Journal of Engineering and Sustainable Development
http://jeasd.uomustansiriyah.edu.iq
https://doi.org/10.31272/jeasd.24.4.4

Vol. 24, No. 04, July 2020
ISSN 2520-0917

HEAT TRANSFER INSIDE BUILDING- CLADDING SOLAR COLLECTOR

Ahmed Mohsin Abdulmalik[1]
Lect. Dr Basima salman khalaf [3]
*** Lect. Dr basim Freegah**[2]
Prof. Dr. Mohammed H. Alhamdo[4]

1) Mechanical Eng، College of Engineering, Mustansirya University , Baghdad, Iraq.
2) Mechanical Eng، College of Engineering, Mustansirya University , Baghdad, Iraq.
3) Mechanical Eng، College of Engineering, Mustansirya University , Baghdad, Iraq.
4) Mechanical Eng، College of Engineering, Mustansirya University , Baghdad, Iraq.

Received 4/9/2019 ***Accepted in revised form 10/30/2019*** ***Published 1/7/2020***

Abstract: In this work numerical and experimental investigation have been adopted to collect and store solar energy in exterior-wall cladding, Various improvements have been investigated inside and outside the duct to increase the efficiency of thermal heating. ANSYS software has been used to simulate current case. Results shows that there is a good agreement between experimental and numerical results and this agreement increases as air velocity increases. The average percentage error for air inside duct at velocity of air 1 m/s, 3 m/s and 5 m/s is 8%, 16.5% and 5% respectively. Several vertical cooper cylinders with 12 mm diameter were added inside basin to increase the surface area. It has been found that the enhancement in temperature of air between this case and Smooth Duct, Smooth Cover base model for air velocity of 1,3and 5 m/s is 3, 4 and 11 % respectively. The effect of increasing surface area by using granular (corrugated) duct on the air temperature distribution along the duct. Have been also investigated the percentage enhancement in temperature of air between this case and previous base smooth duct case for velocities of 1, 3 and 5 m/s is 19.4, 28.6 and 16.5 % respectively. The enhancement in heat transfer when using both granular hollow sphere duct with vertical metal cylinders for air velocity of 1, 3 and 5 m/s is 27.5, 33 and 35.2 % respectively.

Keywords: *Finite element; Experiment; Thermal energy storage; Infrared thermal imaging*

1. Introduction

Recently, the most of researchers have been tried to find new ways for improving heat transfer in several heat applications. Undoubtedly that there are many ways could be utilized to improve the heat of transfer, one and of a most important of these ways is increasing surface area. There are several materials type, one of the most type that widely used is granular duct; many applications such as cryogenics, combustion chambers, and compact of heat exchangers.

Jaurker et. al. [1] conducted experimental study in order to analyze the effect of rib and grooved on the heat transfer and fluid flow characteristics of air flow in the rectangular solar air heater duct. The rectangular- duct has internal dimensions of (2420 mm X 156 mm X 22 mm). In this study, the effect of roughness parameters namely rib height (e), groove position (g) and rib pitch (p) have been determined. The results have been demonstrated that the Nusselt number with the grooved roughness is higher as compared to smooth duct

**Corresponding Author*: *ahmed1975414@gmail.com*

by about 2.7 times. Ahn, S. (2001). [2] Conducted the effect of roughness types on friction factors and heat transfer in roughened rectangular duct. In this work, several parameters have been investigated such as a fully developed heat transfer is compared with friction factor characteristics has been made with five various kinds of rectangular duct geometry, Reynolds numbers and friction factors are also investigated in the rough and smooth channel. The results were showed that the friction factors for the square-shaped geometry has the highest value because of its strongest turbulence mixing caused by the ribs. The Nusselt numbers in rectangular and triangular shapes are much higher than other Cases. It is because the radial turbulent fluctuating components closely related to heat transfer mechanisms become more significant in the sharp square- and triangle-shaped geometries. Goel, M., et al. (1994). [3] Investigated the Laminar forced convection heat transfer in micro capsulated. The primary parameters in the study were the bulk Stefan number and the volumetric concentration. In addition, a few experiments were conducted to evaluate the effect of particle diameter and degree of homogeneity of the suspension. The results were showed that, the bulk Stefan number, which is the ratio of the sensible heat capacity of the suspension to its latent heat capacity, was found to be the most dominant parameter. The volumetric concentration does not have a significant effect directly on the heat transfer. However, it is included in the definition of the Stefan number, and thus has an indirect effect. The microcapsule to duct diameter ratio had a noticeable the effect on the heat transfer. Gräsel, J., et al. (2006). [4] The Parametric inter turbine duct design and optimization. The objective of the present research is to develop a fast and robust tool for the parametric design and optimization of S-shaped ducts. In contrast to the inherently three-dimensional nature of the duct flow, current design practice for inter-turbine ducts still employs standard performance maps derived from annular diffusers. As a consequence these low fidelity methods produce duct shapes, which are far from the optimum in terms of minimum duct length and maximum area ratio.

