NUMERICAL INVESTIGATION OF THE EFFECT OF AIR FLOW RATE ON FLOW AND PRESSURE DISTRIBUTION IN A TURBOCOMPRESSOR HEAT EXCHANGER ROOM

Ogan Karabas*,[1], K. Suleyman Yigit*, Ahmet Tasci, Gökay Guney**, Umitcan Guler,****
*Faculty of Eng. Dep. of Mechanical Engineering Kocaeli University 41380 Kocaeli, Turkey,
ogan.karabas@kocaeli.edu.tr, kyigit@kocaeli.edu.tr
** IHI DALGAKIRAN Machinery Industry and Trade Inc. 41455 Kocaeli, Turkey,
a.tasci@ihidalgakiran.com, g.guney@ihidalgakiran.com, u.guler@ihidalgakiran.com

Abstract: Cooling the air in the heat exchanger room between compression stages in turbocompressors affects the general performance and efficiency of the turbocompressors. One of the determining factors of cooling performance in this regard is the flow distribution in the heat exchanger room. Uniform flow distribution in the heat exchanger room inlet region increases the air cooling performance of the heat exchanger. In this study, the heat exchanger room of a turbocompressor designed by IHI DALGAKIRAN was modelled in 3D and the effect of the inlet air flow rate on the flow distribution in the heat exchanger room was investigated with CFD analyses performed with ANSYS Fluent fluid simulation software for 3 different inlet air flow rates determined according to the operating conditions. By obtaining the path lines of air particles in the heat exchanger room, the effect of the inlet air flow rate on the flow distribution in the heat exchanger room was examined. Moreover, the pressure distributions at the outlet region of the heat exchanger room were obtained by CFD analyses, and the effect of the inlet air flow rate to the heat exchanger room on the outlet pressure of the exchanger room was also examined.
Keywords: Turbocompressor, Heat exchanger room, CFD, Air flow rate, Flow distribution, Pressure distribution

BİR TURBOKOMPRESÖRDE HAVA DEBİSİNİN ISI DEĞİŞTİRİCİ ODASINDAKİ AKIŞ VE BASINÇ DAĞILIMINA ETKİSİNİN NÜMERİK İNCELENMESİ

Özet: Turbokompresörlerde sıkıştırma kademeleri arasında yer alan ısı değiştirici odasında havanın soğutulması turbokompresörlerde genel performansı ve verimi etkiler. Bu bağlamda soğutma performansını belirleyici faktörlerden biri de ısı değiştirici odasındaki akış dağılımıdır. Isı değiştirici odası giriş tarafında akış dağılımının düzenli olması ısı eşanjörünün hava soğutma performansını iyileştirir. Bu çalışmada IHI DALGAKIRAN tarafından üretilmiş bir turbokompresör 3 boyutlu modellenmiş ve çalışma şartlarına göre belirlenen 3 farklı hava debisi için ANSYS Fluent akışkan simülasyon yazılımı ile HAD (Hesaplamalı akışkanlar dinamiği) analizleri gerçekleştirilerek ısı değiştirici odasına giren hava debisinin ısı değiştirici odasındaki akış dağılımına etkisi araştırılmıştır. Isı değiştirici odasındaki hava partiküllerinin akım çizgileri elde edilerek giriş hava debisinin ısı değiştirici odasındaki akış dağılımına etkisi incelenmiştir. Ayrıca, HAD analizleri ile ısı değiştirici odası çıkış tarafında basınç dağılımları da elde edilerek ısı değiştirici odasına giren hava debisinin ısı değiştirici odası çıkış basıncına etkisi de incelenmiştir.
Anahtar Kelimeler: Turbokompresör, Isı değiştirici odası, HAD, Hava debisi, Akış dağılımı, Basınç Dağılımı

INTRODUCTION

Centrifugal turbo compressors are applied to industrial compressors because of their high flow rate, high efficiency, reliability, relatively wide operating range and cost benefits (Ibaraki et al., 2014). Typical centrifugal plant air compressors have more than one stage as shown in **Figure 1**, because they typically can compress about 2:1 to 3:1 per stage, and a plant air system needs about 4:1 or more pressure rise.



Figure 1. Turbocompressor stages

Between those stages pressurized air whose temperature is increased, needs to be cool down before it enters the next stage. Therefore, between the stages, the sections shown in **Figure 2** include intercooler.



Figure 2. Locations of air coolers

As seen in the P-V diagram in **Figure 3**, this pressurizing and cool down process is repeated until air reaches desired outlet pressure. Today, according to desired flow rate and outlet pressure, turbo compressors with 3 or 4 stages are mostly preferred. This method is the most efficient and feasible method in compressing theory.

