Computational study of the turbulent flow inside a heat exchanger using an incurved diffuser

S. FRIKHA[a], Z. DRISS[b], M.A. Haggui[c]

a. Frikha_sobhi@yahoo.fr
b. Zied.Driss@enis.rnu.tn
c. Aymenhaggui@yahoo.fr

Laboratory of Electro-Mechanic Systems (LASEM), National School of Engineers of Sfax (ENIS), University of Sfax (US), B.P. 1173, Road Soukra km 3.5, 3038 Sfax, TUNISIA

Résumé :

L'objectif de ce travail est d'étudier l'écoulement turbulent dans un échangeur de chaleur contenant un diffuseur incurvée. Pour cela, nous avons développpé des simulations numériques avec un code CFD. Le modèle numérique est basé sur la résolution des équations de Navier-Stokes couplées avec le modèle de turbulence k-ε. Ces équations sont résolues par discrétisation volumes finis. Particulièrement, nous sommes intéressés à visualiser la température, la vitesse, la pression totale, la pression dynamique, la vorticité et les caractéristiques de la turbulence.

Abstract:

The objective of this work is to study the turbulent flow inside a heat exchanger using an incurved diffuser. For thus, we have developed a numerical simulation using CFD code. The numerical model is based on the resolution of the Navier-Stokes equations in conjunction with k-ε turbulence model. These equations were solved by a finite volume discretization method. We are particularly interested to visualize the temperature, the velocity, the total pressure, the dynamic pressure, the vorticity, the turbulent kinetic energy, the turbulent dissipation rate and the turbulent viscosity.

Key words: heat exchanger, incurved diffuser, turbulent flow, CFD.

1 Introduction

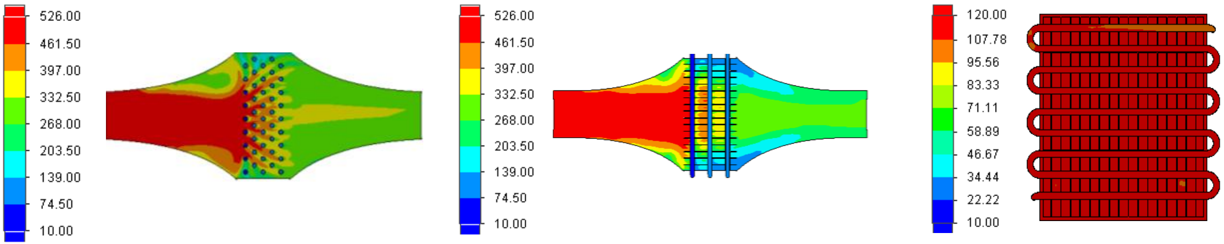
In industrial manufacturing processes, a large amount of thermal energy is directly dumped into the environment. This results in a significant waste of energy. Researchers confirm that more than 30-40% of fuel energy wastes from the exhaust and just 12–25% of the fuel energy convert to useful work [1–3]. The dumped thermal energy can be recovered using a heat exchanger in the exhaust. A heat exchanger is a heat transfer device that is used for transfer of internal thermal energy between two or more fluids available at different temperatures. The heat exchanger is an essential element to any policy of energy conservation. It is used mainly in industrial sectors (chemicals, petrochemicals, steel, food processing, energy production) and transportation (automotive, aeronautics), but also in the residential sector and tertiary (heating, air conditioning). The choice of a heat exchanger, for a given application depends on many parameters such as field temperature and pressure of fluids, and physical properties of these fluids, maintenance, cost and space. The size of the heat recovery system is strongly associated with heat recovery potential and fluid temperature. In the literature, relevant studies on heat exchanger networks [4-10] can be found. CFD numerical simulations have been used in the study of heat exchangers. For example, Zhang and Li [11] proposed a structure of two-stage-distribution and the numerical investigation shows that the flow distribution in plate-fin heat exchanger is more uniform if the ratio of outlet and inlet equivalent diameters for both headers is equal. Wen et al. [12] employed CFD technique to simulate and analyze the performance of fluid flow distribution and pressure drop in the header of plate-fin heat exchanger. Wasewar et al.[13] studied the flow distribution through a plate-fin heat exchanger. A modified header is proposed and simulated using CFD. The modified header configuration has a more uniform flow distribution than the conventional header configuration. In addition, the efficiency of the modified heat exchanger is seen to be higher than that of the conventional heat exchanger. Gan et al. [14] performed a CFD simulation on the tubes of a heat exchanger used in closed-wet cooling towers. Pressure drop was found to depend on the tube configurations and water to air ratio. The predicted pressure loss coefficient was found inversely proportional to transverse pitch, but was in direct relationship with water to air mass flow rate. In this context, we are interested in studying the turbulent flow inside a heat exchanger with an incurved diffuser. For thus, we make numerical simulations of the turbulent flow and we present all the results, such as temperature, pressure, velocity and turbulent characteristics.

