An experimental and numerical study for two phase flow (water-air) in rectangular ducts with compound turbulators

M W Al – Jibory¹, R S Al – Turaihi ², and H N Al – Jibory³

¹,³Department of Mechanical Engineering, University of Kerbala, Kerbala, Iraq
²Department of Mechanical Engineering, University of Babylon, Hilla, Iraq

Abstract. The improvement of heat transfer by using multiphase flow is a key focus in the energy industry and related applications such as steam turbines (power plants), steam generators and condensers, refrigeration, food manufacturing, and heating and cooling systems. Dispersed bubble two-phase flow (water-air) with heat transfer in a rectangular turbulated vertical canal with dimension 5 × 3 × 70 cm was thus studied in the current work and experimental and numerical studies were performed to test the influence of the superficial inlet velocity of air and water and the position of grooves at constant heat on the heat transfer coefficient and the temperature distribution along the test section. Water superficial inlet velocities were (0.0987, 0.1974, 0.296, and 0.395 m/s), while air superficial inlet velocities were (1.4609, 2.923, and 4.384 m/s), and the heat power was a constant (109.65 W). The results indicated that the local coefficients of the heat transfer for the experimental and numerical study were raised as the superficial inlet velocity rose. The opposite effect was indicated in terms of the temperature distribution along the test section which dropped as superficial inlet velocities rose. The presence of compound turbulation led to an enhancement of the experimental and numerical heat transfer coefficients over those seen in the smooth channel by (56.5% and 54.7%), respectively, for g/p = 0.55 and water and air superficial velocities of (0.395 m/s and 4.384 m/s). Good agreement was found between the experimental and numerical data, with the percentage deviation between the experimental and numerical results being only (5.77%).

Key words: Heat transfer, vertical channel, two-phase flow, turbulators, compound turbulator, CFD, Ansys Fluent.

Nomenclature

| Symbol | Description | Unit |
|--------|-------------|------|
| ΔT     | Temperature difference | K    |
| μ      | Dynamic viscosity | kg/m. s |
| A      | Cross-sectional area | m²    |
| a      | Air | - |
| E      | Energy | J    |
1. Introduction

A phase is defined as one of the states of matter, whether liquid, solid, or gas. Multiphase flow is thus the synchronous flow of numerous phases. The reporting of multiphase flow is fundamental to industries related to energy generation and its applications (Awad Chapter 11) [1] such as steam turbines (power plants), steam generators, condensers, refrigeration, coal fired furnaces, pipelines carrying oil and natural gas, food manufacturing, nuclear power generation units, and heating and cooling systems, due to several characteristics such as temperature homogeneity, high rates of mass and heat transfer between phases, and good mixing of phases. One recognized method for increasing the rate of heat transfer from a surface is to roughen the surface either with designed roughness elements or randomly by using sand grains. An increase in heat transfer is generally accompanied with an increase in the resistance to fluid flow.

Gas–liquid flow has been studied in several numerical and experimental research projects. Xu et al. (1999) [2] conducted an experimental investigation of air-water flow in straight rectangular canals, observing flow regimes by utilising a CCD camera and scanning the images. Vlasogiannis et al. (2002) [3] experimentally defined the influence of flow regime on the coefficient of heat transfer in vertical channels and reported visual observations by video camera to create a flow regime map. Hetsroni et al. (2003) [4] performed a study on flow regimes and warmth transfer for air-water flow in an inclined tube with angle of 8°. Asano et al. (2004) [5] experimentally clarified the characteristics of two-phase flow in a plate warmth exchanger with a vertical single canal by visualisation using neutron radiography to measure the two-dimensional void fraction allocation. Rozenblit et al. (2006) [6] experimentally investigated the flow patterns of air-water flow by using surfactant in a vertical

\[ \varepsilon \text{ Turbulent dissipation rate } \text{m}^3/\text{s}^3 \]

\[ f \text{ Fluid } \]

\[ g \text{ Acceleration of gravity } \text{m/s}^2; \text{ standard value } = 9.80665 \]

