Two-Dimensional Numerical Investigation on Mixed Convection Nanofluids Flow in a Duct having Backward-Facing Step under Laminar Condition

Saad Abdulwahab Razuqi

Asst.Lecturer, Mechanical Engineering Department, Al-Mustansiriayah University /Baghdad, Iraq.

Received 12 June 2017; Accepted 15 September 2017; Available online 20 September 2017

Address For Correspondence:
Saad Abdulwahab Razuqi, Asst.Lecturer, Mechanical Engineering Department, Al-Mustansiriayah University /Baghdad, Iraq
E-mail: Saad_abd59@yahoo.com

Copyright © 2017 by authors and American-Eurasian Network for Scientific Information (AENSI Publication).
This work is licensed under the Creative Commons Attribution International License (CC BY).
http://creativecommons.org/licenses/by/4.0/

ABSTRACT
A numerical study of heat transfer and laminar nanofluids flow over backward-facing steps in a two-dimensional horizontal rectangular duct is numerically simulated. In this work, the nanoparticles Al$_2$O$_3$ and SiO$_2$ are studied with different volume concentrations at a range from 0% to 4% were considered. The numerical results were carried out for constant step height of 0.03 m. The bottom wall is subjected to constant heat flux ($Q = 60 \text{ W/m}^2$) and the other three walls at adiabatic conditions. The simulation is conducted in the Reynolds number range of ($150 \leq Re \leq 550$). The governing partial differential equations in two dimensions was solved by numerical analysis was based on finite volume numerical techniques with SIMPLE algorithm, using ANSYS FLUENT commercial CFD software, to study the effect of Reynolds number also the concentration of nanofluids on the heat transfer enhancement and flow characteristics. The results reveal that the nanofluid with SiO$_2$ has the best enhancement. The increasing in volume concentration and Reynolds number lead increasing in Nusselt number value.

KEYWORDS: Twisted tape; Heat transfer; Nusselt number

INTRODUCTION
The presence of flow subsequent reattachment and separation because of a sudden contraction or expansion passages of the flow, such as forward-facing steps and backward-facing, playing the main role in the design of most engineering applications where cooling or heating is taken into consideration. These applications appear in high performance heat exchangers, cooling systems for electronic equipment, environmental control systems, combustion chambers and so on [1].

In fact, the study of heat transfer near to backward and forward facing steps using nanofluids has not been seen richly in the published literature, with the importance that the geometries appear usually in many industrial heat transfer devices, as in the heat exchangers. The heat flux in thermal devices has become more and vaster due to the increasing main requirements for the respective systems. There are rather limited studies that have dealt with how to convert the flow structure and temperature distribution, and then to attain the high heat transfer performance.

