Heat Transfer Enhancement in Air Cooled Gas Turbine Blade Using Corrugated Passages

Dr. Farhan Lafta Rashid , Dr. Haider Nadhom Azziz , Dr. Emad Qasem Hussein
Kerbala University, Petroleum and Petrochemical Engineering Department
Corresponding Authot E-Mail: engfarhan71@gmail.com

Abstract

In this paper, an investigation of using corrugated passages instead of circular cross section passages was achieved in conditions simulate the case in the gas turbine blade cooling using ANSYS Fluent version (14.5) with Boundary conditions: inlet coolant air temperature of 300 K with different air flow Reynolds numbers (191000, 286000 and 382000). The surrounding constant hot air temperatures was (1700 K). The numerical simulations was done by solving the governing equations (Continuity, Reynolds Averaging Navier-stokes and Energy equation) using (k-ε) model in three dimensions by using the FLUENT version (14.5). The present case was simulated by using corrugated passage of 3 m long, internal diameter of 0.3 m, 0.01 m groove height and wall thickness of 0.01 m, was compared with circular cross section pipe for the same length, diameter and thickness. The temperature, velocity distribution contours, cooling air temperature distribution, the inner wall surface temperature, and thermal performance factor at the two passages centerline are presented in this paper. The coolant air temperature at the corrugated passage centerline was higher than that for circular one by (12.3%), the temperature distribution for the inner wall surface for the corrugated passage is lower than circular one by (4.88 %). The coolant air flow velocity seems to be accelerated and decelerated through the corrugated passage, so it was shown that the thermal performance factor along the corrugated passage is larger than 1, this is due to the fact that the corrugated walls create turbulent conditions and increasing thermal surface area, and thus increasing heat transfer coefficient than the circular case.

Keywords: Blade cooling, heat transfer enhancement, gas turbine, rib turbulator cooling.

تحسين الانتقال الحراري لريشة التوربين الغازي المبردة بالهواء باستخدام الممرات المموجة

د. فرحان لفته رشيد ، د. حيدر ناظم عزيز ، د. عماد قاسم حسين

قسم هندسة النفط والبتروكيمياويات/ جامعة كربلاء

الخلاصة:

تم انجاز البحث الحالي باستخدام الممرات المموجة بدلا من الممرات ذات المقطع العرضي الدائري في ظروف تشابه ظروف تبريد ريشة التوربين الغازي باستخدام برنامج الانسز فلونت 14.5 وكانت الظروف الحدودية: درجة حرارة دخول هواء التبريد 300 درجة مطلقة ومعدل جريان هواء التبريد كرقم رينولد (191000, 286000, 382000) وكانت درجة حرارة الهواء الحار المحيط 1700 درجة مطلقة. التشبيه العددي تم بحل المعادلات الحاكمة (الاستمرارية، نافير ستوك المعدل رينولد، ومعادلة الطاقة) باستخدام نموذج (k-ε) الثلاثي الابعاد. الحالة الحالية تم تشبيهها باستخدام ممر متموج بطول 3 م وقطر داخلي 0.3 م وبارتفاع الاخدود 0.01 م وسمك جدار الممر 0.01 م ، وقد تمت مقارنة هذه الحالة مع حالة ممر بمقطع عرضي دائري وبنفس طول وقطر وسمك الممر المتموج. استحصلت النتائج للتوزيع الكنتوري لدرجة الحرارة والسرعة وتوزيع درجة حرارة هواء التبريد ودرجة حرارة السطح الداخلي لجدار الممر ومعامل الاداء الحراري على طول مركز كلا الممرين ، وقد كانت درجة الحرارة على طول مركز الممر المتموج اعلى من درجة الحرارة على طول مركز الممر ذي المقطع العرضي الدائري بمقدار 12.3 % وكانت درجة حرارة السطح الداخلي للمر المتموج اقل من درجة حرارة السطح الداخلي للمر ذي المقطع العرضي الدائري بمقدار 4.88 % ، سرعة جريان هواء التبريد تتعجل وتتباطى خلال الممر المتموج وان معامل الاداء الحراري على طول الممر المتموج اعلى من 1 وهذا ناتج عن حقيقة ان الجدران المتموجة تخلق ظروف اضطراب وتزيد من مساحة الانتقال الحراري وبذلك تؤدي الى زيادة معامل الانتقال الحراري للممر المتموج عن قيمته للمر ذو المقطع العرضي الدائري.

Introduction

Gas turbines are strongly efficient engineered main movers that transferring energy from thermal (at combustion stage) to mechanical form – are excessively used for propulsion and power production systems. Typically, gas turbines operate at high temperatures that often range from (1644 to 2199) K [1,2]. Engineers raised the gas turbine inlet temperatures in order to receive larger thermal efficiency and larger production of power, and advanced in constructing the temperatures to higher magnitudes as yet remains. The engines gas turbine components are mainly deal with the largest values of temperature for which are quite behind the permissible material limit [3]. The turbine blades often cooled by the air removed from the engine compressor. The duty is reservation largest metal temperature under the magnitudes determined by material potency and avoiding the high temperature changing according to the turbine blade in order to take part of reasonable life and operational needs by excerpt the minimal cooling air potential. To attain this duty, individual techniques are performed to optimize the turbine blade cooling. The main cooling internal zones in a turbine blade are: regions close blade leading edge usually cooled by impingement cooling while at the central regions, air is ducted by overelaborate passages which are often ribbed, and the trailing edge zone is usually armed with pin-fins arrangement as shown in figure (1) [4].

