Numerical and experimental study for assessment the effect of baffles in a grooved cavity

Aya H. Yasin¹, Nabil J. Yasin¹
¹Engineering Technical College-Baghdad, Middle Technical University(MTU), Baghdad, Iraq
E-mail: NabilYasin@toc.edu.iq

Abstract. The present study is concentrated on the effect of using baffles (as abstraction means) in a cavity to enhance the heat transfer. For this purpose, an experimental and numerical study was carried out. In the experimental study, a test rig consists of a groove rectangular duct and square baffle at three different positions at the top wall with a constant heat flux applied at the bottom wall was built up and used. In the numerical study, mesh generation and finite volume analysis were performed using Ansys. 18 with (k-ε Relazabel) turbulent model to calculate the pressure drop, temperature, and heat transfer rate. The study was concentrated on the effect of baffle position, and number for the range of Reynolds number (4000-16000) and Grashof’s number between (10⁷-10⁸) using air with (Pr=0.7) as a working fluid. The results show that inserting a square baffle at the right position of the cavity leads to increasing of Nusselt number by (73%) comparing to that cavity with no baffles and (60%), (67%) with that at left and center position for the baffle. Comparing numerical and experimental shows a good agreement between them with a maximum deviation (12%), also a close agreement notice when comparing the present results with some of the previous works.

1. Introduction
The heat transfer process is involved in many industrial commercial and domestic applications via convection, utilization, and energy recovery. Enhancement of heat transfer is vital in many thermal industrial applications, such as electrical application, cooling, microelectronic equipment, refrigerant evaporation and condensation, and waste heat recovery applications[1]. The convection in enclosure and cavities have been of interest to engineering application. It is well known that the convection in an enclosure is due to either vertically or horizontally imposed temperature difference [2]. Many types of turbulators such as ribs, fins, baffles, winglets, are installed into a channel to increase the heat exchange rate. The design of turbulators and their parameters, shape, attach angle, height, space between them and location has been the subject of attention of many researchers [3]. How and Hsu, (1998) [4], investigated numerically 2-D simulation for the mixed convection under transient laminar flow in enclosure divided by a conducting baffle. A heat source was mounted on a vertical wall in the enclosure. The results included that the rate of transient heat transfer depends strongly on the location and height of the baffle.

Radhakrishnan et al. (2009) [5], investigated experimentally and numerically the mixed convection from a heat-generating element in a vented cavity without and with baffle configuration. The study was focused on the best location of the baffle on the rectangular chamber walls and the effect baffle heights on the heat transfer enhancement from the heater. The results illustrated that the enhancement in heat transfer...
be maximum when the baffle posited in the middle of the lower wall. Asif et al., (2011) [6], carried out a numerical study for 2-D investigation for mixed convection flow in a vertical enclosure where the baffles were heated isothermally and mounted on both at the right and left sides walls of the enclosure, all other walls assumed to be adiabatic. Successive under Relaxation (SUR) method was used in the discretizing equations with specified boundary conditions. They concluded that when the Richardson and Reynolds numbers are higher that’s led to give higher heating efficiency in the enclosure and maximum heating efficiency.

Sriromreun et al., (2012) [7], investigated numerically and experimentally the effect of baffle turbulators on the augmentation of heat transfer in rectangular channel. Z-shaped baffles are located in zigzag shapes which are aligned in series on the isothermal-fluxed upper wall. It was found that, increasing in (Nu) by about 35% when (Re) and the height of baffle increases while the thermal performance enhancement (TEF) tends to decrease. Nardini et al., (2015) [8], analyzed the natural convection in a square cavity with vertical and horizontal Plexiglas walls and separated sources. The vertical walls are attached with two Plexiglas baffles symmetrically between the sources. The results pretend clearly different baffle lengths have a remarkable influence on the characteristics of flow and heat transfer of the fluid. Sharma et al., (2015) [9,10], investigated a 2-D numerical simulation for the characteristics of heat transfer and the structure of the flow in a baffled grooved channel. The study included a channel without and with a baffle in the middle of the enclosure with heating differentially from the sides. The outer walls were considered to be adiabatic. The results showed that, for any baffle location and fluid flow direction, the pressure drop and pumping power increases with increasing of baffle height. Menni et al., (2016) [11], carried out a 2-D numerical investigation for a rectangular baffled channel for knowing the thermo-hydraulic behaviour of convection heat transfer under turbulent forced flow. Yasin et al., (2018)[12], carried out a numerical and experimental investigation for determining the influence of vertical unheated baffle on the mixed convection fluid flow in square-grooved- cavity with the heating source from three sides.