\[ h_x \text{ Local heat transfer coefficient } \text{W/m}^2.\text{K} \]

\[ l \text{ Turbulent intensity } \]

\[ m \text{ Inlet } \]

\[ n \text{ Thermal conductivity } \text{W/m.K} \]

\[ \mathcal{Q} \text{ Mass flow rate } \text{Kg/s} \]

\[ q \text{ Heat flux } \text{W/m}^2 \]

\[ Q \text{ Volume flow rate } \text{m}^3/\text{s} \]

\[ r \text{ Rib } \]

\[ s \text{ Reynolds number } \]

\[ t \text{ Inlet } \]

\[ T \text{ Temperature } \text{K} \]

\[ \nu \text{ Turbulent intensity } \]

\[ \mathcal{U} \text{ Superficial velocity } \text{m/s} \]

\[ \mathcal{V} \text{ Mass averaged velocity } \text{m/s} \]

\[ \mathcal{W} \text{ Wetted perimeter of the channel } \text{m} \]

\[ \mathcal{W}_\text{water} \text{ Water } \]

\[ x, y \text{ Variables Coordinate } \]

\[ \alpha \text{ Volume fraction } \]

\[ \rho \text{ Mass density } \text{kg/m}^3 \]

\[ \kappa \text{ Turbulent kinetic energy } \text{m}^2/\text{s}^2 \]

\[ c_p \text{ The specific heat of fluid } \text{kJ/kg.k} \]
upward tube. Layek et al. (2007) [7] offered an experimental report on the characteristics of warmth and liquid flow for turbulent flow in a duct having rectangular shape and rib-groove turbulator and compared it with flow in a smooth duct. Wang and Sun (2007) [8] experimentally investigated the features of friction and warmth transfer in a duct with a square shape roughened using various shaped ribs on one side of the wall. Bilen et al. (2008) [9] performed an experimental investigation into the characteristics of friction and surface heat transfer for turbulent air flow in various grooved tubes, including circular, square, and trapezoidal cross-sections. Eiamsa-ard and Promvonge (2009) [10] examined the shared impacts of turbulators containing ribs and groove on the turbulent forced convection warmth transfer and friction features in a rectangular duct under identical heat flux limit conditions. Hong and Liu (2010) [11] developed techniques for exploring flow patterns of two-phase flow by means of high-speed image acquisition. Bolotnov et al. (2011) [12] performed a numerical study of two phase flow DNS simulations by using the finite element method to examine turbulent bubbly channel flow. Sur and Liu (2012) [13] concentrated on an experimental study of two-phase flow patterns, examining the frictional pressure drop of air-water and flow regime transitions in circular microchannels. Ansari and Arzandi (2012) [14] explored the effects of using ribs of different heights on regime boundaries in smooth and ribbed rectangular ducts for two-phase flow (air-water) and presented flow map diagrams. Habeel and Al-Turaihi (2013) [15] experimentally and numerically studied the two-phase flow phenomena for different-shaped obstacles (circular-, square-, and triangular-section) in a rectangular channel for two-phase flow with different air and water flow rates. Al-taae and Jurmut (2014) [16] conducted an experimental and numerical examination of the warmth transfer characteristics in a circular pipe with ring ribs. Jalghaf et al. (2016) [17] experimentally and numerically explored the two-phase flow for a heated body (circular-cylinder) placed horizontally to create unsteady flow in a rectangular channel. Kong and Kim (2016) [18] presented an experimental study characterising horizontal two-phase flow in a round pipe where the visualisation of flow regimes was performed using a video camera under a broad range of two-phase flow conditions. Al-Turaihi and Kareem (2016) [19] experimentally and numerically investigated the characteristics of air-water two-phase flow in a rectangular channel set in a vertical plane by utilising internal ribs and different discharge rates of water and air, under constant heat, showcasing the effects of varying the discharge of water and air on the coefficient of heat transfer.

