipped with a centrifugal compressor and a radial turbine; heat exchanger/recuperator, generally used to increase the overall efficiency, the compressed air from the compressor taking part of the thermal energy of the flue gases exiting the turbine.

2. METHOD

The objective of the work presented in this section is to study the heat transfer inside a recuperator channel for estimation of the performance of its cold channel, in terms of exit total temperature and pressure loss.

In order to investigate the impact of the geometry of a heat exchanger on heat transfer and pressure losses, several numerical simulations were performed on a simplified shape of a recuperator, using the CFD software Ansys CFX. The configurations used for the computations are built on variation of:

- The cross-section area of the channel of the recuperator.
- The length of the channel of the recuperator.
- The number of the channels of the recuperator.
- The width of the solid domain.

The numerical simulations provided the total temperature and total pressure, at the exit of the so-called “cold channel”, parameters that were than considered to be used as input data for the gas turbine cycle calculations.

Considering the functioning of the recuperator, as shown in Fig. 1, the air coming from the compressor of the gas turbine passes through the so-called “cold channel” and is heated by the solid domain. The solid domain is also heated by the hot gases that come from the turbine of the gas turbine and pass through the so-called “hot channel”.

Regarding its design, a parallel flow can lead to a good performance, but in this case the exit temperature of the “cold channel” is always lower than the exit temperature of the “hot channel” [5]. In counter-current design, the restriction is more relaxed meaning that the exit temperature of the “cold channel” can surpass the value of the exit one from the “hot channel”, the heat transfer being limited by the inlet temperature of the cold stream. Hence, the chosen configuration for the relative position of the channels was counter flow, in order to achieve a greater heat recovery.

The height of the fluid channels is equal to the one of the solid domain and has a value of 50 mm. This was kept constant for each of the 13 Cases as, in this configuration, the temperature varies very little perpendicular to the flow. For each Case, the cross-section of the “hot channel” is equal to the cross-section of the “cold channel”, and its shape is rectangular. For the solid domain, all 13 Cases used as material: steel with a density of

7854 [kg/m^3], specific heat capacity of 434 [J/(kg*K)] and thermal conductivity of 60.5 [W/(m*K)] [6]. For each simulation, a computational domain containing three blocks (two fluid ones and a solid one) - presented in Fig. 2- was defined. These domains were meshed by means of an unstructured grid into 852642 elements for the fluid block and 121806 elements for the solid block.

Table 1- The geometrical characteristics of the channel used in numerical simulations

Case No.	Fluid Channels		Solid Domain	
	Width [mm]	Length [mm]	Width [mm]	Length [mm]
1,2,3,4	10	1000	2	1000
5,6	2	1000	2	1000
7,8,9	2	1000	1	1000
10	2	1500	1	1500
11,12,13	2	2000	1	2000