The results showed that, when the baffle length is full, Nusselt numbers reached to maximum values. Most of the previous studies about the convection heat transfer in enclosures and cavities were achieved using the numerical methods for the solution of the governing equations. While a few little of these studies use the experimental method. None of the above literature determined the best baffle position or the preferred geometrical shapes of baffles. The present work was divided into two parts, numerical and experimental study for a cavity having a baffle located at the top of the cavity while at the bottom of the cavity, constant heat flux was applied on the horizontal bottom surface. The experimental study of this work claimed to be from the few attempts in the field of the heat transfer by convection within the cavity having an obstruction

2. Experimental set up and the experimental Procedure
The schematic diagram of the test rig is shown in Figure. (1) which consists mainly of a rectangular duct, cavity, baffle, heater, fan, measurement tools, and devices. The duct is fabricated from aluminium with dimensions of (250x250) mm, thickness (2mm) and (2000) mm length, rectangular cross-section area, and the cavity was made from an aluminium sheet with a dimension of (250*250) mm and (5 mm) thickness. The cavity bottom wall was heated using a plate heater. The baffle is a square plate made from aluminium having a dimension of (250x250) mm and (6 mm) thickness, inserting vertically downward inside the cavity from the top wall. The baffle was placed at three positions (left, right and middle) in the top wall of the cavity as shown in Figure (2).
The cavity bottom wall was heated using a plate heater. The baffle is a square plate made from aluminium having a dimension of (250x250) mm and (6 mm) thickness, inserting vertically downward inside the cavity from the top wall. The baffle was placed at three positions (left, right and middle) in the top wall of the cavity as shown in Figure (2).

Many thermocouples were distributed along and around the test rig to measure the air temperature shown in Figure. (3). A hot-wire anemometer, model (TES 1341) was used to measure the average air velocity inside of the test section by recording the linear movement of air flow. The air pressure drop occurs through the cavity was measured using manometer (PM-9102 Pressure Manometer). The assembled rig was located in a space far from environmental effects such as direct sunlight, vibrations, wind, and temperature fluctuation to prevent their effect on the experimental results. Before the experiments started, all measurement devices used in the experiment were calibrated according to the general standard schemes. To start the experiments a primary test was carried out to check the air leakage, heater On-Off.
operation, controlling of the fan to provide the desired flow rate at a given speed then the experiments were started as follows:

1- Setting the baffle at the required position and height.
2- Switching (ON) the blower to allow air flowing continuously inside the duct through the test section and wait for a period of time to be sure that the flow is stable and reached the specified required value.
3- Switching (ON) the heater located at the bottom wall of the cavity.
4- Recording the inlet and outlet of the flow rate of air flowing and the pressure drop through the cavity.
5- The temperatures of all thermocouples were recorded by using the data logger (12 thermocouples) every (10) minutes until reaching steady state.
6- The voltage and current were recorded when the time steady, and the air temperatures within the cavity was measured by thermocouple along a period of time step about 10 min between each reading.

All previous procedure was repeated for a velocity inlet values(0.3,0.5,0.8 and 1m/s) and applied heat flux of (300 and 500 W) at the bottom wall with and without baffles, using different baffle heights and positions (left, right, center).