2 Numerical results

The CFD code “SolidWorks Flow Simulation” was used for calculation. Two longitudinal planes defined by z=0 m and y=0 m and one transverse plane defined by x=0 m are considered.

2.1 Temperature

Figure 1 presents the distribution of the temperature in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, it is clear that the temperature is at its maximum at the entry of the diffuser, which is the value of the inlet boundary condition. The temperature distribution shows a decrease on the sides of the first diffuser and through the heat exchanger. Out of the heat exchanger, the temperature of the gas decreases, but still represents important values of more than 250 °C. This fact is due to the heat loss through the heat exchanger. According to the plane defined by x=0, the water reaches a temperature of 120 °C.

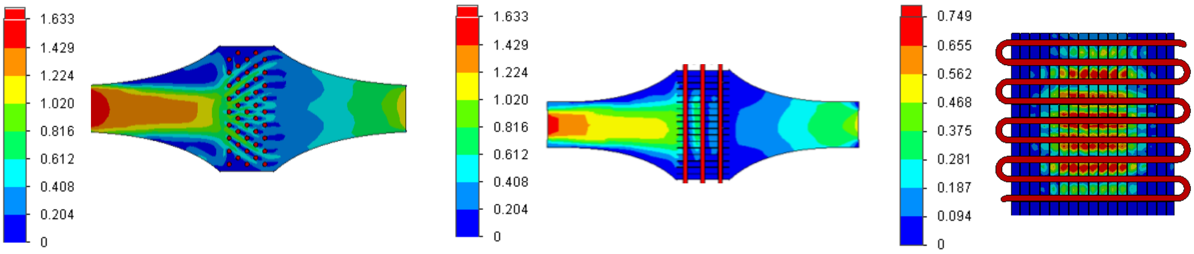


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig. 1. Distribution of the temperature

2.2 Magnitude velocity

Figure 2 presents the distribution of the magnitude velocity in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, the maximum value of the velocity appears in the gas inlet which is imposed by the boundary conditions. The velocity distribution shows a decrease in the first diffuser. Also, a decrease has been noted on the sides of the first diffuser which there is an important drop of the velocity. Out of the heat exchanger, the velocity decreases in the second diffuser. At the end of the diffuser, an increase of the velocity has been observed due to the reduction of the diffuser section. Indeed, it is clear that through the reduction of the size of the diffuser, the velocity value increases at the end of the diffuser. According to the transverse plane defined by x=0 m, the velocity in the tubes is constant. An increase of the magnitude velocity has been observed in the middle. Along the sides, it has been noted a decrease of the magnitude velocity.

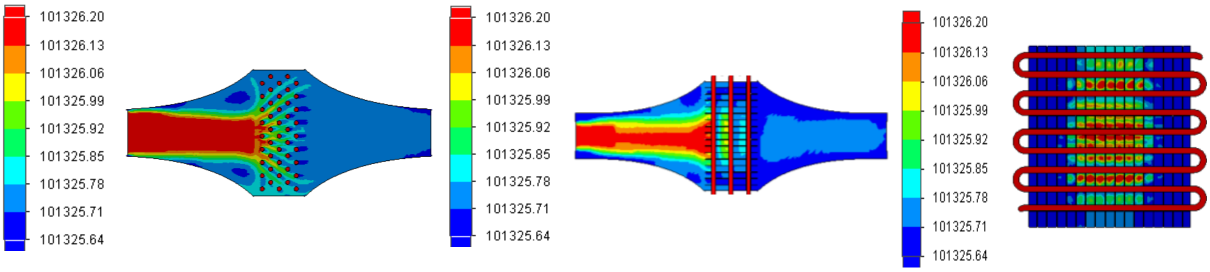


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig. 2. Distribution of the magnitude velocity