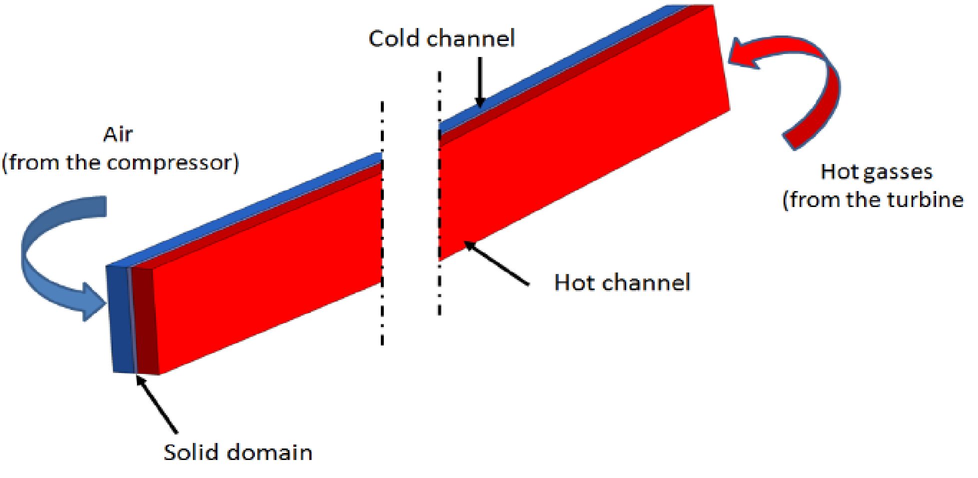

Fig. 1 – The geometry used in numerical simulations

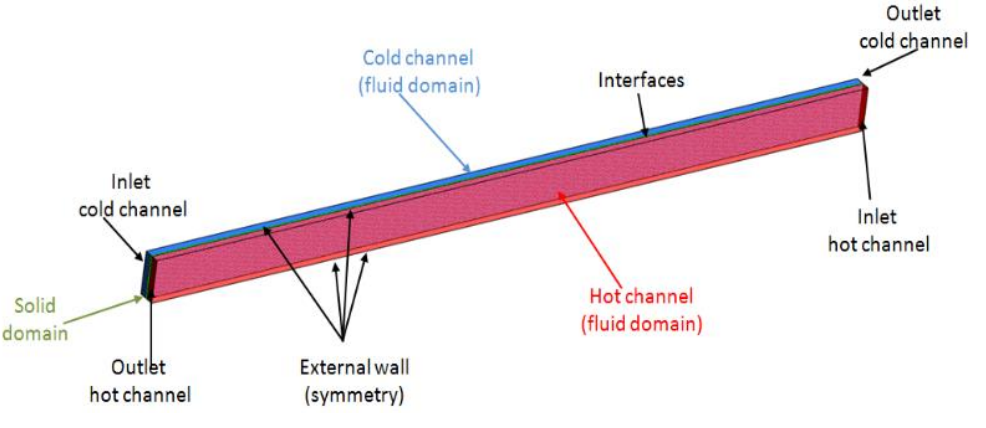

Fig. 2 – Computational domain and boundary conditions

The turbulence model used for both cold and hot channels was Shear Stress Transport and the simulations were carried out until convergence of the results was reached. The convergence criterion was a Normalized Residual level of the order $1e^{-5}$ [6].

The following boundary conditions were applied on the surfaces indicated in Fig. 2 for each of the 13 numerical simulations:

- Inlet Cold Channel: Subsonic inlet boundary conditions with the total pressure of 3.54637 bars and the total temperature of 478 K.
- Outlet Cold Channel: Subsonic outlet boundary condition, with the mass flow in Table 2.
- Inlet Hot Channel: Subsonic inlet boundary conditions with the total pressure of 1.05413 bar a total temperature of 878 K.
- Outlet Hot Channel: Subsonic outlet boundary condition, with the mass flow in Table 2.
- "External" wall: Symmetry type boundary conditions.
- Interfaces wall: interfaces between the fluid and the solid blocks.

The inlet total pressures and total temperatures, for both cold and hot channel were chosen based on the results of the gas turbine cycle calculation. The corresponding mass flows for these operating conditions would be 0.307455 kg/s for the cold channel and 0.31 kg/s for the hot channel. As a recuperator has more sequential channels, for one channel the mass flow is the total mass flow divided by the number of channels.

Usually, in a recuperator, the cold stream number of channels is equal to the hot stream one, but this study focuses on the influence of the performance of the recuperator on the overall performances of the gas turbine.

As the main influence in the overall performances of a gas turbine is the flow in the cold channels (that re-enters in the main flow of the gas turbine) the correlation between the number of channels of the two streams is not the object of the present work, but it will be considered for future studies.

Table 2 – Boundary conditions for the numerical simulations

Case No	Cold Channel		Hot Channel	
	Outlet Mass Flow [kg/s]	Corresponding number of channels	Outlet Mass Flow [kg/s]	Corresponding number of channels
1	0.079	4	0.018	17
2	0.045	7	0.009	33
3	0.069	4	0.013	24
4	0.033	9	0.009	33
5	0.007	45	0.001	212
6	0.006	52	0.002	174
7	0.006	52	0.002	174
8	0.004	77	0.002	174
9	0.002	154	0.002	174
10	0.004	77	0.002	174
11	0.004	77	0.002	174
12	0.002	154	0.002	174
13	0.001	307	0.002	174

3. NUMERICAL SIMPLIFICATION

In order to reduce the numerical simulations' computational time, a series of simplifications were made. They are used in the same way for all the cases and they are as follows:

- For both fluid domains, Air Ideal Gas was set as the material used.
- There are no heat losses with the environment. In other words, the hot stream heat (Q_{hot}) is entirely transferred to the cold stream (Q_{cold}), so the next equation can be written:

$$\frac{Q_{cold}}{Q_{hot}} = 1 \tag{1}$$

In each simulation only one "hot channel", one "cold channel" and the solid domain between them was meshed and simulated.