The results of the DOE study show that the optimization of the clean duct resulted in a significant increase of pressure recovery without degradation of the pressure loss. However, a 3D CFD calculation has to be performed in order to verify the optimization results. Potter Jr, A. E. and E. L. Wong (1958). [5] Conducted the effect of pressure and duct geometry on blow off body flame stabilization**.** In this study, the parameters were studied the Blow off velocities and recirculation-zone lengths of propane-air flames stabilized by cylindrical flame holders were measured as a function of pressure (0. 25 to 0. 8 atm), cylinder diameter (3/8 to 1.0 in.), fuel air ratio, and tunnel geometry (3 by 3 and 1 by 3 in.) for Reynolds numbers ranging from 0.64XI04 to 17. 3XI04. The output were obtained of this research support the view that the variation of blow off velocity with flame holder size and tunnel geometry is largely the result of changes in the recirculation-zone length, and that the variation of blow off velocity with pressure is principally the result of variation of the critical time with pressure. And the value of the blow off velocity for a given pressure and flame holder diameter is greater in the 3- by 3-inch than in the 1- by 3-inch tunnel. Hooman, K. (2007) [6]. Carryout the Entropy generation for the microscale forced convection: effects of different thermal boundary conditions, velocity slip, temperature jump, viscous dissipation, and duct geometry. The parameters were be studied, the fully developed temperature distribution and entropy generation due to the forced convection in the micro electro mechanical systems (MEMS) in the Slip-flow regime and two different thermal boundary conditions are investigated, being isothermal and isoflux walls. The results has been showed that the effect of the thermal boundary condition is more pronounced than that of the duct cross-section. The reason is that the heat transfer aspects of the fully developed region in a duct of isothermal wall are substantially different from those of the isoflux case as reflected by the presence of the Péclet

number in *Ns* for the isoflux case and the absence of *Br* in the Nusselt number expression for the isothermal case.

The main target of current work is to enhance the thermal performance of solar collector building – cladding system. A survey for available published of literature reveals a lack of the information about the thermal behaviors of combined two or more methods (at the same time) for both inside and outside system fluid ducts. Therefore, to cover this gap, two heat transfer enhancement methods shall be tested at the same time on a developed wall cladding system for the purpose of both collect and store solar energy.

2. Numerical Analysis

2.1 Physical Model

The current study investigated a solar collector and storage system by using compressed fluid (air) as a working fluid inside smooth copper duct with dimensions of (50 × 25 × 800) mm and the other granular duct with semi spheres of 10 mm diameter that placed in (150× 600× 60) mm copper insulated basin as shown in figures 3and4. Two types of copper cover both with dimension of (70×160) mm and 0.5 mm thickness plate have been used. The first cover were smooth, while the second have vertical copper cylinders with 12mm diameter and 40 mm length that extended inside the base.

2.2 Numerical mode

The first model (base line case) consider smooth duct and cover (SDSC model) as shown in Figure 1. The second one has been added a cooper cylinders (SDY model) as shown in Figure 2. The third model has been changed the smooth duct by granular duct (GDSC model) as shown in Figure 3. The fourth model were added cylinders with granular duct (GDY model) as shown in Figure 4.