2.3 Total pressure

Figure 3 presents the distribution of the total pressure in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, a compression zone characteristic of the maximum values of the total pressure has been observed in the heat exchanger upstream and the middle of the diffuser. A progressive decrease has been observed on the sides of the first diffuser. A decrease of the total pressure has also been noted at the second diffuser and the heat exchanger downstream. The total pressure decreases out of the heat exchanger. In fact, the total pressure has approximately the same value and do not have a great value change in all the length of the diffuser. According to the transverse plane defined by x=0 m, the distribution of the total pressure is uniform in the tube and the difference is located in the heat exchanger. An increase of the total pressure has been observed in the middle. Along the sides, it has been noted a decrease of the total pressure.

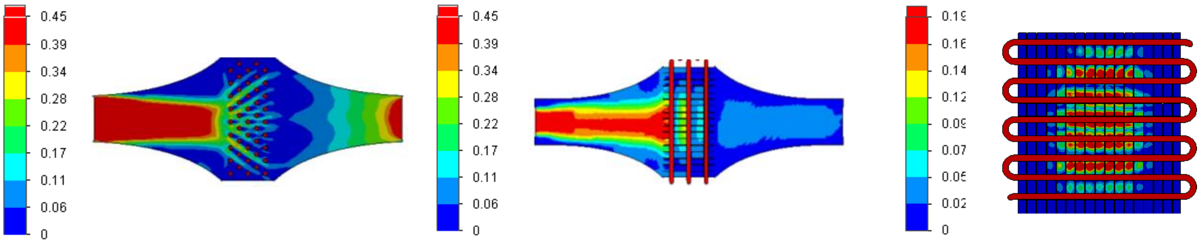


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig. 3. Distribution of the total pressure

2.4 Dynamic pressure

Fig. 4 presents the distribution of the dynamic pressure in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, the compression zones characteristics of the maximum values of the dynamic pressure are localized in the heat exchanger and the middle of the diffuser. A decrease of the dynamic pressure values appears in the sides of the diffuser and in the second diffuser. In the second diffuser and through the gas flow, the dynamic pressure decreases. This fact is due to the difference in the section and size of the diffuser. In the end of the second diffuser, a progressive increase of the dynamic has been observed. In the transverse plane defined by x=0 m, the distribution of the dynamic pressure is uniform in the tube. An increase of the dynamic pressure has been observed in the middle of the heat exchanger. However, along the sides, it has been noted a decrease of the dynamic pressure.

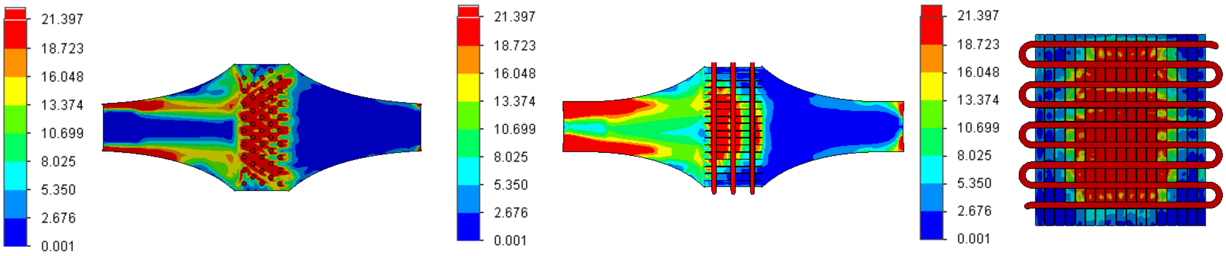


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig.4. Distribution of the dynamic pressure

2.5 Vorticity

Figure 5 presents the distribution of the vorticity in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, a wake characteristic of the maximum values of the vorticity have been observed in the entry of the gas inlet. In the sides and in the middle of the first diffuser, the values of the vorticity are very low. The vorticity presents an increase in the middle of the heat exchanger. In the heat exchanger downstream, the vorticity decreases. An increase of the vorticity has been noted in the outlet of the second diffuser. According to the transverse plane defined by x=0 m, the vorticity is constant in the tube. In the heat exchanger the vorticity is quite high in the middle and starts to decrease on the sides.