The parameters of interest in these calculations are the total pressure and temperature resulted at the outlet of the cold stream.

As, pressure and temperature are intensive properties of the fluid [7], it can be assumed that the flow will be similar in all the channels, hence the computations having only one channel of each stream should have similar results with the ones having the entire geometry with all the channels.

- Change in number of channels is "translated" as a change in the imposed mass flow. A mass flow ($\dot{M}$) is related to the density (ρ), velocity (v) of the fluid and area through which the fluid passes (S) [8]. As the total area of a recuperator is the number of channels (n) multiplied with the cross-section area of a channel (S_c), it can be written:

$$\dot{M} = \rho \cdot v \cdot n \cdot S_c \tag{2}$$

and for one channel of the stream, the mass flow will be $\dot{M}_c$, defined as:

$$\dot{M}_c = \rho \cdot v \cdot S_c \tag{3}$$

Dividing the equation (eq. 2) by (eq. 3), it is obvious that the number of channels can be obtained. Therefore, in the presented numerical calculations, the mass flow variation is made for only one channel and it can be interpreted as changing the channels number.

4. RESULTS AND DISCUSSION

In order to verify if the computational domain is well chosen, the velocity vectors at both inlets and outlets for each case were analysed. As presented in Fig. 3 - for Case 11 - the phenomena of inflow or outflow hadn't occurred.

A second check was made for each case, and consisted in verifying the Equation (eq 1). The heat (Q) for each stream was computed as:

$$Q = \dot{M} \cdot c_p \cdot \Delta T^* \tag{4}$$

where $\dot{M}$ is the mass flow through one channel, c_p is the specific heat of the used fluid and ΔT^* is the difference between inlet and outlet temperatures of that channel [9].

The heats calculated in this way, for each of the 13 Cases, check out the Equation (eq. 1), with errors less than 5 %.

This is considered another way to verify that the numerical simulations have a correct imposed setup and have reached convergence.

In Table 3, the averaged numerical simulation results are presented for each case. The total temperature and total pressure obtained at the outlet of the cold channel are used in the gas turbine cycle calculation to calculate the overall performances of the gas turbine.