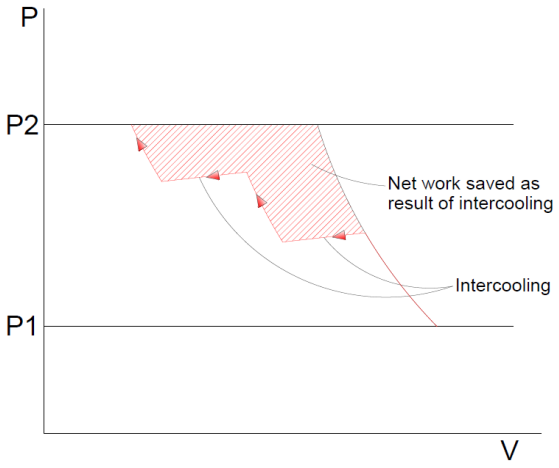
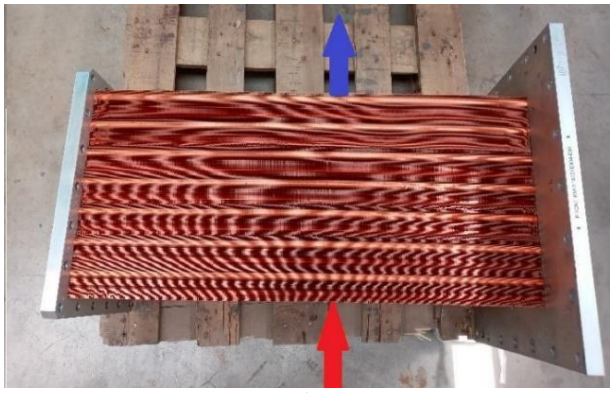
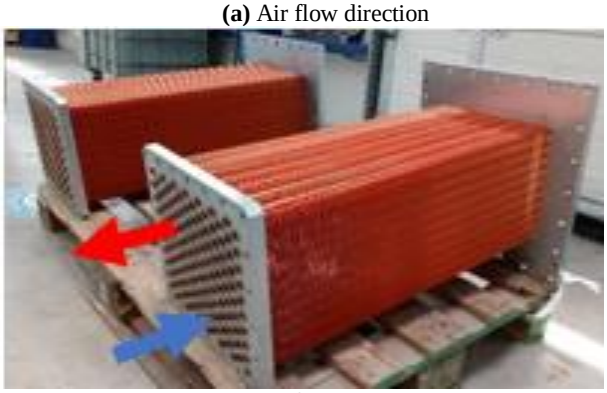


Figure 3. P-V diagram for multistage compression

Generally, enhanced fin and tube heat exchangers are used as intercooler in turbocompressors. While the hot air passes between the fins on the direction shown in **Figure 4a**, the coolant (eg water) passes through in the pipes on the direction shown in **Figure 4b** and the hot air cool down.



(a) Air flow direction



(b) Coolant flow direction

Figure 4. (a) Air, **(b)** Coolant flow direction in heat exchanger

During that travel from inlet to outlet indicated in **Figure 5**, air needs a cover to be directed and passed on.

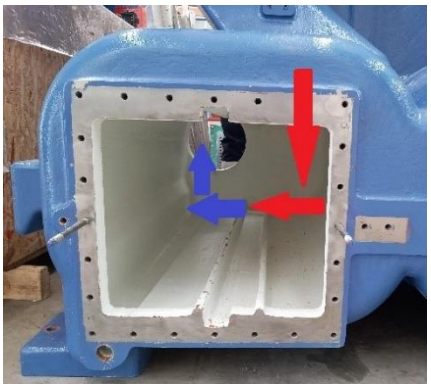


Figure 5. Air flow in heat exchanger room

There are casted parts in the structure of turbocompressors. Form of that casted parts are quite important to prevent of pressure drops etc. since it affects the flow and designing those parts are one of the most challenging engineering problems in turbocompressor technology. Since casting and machining process of those

parts need too much time and investment, the final product design must be completed by simulating the air flow in the turbocompressors.

Also, flow rate affects to performance of heat exchanger. In a fixed heat exchanger room dimensions, heat exchanger also has limited dimensions. Thus dimension between fins and flow rate affects the performance of heat exchanger. If space between fins is very small, air speed increases. High speed of air causes low heat exchanging efficiency. If dimension between fins is bigger, some amount of air can pass with less heat exchanging ratio between fins.

Aero design of turbocompressor is very important to catch requested flow rate and pressure ratio with in efficient line. There are many factors to affect performance and efficiency of turbocompressor which are as suction pressure, temperature, humidity, cooling water temperature, flow rate, heat exchanger performance etc. Aero performance of all stages affects to total performance. If cannot catch requested aero specifications between stages, at least cannot reach to requested pressure ratio or flow rate in efficient line. All aero parts are calculated and designed in accordance with some air specification. From suction to discharge, all stages are related with other one. Therefore, condition of air is very important throughout compression and cooling line.

Several studies have been made during the period 1950 to 1980 to calculate compressors performance accurately. As a result of those studies, some mathematical models have been derived and developed. Since the most undesired staff is pressure drops in turbocompressors due to its nature, researchers focused on the components and designs causing pressure drops.