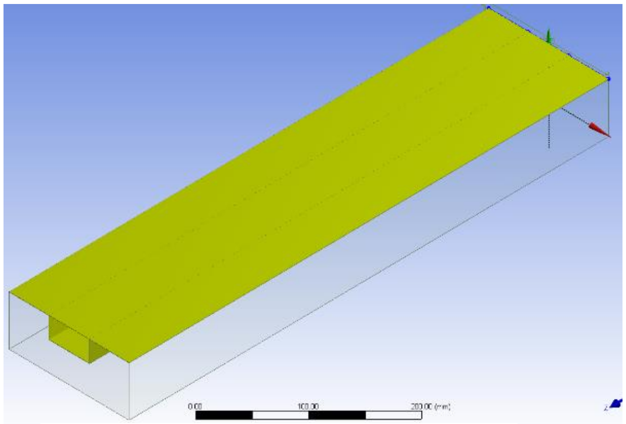

Figure 1. Geometry form of base line SDSC model

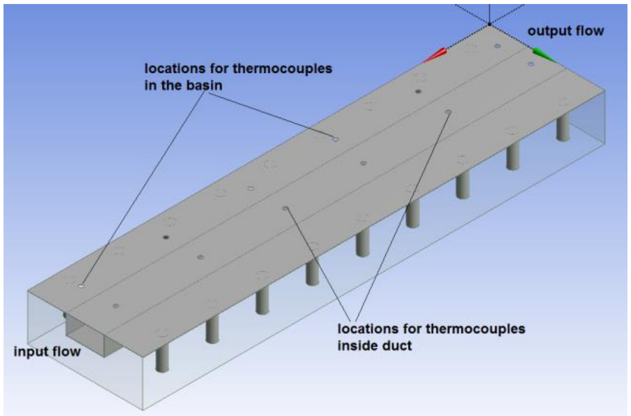

Figure 2. Geometry form of SDY model

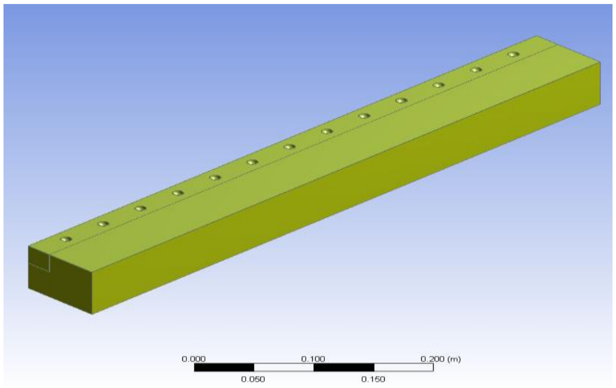

Figure 3. Geometry form GDSC model

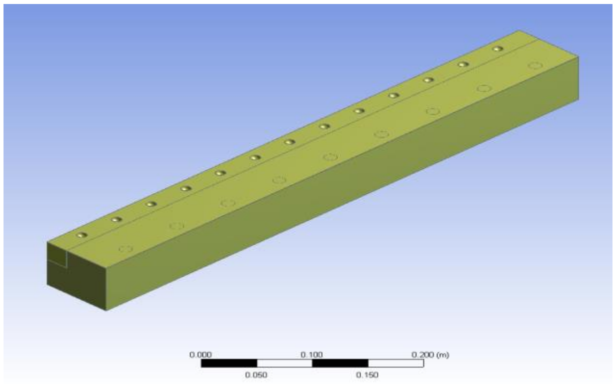

Figure 4. Geometry form of GDY model

2.3 Governing Differential Equation

2.3.1 Momentum equation

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

2.3.2 Energy equation

$$\rho C_p \left\lfloor U\frac{\partial T}{\partial X} + V\frac{\partial T}{\partial y} + W\frac{\partial T}{\partial z}\right\rfloor = \left\lfloor U\frac{\partial p}{\partial X} + V\frac{\partial p}{\partial y} + W\frac{\partial p}{\partial z}\right\rfloor + K\,\nabla^2\, T + \mu\,\phi$$

2.3.3 continuity equation

$$\frac{\partial\,(\rho\, u)}{\partial\, X} + \frac{\partial\,(\rho\, V)}{\partial\, Y} + \frac{\partial\,(\rho\, W)}{\partial\, Z} = 0$$

2.4 Meshing

The Multizone mesh have been generated for SDSC model and that gave a number of element 3185185 with element size 0.003 so, there's no great variation in output accuracy as shown in Figure 5.

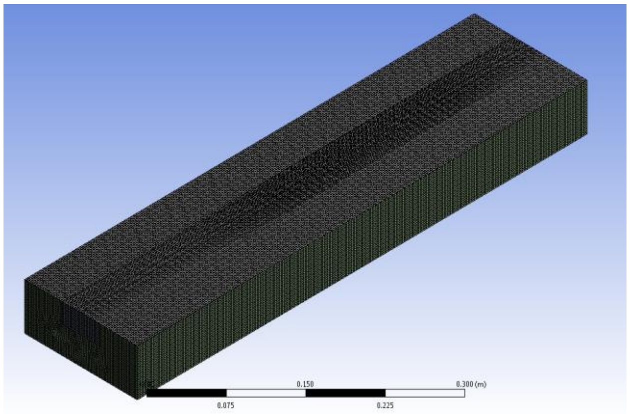

Figure 5. Mesh form of base line for SDSC model

In SDY model, there is an increase in number of element because of adding the copper cylinders to the geometry form and it reaches about 3300106 with output accuracy as shown in Figure 6.