3. Data Analysis

The heat delivered by the heaters can be known by:

\[ Q_{heater} = \text{Power} = I \cdot V \]  

(1)

The heat gained by air in the test section was calculated as

\[ Q_{air} = \dot{m}_a \cdot c_p a \cdot \left( T_{out} - T_{in} \right) \]  

(2)

The heat transfer by convection can be calculated by using:

\[ Q_{conv} = h_{ave} \cdot A_s \cdot \left( T_W - T_{ave} \right) \]  

(3)

By using energy balance, assuming no heat losses, the heat transfer of convection equals to the heat gained by air, and then the heat transfer rate was calculated as:

\[ h_{ave} = \frac{Q_{conv}}{A_s \cdot (T_W - T_{ave})} \]  

(4)
$T_{ave}$ was obtained from the experiments by averaging six locations of measured temperatures along the midline of the cavity. All the physical properties for the air were taken at mean film temperature ($T_f$).

$$T_f = \frac{T_{out} + T_{in}}{2}$$  \hspace{1cm} (5)

Average Nusselt number is calculated from

$$Nu_{ave} = \frac{h_d}{R}$$  \hspace{1cm} (6)

Uncertainties and error experimentally can be arising from calibrations, measurements, kinds of instruments and readings. Nusselt number as in equation (6) is a function of many independent variables. According to the uncertainty analysis given by Doebelin [13], Nusselt number uncertainty analysis shows that the maximum error is ($\pm 0.352$).

4. CFD MODELING AND SIMULATION

4.1 Geometry

The geometry considered in this study is a three-dimensional duct-cavity configuration Horizontal square duct has dimensions of (H = 250 mm) and exit length (2H), the square cavity depth equal to (H). The geometric dimensions of the duct are described in Figure (4). The height of the inflow and outflow openings is equal to the depth of the cavity. The heat source was mounted at the bottom wall while the remaining walls of the duct and cavity are considered to be adiabatic. The baffle is mounted at the top wall of the cavity in three different locations. The flow is considered to be turbulent, incompressible and the have a negligible viscous dissipation. All the thermophysical properties of the fluid are assumed constant except for the variation in density with temperature (Boussinesq approximation) giving rise to the buoyancy forces. The thermophysical properties of the fluid are evaluated at constant ambient temperature, ($T_i$) which is fixed at 295 K in all cases. Three dimensional governing equations of energy, continuity and Navier-Stokes equation was used by FLUENT to solve the turbulent flow field. A square geometrical shape of the baffle was considered in the present simulation. The baffle has a dimension of (250*250) mm with a thickness (6 mm). Table(1) shows the geometrical parameters of the baffle. A total of eight cases were considered in the present numerical simulation. The details of the cases are listed in Table (2). All cases were simulated under the same boundary and initial conditions as in the experimental analysis.

Figure 4. Physical domain and geometric dimensions of the duct employed in the numerical solution (all dimension in mm)
**Table 1. The geometrical dimension of the baffles.**

| Type of Baffle | Parameter | Symbol | Value (mm) |
|----------------|-----------|--------|------------|
| Square Baffle  | Length    | H      | 250        |
|                | Width     | W      | 250        |
| Baffle         | Height ratio | h     | 0.25-0.75 H |
|                | Thickness | b      | 6          |

**Table 2. Cases under numerical simulation**

| Case No. | No. baffle | Position        | Velocity (m/s) | θ°  | h       |
|----------|------------|-----------------|----------------|-----|---------|
| 1        | 1          | Right           | 0.3, 0.5, 0.8, 1 | 90° | 0.75 H  |
| 2        | 1          | Center          | 0.3, 0.5, 0.8, 1 | 90° | 0.75 H  |
| 3        | 1          | Left            | 0.3, 0.5, 0.8, 1 | 90° | 0.75 H  |
| 4        | 2          | Right and left  | 0.5            | 90° | 0.75 H  |
| 5        | 3          | Right-center-left | 0.5      | 90° | 0.75 H  |
| 6        | 1          | Center          | 0.3, 0.5, 0.8, 1 | 60° | 0.75 H  |
| 7        | 1          | Left            | 0.3, 0.5, 0.8, 1 | 60° | 0.75 H  |
| 8        | 1          | Center          | 0.5            | 90° | 0.75, 1.1, 1.2, 1.5 H |

4.2 *Boundary Conditions*

The boundary conditions and domain physics for cases under study are listed in Tables (3) and (4). The other boundary condition is a non-slip condition, heater with wall condition on bottom surfaces with (500) Watt, three position of the baffle (left, right and center). On the duct surface \( (U_x=U_y=0) \) where 1054000 elements tetrahedral mesh of finite volume method cell has been selected. Checking the numerical convergence has also been done.