In this paper, an experimental and numerical exploration of the two-phase flow (water-air) in a straight up rectangular canal with a compound turbulator is investigated and compared with similar flow in a smooth channel in terms of maximum heat transfer enhancement. A computational fluid dynamics simulation is performed for water-air flow in turbulated and smooth channels, with three positions used for the groove (g/p = 0.37, 0.55, and 0.73). The results are compared with the smooth channel to show the turbulator’s effect on the heat transfer coefficient.

2. Experimental test

Experimental equipment and procedure:
An experimental rig was constructed to study the enhancement of heat transfer for operational fluid inside a rectangular canal by use of a compound turbulator along with two-phase flow. The equipment used for the experimental testing and the measuring systems are shown in figure (1). These consisted of
1. A pump connected to a flow meter. The water pump (marquees) was used to pump water into the channel at a maximum discharge of 30 L/min, under a voltage of 220 volts, and a maximum head of 30 m. The flow meter’s flow rate ranged from 1.8 to 18 L/min.
2. A ROSY Z-0.036/8 Air Compressor, used to supply air (gas phase) with a working pressure of 0.8 MPa, a voltage of 220 V, power of 1 HP, and a frequency of 50 Hz
3. An air flow meter used to measure the average air inflow entering the channel, with a range of 5.833 to 58.33 L/min.
4. A test section as shown in figure (1) and (2), with a rectangular cross section of 5 × 3 cm and a length of 70 cm to examine the conduction of the two-phase flow (gas and liquid) over the heated plate (ribs and groove).
5. A power supply with a maximum voltage of 220 V.
6. Two heaters with 400 Watt capacity and voltages of 220 V.
7. Thermometer to verify the temperature readings at various positions within the examination canal, along with five thermocouples (K-type), one on the surface rib and the others spread throughout the examination canal.
8. A Sony digital camera recorder (DSC-W220 model), utilised to show the behaviours of the two-phase flow through the channel.

Various operational conditions were investigated in terms of the influence of these conditions on temperature profiles in the rectangular channel with the compound turbulator; these included changes to air flow rate and water flow rate and constant values of heat power. The chosen boundary conditions are displayed in table (1).

Figure 1. Schematic Diagram of the Two-Phase Flow
Figure 2. (a, b and c) display different compound turbulators ($g/p = 0.55$, $g/p = 0.73$, and $g/p = 0.37$, respectively)

Table 1. The operational conditions used in the experimental work.

| Heat power $q$ (watt) | Water discharge $Q_w$ (l/min) | Air discharge $Q_a$ (l/min) |
|-----------------------|-------------------------------|-----------------------------|
| 109.65                | 3                             | 8.33                        |
| ---                   | 6                             | 16.67                       |
| ---                   | 9                             | 25                          |
| ---                   | 12                            | ---                         |
The experimental procedures was as follows:
Various values of flow rate (four values for water, three values for air) and one value of electric power as shown in Table (1) were tested. For the first test, the following procedure was followed:

1- Installing the heat plate (first model) inside the channel.
2- Setting the centrifugal pump for water to the first flow rate of (3 L/min).
3- Connecting electric power to the heater.
4- Waiting for until the heated plate arrives at the required temperature (between 5 and 10 mins.).
5- Running the compressor on its primitive value (8.333 L/min).
6- Taking the temperature at the five sensors (four along the channel wall and one on the heated plate), while simultaneously recording video to monitor the nature of the fluid flow
7- Changing the discharge of the airflow while maintaining the discharge of the water and repeating the previous steps.
This process was repeated until all air discharges were tested.
8- Changing the water flow rate and repeating the previous steps.
This sequence was repeated for all water discharge rates, as displayed in table (1).
9- The previous steps were repeated for the remaining two models.