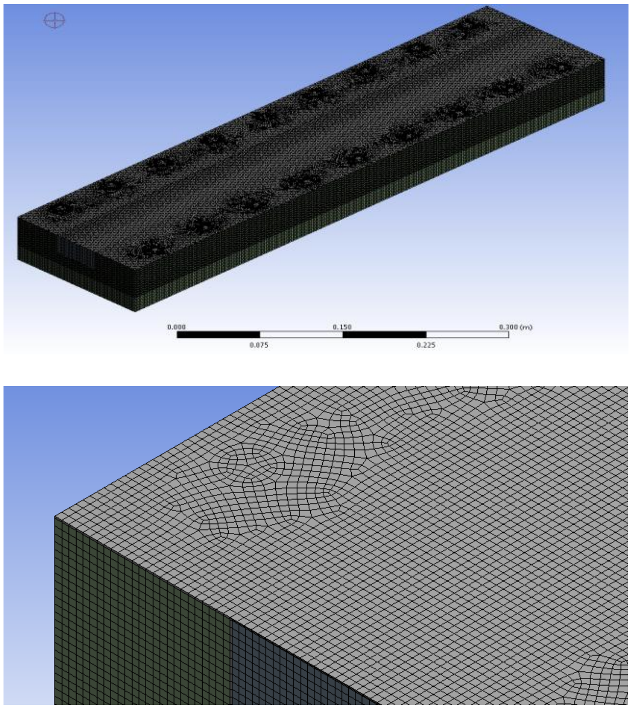

Figure 6. Mesh form SDY model

The granular (GDSC model) is need a new technic for mesh building by symmetric half of the geometry model because the high number of element (more than 16 million element). This number is big for building the mesh whereas, the number of element were 7315652 as shown in Figure 7.

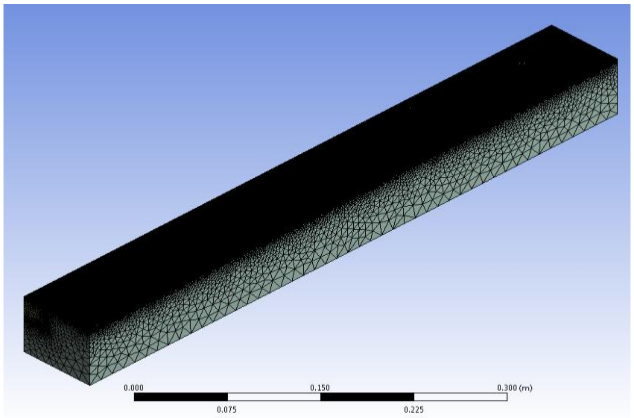

Figure 7. Mesh form for GDSC model

The GDY model has been added cylinders so the number of element and the time were take more than previous model. Also the granular duct push to use the symmetrical technic because the high number of mesh element as shown in Figure 8. The number of element were 8834854.

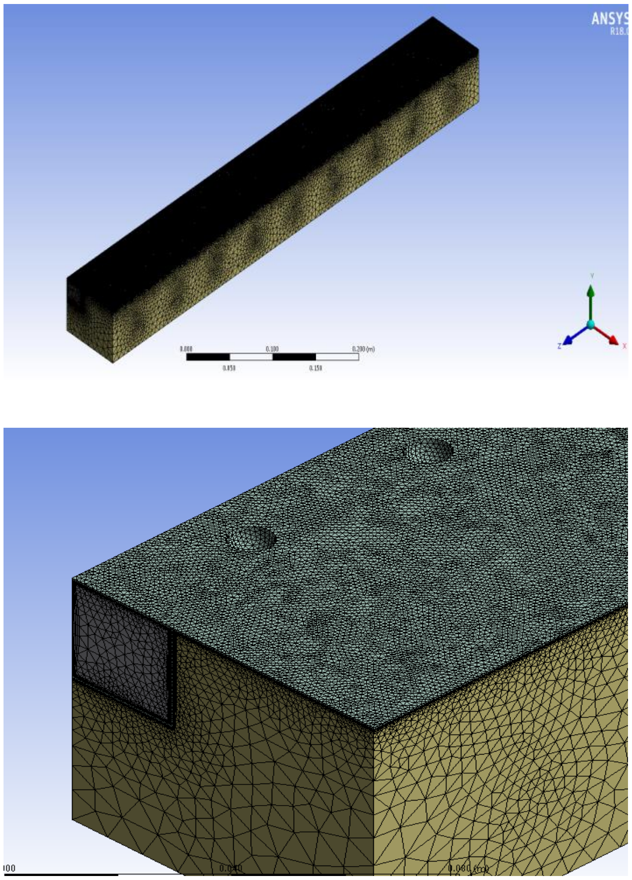

Figure 8. Mesh form GDY model

The granular duct take more complication form, so it have been taken half geometry to satisfy and support the mesh geometry as shown in figure 9.