5. *Results & Discussion*

Figure (5) shows the temperature distribution within the cavity for different air velocities and constant heat flux at the bottom wall (300 watts) at a fixed baffle height \( (h=0.75 \text{ H}) \). The Figure shows that when air speed increase then the temperature decreases and maximum temperature value occurs in front of the heater, this is due to the increase of turbulent flow leads to a decrease of interconnection between air molecules. Moreover, the Figure shows that the difference between the temperature values at a different position along the midline of the cavity seems to be uniform. This is due to the relative smoothness of air passing the cavity. A comparison for the values of the measured air temperature between the cavity with and without baffle shows clearly that the level of the temperature when using baffle is lower especially at the right baffle position that is due to baffle effect and the turbulence occurred in the zone.
### Table 3. Boundary condition

| Boundary | Description | Type | Value |
|----------|-------------|------|-------|
| Air inlet | Velocity inlet | Type | Velocity inlet |
|          | Static temperature | Value | 22 °C |
|          | Heat transfer | Type | Static temperature |
|          | Inlet velocities | Value | 0.3, 0.5, 0.8, 1 m/s |
| Air outlet | Pressure outlet | Type | Pressure outlet |
|          | | Value | 0 pa |
| Wall | Heat transfer | Type | adiabatic |
|          | Mass and momentum | Type | No slip wall |
| Heat flux | Wall | Type | Bottom surface of cavity |
|          | Heat transfer | Value | Constant heat flux |
|          | power | Value | 500 W |

### Table 4. Domain physics

| Fluid | Type | Value |
|-------|------|-------|
| Analysis state | Type | steady |
| Material | Type | Air at 22°C |
| Thermal conductivity (K) | Value | 0.0242 W/m.K |
| Density (ρ) | Value | 1.225 kg/m³ |
| Specific heat (Cp) | Value | 1006.43 J/kg.K |
| Type flow | Type | Turbulent |
| Type | Type | Solid |
| Analysis type | Type | Solid |
| Material | Type | Aluminum |
| Domain motion | Type | Stationary |
| Thermal conductivity (K) | Value | 250 W/m.k |
| Type | Type | Solid |
| Analysis type | Type | Steady state |
| Material | Type | Aluminum |
| Domain motion | Type | Stationary |
| Thermal conductivity (K) | Value | 250 W/m.k |
Figure 5. Variation of temperature with the midline of the cavity different \( (\text{Re}) \) and baffle position \( (q=300 \text{ W}, h=0.75 \text{H}) \)

Figure. (6) shows the effect of the baffle height on air temperatures inside the cavity for the three-positions of the baffle, for \( \text{Re}=13000 \) and heat flux corresponding to \( (\text{Gr}=10^8) \). In general, it is observed that at any baffle height and location, the air temperatures seem to be decreasing due to the baffle effect. When the baffle height is increased from that without baffle \( (h=0) \) to \( (h=0.75 \text{ H}) \) the hot streams pushes the circulating cell which formed in the groove of the cavity, hence the external flow is guided along with the baffle and make a distribution for the circulating cell which developed inside the groove of the cavity.