3. Numerical simulation
CFD software was applied for the numerical work on adiabatic air-water flow characteristics through a vertical canal including a heated plate. Construction of the numerical field and its examination were performed via GAMBIT and FLUENT (ANSYS 15.0) CFD codes, respectively. In this study, the geometry of the rectangular canal was divided into elements (quadrilateral structured grid) by using, Meshing with the greatest and smallest size both equivalent to (0.001 m). The number of elements were 21,520, 21,511, 21,525, and 21,793 for g/p = 0.37, g/p = 0.55, g/p = 0.73, and smooth channel, respectively. The model governing equations were solved for each element of the model's geometry. Figure (3) shows the mesh of the smooth channel and the channels with three types of compound turbulators.
3.1. Governing equations

A mixed model was used to solve the conservation equations of continuity, momentum, and energy for each phase, which is particularly useful where the phases move at different velocities. The governing equations can be written as in Fluent User’s Guide [21]

I. Continuity Equation

The common form of this equation is given by

\[
\frac{\partial}{\partial t} \left( \rho_m \right) + \nabla \cdot \left( \rho_m \bar{v}_m \right) = 0
\]  \hspace{1cm} (1)

The mass-averaged velocity \( \bar{v}_m \), is represented as

\[
\bar{v}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \bar{v}_k}{\rho_m}
\] \hspace{1cm} (2)

and \( \rho_m \) is the density of the mixture:

\[
\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k
\] \hspace{1cm} (3)
\( \alpha_k \) is the volume fraction of phase \( k \).

**II. Momentum Equation**

The common form of this equation is given by

\[
\frac{\partial}{\partial t} \rho_m \bar{v}_m + \nabla \cdot (\rho_m \bar{v}_m \bar{v}_m) = -\nabla p + \nabla \cdot \left[ \mu_m \left( \nabla \bar{v}_m + \nabla \bar{v}_m^T \right) \right] + \rho_m \bar{g} + \bar{F} + \nabla \cdot \left( \sum_{k=1}^{n} \alpha_k \rho_k \bar{v}_{dr,k} \bar{v}_{dr,k} \right)
\]

(4)

Where \( \mu_m \) is the viscosity of the mixture \( \bar{F} \) is a body force, \( n \) is the number of phases; \( \mu_m \) is given by

\[
\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k
\]

(5)

where \( \bar{v}_{dr,k} \) is the drift velocity for secondary phase \( k \)

\[
\bar{v}_{dr,k} = \bar{V} - \bar{V}_m
\]

(6)

**III. Energy Equation**

The common form of this equation is given by

\[
\frac{\partial}{\partial t} \sum_{k=1}^{n} (\alpha_k \rho_k E_k) + \nabla \cdot \sum_{k=1}^{n} \left( \alpha_k \rho_k (\bar{v}_k + p) \right) = \nabla \cdot \left( \kappa_{eff} \nabla T \right) + S_E
\]

(7)

The first term in the right side of equation (3.7) represents the energy transfer due to conduction. \( S_E \) includes all other volumetric heat sources, where \( \kappa_{eff} \) is the effective conductivity \( \left( \sum \alpha_k (k_k + k_t) \right) \) and \( k_t \) is the turbulent thermal conductivity.

**IV. Turbulence Model**

Based on the difference between the numerical and the experimental results, the turbulence \( k - \varepsilon \) of the standard mixture model was set for the two-phases model; this was based on the following equations [20]:

\[
\frac{\partial}{\partial t} (\rho_m k) + \nabla \cdot (\rho_m \bar{v}_m k) = \nabla \cdot \left( \frac{H_{lm}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \varepsilon
\]

(8)
\[
\frac{\partial}{\partial t}(\rho_m \vec{e}) + \nabla \cdot (\rho_m \vec{v}_m \vec{e}) = \nabla \cdot \left( \frac{\mu_{l,m}}{\sigma_e} \nabla \vec{e} \right) + \frac{\vec{e}}{k} (C_{1e} G_{k,m} - C_{2e} \rho_m \vec{e})
\]

(9)

where \( \vec{e} \) is the turbulent dissipation rate, \( \sigma \) is the turbulent Prandtl number for \( k \) and \( \vec{e} \), and \( G_k \) is the generation of turbulence kinetic energy.