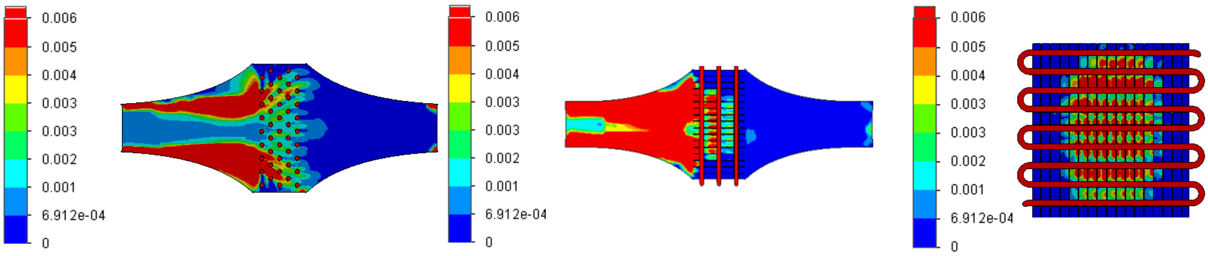


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig.5. Distribution of the vorticity

2.6 Turbulent kinetic energy

Fig. 6 presents the distribution of the turbulent kinetic energy in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, a wake characteristic of the maximum value of the turbulent kinetic energy has been observed in the first diffuser. The results show also a decrease in the value of the turbulent kinetic energy almost at the end of the heat exchanger and at the entry of the second diffuser. After then, the turbulent kinetic energy decreases progressively. The transverse plane defined by x=0 m presents a wake characteristic of the maximum value of the turbulent kinetic energy. In the sides, a progressive decrease of the turbulent kinetic energy has been observed. However, the turbulent kinetic energy in the tubes is constant.

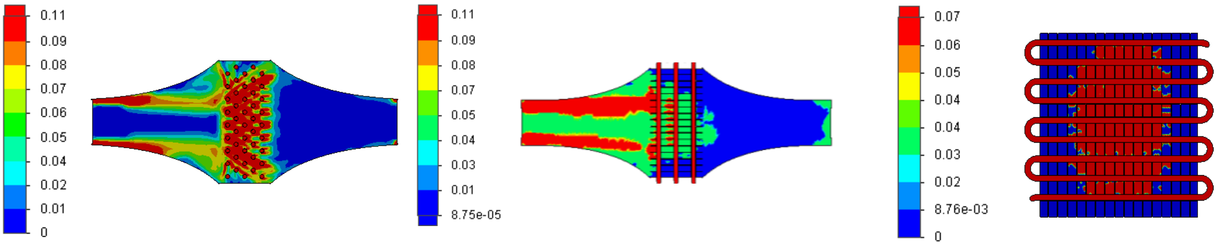


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig.6. Distribution of the turbulent kinetic energy

2.7 Dissipation rate of the turbulent kinetic energy

Figure 7 presents the distribution of the dissipation rate of the turbulent kinetic energy in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, a wake characteristic of the maximum values of the dissipation rate of the turbulent kinetic energy has been observed in the sides of the diffuser. In the middle of the diffuser and in the gas inlet, the dissipation rate of the turbulent kinetic energy decreases. The wake extension has been observed until the gas leaves the heat exchanger where there is a great drop on the dissipation rate of the turbulent kinetic energy values. The plane x=0 m shows a wake characteristic of the maximum values of the dissipation rate of the turbulent kinetic energy in the middle of the planes. The dissipation rate of the turbulent kinetic energy decreases in the other sides of the diffuser. However, the dissipation rate of the turbulent kinetic energy is constant in the tubes.

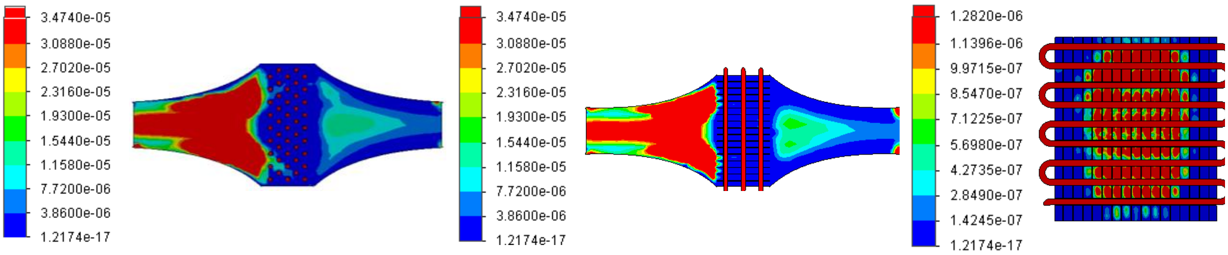


(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig.7. Distribution of the dissipation rate of the turbulent kinetic energy