For example, for the right baffle with baffle height \( (h=0.75 \text{ H}) \), it is noticed that the maximum temperature inside the cavity is lower by a rate of 50.9% as compared to that without baffle. At the left baffle position, the rate will be less by (37.7%) and (54.7%) at center baffle condition Figure (7) shows the relation between the \( (\text{Re}) \) against friction factor at the three positions of the baffle and different height \( (0.25H, 0.5H, 0.75H) \) with \( q=500 \text{ W} \) corresponding to \( (\text{Gr}=10^8) \). The Figure shows that when \( (\text{Re}) \) increases the friction factor decrease that is due to the high speed of air which causes an air separation from the wall and creates a negative pressure gradient. Notice from the Figure that maximum value for friction factor at height \( h=0.75\text{H} \) equals to for the left, 0.52 for the center and 0.28 for the right baffle position at \( \text{Re}=4000 \) while the minimum friction factor was 0.05 for all positions at \( (h=0.25\text{H}) \) when \( \text{Re}=16000 \). In general, the Figure shows that at same position of the baffle, the value of the friction factor increases with increasing of the baffle height for any baffle position.
As shown from Figure (8) the friction factor and pressure drop are at maximum when baffle position is at (center), hence the heat transfer increased with baffle height with increasing of friction factor and with more pumping power. The effect of variation in Nusselt number with Richardson number shows in Figure (9), which shows that (Nu) increase with decreasing of (Ri), and there is a marginal heat transfer enhancement when the baffle is at the right position, it is maximum value was when Ri < 1, the same observation is found for the left and center position with extremely lower value. Richardson number is the ratio between natural convection to force convection (Ri=Gr/Re^2). If the Richardson number is of order unity, then the flow is likely to be buoyancy-driven, the energy of the flow derives from the potential energy in the system. [14]. The thermal performance(TP) of the cavity is defined as the ratio of (Nu) of the cavity with baffle over that is without baffle. Whenever the value increase that indicated a good performance for that geometry. Figure (10), show the comparison between three positions of baffle with height (h=0.75 H) and a Gr=108. That is clear from the Figure that the right baffle position has the highest value of thermal performance of (5.5). Such Figure is a good tool to assess the enhancement and suggest a suitable location and height of baffle on the external flow direction to achieve the best heat transfer enhancement from the different considered variations of the baffled cavity.

Figure 6. Temperature Variation with Different Baffle Position and Height at Re=13000, q=500 W.

Figure 7. Variation of Friction Factor with (Re) for different baffle position and heights at, q=500 W.
Figures (11) and (12) shows the effect of the baffle position on the shape of vortices, depending mainly on the flow direction. From the isothermal contour, it can be noticed that the maximum temperature located at the right wall of the cavity as an opposite flow that is exactly noticed from experimental work. The isothermal lines help to visualize the temperature distribution in the cavity. It can be observed from the isothermal line, that thermal boundary layer stacked alignment to the right wall of the cavity; this is due to opposing forced flow appreciated. The isothermal lines are almost uniformly distributed especially at (Re=4000) but with increasing of (Re) to (16000) their shape's characteristics are getting to be more complex. From above, it is clear that the highest value of the temperature gradient is located near to the edge right groove wall and the smallest value in the left groove wall. The thermal boundary layer that clustered from the bottom wall (where heater located) is moved as wavy shape towards up and heated fluid going in from the cavity. The velocity streamlines shown in both Figures, as the cavity is heated bottom wall, the cold air will force the flow through the cavity, hence, a small circulating cell is formed.
entire the cavity. The second air circulating is appearing behind the baffle (at any position) at the top tip, this is due to the isolated region created by the baffle, and reversed flow has happened. The width of the circulating cell behind the baffle is different and depending mainly on the baffle position. The third circulating cell appears at the down wall of the ended channel, it has shown clear at the center baffle position, and appears at the right position only when the velocity increases and it is not found at the left position due to the highest speed applied perpendicularly towards air outlet. In general, it can be noticed that increasing of (Re) lead to form a large circulating zones above the mainstream due to the fact that at a high (Re), the flow velocity is high while it is known that a low value of (Re), the induced flow has little energy and expand suddenly due to the pressure rise in the cavity. Figure (13) shows the variation of friction factor with different (Re) at h=0.75 H and q=500 W estimated from the numerical data which shows that the friction factor starts to increase at the zone which baffle was inserting. Figure (14) shows the variation of (Nu) with (Re) and (Ri) for the three different baffle positions from the numerical data. From the Figure, we may observe the increase of (Nu) by increasing (Re) and the dominated process is the forced convection as (Ri) is less the unity. Figure (15) show temperature and pressure gradient within the multi baffle cavity estimated numerically, the Figure shows that maximum average temperature values have happened when using three baffles this is due to the first and second baffle push the hot air carried out from the cavity thus air attack with the third baffle hence, the maximum the temperature was presented. While the pressure gradient at the mid horizontal line within the cavity will decrease when used. two baffles as the negative pressure occur inside the cavity. Figure (16) shows the variation of the effect of baffle angle on the value of friction factor which shows that the while the pressure drop and friction factor at baffle inclined angle of (θ=90°) is higher than at (θ=60°) for both at left and center baffles. Figure (17) shows the relation between (Nu) with (Re) and (Ri) estimated from the numerical results at the baffle angle of (θ=60°) which shows that the best location of inclined baffle was found to beat center position at this angle as it gives highest Nusselt number.