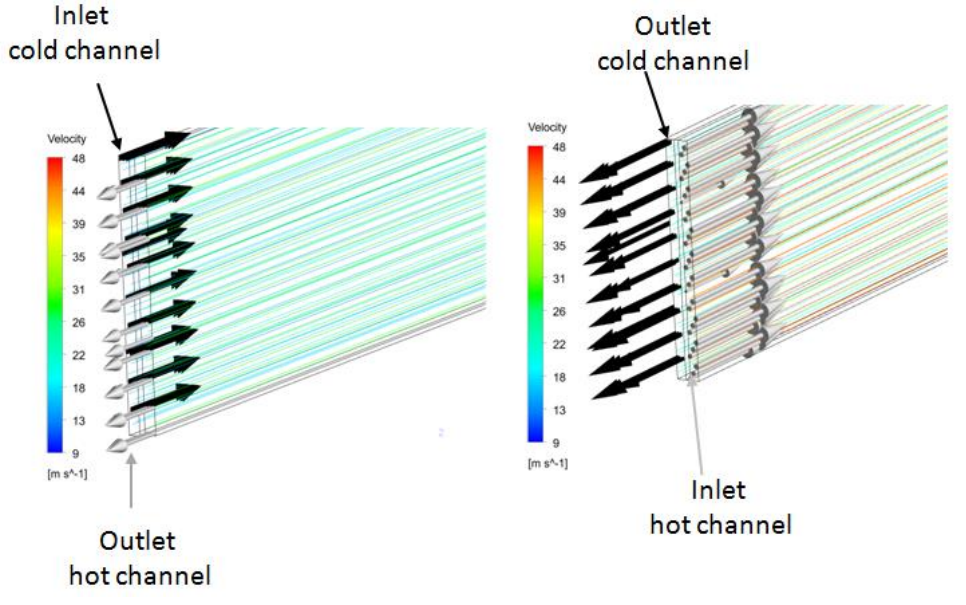

Fig. 3 – Velocity vectors and streamlines obtained in Case 11

Table 3 – The averaged numerical simulations results

Case No.	Cold Channel Outlet			Hot channel Outlet		η
	Total Temperature [K]	Total Pressure [bar]	Velocity [m/s]	Total Temperature [K]	Velocity [m/s]	
1	496	3.52818	65	801	80	0.044
2	497	3.53993	37	788	41	0.047
3	494	3.53250	56	792	56	0.040
4	500	3.54256	27	793	41	0.055
5	555	3.51448	32	529	22	0.194
6	578	3.52179	28	547	28	0.252
7	579	3.52176	28	547	28	0.255
8	614	3.53322	22	567	29	0.343
9	674	3.54202	11	614	31	0.492
10	641	3.52650	21	531	27	0.409
11	647	352018	21	508	26	0.426
12	713	3.53773	12	534	27	0.590
13	790	354354	7	626	32	0.780

The evaluation of the different Cases is based on effectiveness of each configuration, defined as:

$$\eta = \frac{T_{21}^{*} - T_{2}^{*}}{T_{4}^{*} - T_{21}^{*}} \tag{5}$$

where T_{21}^{*} is the total temperature at the inlet of the cold channel, T_{2}^{*} is the total temperature resulted at the outlet of the cold channel and T_{4}^{*} is the total temperature at the inlet of the hot channel [10].

After an overall analyse of the results presented in Table 3, it can be observed that the cross section area (varied by width) and area of transfer (varied by length) have a great influence on the effectiveness (Case 4 compared with Case 6 and Case 9 compared with Case 10, Case 11) while the width of the solid domain has a very small influence in these configurations (Case 6 compared with Case 7).

Fig. 4 shows the total temperature distributions plotted on the middle of the computational domain, for Cases 4 and Case 11.

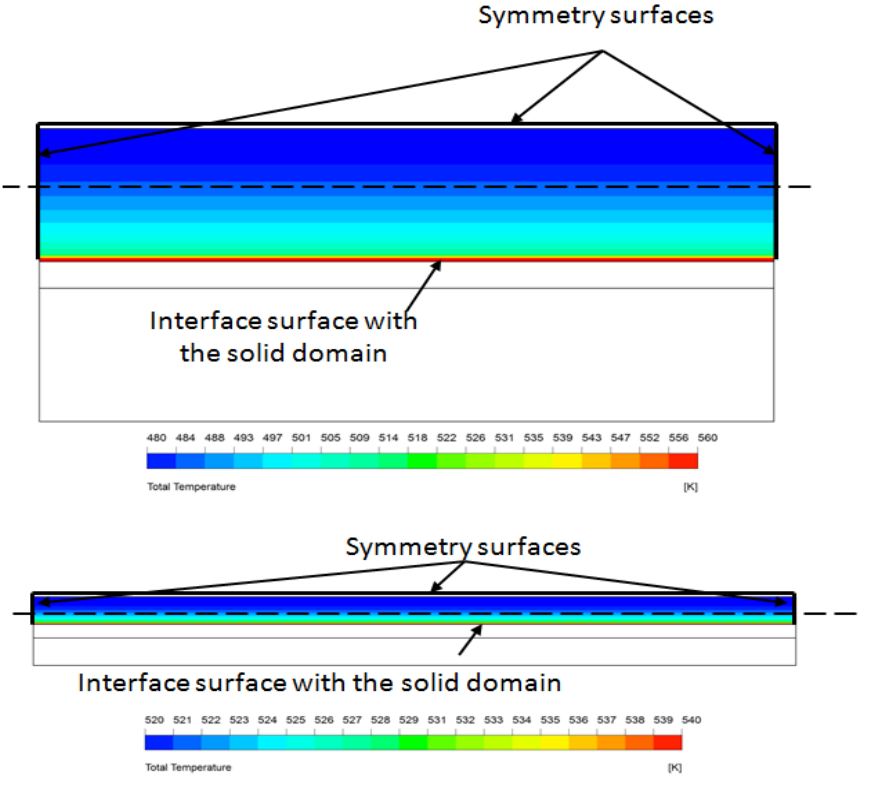

Fig. 4 – Total temperature distributions for Case 4 (up) and Case 12 (down)

It is noticeable, that regardless of configuration, the temperatures vary in width and the flow remains similar at different heights.