The density and the velocity of the mixture are computed as follows:

\[
\rho_m = \sum_{i=1}^{n} \alpha_i \rho_i
\]

(10)

\[
\vec{v}_m = \frac{\sum_{i=1}^{n} \alpha_i \vec{v}_i \rho_i}{\sum_{i=1}^{n} \alpha_i \rho_i}
\]

(11)

The production of turbulence kinetic energy, \( G_{k,m} \), and turbulent viscosity, \( \mu_{l,m} \), are calculated as follows:

\[
\mu_{l,m} = \rho_m C_{mu} \frac{k^2}{\vec{e}}
\]

(12)

\[
G_{k,m} = \mu_{l,m} \left( \nabla \vec{v}_m + \left( \nabla \vec{v}_m^T \right) \right) : \nabla \vec{v}_m
\]

(13)

The model constants of the Ansys simulation can be seen in Table (4) (Fluent 15.0 program).

**IV. Heat Transfer Coefficient**

The rate of heat transfer is the amount of heat that is transferred per unit of time (usually per second). The local heat transfer coefficient between the surface of the heated plate (rib) and the two-phase flow mixture can be written as in Fluent User’s Guide [21]:

\[
h_s = \frac{q}{A_s (T_s - T_f)}
\]

(14)

3.2. **Problem assumptions**

With a view to reproducing the two-phase flow (water-air) in addition to modelling heat transfer, the following assumptions were selected:

1. Steady state flow
2. Turbulent flow
3. Pressure based solver
4. Two-dimensional Zone
5. Incompressible flow
6. The acceleration of gravity in the Y direction is (-9.81 m/s\(^2\))

| Zone                  | Boundary Type            |
|-----------------------|--------------------------|
| Channel inlet W       | Water superficial velocity|
| Channel inlet A       | Air superficial velocity  |
| Channel outlet        | Outlet pressure          |
| Channel side          | Heat flux                |
| Channel content       | Water – Air              |

| Variable              | Relaxation factors       |
|-----------------------|--------------------------|
|                       | Two phases (Water-Air)   |
| Pressure              | 0.3                      |
| Momentum              | 0.5                      |
| Volume fraction       | 0.5                      |
| Energy                | 0.8                      |

| Constant              | Value    |
|-----------------------|----------|
| \(\sigma_k\)          | 1.00     |
| \(\sigma_e\)          | 1.30     |
| \(C_{1\varepsilon}\)   | 1.44     |
| \(C_{2\varepsilon}\)   | 1.92     |
| \(C_{\mu}\)           | 0.09     |

| Property             | Water – Liquid (Ansly Fluent 15.0 Database) | Stainless steel – Solid (Ansly Fluent 15.0 Database) | Air – Gas (Ansly Fluent 15.0 Database) |
|----------------------|---------------------------------------------|-----------------------------------------------------|--------------------------------------|
| Density (kg/m\(^3\)) | 998.2                                       | 2719                                                | 1.225                                |
| Thermal conductivity (W/m.K) | 0.6                          | 202.4                                               | 0.0242                               |
| Viscosity (kg/m.s)    | 0.001003                                   | -                                                   | 1.7894e-05                           |
4. Results and discussion

The impacts of varying $U_w$ between (0.0987 m/s and 0.395 m/s), and $U_a$ between (1.4609 m/s and 4.384 m/s), at a constant heat power of (109.65 W), and the effects of the groove position on the temperature distribution and coefficient of heat transfer were investigated experimentally and numerically. The results of the three different positions of grooves in the heated plate were compared graphically, where every line represents a different position for the groove. A comparison was also made between turbulated and smooth channel. The results of the numerical work matched the experimental results well, with a deviation of about (5.77 %) between them. The diversity between the experimental photographs and numerical results arises because of two causes. The first is that the numerical model is a two-dimensional model, while in the real case it is a three-dimensional model, and thus the water and air flow rates input into the test channel are different from the superficial velocities. The second is that the photographs were taken generally from the experimental images for each situation and the distinctions might be less for other images for similar cases.

4.1. Experimental results

Figure (5) displays the impact of increasing the superficial velocity of water on the local coefficient of heat transfer for different positions of the groove in a compound turbulator for diverse values of air superficial velocity and constant heat power.