Impellers and diffusers are the key components of pressurizing process. Therefore, they were initially the most widely researched components. Khoshkalam et al., 2019, proposed a method to calculate losses in impeller and diffuser. Besides, they compared the importance of detected losses in different rotational speeds and operating range of turbocompressor.

Doustmohammadi et al., 2013, developed an analytical model for centrifugal compressors. In this study, they investigated loss mechanisms in impeller, diffuser and volute separately for wide operating rotational speeds of the compressor. They used entropy generation approach to determine compressor efficiency. The results of those experiments have verified with test results.

Hazizi et al., 2019, developed another numerical model and analysed whole compressor from inlet to outlet with components by using CFD. They also verified their results with experiments.

After researchers performed many studies about these key components, the efficiency of those components has sharply been improved. Recently, it has been determined that the most important sources of loss are heat exchangers between stages. After that researchers focused on that area to improve compressors performance.

Cooling performance and turbocompressor outlet parameters are strongly affected by exchanger design parameters such as fin structure, fin pitch and number of tube rows. Therefore, both theoretical and experimental studies have been made. In addition, by understanding better the air flow around exchangers, researchers aimed to design an exchanger with maximum cooling performance and minimum pressure drop.

In many studies, to simplify the model cross flow heat exchangers have been modelled as porous media. Banerjee et al., 2015, determined the permeability and inertial coefficients by pressure drop experiments. They measured pressure drop for per unit length of exchanger and applied Darcy–Forchheimer equation to determine these parameters.

Shen and Zhang, 2017, have investigated the effect of air flow on cross flow heat exchangers. They have created a mathematical model to define viscous and inertial resistances through the exchanger. Then they compared their results with experimental data. The results have shown that their mathematical model is quite close to real situation.

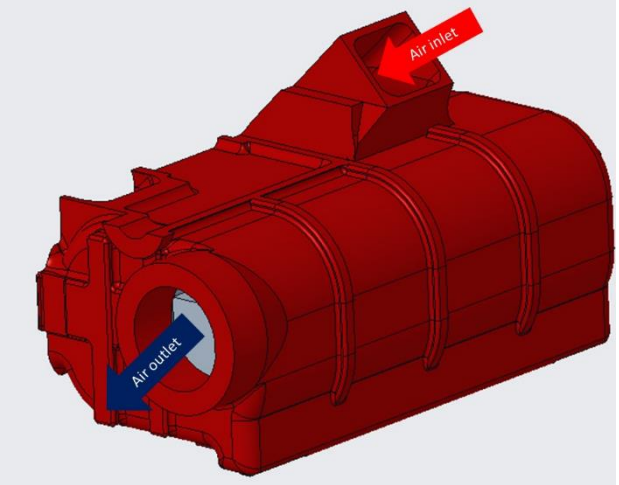
Yaïci et al., 2014, investigated the distribution of air flow inside a fin and tube heat exchanger and discussed the effects of the flow for the heat transfer performance of the heat exchanger

In this study presented, CFD analyses were carried out at 3 different flow rates determined according to the operating conditions in order to examine the effect of air flow rate on flow distribution in heat exchanger room of a turbocompressor designed by IHI DALGAKIRAN. Path lines were obtained to compare the flow distributions in the heat exchanger room for these 3 cases. In addition, pressure distributions were obtained for all 3 cases in the outlet region of the heat exchanger room.

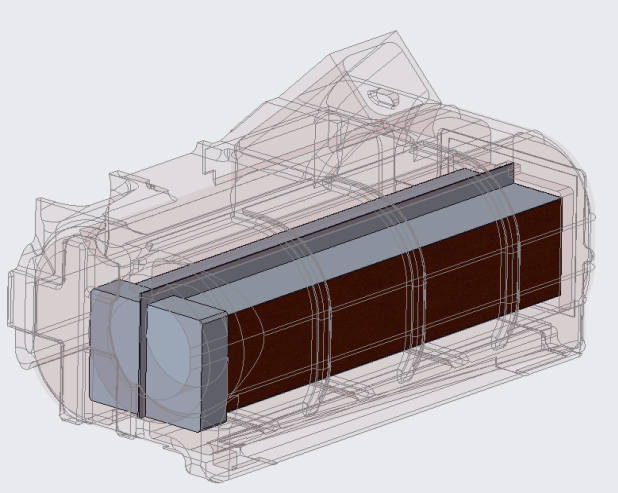
METHOD

Modelling Exchanger Room

The exchanger room to be analysed for CFD has been modelled in 3D as shown in the **Figure 6a**. Also, the heat exchanger is clearly seen in this figure, which shows the air inlet to the heat exchanger room and the air outlet from the heat exchanger room. Moreover, the location of the heat exchanger in the heat exchanger room is shown in **Figure 6b**.



(a)3D Analysis model



(b) Transparent view of analysis model

Figure 6. Analysis model **(a)** 3D, **(b)** Transparent

The discretization of the exchanger room to 14672626 elements, excluding heat exchanger, for CFD analyses is shown in **Figure-7**.