To understand the effects of flow separation that results from abrupt changes of the geometry in an open flow setting the flow over a backward-facing step presents the important prototype. The geometry is well known in engineering applications and it is used as an original model separated flow in basic studies of flow control, and of turbulence in separated flows, which may further be with the assessment of turbulence models. Interest in
studying the reattachment phenomena over the backward-facing step continued in the 1970’s and in the 1980’s with the development of more sophisticated visualization techniques, the engineering community produced a large quantity of publications referring to experimental and numerical results obtained for forced convection flow regime by Armaly et al. [2]. Denham and Patrick [3] discussed the measurements of two-dimensional laminar flow over a down-stream facing step. They observed that there are a small fluctuations occurred at Re = 229, these fluctuations are observed to be the onset of transition to the separated boundary layers. Also, they found that there is a flow tendency toward secondary separation at the upper wall produced because of the inflection of the velocity profiles at highest Reynolds number. The studies on flow over backward-facing step were not only focused on the fluid flow problem but also on the heat transfer phenomena. Sparrow and Chuck [4] performed the first numerical work dealing with heat transfer and fluid flow over a backward-facing step. They implemented a numerical finite difference for studying the airflow phenomenon over a two-dimensional channel heated at constant temperature for the bottom wall from the foot of the step to the end of the channel. They observed that the local Nusselt number distribution begins with a minimum value at the step, and then it increases with a maximum value near the reattachment point position. Beyond the maximum, the local Nusselt number monotonically decreases towards the fully develop value. This behavior reflects the separation, reattachment, and redevelopment experienced by the flow. The flow with the three-dimensionality over backward-facing step has been numerically and experimentally explored widely in the last decade by Shih and Ho [5] and Tylli et al. [6]. An experimental and numerical studies presented by Tylli et al. [6] to investigate the sidewall effects for 3D water flow over a backward facing step. The numerical simulation was used just for laminar flow at Reynolds number up to 700. They found that good agreement between the three dimensional simulation and experiments are obtained when Re = 650. Also, they saw that at low Reynolds numbers, the sidewall does not affect the structure of laminar flow in the channel. They observed that with the higher values of Reynolds numbers, the flow does not infiltrate up to the mid plane of channel. In addition, the intensity of the wall-jet and three dimensional flow increased with Reynolds number at laminar flow, also it decreased at transitional flow. The is a clear variation noticed in the three dimensional turbulent flow field. Armaly et al. [7] measured the three dimensional laminar velocity for separated airflow adjacent to a backward facing step. They found that the Xu-lines show strong variations in the span-wide direction. The recirculation region size increased with Reynolds number increment. They appointed that for Re < 98.5 there is no flow region recirculation adjacent to the side-wall. However, at a range of Reynolds number (Re < 190), a mild region for recirculation flow was found at the upper corner of the side-wall. They deduced that the transverse velocities at the peak value move to the duct center as the distance from the step increased. Chen et al. [8] conducted numerical simulation of forced convection turbulent flow at adjacent to a two-dimensional backward-facing step. The step height effects on heat transfer and turbulent separated flow were obtained. They also noticed that the primary recirculation region size near to the backward facing step increases both in length and height with the increase of step height. The higher turbulent kinetic energy developed along the separating shear layer and near the reattachment region. The magnitude of friction factor became smaller with the step height increasing. Moreover, they found that when the step height increases cause to increase the maximum turbulent kinetic energy and temperature magnitude. Iwai et al. [9] presented a numerical analysis for 3D mixed convection flow over a backward facing step at low range of (Re) to study the duct aspect ratio effect. They observed that the ratio of (16) is wanted for 2D region of the duct at Re = 250. Furthermore, they found that when the aspect ratio and Reynolds number increased the maximum Nusselt number increased. There is an analogous between the coefficient of skin friction and the case of Nusselt number distribution. Nie and Armaly [10] conducted a numerical investigation for three dimensional laminar flow adjacent to a backward facing step in a rectangular duct. The effects of the step height on the flow and heat transfer characteristics represented the main objectives of their work. The flow condition, thermophysical properties, and the geometry were done on the available measurements investigated by Armaly et al. [11] for validation. They found that the increasing of step height leads to increase the Nusselt number and the size of the sidewall reversed flow region. Moreover, the increasing in the step height made the displacement between the point and step were the impingement flow reach the downstream wall increased, and the secondary recirculation region of the flow developed adjacent to the step bottom corner. They also found that for small step height of the case, the minimum value of (Nu) was existed near the bottom corner between the duct centerline and the steps. When the step height increased, the minimum value of Nusselt number was appeared to move in the direction towards the bottom corner of the sidewall and the step. They also concluded that the friction factor increased along the centerline of the duct inside the primary region of the recirculation flow, but it decreased outside the region of recirculation with the increasing step height. Saldana et al. [12] presented numerical simulation for forced convective air flow over 3D backward facing step. The distribution of local (Nu) along the bottom wall was found to lie in the vicinity of the points of intersection between Xu-line and Xw-line where the shear stress was zero. The velocity profiles revealed that for (Re ≥ 343) the flow at the exit did not reach fully developed conditions.