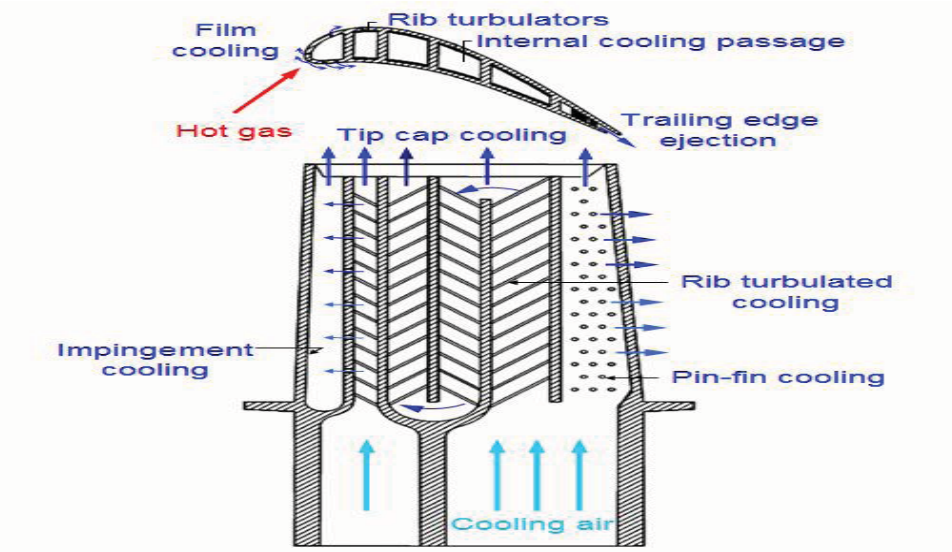

Fig. (1) A modern cooling concepts of the gas turbine engine [4]

Kittisak et al. [5] studied the transfer of heat using turbulators as nozzle and generator as swirl in the constant heat flux pipe. In the experimental part, the turbulators of nozzle were inserted by fitting nozzles inside the test pipe with different pitch ratios of PR = 2.0, 4.0, and 7.0, while the generator of snail swirl was putted at the entry of test pipe. Safely, largest friction factor and heat transfer than the case of plain pipe at the same conditions were found. It was discovered that the rate of heat transfer and friction factor rises with decreasing the pitch ratio. For the range examined, the highest value of Nusselt numbers for the case of using the enhancement apparatus with PR = 2.0, 4.0 and 7.0 are 374%, 342% and 309%, respectively, if compared with the plain pipe.

Hiral Prajapati [6] reported studies of CFD for Nusselt number, friction factor and enhancement of heat transfer with the Re-Normalization Group (RNG) and (k-ε) turbulence model for a circular tube equipped with wire coil insert. Result exhibited that the wire roll with pitch/diameter of (0.434, 0.651 and 0.868) able to consolidate the heat transfer up to (4.98, 5 and 4.3) times, respectively and values of friction factors up to (5.82, 4.06 and 3.3) times, respectively, if compared with plain pipe. Wire roll with rib height/diameter of (0.038, 0.128 and

0.171) gave enhancement in heat transfer (4.99, 5.92 and 7.6) times, respectively and enhancement in friction factor (5.82, 19.97 and 35.7) times, respectively that plain tube.

Onur and Ali [7] achieved a CFD study of flows over walls of heated ribbed with the effect of the Reynolds number and height of rib. Laminar and turbulent flow with fixed thermo physical properties was presumed for air at two values of the initial stream wise Reynolds number of ($2.7x10^5$ and $3.4x10^6$). The method of finite-volume was used to solve the governing equations, coupled with the (k-ε) turbulence model with near-wall treatment. Results presented that the presence of the ribs can effectively promote the heat transfer. The values of enhancement in heat transfer was raised with rib height and be larger in laminar than that of turbulent flows.

Arkan Al Taie et al**.** [8] presented an investigation of heat transfer characteristics and thermal performance in 50 cm stainless steel tube length, the outside diameter was 60 mm with inside diameter of 30 mm and constant surrounding hot air temperature ranged 1000, 1200 and 1400 K. The (k-ε) model is used in ANSYS - FLUENT 14.5 to simulate turbulence. The special arrangement of ribs (triangular cross section of 5x5 mm) were putted in the tube with pitch of 8 cm. Results of distribution for temperature and velocity along the tube centerline for the circular pipe with ribs were compared with that of smooth case (without ribs) , these results indicate that the use of internal ribs increase the rate of heat transfer and found to rejoice the highest performance factors for turbulent flow.

Arkan Al Taie et al**.** [9] presented an investigation of coolant air flow forced convection (10 m/s velocity) in 50 cm long stainless steel circular pipe with 60 mm outer diameter and 30 mm inner diameter at constant hot air surrounding temperature of (1000, 1200 and 1400 K). The well-defined model (k-ε) is used to simulate turbulence in ANSYS - FLUENT 14.5. A special configuration of opened circular rib with (5x7 mm cross section) is putted in the tube circular pipe and spaced by 8 cm pitch. The results of distribution for temperature and velocity along the pipe centerline for the tube with ribs were compared with the smooth one , the use of internal ribs increases the heat transfer and thermal performance factors.

Mohammed W. Al-Jibory et al**.** [10] achieved an experimental and numerical investigation for using circular ribs in rectangular passages for inlet coolant air temperature of 300 K, surrounding hot air temperature of 473 K and coolant air flow Reynods number (7901, 9500 and 11500). Researchers achieved the numerical part by solving the governing equations (continuity, Reynolds averging Navier-Stokes and energy) in turbulent regime with (K-ε) model in three dimensions by using the fluent version (15). The used ribs with height (e=0.005 m) and rib-rib spacing (P=0.05 m) fitted in 60x30 mm rectangular cross section passage with 0.5 long. The coolant air temperature for rectangular passage with ribs found to be higher than for the smooth case by (6.4 %). The temperature distribution for the inner wall surface with ribs is lower than for the smooth one by (3.21 %).