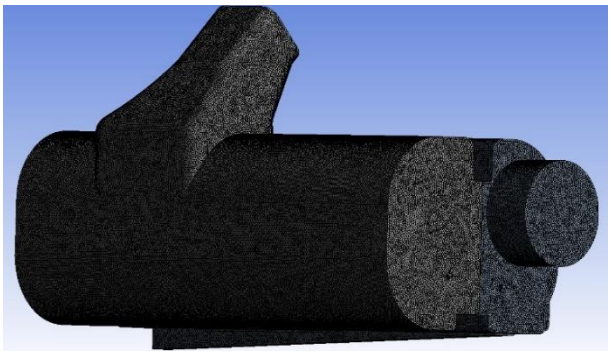


Figure 7. Discretization of heat exchanger room

Modelling Heat Exchanger

Another critical issue is modelling the exchanger. In original design, exchanger has more than 1500 fins. As shown in **Figure 8**, these fins thicknesses (0,6 mm) and the space between fins (1 mm) are quite small.

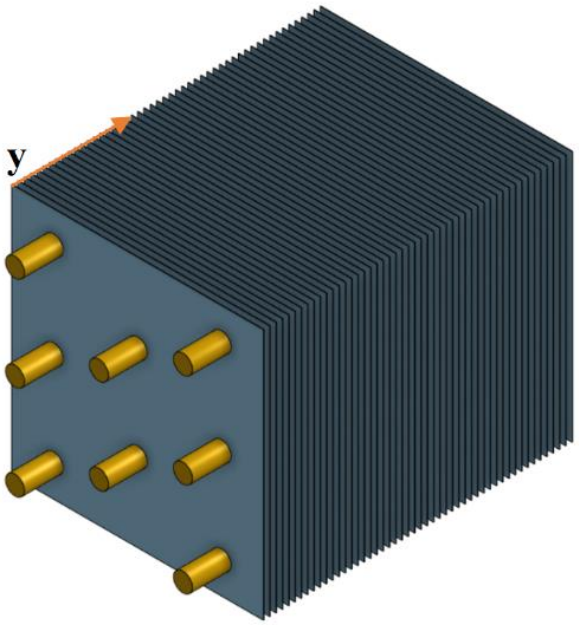


Figure 8. Structure of heat exchanger

Therefore, meshing and solving the original geometry is quite difficult. For this reason, exchanger has been drawn as blocks and modelled as porous media to define vacancies inside defined volume (Banerjee et al., 2015). Considering the vacancy volume in the heat exchanger, the permeability is determined 0,67. The exchanger modelled in that study is known as cross flow exchanger in literature. Air pass through between fins but cannot pass through normal to fins surface. Therefore, to restrict air flow on y-direction, so high viscous and inertial resistance are defined on y-direction to porous media. For this reason, as seen in the section of the exchanger room in **Figure 9**, heat exchanger is designed as a bulk and modelled in 3D as porous media to define vacancies.

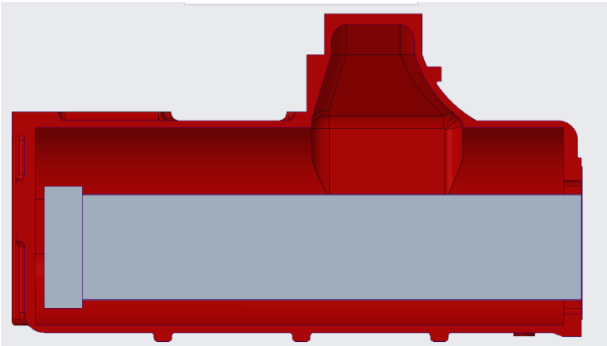


Figure 9. Sectioned view of analysis model

The discretization of the heat exchanger to 1271284 elements for CFD analyses is shown in **Figure 10**.

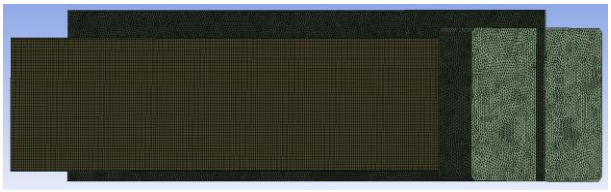


Figure 10. Discretization of heat exchanger

CFD Methodology

In this study, the air flow in the heat exchanger room was simulated steady by using Ansys Fluent Fluid Simulation software, with air at the same pressure and temperature at different flow rates. As turbulence model, realizable k-epsilon model was used. In this model, an improved method is used to calculate turbulent viscosity. Turbulent viscosity is not considered as constant just as k-epsilon model (Shih et al., 1995). Besides, boundary layer characteristic in a large pressure gradient is predicted better (Shaheed et al., 2019). The governing equations used to carry out the CFD analyses continuity **Eq.(1)**; conservation of momentum **Eq.(2)**; conservation of energy **Eq.(3)**; turbulence kinetic energy rate **Eq.(4)**; turbulence dissipation rate **Eq.(5)**

$$\frac{\partial \rho}{\partial t} + \rho\left(\nabla . \vec{V}\right) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \tag{2}$$

$$\rho\left[\frac{\partial h}{\partial t} + \nabla.(h\vec{V})\right] = -\frac{Dp}{Dt} + \nabla.(k\nabla T) + \varphi \tag{3}$$