![Isothermal lines for Cavity with different position of Baffle (h=0.75 H), q=500 W.](image)

Figure 11. Isothermal lines for Cavity with different position of Baffle (h=0.75 H), q=500 W.
a) $Re=4000$  

b) $Re=16000$

**Figure 12.** Velocity lines for Cavity with different Position of Baffle ($h=0.75H$), $q=500$ W.

![Velocity lines for Cavity with different Position of Baffle](image)

**Figure 13.** Friction factor with ($Re$) for baffle height ($h=0.75H$), ($q=500$ W) from numerical data.

![Friction factor with ($Re$) for baffle height](image)
Figure 14. Variation (Nu) with (Re) and (Ri) for square baffle (h=0.75 H), q=500 W numerically.

Figure 15. Variation Temperature value with x-axis for multi of baffle at Re=8000 and q=500 W Numerically.

Figure 16. Variation of the friction factor with (Re) for different baffle angles, estimated numerically (q=500W).
Figure 17. (Nu) With (Re) and (Ri) for cavity with inclined Baffle ($\theta=60^\circ$, $q=500$W), estimated numerically.

Figure (18) shows the variation of the average Nusselt number with the Reynolds number for present work experimentally and numerically. The Figure shows a good agreement between them with the maximum deviation was (17%), which attributed mainly to the assumption used in the numerical estimation. Figure (19) shows the variation of the local Nusselt number with the dimensional axial distance for the present work (Cavity without baffle with $(Re=4000)$ and $q=500$ W) and the results of another researcher. Clearly, from this Figureure, there is the same trend between present work and Stiriba [8], Yasin [25].
6. Conclusions
The following conclusion can be derived from the present study:

1. Using the baffle in a cavity lead to enhance the heat transfer process within it, where the temperature of air inside the cavity is decreasing when Reynolds number increases.

2. Using the baffle in the cavity lead to increase the Nusselt number, by (83.6 %) comparing with that of no baffle, with a considerable increasing of pressure drop by (0.213 Pa).

3. Nusselt number for the air inside the cavity with baffle position to the right position is found to be more by (18.5 %) and (33.7 %) comparing with the cavity having baffle at the center and left position respectively with pressure difference of (0.213 pa), (0.116 pa) and (0.208 pa) for right, center and left baffle positions respectively.

4. Using Inclined baffle by (60°) at center and left positions, lead to increasing of Nusselt number by about (25.7 %) and (29.1 %) from that at θ=90°. pressure difference about (15.6 %), (62.2 %) for center and left baffle positions respectively.

References
[1] Ghorab, Mohamed G. 2015 Forced convection analysis of discrete heated porous convergent channel. J.of Heat transfer Engineering 36(9): 829-846.
[2] Hasnaoui, M., E. Bilgen, and P. Vasseur. J.Eng 1992 Natural convection heat transfer in rectangular cavities partially heated from below. J. of Heat transfer 6(2): 255-264.
[3] Jedsadaratanachai, Withada. 2014 Mathematical Investigation on Laminar Forced Convection in Square Channel with U-Shaped Rib Turbulator. J.of Heat transfer Engineering 17(1): 36.
[4] How, Sey-Ping, and Tsan-Hui Hsu. 1998 Transient mixed convection in a partially divided enclosure. J. of Computers & Mathematics with Applications 36(8): 95-115.
[5] Radhakrishnan, T. V., Joseph, G., Balaji, C., & Venkateshan, S. P. 2009 Effect of baffle on convective heat transfer from a heat generating element in a ventilated cavity, J. of Heat and mass transfer, 45(8): 1069-82.
[6] Asif MR, Hossain MS, Hossain KA. 2011 Heat Transfer in a rectangular enclosure with baffles. ARPN J. Eng. Appl. Sci. 6: 29-41.
[7] Sriromreun, Parkpoom, Chinaruk Thianpong, and Pongjet Promvonge. 2012 Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles. International Communications in Heat and Mass Transfer 39(7): 945-952.
[8] Nardini, Giorgia, Massimo Paroncini, and Raffaella Vitali. 2015 Natural convection in a square cavity with two baffles on the vertical walls: experimental and numerical investigation. International Journal of Mechanics 9: 120-127.
[9] Sharma, Aayush K., Pallab S. Mahapatra, Nirmal K. Manna, and Koushik Ghosh. 2015 Mixed convection in a baffled grooved channel. Sadhana 40(3): 835-849.
[10] Sharma, Aayush K., Pallab S. Mahapatra, Nirmal K. Manna, and Koushik Ghosh. 2015 Mixed convection heat transfer in a grooved channel in the presence of a baffle. J. of Numerical Heat Transfer, Part A: Applications 67(10): 1097-1118.
[11] Menni, Y., A. Azzi, C. Zidani, and B. Benyoucef. 2016 Low-Reynolds-number turbulent forced-convection flow over graded baffle plates. Revue des Energies Renouvelables 19(2): 235-250.
[12] Yasin, Nabil Jamil, Kadhum Audaa Jehhef, and Asraa Mahdi Shaker. Assessment of the Baffle Effects on the Mixed Convection in Open Cavity. International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS 18(4).
[13] Doebelin, E. O., Measurement Systems, McGraw-Hill, 1975.
[14] Bergman, Theodore L., Frank P. Incropera, David P. DeWitt, and Adrienne S. Lavine. 2011 Fundamentals of heat and mass transfer. John Wiley & Sons, sixth Edition.
## Nomenclature