When the water superficial velocity increased, the heat transfer coefficient also increased due to the decrease in the temperature difference and increase in the amount of water passing above the heat plate inside the channel. Adding to the surface area of the heated plate (ribs and groove) also increases the surface area for heat transfer, interrupting the expansion of the boundary layer of flow, create strong turbulence inside the channel leading to faster mixing between the wall and core flow.

The temperature difference decreases as the water superficial velocity increases according to the following relationship:

\[ q = m^* \cdot cp \cdot \Delta T \]  \hspace{1cm} (15)

Where

\[ m^* = \rho v A \]  \hspace{1cm} (16)

From Equation (15), it can be observed that the temperature difference varies inversely compared with the water superficial velocity.

The heat transfer coefficient increases as the water superficial velocity increases and the temperature difference decreases as per Equations (15) and (14). These equations show that the coefficient of heat transfer is directly related to water superficial velocity and inversely related to the temperature difference.

Figure (6) display the impact of increasing $U_a$ on the local coefficient of experimental heat transfer with different positions of groove in the heated plate for diverse values of $U_w$ and constant heat power.

At the point when the air superficial velocity increases, the coefficient of heat transfer increases due to the reduction in the temperature difference and reduction in the time residence of mixture (water and air). Adding to the heated plate (ribs and groove) again increases the surface area for heat transfer, interrupting the expansion of the boundary layer of flow, creating strong turbulence intensity inside the channel and leading to fast mixing between the wall and core flow. The air superficial velocity ranged from 1.4609 to 4.384 m/s for channels fitted with different grooves, g/p = 0.37, 0.55, and 0.73, for dissimilar values of $U_w$. As the $U_w$ increased from (0.0987 m/s to 0.395 m/s) at a constant (109.65 W), the value of the experimental heat transfer coefficient increased from (798.45 W/m².K to 903.39 W/m².K) for g/p = 0.37, (828.52 W/m².K to 953.65 W/m².K) for relative groove position g/p = 0.55,
and from (812.966 W/m².K to 922.88 W/m².K) for g/p = 0.73. The maximum difference between an experimental and smooth channel was (56.5%).

4.2. Numerical Results
The behaviour of the numerical results was similar to that of the experimental results, with little difference between them. Figure (7) displays the impact of increased $U_a$ on the local coefficient of heat transfer with different relative groove positions in a compound turbulator for a constant value of heat power and different levels of water superficial velocity. Figure (8) displays the impact of increased $U_w$ on the local coefficient of heat transfer results with different relative groove positions in a compound turbulator for a constant value of heat power and different levels of $U_a$. The maximum difference between the numerical result and the experimental smooth channel was 55.2%.

All results displayed the same type of increases, from 0.0987 m/s to 0.395 m/s and 1.4609 m/s to 4.384 m/s, respectively. The coefficient of heat transfer also increased from 787.5 W/m².K to 892.19 W/m².K for relative groove position g/p = 0.37, from 812.36 W/m².K to 941.16 W/m².K for g/p = 0.55, and from 791.4 W/m².K to 907.94 W/m².K for g/p = 0.73. This was due to the decrease in the temperature difference and decrease in the time residence of the mixture (water and air) and the increase of the amount of water inside the channel. Adding to the heated plate (ribs and groove) increased the surface area for heat transfer, interrupting the expansion of the boundary layer of flow, creating strong turbulence inside the channel leading to faster mixing between the core and wall flows.

Figures (9) and (10) numerically demonstrate the contours of temperature allocation for the three positions of the groove in the channel. The activity of the ribs and groove inside the channel increases as the water superficial velocity increases, which produces an increase in the turbulence of the flow and its recirculation, causing a decrease in the temperature inside the channel that explains the observed increase in the rate of heat transfer.

4.3 Comparison between experimental and numerical results
From all experimental and numerical findings, the coefficient of heat transfer increase by increasing the superficial velocity was greatest at $U_w = 0.395$ m/s and $U_a = 4.384$ m/s; the best g/p was at 0.55. The cause of this effect was presence of the ribs and groove in the channels, where the turbulator provides additional surface area for heat transfer and creates strong turbulence intensity inside the channel by breaking the laminar-sub layer and promoting faster mixing between the wall and core flow. As a result of this effect, the coefficient of heat transfer increases because of both the increase in surface area and increasing turbulence.