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{x_j}\left(\rho k u_j\right) = \frac{\partial}{x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{x_j}\left(\rho\varepsilon u_j\right) = \frac{\partial}{x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{x_j}\right] + \rho C_1 S\varepsilon + \rho C_2 \frac{\varepsilon^2}{k + (\nu\varepsilon)^{0.5}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon \tag{5}$$

where ρ, t, $\vec{V}$, $\bar{\bar{\tau}}$, $\vec{f}$, h, k, T, φ, k, ε,μ, μ_t are; density, time, velocity vector, viscous stress tensor, body force per unit mass, enthalpy, thermal conductivity, temperature, dissipation function referring to work done against viscous forces, turbulence kinetic energy, generation of turbulence kinetic energy, dissipation rate, viscosity and turbulent viscosity, respectively. G_k and G_b represents the generation of turbulence kinetic energy due to the mean velocity gradients and buoyancy. Y_M indicate contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. C_2 and $C_{1\varepsilon}$ are model constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε. S_k and S_ε are user-defined source terms. C_1 is determined by **Eq.(6)** and **Eq.(7)**. S is the modulus of the mean rate-of-strain tensor, defined as **Eq.(8)**. (Ansys, 2021)

$$C_1 = \max\left[0{,}43, \frac{\eta}{\eta + 5}\right] \tag{6}$$

$$\eta = S\frac{k}{\varepsilon} \tag{7}$$

$$S = \sqrt{S_{ij}S_{ij}} \tag{8}$$

Boundary Conditions

Boundary conditions for CFD analyses are given in **Table 1**. According to the operating conditions, 3 inlet flow rates were determined as 21500 m^3/h, 15050 m^3/h and 10535 m^3/h for 3 cases. These flow rates were defined as uniform and perpendicular to the inlet cross-section. For all 3 cases, the air pressure at the inlet to the exchanger room was 1,5 bar(G) and its temperature is 121,3 °C.

Table 1
Boundary Conditions

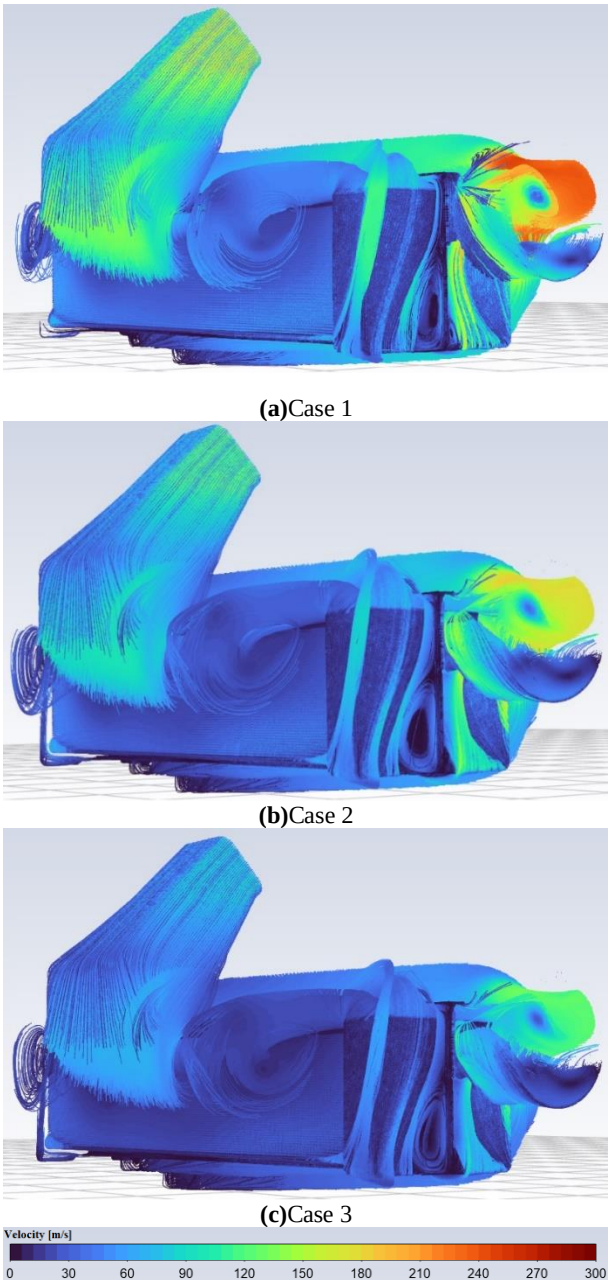
	Case 1	**Case 2**	**Case3**
$\dot{Q}_{in}$ [m^3/h]	21500	15050	10535
P_{in} [bar(G)]	1,5	1,5	1,5
T_{in} [°C]	121,3	121,3	121,3

RESULTS

Flow Distribution

A numerical study was performed to determine the effect of the inlet flow rate at the heat exchanger room on the general performance of the turbocompressor. In this regard, the flow distribution in the heat exchanger room is one of the important factors. According to the results of CFD analyses, the flow distributions in the heat exchanger room by obtaining the velocity magnitude of the fluid particles in the control volume are presented in this section.