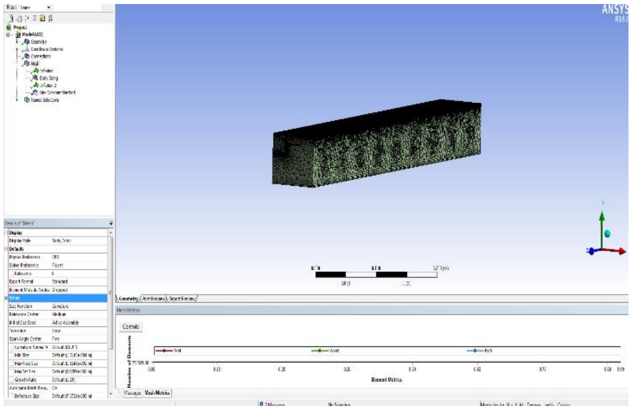

Figure 9. mesh independent program steps for GDYF model

The adoption of the initial and boundary conditions in the simulation program (Ansys) include heat flux, entry air velocity within the duct and the method of insulation. Moreover, the flow velocity that taken for this research were 1, 3 and 5 m/s respectively.

3. Experimental Work

A test rig has been designed, fabricated and installed to complete a requirements of this study and to validate the numerical model as shown in figure 10.

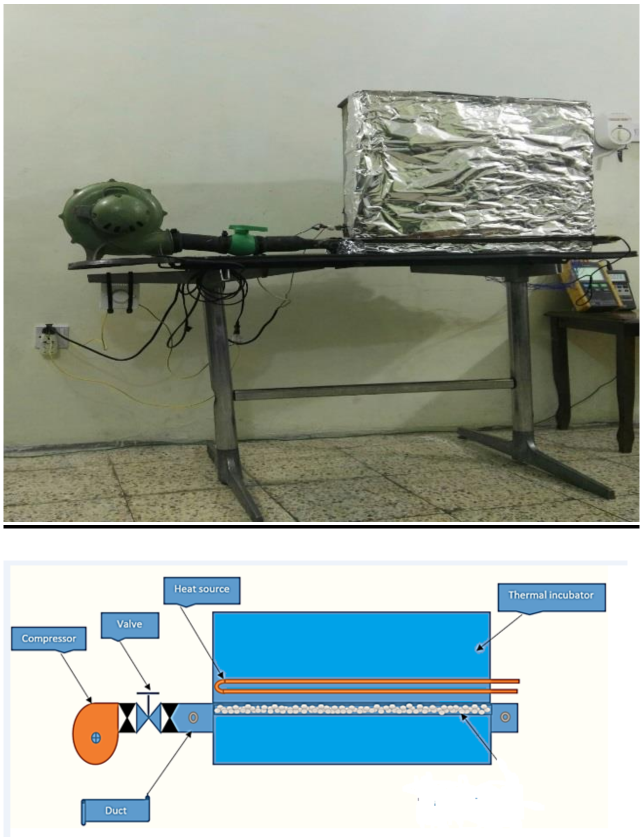

Figure 10. Test rig for the experimental facilities

The heat source has been supplied from the top cover of copper basin. The control electrical system (regulator) has been used to manage the flow within the duct for both types smooth and dimple duct.

3.1 Geometry of wall- cladding system

- The copper basin is the main part for simulating external wall cladding that proposed as a solar collector and storage unit, with a dimensions (150 × 600 × 60) mm. The basin plate thickness 0.5 mm as shown in figure 11.
- Two types of copper covers have been used as a solar collector surface. The first one is normal smooth horizontal plate with, a thickness of 0.5 mm. The second type is horizontal plate with 18 pieces of copper cylinders that have been welded in lower surface of cover. The cylinders diameters is 12 mm and 40 mm in length oriented vertically inside duct in two lines 9 cylinders in each line.

Figure 11. Copper basin

Figure 12. cladding cover with copper cylinders

In this work, two types of air copper ducts were fabricated. The first duct has the smooth surface and the other have a granular dimpled surface with a radius of 10 mm diameter each with a total dimples of 80 half granular spheres, as shown in Figures 13. The second is smooth copper plate as shown in figure 14.