The results of the numerical simulation were based on the water and air superficial velocities and heat power input into the channel, the channel geometry and position of the groove, the volume fraction, and the channel outlet gage pressure; this led to a difference from the experimental results, with a deviation of about 5.77 % at $U_w = 0.395$ m/s and $U_a = 4.384$ m/s between the experimental and the numerical results. This is due to minor variations in the diverse constraints assumed during the simulation; these values enable further changes to be made based on different assumptions, as shown in figure (11).

The heat transfer coefficient was affected by the same ratio when the water or air velocity changed, but at a lower percentage of about 2.28% for the latter in a channel fitted with relative groove position 0.55, as water is the primary phase and has a thermal conductivity higher than the thermal conductivity of air.

5. Program Validations
In order to demonstrate the validity and precision of the turbulence model, numerical testing of previous work was undertaken. (Kareem (2017)) [19], which investigated the heat transfer coefficient for two-phase flow (water- air) inside a rectangular vertical channel with rectangular ribs for three
values of air superficial velocity (1.425, 1.644, and 2.193) and three values of water superficial velocity (0.0421, 0.0842, and 0.1158) and a constant heat power (120 W) was calculated as shown in figure (4). The average percentage error was 4.2%.

Figure 4. Validation of local coefficient of heat transfer vs. air superficial velocity with varying water superficial velocities and constant heat power

6. Conclusions
1-The presence of ribs and grooves in the test section leads to an increase in the experimental heat transfer coefficient through a turbulated canal over that seen in a smooth channel, with a maximum enhancement of 48.2% for a rectangular turbulated channel with g/p = 0.37, 56.5% for g/p = 0.55, and 51.4% for g/p = 0.73. This was shown at a constant heat flux of 109.65 W.
2-The results of the numerical heat transfer coefficient show a maximum enhancement of 47.1% for g/p = 0.37, 54.7% for g/p = 0.55, and 49.7% for g/p = 0.73 compared with the smooth channel.
3-A compound turbulator with a relative groove position (g/p) = 0.55 is thus the best model to enhance heat transfer.
Figure 5. The impact of superficial velocity of air on the coefficient of heat transfer at varying water superficial velocities (0.0987, 0.1974, 0.296, and 0.395 m/s)
Figure 6. The impact of superficial velocity of water on the coefficient of heat transfer at varying air superficial velocities (1.4609, 2.923, and 4.384 m/s)
Figure 7. The impact of superficial velocity of air on the coefficient of heat transfer at varying water superficial velocities (0.0987, 0.1974, 0.296, and 0.395 m/s)
Figure 8. The impact of superficial velocity of water on the coefficient of heat transfer at varying air superficial velocities (1.4609, 2.923, and 4.384 m/s)
Figure 9. The impact of groove position on temperature distribution at 0.0987 m/s and 2.923 m/s water and air superficial velocities for constant heat power

Figure 10. The impact of groove position on temperature distribution at 0.395 m/s and 4.384 m/s superficial velocities of water and air for constant heat power
Figure 11. A comparison between experimental and numerical results at water superficial velocity 0.395 m/s

References

[1] Awad, M. M. 2007 Two-Phase Flow Modeling in Circular Pipes, Ph.D. Thesis Memorial University of Newfoundland, St. John's, NL, Canada.

[2] J.L. Xu, P. Cheng and T.S. Zhao 1999 Gas-liquid two-phase flow regimes in rectangular channels with mini/micro gaps International Journal of Multiphase Flow, pp 411-432.

[3] P. Vlasogiannis, G. Karagiannis, P. Argyropoulos, V. Bontozoglou 2002 Air–water two-phase flow and heat transfer in a plate heat exchanger International Journal of Multiphase Flow, pp 757–772.