This phenomenon is expected, as the imposed conditions do not change in this direction. The limit of the thermal boundary layer for the cold channel -which is indicated in Fig. 4 with a dotted line is far from the surface that limits the computational domain (as this surface is set as Symmetry, it delimits the fluid domain in half).

This fact indicates that a possible improvement in the configuration's effectiveness would further decrease the width of the cold channel.

The width being an important and sensitive parameter of the channel is also sustained by Fig. 5, where it is obvious that at similar velocities, the smaller width configurations have better effectiveness.

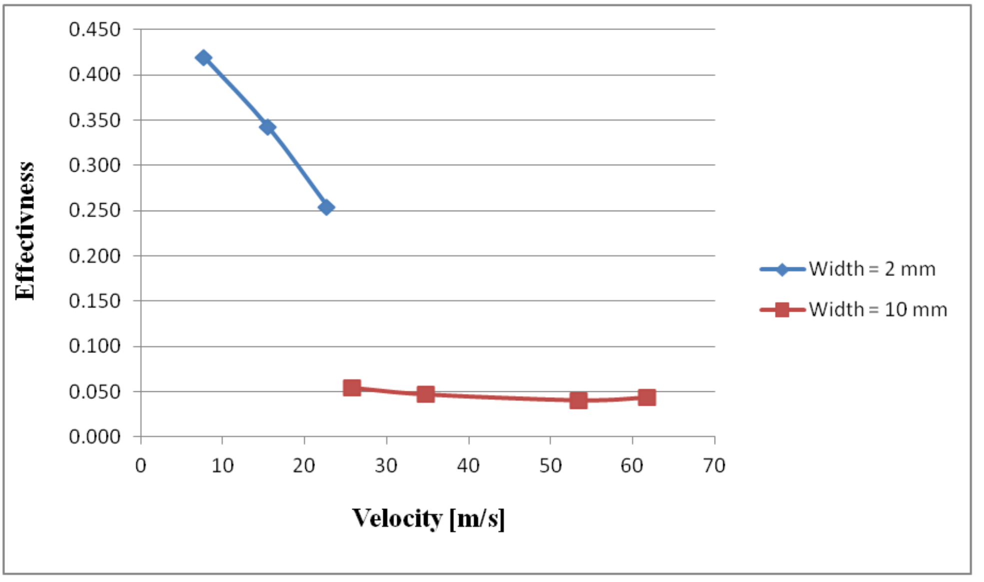

Fig. 5 – Effectiveness versus velocity at different cross section areas

The influence of the length on the configuration's effectiveness - presented in Fig. 6 - is smaller, but still, the transfer area has an important effect on the outlet total temperature of the cold channel.