Conventional fluids such as air and water are used in many cooling system. The increase in heat loads needs to increase the amount of working-fluid which is not desirable in limited space systems. The researchers made
many trails since last century to increase the use of fluids with higher thermal conductivity than the traditional fluids. In addition, solid particles were dispersed in base fluids to enhance the thermal conductivity of working-fluids. Finally, the concept of dispersing nanoparticles in base fluid solved the agglomeration and stability of the solid particles, and the thermal conductivity increases more than 40% compared to traditional fluids [13]. The importance of nanofluids studying have been more interested since Choi’s concept presented, many papers were investigated in different applications to realize the mechanisms and other characteristics of nanofluids. Some researchers made review papers such as Wong and Castillo [14], Das et al. [15], and Yu et al. [16]. They stated that a new generation of cooling technology of nanofluids will be appeared, and more researches are needed to avoid cluster phenomena in nanofluids and improve the stability.

A comprehensive review indicated that a large quantity of publications have been dedicated to study the fluid flow and heat transfer phenomena over a backward and forward facing steps. However, this problem is not clearly understood because of its complexity and the strong three-dimensional flow behavior. The influences of several factors of the geometry such as the types of fluids, boundary conditions and step heights were studied. It is also important to remark here that the horizontal backward and forward facing steps analysis of mixed convection have received limited attention. Thus, this investigation is conducted in the way to contribute accurate numerical data for convective laminar flow over the 2-D horizontal backward-facing step. The numerical procedure and methodology used in this study will be presented in the following.

2. Physical Model and Assumptions:

2-D laminar mixed convection (combination of forced and free convections) flows over backward facing steps in a rectangular duct heated from the bottom is numerically simulated. The upstream duct height ($D_2$) is 0.03 m; the downstream duct height ($D_1$) is 0.06 m. This geometry gives a backward-facing steps height ($H$) of 0.03 m, an expansion ratio of $ER = D_1/(D_1-H) = 2$. The computational flow domain length is 0.05 m and 0.3 m downstream and upstream of the sudden expansion respectively, i.e., $-3 \leq x/H \leq 50$. The computational domain is schematically shown in Figure (1). The following assumptions are used in the model:

1. Steady two dimensional laminar flow in the inner side (working fluid) with single phase fluid (Homogenous model).
2. Incompressible and Newtonian fluid.
3. Constant heat flux and neglect viscous dissipation.
4. The kinetic and potential energies changes are negligible.
5. Radiation heat transfer is not considered.
6. Buoyancy effect is assumed to be taken into account.
7. No heat generation included in this study.
8. The nanofluids are assumed with constant physical properties and incompressible.
9. The nanofluid thermophysical properties are assumed to be dependent on the nanoparticles concentration.
10. At zero relative velocity between the nanoparticles and the fluid phase, the base fluid (distilled water) is assumed to be in thermal equilibrium
11. The size of nanoparticles are very small so there is no slip occurs between them and an ultrafine (< 100 nm) solid particle has been considered.

The no-slip conditions have been considered for all the solid walls, and all wall in adiabatic assumption except the heating section of the bottom wall. The flow has a fully developed velocity profile and uniform temperature at the duct inlet far upstream of the steps.

![Fig. 1: Schematic diagram of the computational domain.](image-url)
3. Thermophysical Properties of Nanofluid:

The thermophysical properties of nanofluids like density and specific heat are calculated using different equations presented in the literature as shown below [16]:

$$\rho_{nf} = \left( \frac{m}{V} \right)_{nf} = \rho_{bf}\frac{V_{nf} + \rho_{bf}V_{p}}{V_{bf} + V_{p}} = (1 - \varphi)\rho_{bf} + \varphi\rho_{p}$$  \hspace{1cm} (1)

$$C_{p_{nf}} = \frac{(1 - \varphi)C_{p_{bf}}\rho_{nf} + \varphi\rho_{p}C_{p_{p}}}{\rho_{nf}}$$  \hspace{1cm} (2)

$$\mu_{nf} = (1 + 7.3\varphi + 123\varphi^2)\mu_{bf}$$  \hspace{1cm} (3)

$$k_{nf} = k_{bf}(1 + 7.74\varphi)$$  \hspace{1cm} (4)

Where:

- $\varphi$: the volume concentration
- $\mu$: the dynamic viscosity.
- $nf$: nanofluid
- $bf$: base fluid.

The thermophysical properties of nanofluids and materials used for simulation are calculated according to the equations (1-4), as shown in tables (1-3).