In this paper, the effect of using corrugated passages instead of circular cross section passages in gas turbine blade with different internal flow of coolant air and uniform wall surface temperature will be examined using Fluent 14.5.

Governing Equations

The basic equations that describe the flow and heat by using ANSYS- Fluent 14.5 in 3D are momentum conservation, mass, and energy equations. These, equations numerically describe three-dimensional, turbulent and incompressible flow [11].

The assumptions that used for the instantaneous equation are:

1- Steady, three-dimensional, incompressible flow, single-phase flow, no-slip, irrotational.

2- Thermal-equilibriums.

(i) Conservation of Mass:

$$\frac{\partial \rho}{\partial t}+\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \ldots\ldots(1)$$

(ii) Momentum Equations:

a) Momentum equation in the X-direction:

$$\frac{\partial \rho u}{\partial t}+\frac{\partial(\rho u u)}{\partial x}+\frac{\partial(\rho v u)}{\partial y}+\frac{\partial(\rho w u)}{\partial z}=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right] \quad \ldots\ldots(2)$$

b) Momentum equation in the Y-direction:

$$\frac{\partial \rho v}{\partial t}+\frac{\partial(\rho u v)}{\partial x}+\frac{\partial(\rho v v)}{\partial y}+\frac{\partial(\rho w v)}{\partial z}=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right] \quad \ldots(3)$$

c) Momentum equation in the Z-direction:

$$\frac{\partial \rho w}{\partial t}+\frac{\partial(\rho u w)}{\partial x}+\frac{\partial(\rho v w)}{\partial y}+\frac{\partial(\rho w w)}{\partial z}=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \quad \ldots\ldots(4)$$

(ii) Energy Equation

$$\frac{\partial}{\partial x}\left(\rho uh - \Gamma\frac{\partial h}{\partial x}\right) + \frac{\partial}{\partial y}\left(\rho vh - \Gamma\frac{\partial h}{\partial y}\right) + \frac{\partial}{\partial z}\left(\rho wh - \Gamma\frac{\partial h}{\partial z}\right) = 0 \quad \text{..........(5)}$$

Where;

$$\Gamma = \mu/Pr \quad \text{..(6)}$$

(iii) Turbulence Model

The turbulence model that utilized in this analysis one of the well-defied turbulence models (the two-equation model) of kinetic energy (K) and its dissipation rate (E) [12, 13].

(i) Turbulence Energy, (k):

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \quad \text{........................(7)}$$

(ii) Energy Dissipation Rate, (ε) [13]:

$$\frac{\partial \rho\varepsilon}{\partial t} + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \left(\rho C_{1\varepsilon}\frac{\varepsilon}{k}G_k\right) - \left(\rho C_{2\varepsilon}\frac{\varepsilon^2}{k}\right) \quad \text{.......(8)}$$

Where;

The turbulent (or eddy) viscosity,(μ_t):

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad \text{..(9)}$$

Also, the breeding of turbulence kinetic energy caused by the mean velocity gradients (**G_k**):

$$\mathbf{G_k = \mu_t S^2} \quad \text{..(10)}$$

S: IS THE MEAN STRAIN TENSOR RATE, DEFINED AS:

$$\mathbf{S = \sqrt{S_{ij}S_{ij}}} \quad \text{..(11)}$$

$$\mathbf{S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)} \quad \text{..(12)}$$

Mesh Generation

The computational mesh genration that is adequate for the discretized solution of three dimentional conservation energy, continuity, momentum equations have been the matter of heavy researches. In this work, the shape of elements were prism with triangular base, and cube shape (hexahedron), as shown in figure (2).

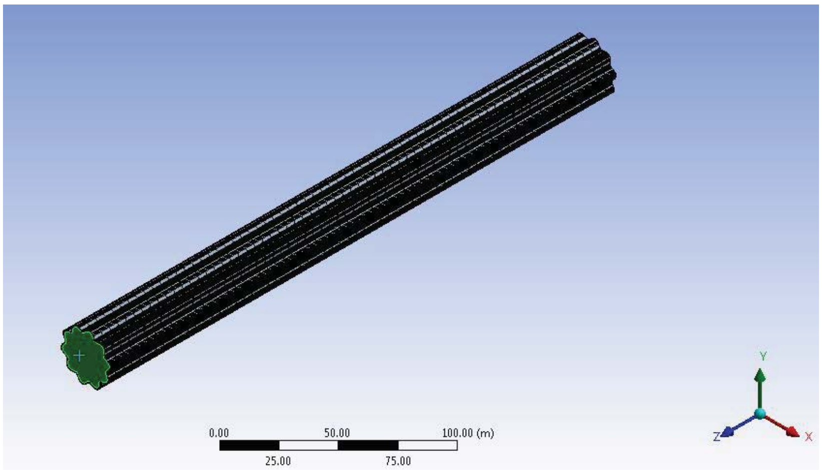

Fig. (2) Mesh generation of the corrugated passage

Results and Discusion

Figures (3) and (4) show a focused view of the temperature contours along the anterior section (Z-axis), of the circular cross section and corrugated passage, respectively. The temperature at an inlet temperature of (300 K) with flow Reynolds number of (191000). The surrounding hot air temperature was 1700 K.