Flow distributions in the perspective view of the heat exchanger room for each of the 3 cases are shown in **Figure 10**. In this figure, it is seen that the air particles follow almost the same path lines in the heat exchanger room in every 3 cases until they pass through heat exchanger. Also, it is clearly seen in this figure that the air particles follow the same path lines at different speeds.



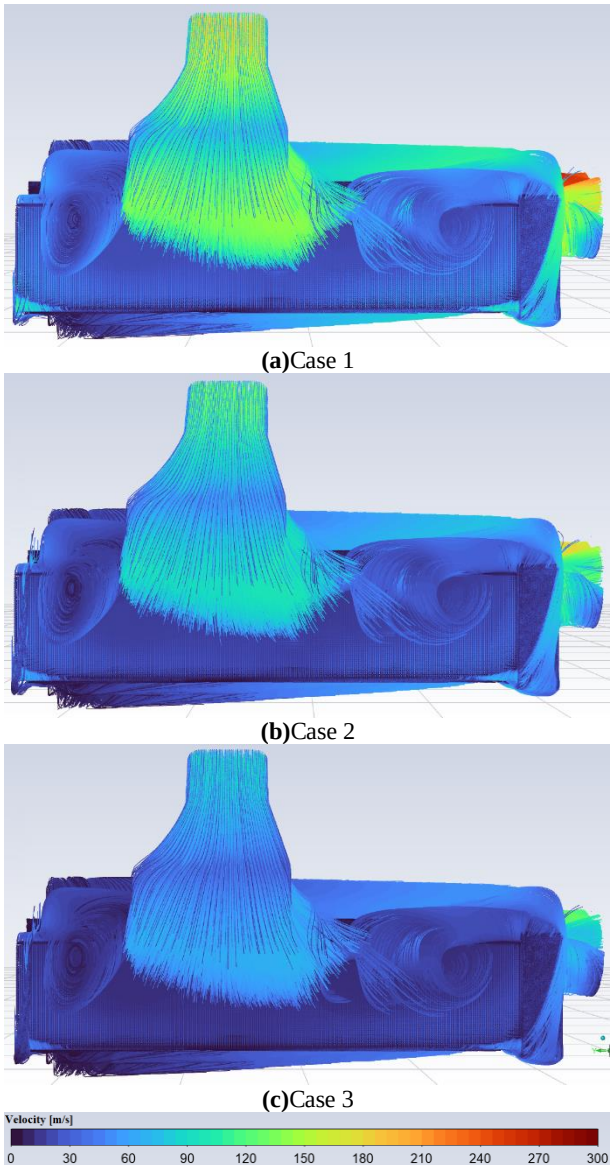
(a) Case 1

(b) Case 2

(c) Case 3

Figure 10. Flow distribution in heat exchanger room **(a)** in Case 1, **(b)** in Case 2, **(c)** in Case 3

The flow distributions in the heat exchanger room inlet region are shown in **Figure 11** normal to the heat exchanger passage. According to this figure, it is seen in all 3 cases that the air entering the heat exchanger room cannot pass through the heat exchanger uniformly. It is obvious that the air does not pass through the entire heat exchanger and there are areas of the heat exchanger that cannot be used for air cooling. Although the velocities of air particles are different in these areas, it is clearly seen that the areas that cannot be used for each cases are almost the same.