2.8 Turbulent viscosity

Figure 8 presents the distribution of the turbulent viscosity in the considered planes. According to the longitudinal planes defined by z=0 m and y=0 m, a wake characteristic of the maximum values of the turbulent viscosity has been observed in the first diffuser. The turbulent viscosity values decrease importantly in the heat exchanger. The results show also an increase in the middle of the second diffuser and a decrease in the sides of the diffuser. In the plane defined by x=0 m, an increase in the middle of the heat exchanger and a progressive decrease in the sides have been observed.



(a) Plane z=0 m (b) Plane y=0 m (c) Plane x=0 m

Fig.8. Distribution of the turbulent viscosity in the transverse plane x=0 m

3 Conclusions

In this work, numerical simulations have been developed to study the turbulent flow inside a heat exchanger using an inurved diffuser. We present all the results from simulation, such as temperature, velocity, total pressure, dynamic pressure, vorticity, turbulent kinetic energy, turbulent dissipation rate and turbulent viscosity. According to the numerical results, fluid flow characteristics decrease in the sides of the first diffuser and in the second diffuser out of the heat exchanger. This knowledge will be used in the construction of the heat recovery system.

Acknowledgments

This work is done in collaboration with the Tunisian Drilling Company.

References

[1] M. Hatami, D.D. Ganji, M. Gorji-Bandpy, A review of different heat exchangers designs for increasing the diesel exhaust waste heat recovery, Renew. Sust. Energ Rev. 37 (2014) 168–181.
[2] R. Saidur, N.A. Rahim, H.W. Ping, M.I. Jahirul, S. Mekhilef, H.H. Masjuki, Energy and emission analysis for industrial motors in Malaysia, Energy Policy 37 (9) (2009) 3650–3658.
[3] M. Hasanuzzaman, N.A. Rahim, R. Saidur, S.N. Kazi, Energy savings and emissions reductions for rewinding and replacement of industrial motor, Energy 36(1) (2011) 233–240.
[4] Soylemez M.S., Optimum length of finned pipe for waste heat recovery, Energy Conversion and Management 49 (2008) 96–100.
[5] Stehlik P., Heat transfer as an important subject in waste-to-energy systems, Applied Thermal Engineering 27 (2007) 1658–1670.
[6] Dagdas A., Heat exchanger optimization for geothermal district heating systems: a fuel saving approach, Renewable Energy (2007) 32:1020–1032.
[7] Borekal M.U., Sapkal V.S., Sapkal R.S., Study of optimum design-parameters of a condensing heat exchanger (CHX) for waste heat recovery, Advances in Energy Research (2006) 245–249.
[8] Navarro A.H., Gomez L.C., A new approach for thermal performance calculation of cross-flow heat exchangers, International Journal of Heat and Mass Transfer 48 (2005) 3880–3888.
[9] Sun S.Y., Lu Y.D., Yan C.Q., Optimization in calculation of shell tube heat exchanger, International Communications in Heat and Mass Transfer 20 (5) (1993) 675–687.
[10] Soylemez M.S., On the optimum heat exchanger sizing for heat recovery, Energy Conversion and Management 41 (2000) 1419–1427.
[11] Zhang, Z., & Li, Y. Z., CFD simulation on inlet configuration of plate-fin heat exchanger. Cryogenics, (2003) 43(12), 673–678.
[12] Wen, J., Li, Y. Z., Zhou, A., Zhang, K., & Wang, J., Study of flow distribution and its improvement on the header of plate-fin heat exchanger. Cryogenics, (2004) 44, 823–831.
[13] Wasewar, K. L., Hargunani, S., Atluri, P., Kumar, N., CFD simulation of flow distribution in the header of plate-fin heat exchanger. Chemical Engineering and Technology, (2007) 30(10), 1340–1346.
[14] Gan G., Riffat S.B., Shao L., CFD modelling of pressure loss across tube bundles of a heat exchanger for closed-wet cooling towers. International Journal Ambient Energy (2000) 21, 77-84.