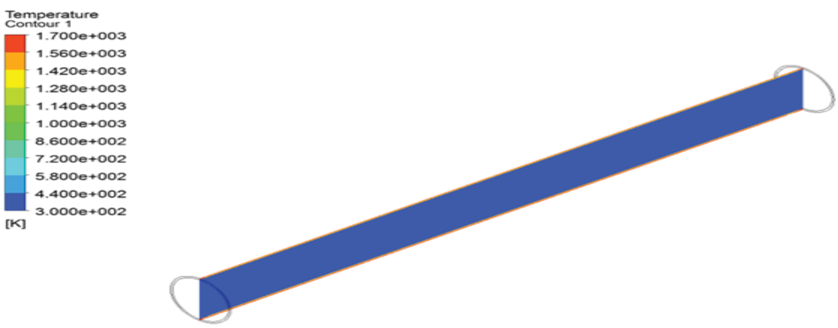

Fig. (3) Temperature contour in (K) at (Z-axis) along the circular cross section passage

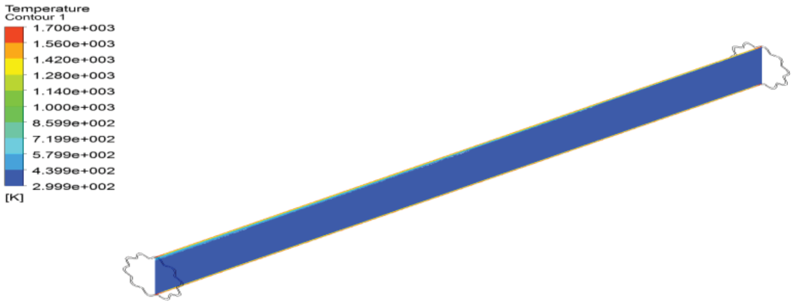

Fig. (4) Temperature contour in (K) at (Z-axis) along the corrugated passage

Figure (5) shows contour of temperature distribution for circular cross section passage for surrounding hot air temperature of (1700 K) , inlet air temperature (300 K) and inlet air flow Reynolds number (Re=191000). It was shown that the cooling air temperature at passage centerline remained constant throughout the it, while the coolant air temperature was unaffected in the channel centerline till (10%) of the passage length and then gradually increased as shown in figure (6). This is due to the effect of circulation generated by the corrugated wall which clearly enhances the transfer of heat between the hot wall and the coolant air flow stream.

Fig. (5) Temperature distribution contour through circular cross section passage

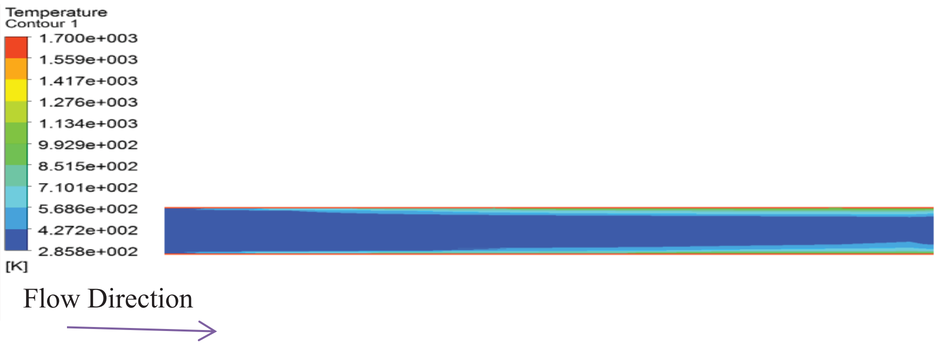

Fig. (6): Temperature distribution contour through corrugated passage

Figure (7) reveals that the cooling air velocity at circular cross section passage centerline increased downstream constant throughout it, while figure (8) presents the velocity distribution contour through the corrugated passage ,The coolant air flow was accelerated and decelerated through the passage, due to the corrugated wall.

Fig. (7) Velocity distribution contour through circular cross section passage

Fig. (8) Velocity distribution contour through corrugated passage

Figures (9) and (10) display contours of temperature distribution across the wall and passage in (K) at locations of (Z= 37.5, 75, 112.5, 150, 187.5, 225, 262.5 and 300) cm along the test section for constant surrounding hot air of 1700 K and coolant air flow Reynolds number of (191000). It can be seen that the local section of corrugated passage seems to have higher heat transfer rate from the wall surface to the coolant air stream than the circular cross section.