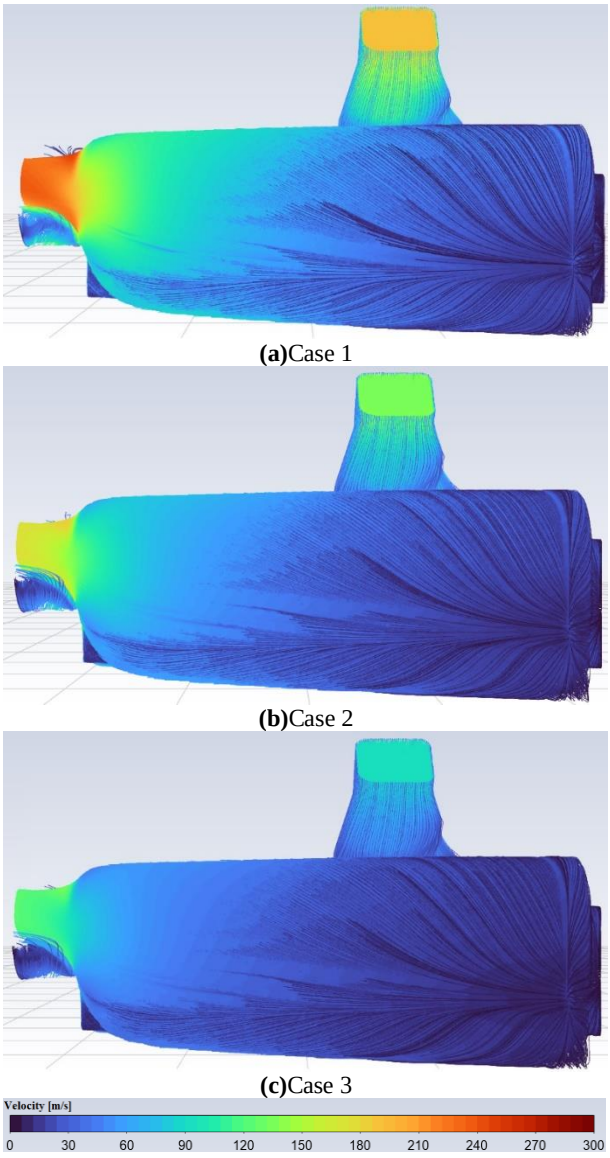
(a) Case 1

(b) Case 2

(c) Case 3

Figure 11. Flow distribution in heat exchanger room inlet region **(a)** in Case 1, **(b)** in Case 2, **(c)** in Case 3

The flow distributions in the heat exchanger room outlet region are shown in **Figure 12** normal to the heat exchanger passage. In this figure, it is seen that the air particles coming out of the heat exchanger follow the same path lines in every 3 cases until the outlet of the heat exchanger room. In addition, it is seen that the outlet velocity of the air particles from the heat exchanger room is directly proportional to the inlet flow rate, as expected.



(a) Case 1

(b) Case 2

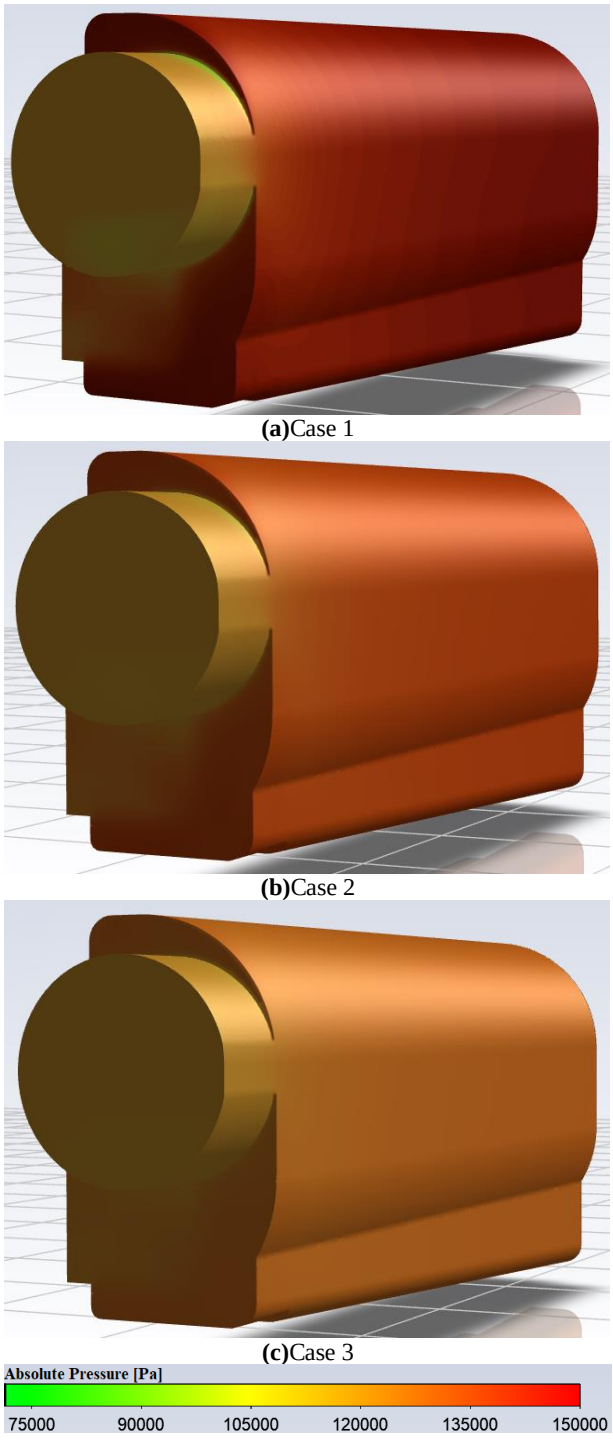
(c) Case 3

Figure 12. Flow distribution in heat exchanger room outlet region **(a)** in Case 1, **(b)** in Case 2, **(c)** in Case 3

Pressure Distribution

Another important factor in the results of the CFD analyses to evaluate the general performance of the turbocompressor is outlet pressure. Pressure distributions according to the results of CFD analyses for all 3 cases in the outlet region of the heat exchanger room, which is determined as the control volume, are presented in this section.