Figure 13. The granular duct

Figure 14. the smooth duct

Electrical Heat source of 1200 W has been insulated in variable altitude stand and applied on the upper plate of duct and cover of basin. 600 W variable capacity air compressor capacity has been used in the current work with rpm around (0-1300). Thermocouples are fixed inside and outside duct. For outside duct the six thermocouples are fixed along the basin line plane between the edge of the duct and copper cylinders with equal spaces between them (60 mm gaps). In addition to that, six others thermocouples along the fluid flow duct and symmetrical spaces with the previous six channels as shown in figure2.

4. Results and Discussion

The model that adopted as a base line fabricated from smooth duct and cover. The average percentage error between experimental and numerical results for air inside duct at velocity of air 1 m /s, 3 m /s and 5 m /s is 8%, 16.5% and 5% respectively. Furthermore, the average percentage error for air outside duct at velocity of air 1 m/s, 3 m/s and 5 m/s is 23%, 25% and 4% respectively. There are several reasons could be behind this error, such as the numerical analysis considered the model to be adiabatic while in real practice there are a little heat losses through insulation, Figures 15, 16 and 17 show these results.

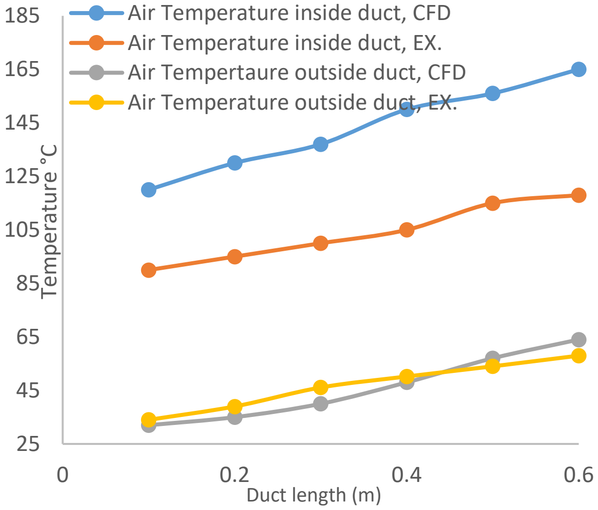

Figure 15 Validation of the experimental results against the CFD results for the temperature of the working fluid at velocity of 1m/s

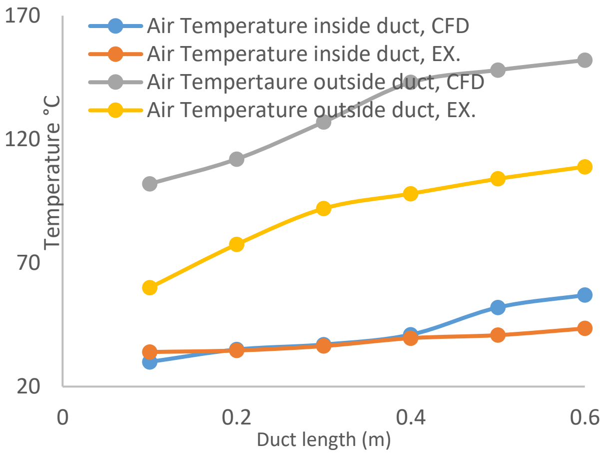

Figure 16. Validation of the experimental results against the CFD results for the temperature of the working fluid at velocity of 3m/s

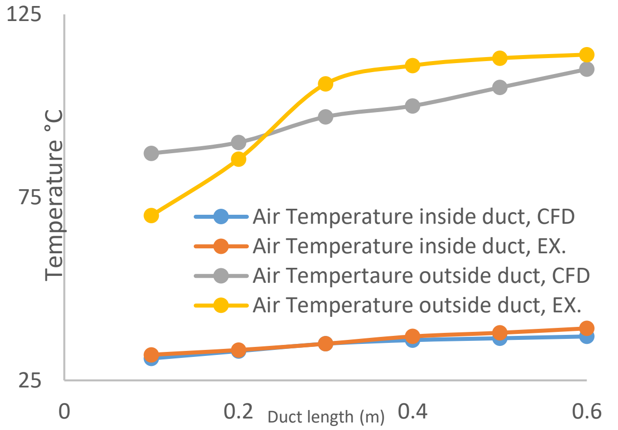

Figure 17. Validation of the experimental results against the CFD results for the temperature of the working fluid at velocity of 5m/s

The effect of adding copper cylinders inside system on the air temperature distribution along the duct at different air velocity are shown in figures 18 to 20. These figures show air temperature distribution for both inside and outside of the duct. The diameter of cylinders which added to the basin is 12 mm and the material is copper. The average percentage enhancement in temperature that compared with the SDSC model with 1, 3 and 5 m/s is 3, 4 and 11% respectively. This increase is justified by the increased surface area of thermal transfer.