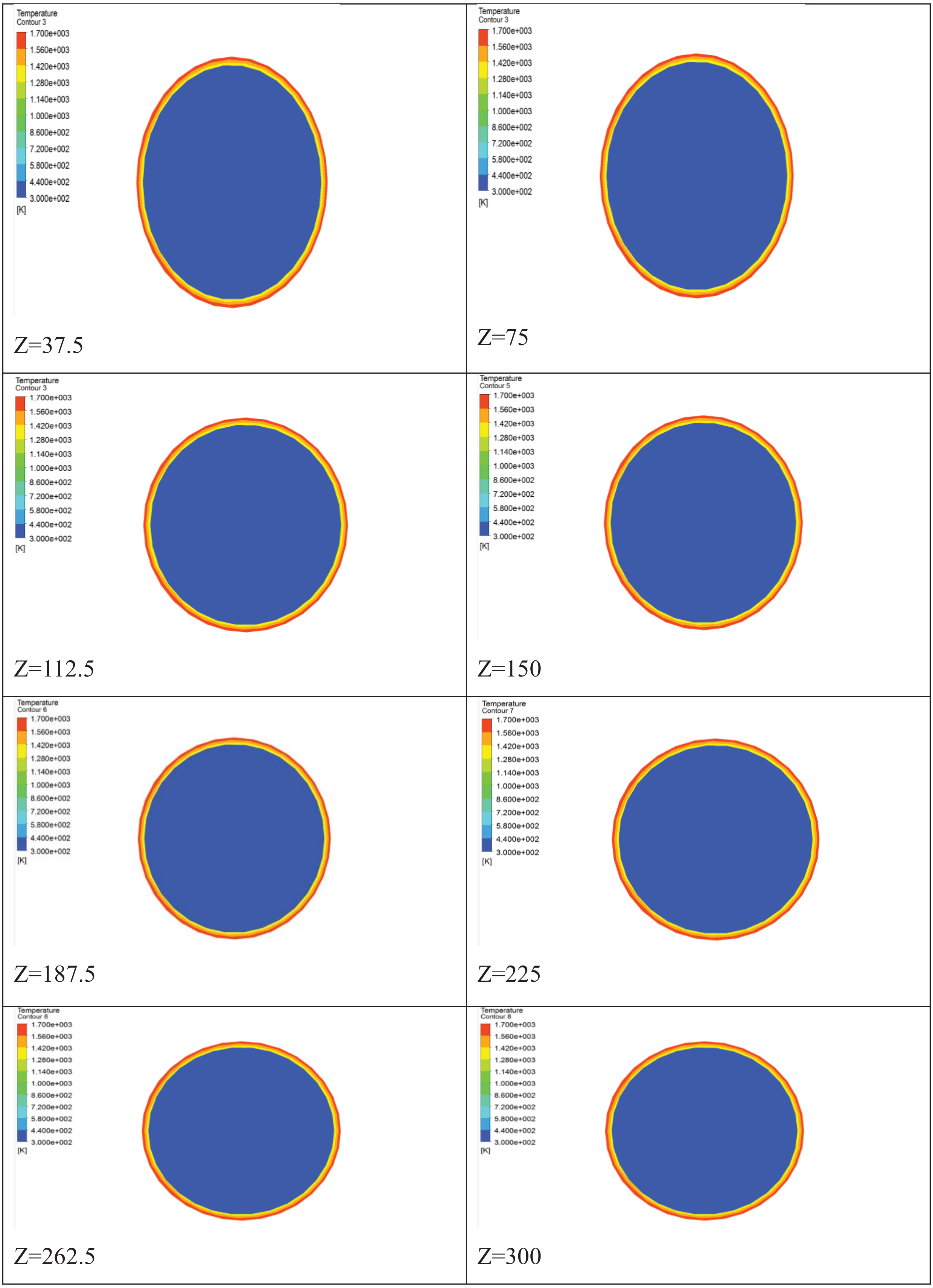

Fig. (9) Local temperature distribution contours through (Z-axis) of circular passage

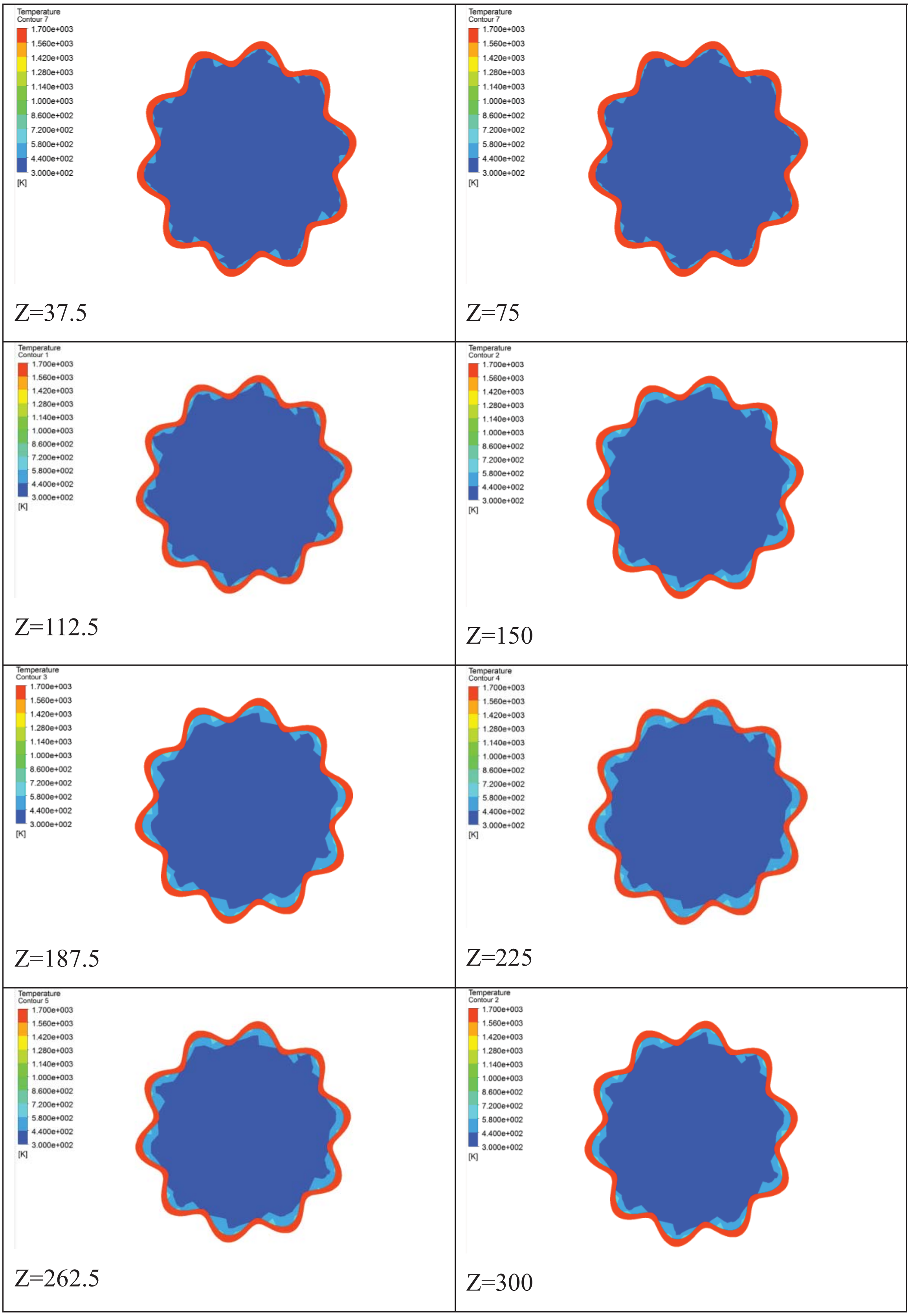

Fig. (10) Local temperature distribution contours through (Z-axis) of corrugated passage