| Symbol | Description                                      | Unit             |
|--------|--------------------------------------------------|------------------|
| A      | Area                                            | m²               |
| b      | Baffle width                                     | m                |
| g      | Acceleration due to gravity                     | m/s²             |
| h      | Height ratio of the baffle                       | m                |
| I      | Current                                         | Amp.             |
| L      | Heated wall length                               | m                |
| Q      | Heat Transfer rate                               | W                |
| TP     | Heat transfer enhancement                       | -                |
| f      | Friction factor                                  | -                |
| K      | Thermal conductivity                            | W/m.k            |
| Cp     | Specific heat at constant pressure              | KJ/kg.k          |
| d      | Diameter                                         | m                |
| H      | Cavity height                                    | m                |
| h_ave  | Average convective Heat transfer coefficient     | W/m².k           |
| h      | Height ratio of the baffle                       | m                |
| T      | Temperature                                      | K                |
| u      | Air Velocity                                     | m/s              |
| V      | Voltage                                          | V                |
| N      | Current                                          | Amp.             |
| A      | Area                                             | m²               |
| R      | Reynolds Number                                  | -                |
| Pr     | Prandtl Number                                   | -                |
| Re     | Reynolds Number                                  | -                |
| Ri     | Richardson Number                                | -                |
| Nu     | Nusselt Number                                   | -                |
| Gr     | Grashof's Number                                 | -                |
| Pr     | Prandtl Number                                   | -                |
| Re     | Reynolds Number                                  | -                |
| Ri     | Richardson Number                                | -                |
| a      | Air                                              | -                |
| ave    | Average                                          | -                |
| conv   | Convection                                       | -                |
| f      | Film                                             | -                |
| d_h    | Hydraulic diameter                               | -                |
| Num    | Numerical                                        | -                |

### Creek symbols

| Symbol | Description                                      | Unit             |
|--------|--------------------------------------------------|------------------|
| ΔP     | Pressure drop                                    | Pa               |
| β      | Thermal expansion coefficient                    | Gr               |
| θ      | Baffle inclination angle                          | Pr               |
| V      | Kinematic viscosity                              | Re               |
| μ      | Dynamic viscosity                                | Ri               |
| P      | Density                                          | Kg/m³            |

### Subscripts

| Symbol | Description                                      | Unit             |
|--------|--------------------------------------------------|------------------|
| in     | inlet                                            | -                |
| out    | Outlet                                           | -                |
| th     | Thermocouple                                     | -                |
| sur    | Surface                                          | -                |
| Exp.   | Experimental                                     | -                |

### Dimensionless Number

- Nu = (h d_h)/K
- Gr = (g β (T_{sur} - T_{ave}) H^3)/v^2
- Pr = (Cp μ )/K
- Re = (ui d_h)/ v
- Ri = Gr/Re^2