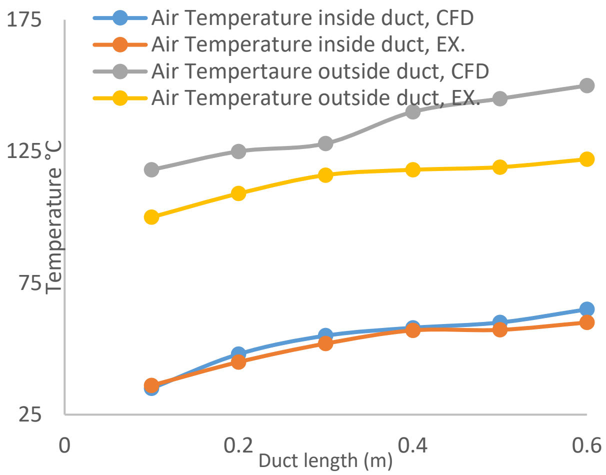

Figure 18. Temperature distribution of air along the duct for SDY model at air velocity of 1 m/s

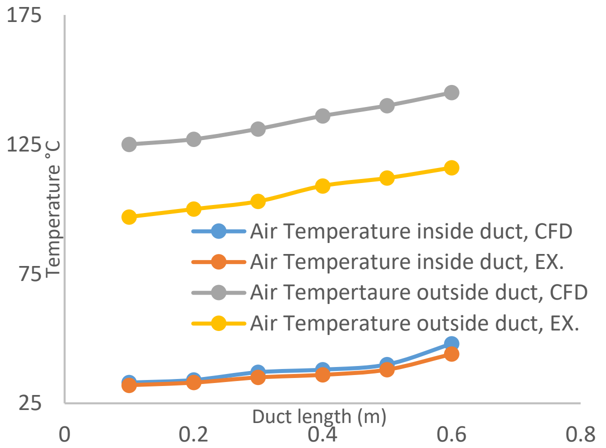

Figure 19. Temperature distribution of air along the duct for SDY model at air velocity of 3 m /s

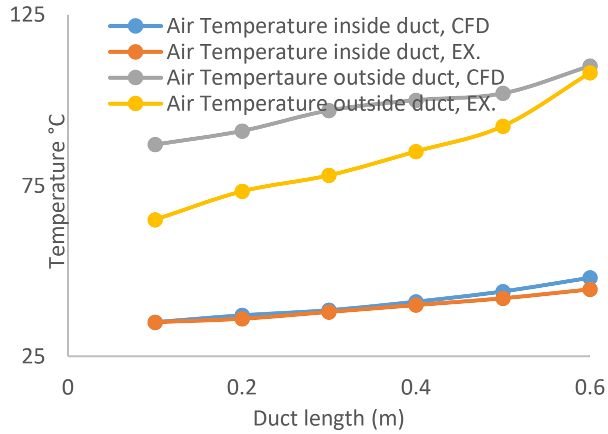

Figure 20. Temperature distribution of air along the duct for SDY model at air velocity of 5 m/s

Figures 21 to 23 show the effect of adding both granular duct and vertical metal cylinders on the air temperature distribution along the duct. It can be concluded that there is enhancement in heat transfer when using granular hollow sphere duct with vertical metal cylinders. The percentage enhancement in temperature of air between this case and SDSC model in velocities 1, 3 and 5 m/s is 27.5, 33 and 35.2 % respectively.

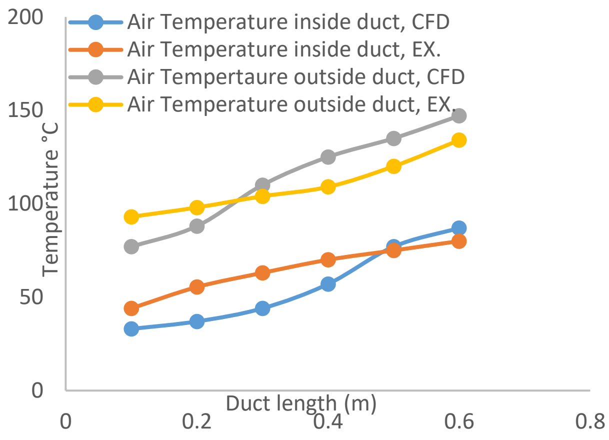

Figure 21. Temperature distribution of air along the duct for GDY model at air velocity of 1 m/s