Figure (11) shows the cooling air temperatures at the passage centerline for a constant surrounding hot air temperature (1700 K) , inlet air temperature (300 K) and coolant air flow of (Re=191000). The coolant air temperature at the corrugated passages centerline seems to be higher than that for circular cross section passage. This because of corrugated wall make vortices and then leads to rise the heat transferred from the channel wall to cooling air stream, and thus lead to increase the coolant air temperature by (12.3%).

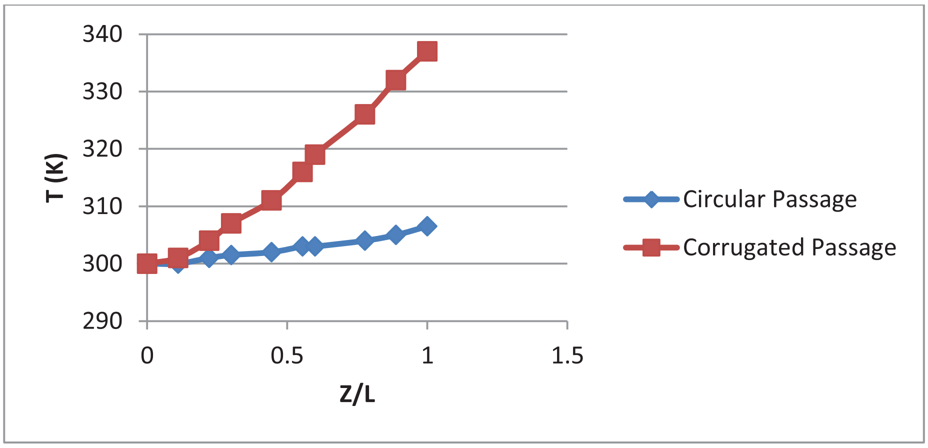

Fig. (11) Coolant air temperature at the passage centerline for air flow velocity of 10 m/s

Figure (12) presents the temperature distribution along the inner passage surface, it can be seen that using corrugated passage reduces the inner wall temperature surface. This is due to that the corrugated wall create wakes and then lead to generate vortices. This lead to enhance the rate of heat transfer from the hot wall to the coolant air stream, i.e. increase the coolant air temperature and decrease temperature in the wall by (4.88%).

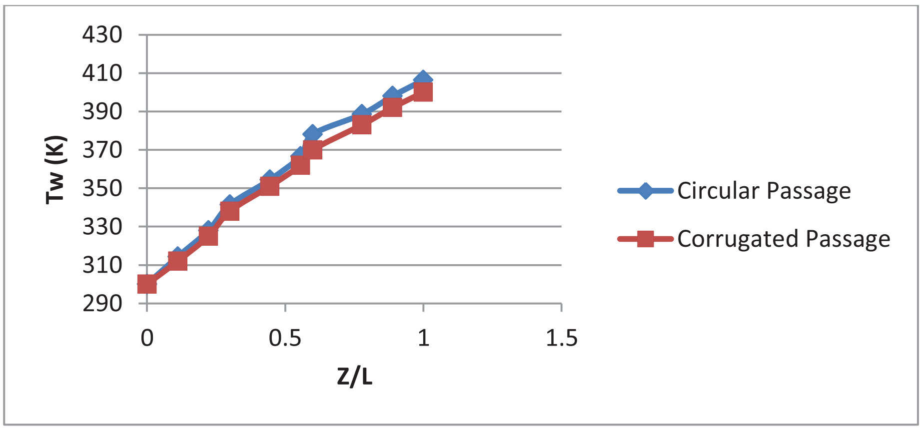

Fig. (12) Temperature at the inner wall surface along the passage for air flow velocity of 10 m/s

Figure (13) presents the thermal enhancement factor (heat transfer coefficient of corrugated passage divided by heattransfer coefficient for the circular cross section), it can be seen that the thermal enhancement factor is always be larger then 1, this means that the corrugated surface lead to give more heat transfer area and so increasing the flow turbuelence which cause in increasing the enhancement factor.

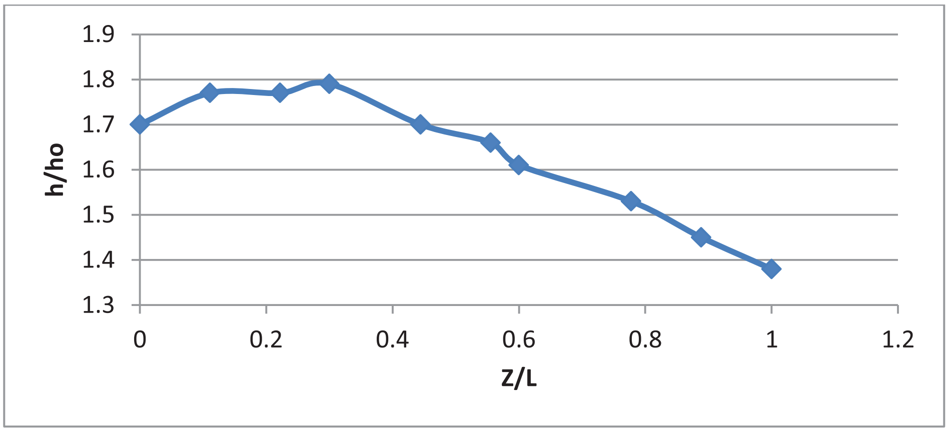

Fig. (13) Enhancement factor along the passage for air flow velocity of 10 m/s

Figure (14) reveals the coolant air flow velocity along the corrugated passage centerline, it can be seen that the coolant air flow velocity was accelerated and decelerated through the passage due to the corrugated wall surface.