The pressure distributions at the heat exchanger outlet region are shown in **Figure 13** as volume average absolute pressure for each 3 cases. It is clearly seen in this figure that according to the results of the CFD analyses performed with the assumed model, the inlet flow rate to the heat exchanger room and outlet pressure of the heat exchanger room are directly proportional.



(a) Case 1

(b) Case 2

(c) Case 3

Figure 13. Pressure distribution at outlet region of heat exchanger room **(a)** in Case 1, **(b)** in Case 2, **(c)** in Case 3

The volume average absolute pressure values calculated for 3 cases are shown in **Table 2**.

Table 2
Outlet Pressure

	Case 1	**Case 2**	**Case3**
Volume Avg. Pressure [Pa]	135377,3	119780,3	110411,7

CONCLUSION

The study results show that in all 3 cases, the fluid particles follow the same path lines in the heat exchanger room. In other words, the air flow rate has no effect on the flow distribution in the heat exchanger room under the specified conditions. Moreover, in all 3 cases it is seen that the air particles are not uniformly distributed in the heat exchanger room. This indicates that the heat exchanger cannot be used at full capacity and means that the air passing through the heat exchanger will cool less than possible. Considering that leaving the air as cold as possible from the heat exchanger room increases the compressor efficiency, it is obvious that one of the issues that needs to be developed for the compressor in which this study is carried out is to provide that the uniform air flow distribution in the heat exchanger room. Considering that the inlet air flow rate to the heat exchanger room has no effect on an air flow distribution, it is clear that design changes are needed to direct the air flow to the heat exchanger uniformly.

Another result obtained from the study is that the pressure drop in the heat exchanger room decreases as the inlet air flow rate increases. It has been determined that as the inlet flow rate to the heat exchanger room decreases, the air particles are exposed to more resistance while passing through the heat exchanger, which is assumed as porous media, so the pressure drop increases. However, it is seen that as the inlet flow rate to the heat exchanger room increases, the pressure drop increases while the air particles pass to the circular cross section at the outlet of the heat exchanger room. This shows that a design change is needed in the transition section of the heat exchanger room to the circular section at the outlet so that the turbocompressor can operate more efficiently with a high inlet flow rate.

It is recommended for further studies to verify the CFD analyses results with experiments, except that the flow distribution in the heat exchanger room is uniform and design changes are made to reduce the pressure drop at the transition to the circular cross section at the heat exchanger room outlet.

NOMENCLATURE

CFD	Computational fluid dynamics
G	Gauge pressure
P	Pressure [Pa]
P_{in}	Inlet pressure [Pa]
$\dot{Q}_{in}$	Inlet flow rate [m^3/h]
T_{in}	Inlet temperature [$^{\circ}C$]
V	Volume [m^3]

REFERENCES

Ansys Inc., 2021, ANSYS FLUENT theory guide, Canonsburg, PA 15317.

Banerjee A., Chandran R. B. and Davidson J. H., 2015, Experimental investigation of a reticulated porous alumina heat exchanger for high temperature gas heat recovery, *Applied Thermal Engineering*, 75, 889-895.

Doustmohammadi A. A., HajilouyBenisi, A. and Mojaddam M., 2013, Experimental and numerical investigation of losses in centrifugal compressor components, *Turbo Expo: Power for Land, Sea, and Air* (Vol. 55249, p. V06CT40A015).

Hazizi K., Ramezanpour A., Costall, A. and Asadi M., 2019, Numerical analysis of a turbocharger compressor, *In E3S Web of Conferences (Vol. 128, p. 06012). EDP Sciences*.

Ibaraki S., Tomita I. S. A. O. and Sugimoto K. O. I. C. H. I., 2015, Aerodynamic design optimization of centrifugal compressor impeller based on genetic algorithm and artificial neural network, *Mitsubishi Heavy Industries Technical Review*, 52(1), 77.

Khoshkalam N., Mojaddam M. and Pullen K. R., 2019, Characterization of the performance of a turbocharger centrifugal compressor by component loss contributions, *Energies*, 12(14), 2711.

Shaheed R., Mohammadian A. and Gildeh H. K., 2019, A comparison of standard k–ε and realizable k–ε turbulence models in curved and confluent channels, *Environmental Fluid Mechanics*, 19(2), 543-568.

Shen K., Zhang Z., Zhang Z. and Yang, Y., 2017, Investigation of effect on cross-flow heat exchanger with air flow non-uniformity under low Reynolds number, *Advances in Mechanical Engineering*, 9(7), 1687814017708088.

Shih T. H., Liou W. W., Shabbir A., Yang, Z. and Zhu J., 1995, A new k-ϵ eddy viscosity model for high reynolds number turbulent flows, *Computers & fluids*, 24(3), 227-238.

Yaïci W., Ghorab M. and Entchev E., 2014, 3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar heat exchangers, *International Journal of Heat and Mass Transfer*, 74, 490-500.