[4] G. Hetsroni, D. Mewes, C. Enke, M. Gurevich, A. Mosyak, R. Rozenblit 2003 Heat transfer to two-phase flow in inclined tubes International Journal of Multiphase Flow pp173–194.

[5] Hitoshi Asano *, Nobuyuki Takenaka, Terushige Fujii 2004 Flow characteristics of gas–liquid two-phase flow in plate heat exchanger (Visualization and void fraction measurement by neutron radiography) Experimental Thermal and Fluid Science, pp 223–230.

[6] R. Rozenblit, M. Gurevich, Y. Lengel, G. Hetsroni 2006 Flow patterns and heat transfer in vertical upward air–water flow with surfactant International Journal of Multiphase Flow pp 889–901.

[7] Apurba Layek, J.S. Saini, S.C. Solanki 2007 Heat transfer and friction characteristics for artificially roughened ducts with compound turbulaters International Journal of Heat and Mass Transfer, pp 4845–4854.

[8] Lei Wang, Bengt Sunde’n 2006 Experimental investigation of local heat transfer in a square duct with various-shaped ribs Lund University, pp. 43:759–766.
[9] Kadir Bilen a,*, Murat Cetin b, Hasan Gul c, Tuba Balta 2009 The investigation of groove geometry effect on heat transfer for internally grooved tubes Applied Thermal Engineering, pp 753–761,

[10] Smith Eiamsa-ard , Pongjet Promvonge 2009 Thermal characteristics of turbulent rib-grooved channel flows International Communications in Heat and Mass Transfer pp 705–711.

[11] Wen peng Hong, Yan Liu 2010 Investigation of Gas-liquid Two-phase Flow Patterns Based on High-speed Imaging Methods 3rd International Congress on Image and Signal Processing.,

[12] Igor A. Bolotnov , Kenneth E. Jansen, Donald A. Drew, Assad A. Oberai, Richard T. Lahey Jr.,Michael Z. Podowski 2011 Detached direct numerical simulations of turbulent two-phase bubbly channel flow International Journal of Multiphase Flow pp 647–659.

[13] Aritra Sur, Dong Liu 2012 Adiabatic airwater two-phase flow in circular microchannels International Journal of Thermal Sciences pp 18-34 ,

[14] M.R. Ansari , B. Arzandi 2012 Two-phase gas–liquid flow regimes for smooth and ribbed rectangular ducts International Journal of Multiphase Flow pp 118–125.

[15] Laith Jaafer Habeeb1, Riyadh S. Al-Turaihi 2013 Experimental Study and CFD Simulation of Two-Phase Flow around Multi-Shape Obstacles in Enlarging Channel American Journal of Mechanical Engineering Vol. 1, No. 8, 470-486.

[16] Arkan Al-taee, Hasasn Ali Jurmut 2014 Experimental and Numerical Investigation of Convective Heat Transfer in a Circular Pipe with Internal Ring ribs Journal of Kerbala University Vol. 12 No.3.

[17] Jalghaf, H. K. , Al-Turaihi, R. S., and Habeeb, L. J. 2016 Air-Water Flow Investigation around Hot Circular Cylinder inside Channel Advances in Natural and Applied Sciences, Vol. 10, No. 12, PP. 16-27.

[18] Ran Kong, Seungjin Kim 2016 Characterization of horizontal air–water two-phase flow Nuclear Engineering and Design.

[19] Riyadh S. Al-Turaihi and Doaa Fadhil kareem 2016 The Effect of the Two Phase Flow (Water–Air) Over Triangle Ribs on Heat Transfer Coefficient in Vertical Rectangular Duct European Journal of Scientific Research Mech. Eng. Department, University of Babylon Hilla, Iraq, Vol. 140 pp 248-262.

[20] Fluent User’s Guide September 29 2006 Modeling Multiphase Flows Fluent Inc.

[21] Riyadh S. Al-Turaihi and Doaa Fadhil kareem 2017 Enhancement the Heat Transfer Coefficient for two phase flow through a Rectangular ribbed Vertical channel M.S.C thesis Mech. Eng. Department, University of Babylon, Hilla, Iraq.