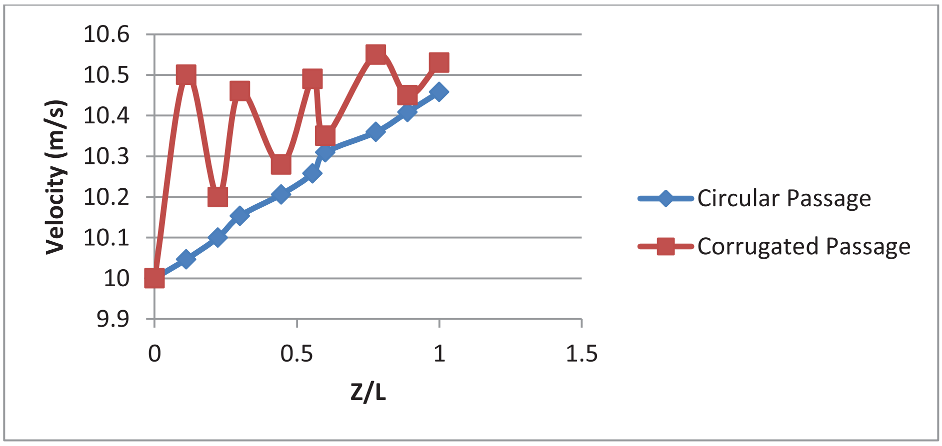

Fig. (14) Velocity distribution along the passage centerline for air flow velocity of 10 m/s

Fig.(15) displays the friction factor ratio (friction factor of corrugated passage divided by its value for circular cross section) against Reynolds number of the coolant air flowed through the passage, the increase in Reynolds number accompanied with decreasing in friction factor ratio, the maximum value at (Re=191000).

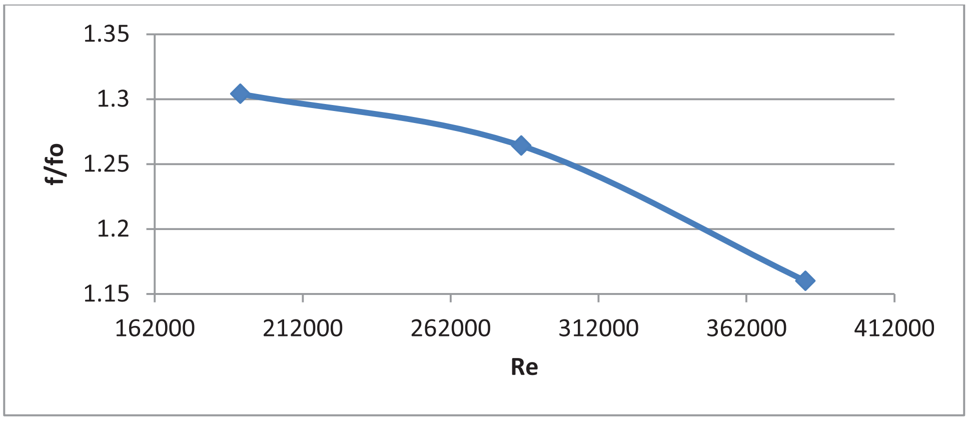

Fig. (15) Friction factor ratio for different air flow Reynolds number

Figure (16) reveals coolant air temperature along the passage centerline for different coolant flow Reynolds number, the increase in Reynolds number increases coolant air temperature and reduces the inner wall passage temperature (as shown in figure (17)), because the increase in flow Reynolds number lead to increase in flow turbulence which lead to enhance the rate of heat transfer from the passage wall to coolant air stream.

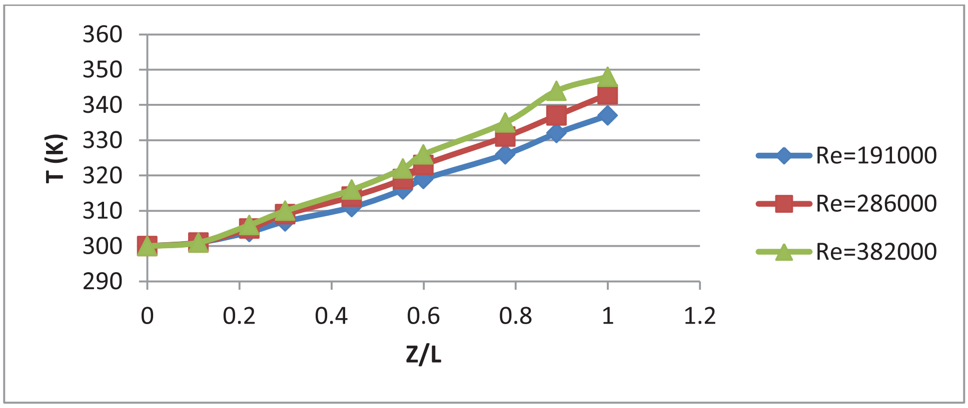

Fig. (16) Coolant air temperature at the corrugated passage centerline for different air flow Reynolds number