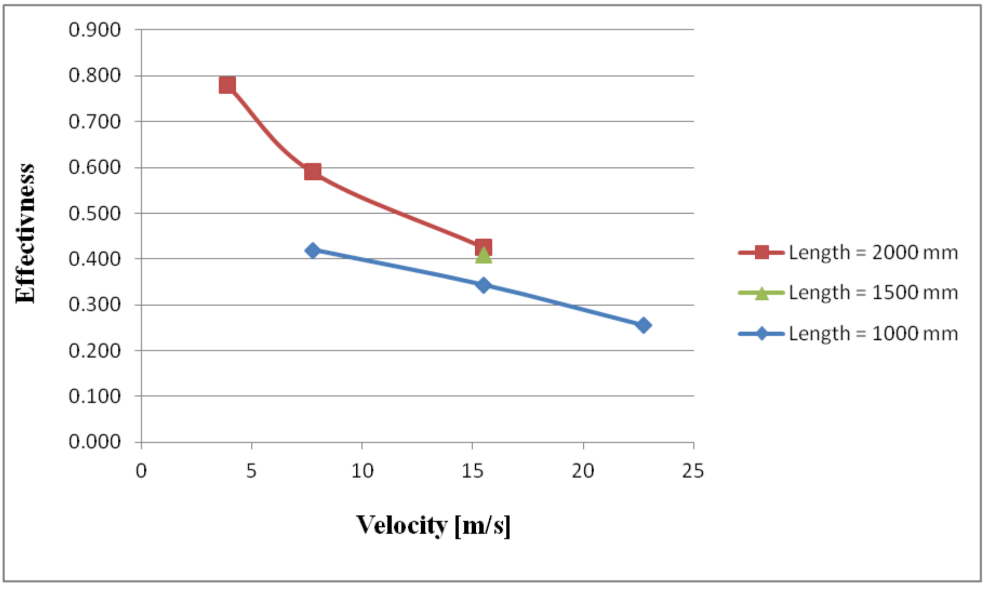

Fig. 6 – Effectiveness versus velocity at different transfer areas

The best configuration obtained is in Case 13 which has a length of 2 m. It is undeniable that the length will be not practical if used in an aviation application. The purpose of this study was to evaluate the influence of the overall dimensions of a cold stream recuperator on its performance and on a gas turbine performance; hence, even if for the presented work, the numerical simulations serve its purpose, to be able to fully design the recuperator, a future work will focus on compacting the fluid channels while keeping the transfer surfaces obtained in the present configurations with similar pressure losses.

5. CONCLUSIONS

A parametric study consisting of 13 RANS numerical simulations was carried out in order to assess the pressure loss through a heat recuperator equipping a gas turbine cogeneration group. A simplified model was used to describe the recuperator geometry. The numerical simulations were carried out while changing the cross-section area of the channel of the recuperator, the length of the channel of the recuperator, the number of the channels of the recuperator and the width of the solid domain. The height of the fluid channels, the recuperator materials and the boundary conditions were fixed for all the 13 simulated configurations.

The simulations used a three blocks computational domain, discretized by an unstructured grid consisting of about 850,000 elements for the fluid domain and about 12000 elements for the solid domain.

The analysis of the numerical simulations results has shown that the computational domain was well defined, as indicated by the flow streamlines. It was also verified that the ratio of the fluxes for the cold and hot streams, as computed through the numerical simulations remains within 5 % of the theoretical ratio of 1.

The averaged total temperature and at the outflow were determined, and the effectiveness of each simulated case was determined. It was found that both the cross section area and the heat transfer area have a strong impact on the recuperator's efficiency. On the contrary, the width of the solid domain has a very small influence.

The temperature gradient in the normal to the flow direction was found to be similar for all cases. Further optimization may be achievable if the width of the channel is decreased.

Finally, the optimal configuration was found to be Case 13, with a length of 2m, a width of 2 mm for the fluid channels, and of 1 mm for the solid domain, and with 307 cold and 174 hot channels.

Future work will focus on achieving a more compact design, while maintaining the current pressure losses.

REFERENCES

[1]* * * *Promovarea producerii de energie in cogenerare (01.04.2012)* http://www.pelifilip.com/wp-uploads/publications/b96e7-publication.pdf

[2] * * * Directiva 2004/8/CE a Parlamentului European si a Consiliului, 11.02.2004.

[3] E. Barbu, R. Petcu, V. Silivestru, V. Vilag, *Centrale Cogenerative cu Turbine cu gaze (Gas Turbine Cogeneration Plants),* INCDT COMOTI.

[4] * * * *The European Educational Tool on Congeneration*, Second Edition, December 2001.

[5] D. Annaratone, *Handbook for Heat exchangers and tube Banks design,* Springer-Verlag Berlin Heidelberg, 2010.

[6] * * * ANSYS, Inc Southpointe CFX Solver modeling-User guide, 2015.

[7] E. Carafoli, V. N. Constantinescu, *Dinamica fluidelor incompresibile (I)*, Editura Academiei, Bucuresti, 1981.

[8] A. Bejan, A.D. Kraus, *Heat transfer handbook*, John Wiley & Sons, 2003.

[9] L. HoSung, *Thermal Design Heat Sinks, Thermoelectrics, Heat Pipes, Compact heat Echangers, and Solar Cells*, John Wiley & Sons, 2011.

[10] E. U. Schlunder, *Heat Exchanger Design Handbook - Heat exchanger theory*, Hemisphere Publishing Corporation, 1983.