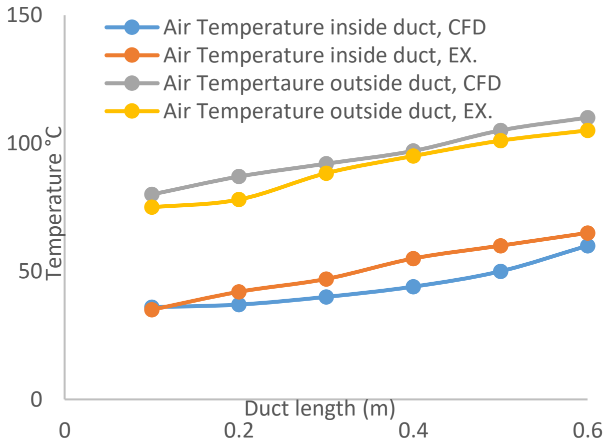

Figure 22. Temperature distribution of air along the duct for GDY model at air velocity of 3 m/s

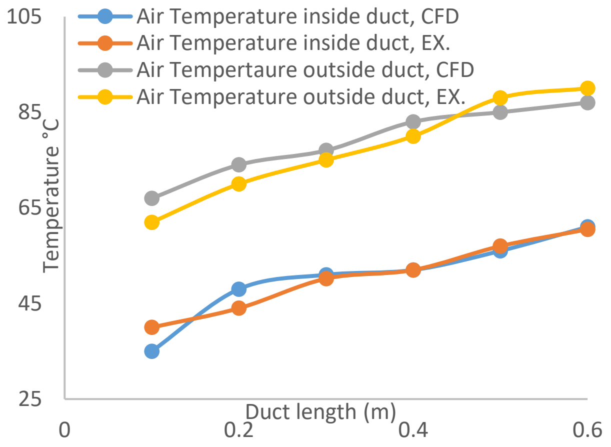

Figure 23 Temperature distribution of air along the duct for GDY model at air velocity of 5 m/s

5. Conclusions

In the present work, a building cladding system has been developed and tested experimentally and numerically. The thermal performance of this system has been improved through building and testing several enhancement methods. According the obtained results, several significant conclusions can be summarized as follows;

1. The use of vertical copper cylinders outside duct led to an improvement in the thermal performance by 3% for the velocity of air at 1 m/s while the percentage increase in heat gain rates were enhanced for the velocity of 3 and 5 m/s by about 4% and 11% respectively.
2. The use of a duct with granulated hollow spheres surface has been found to increase the thermal gain rate fluid flow velocity of 1, 3 and 5m/s by about 19, 28.6 and 16.5% respectively.
3. The thermal enhancement of using both copper cylinders with granular duct (GDY model) is found to increase the rate of general thermal gain by about 27.5%.for 1 m/s and for air velocities 3 and 5 m/s is 33 and 35.2% respectively.

Acknowledgements

This research was supported by [Lec. Ammar A. Hussain]. From [Almustansirya university, college of engineering] who gave access to the laboratory and research facilities. Without his precious support it would not be possible to conduct this research.

Abbreviations

SDSC:	Smooth duct smooth cover.
SDY:	Smooth duct cover cylinders.
GDSC:	Granular duct smooth cover.
GDY:	Granular duct cover cylinders.

6. References

1. A.R. Jaurker a, J.S. Saini b, B.K. Gandhi. (2006). *"Heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness." Solar Energy **80**(8): 895-907.*
2. Ahn, S. (2001). *"The effects of roughness types on friction factors and heat transfer in roughened rectangular duct." International Communications in Heat and Mass Transfer **28**(7): 933-942.*
3. Goel, M., et al. (1994). *"Laminar forced convection heat transfer in microcapsulated phase change material suspensions." International Journal of Heat and Mass Transfer **37**(4): 593-604.*
4. Gräsel, J., et al. (2006*). Parametric Interturbine Duct Design and Optimisation. 25th International Congress of the Aeronautical Sciencies,(ICAS), Hamburg, Germany.*
5. Potter Jr, A. E. and E. L. Wong (1958). "Effect of pressure and duct geometry on bluff-body flame stabilization."
6. Hooman, K. (2007). *"Entropy generation for microscale forced convection: effects of different thermal boundary conditions, velocity slip, temperature jump, viscous dissipation, and duct geometry." International Communications in Heat and Mass Transfer **34**(8): 945-957*