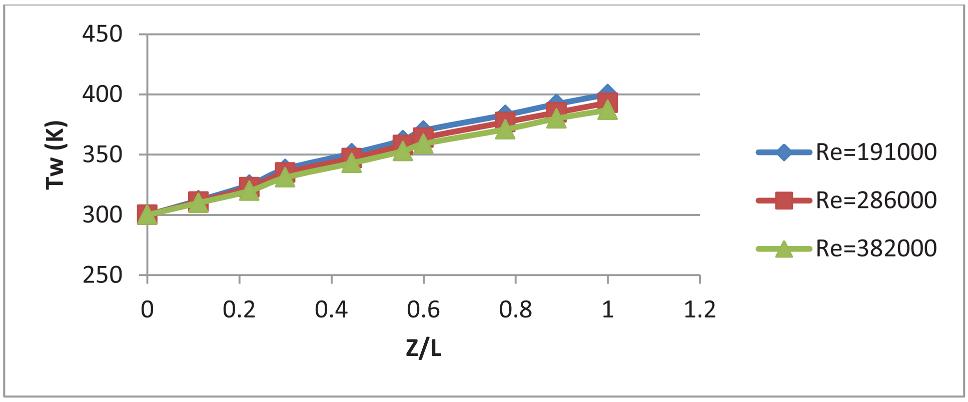

Fig. (17) Temperature at the inner wall surface along the corrugated passage for different air flow Reynolds number

Conclusions

From the present work, the following conclusions can be obtained:

1. The coolant air temperature for corrugated passage was higher than that for circular cross section passage by (12.3 %).
2. The temperature distribution in the inner wall surface for the corrugated passage is lower than that for circular one by (4.88 %).
3. The coolant air flow velocity was accelerated and decelerated through the corrugated passage, due to the corrugated wall.
4. The thermal performance factor along the corrugated passage is larger than 1, meaning that the heat transfer coefficient for corrugated arrangement passage always exceeds the circular case.

References

1. Boyce, Meherwan P., "Gas Turbine Engineering Handbook", 3rd ed. Boston: Gulf Professional Publications (2006).
2. Minking K. Chyu, "Recent Advances in Turbine Heat Transfer-with A View of Transition to Coal Gas Based Systems", International Heat Transfer Conference, IHTC-14(2003).
3. Jonas Bredberg, "Turbulence Modelling for Internal Cooling of Gas-Turbine Blades", PhD Thesis, Department of Thermo and Fluid Dynamics Chalmers University of Technology, se-412 96 Goteborg, Sweden (2002).
4. Han J. C., Dutta S., and Ekkad S.V., "Gas Turbine Heat Transfer and Cooling Technology", Taylor & Francis, Inc., New York (2000).
5. Kittisak Yongsiri, Smith Eiamsa-ard, Chinaruk Thianpong and Pongjet Promvonge ", Effect of Turbulent Decaying Swirl Flow on Heat Transfer Enhancement in a Tube", The 2nd Joint International Conference on “Sustainable Energy and Environment (SEE 2006)".
6. Hiral N. Prajapati,"Numerica Study on Heat Transfer and Pressure Drop Characteristics of a Tube Equiped with Wire Coil Insert", 37th National & 4th International Conference on Fluid Mechanics and Fluid Power (2010).
7. Onur Yemenici and Ali Sakin," Numerical Investigation of Heat Transfer for Laminar and Turbulent Flow over Ribbed Walls", International Journal of Engineering and Advanced Technology (IJEAT), Vol. 2, Issue 6, pp. 163-166 (2013).
8. Arkan Altaie Moayed R. Hasan Farhan Lafta Rashid, “Numerical Investigation in a Circular Tube To Enhance Turbulent Heat Transfer Using Opened Rings -Triangular Cross Section”, Journal of Babylon University/Engineering Sciences/ No.3 Vol. 23 (2015).
9. Arkan Altaie , Moayed R. Hasan , Farhan Lafta Rashid, “Numerical Investigation of Heat Transfer Enhancement in a Circular Tube with Rectangular Opened Rings”, Bulletin of Electrical Engineering and Informatics Vol. 4, No. 1, pp. 18-25 (2015).
10. Mohammed W. Al-Jibory, Farhan Lafta Rashid, Hasan Q. Hussein, " Cooling Enhancement in Gas Turbine Blade Rectangular Passages Using Circular Ribs", 7th Scientific Engineering and 1st International Conference, University of Baghdad-College of Engineering, IEEE Iraq section, 17-18 May (2017).
11. Wendt, J.," Computational fluid dynamics: an introduction", Third ed.: Springer Science & Business Media (2008).
12. Launder, B.E. and D.B. Spalding, "Lectures in mathematical models of turbulence", (1972).

13. Mahmoud, M.S., "Theoretical Study Of Heat Transfer In Circular Holes For Turbine Blade", International Journal of Scientific & Engineering Research, 4(2) (2013).

Nomenclature

Symbol	Description	Dimension
G	Generation of turbulence kinetic energy due to the mean velocity gradients	$kg/m^3.s$
g	Ground accelerate	m/sec^2
h	Specific enthalpy	kJ/kg.K
h	Heat transfer coefficient	$W/m^2.K$
k	Thermal conductivity	W/m. K
P	Pressure	N/m^2
Re	Reynolds number	---
S	modulus of the mean rate -of-strain tensor	---
T	Temperature	^oC
u, v, w	Velocity component in Cartesian coordinate	m/s
X,Y,Z	Cartesian coordinate	m

Greek Symbols

Symbol	Description	Dimension
∂	Partial differential operation	---
μ	Dynamic viscosity	kg/m.sec
ε	Turbulence eddy dissipation rate	m^2/sec^3
κ	Turbulent kinetic energy	m^2/s^2
ρ	Density	kg/m^3
f	Friction factor	---

Subscripts

Symbol	Meaning
i, j, k	The three coordinate direction
s	Surface
t	Turbulent

Abbreviation

Symbol	Description
CFD	Computational fluid dynamic
FLUENT	Fluid And Heat Transfer Code
ANSYS	Analysis System
3D	Three Dimension