**Table 1: Thermophysical properties of pure water and different materials at $T=300$ K**

<table>
<thead>
<tr>
<th>Thermophysical properties</th>
<th>Water</th>
<th>Al2O3</th>
<th>SiO2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$ (kg/m$^3$)</td>
<td>998.203</td>
<td>3970</td>
<td>2200</td>
</tr>
<tr>
<td>$\mu$ (Ns/m$^2$)</td>
<td>2.01×10$^{-3}$</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$k$ (W/m.K)</td>
<td>0.613</td>
<td>40</td>
<td>1.2</td>
</tr>
<tr>
<td>$C_p$ (J/kg.K)</td>
<td>4182.2</td>
<td>765</td>
<td>703</td>
</tr>
</tbody>
</table>

**Table 2: Effective thermophysical properties of nanofluid with Al$_2$O$_3$**

<table>
<thead>
<tr>
<th>Properties</th>
<th>$\Phi = 0.04$ (d$_p$=25nm)</th>
<th>$\Phi = 0.01$ (d$_p$=25nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mu_{eff}$ (Ns/m$^2$)</td>
<td>0.00157361</td>
<td>0.00110490</td>
</tr>
<tr>
<td>$K_{eff}$ (W/m.K)</td>
<td>0.7275921</td>
<td>0.67558618</td>
</tr>
<tr>
<td>$\rho_{eff}$ (kg/m$^3$)</td>
<td>1117.0749</td>
<td>1027.921</td>
</tr>
<tr>
<td>$c_{p,eff}$ (J/kg.K)</td>
<td>3696.4211</td>
<td>4050.2221</td>
</tr>
<tr>
<td>$\beta_{eff}$ (1/K)</td>
<td>0.0001775</td>
<td>0.0001983</td>
</tr>
</tbody>
</table>

**Table 3: Effective thermophysical properties of nanofluid with SiO$_2$**

<table>
<thead>
<tr>
<th>Properties</th>
<th>$\Phi = 0.04$ (d$_p$=25nm)</th>
<th>$\Phi = 0.01$ (d$_p$=25nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mu_{eff}$ (Ns/m$^2$)</td>
<td>0.00157361</td>
<td>0.00110490</td>
</tr>
<tr>
<td>$K_{eff}$ (W/m.K)</td>
<td>0.63852489</td>
<td>0.63139694</td>
</tr>
<tr>
<td>$\rho_{eff}$ (kg/m$^3$)</td>
<td>1046.27488</td>
<td>1010.221</td>
</tr>
<tr>
<td>$c_{p,eff}$ (J/kg.K)</td>
<td>3889.572</td>
<td>4106.432</td>
</tr>
<tr>
<td>$\beta_{eff}$ (1/K)</td>
<td>0.00018914</td>
<td>0.000202</td>
</tr>
</tbody>
</table>
4. **Geometry Meshing:**

A two dimensional geometry is created by using ANSYS Workbench. The simulations are done by employing a non-uniform grid system with quadrilateral elements. The grid of 500x120 is chosen as the best for accuracy and convergence time, as shown in figure (2).

![Fig. 2: Backward - Facing Steps finite set of control volumes (mesh)](image)

4.1 **Governing Equations:**

The set of equations continuity, the three equations of Navier–Stokes and energy are solved numerically using finite volume scheme to achievement the simulation of the thermal and flow fields [17].

4.2 **Continuity equation:**

\[
\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0
\]  

(5)

4.3 **Momentum equations:**

\[
\frac{\partial}{\partial x}(\rho u u) + \frac{\partial}{\partial y}(\rho u v) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)
\]  

(6)

\[
\frac{\partial}{\partial x}(\rho u v) + \frac{\partial}{\partial y}(\rho v v) = -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)
\]  

(7)

4.4 **Energy equation:**

\[
\frac{\partial}{\partial x}(\rho C_p u T) + \frac{\partial}{\partial y}(\rho C_p v T) = K \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)
\]  

(8)

Where (p) is the pressure, (T) is the temperature, and (u, v) are the velocity components in coordinate directions of (x) and (y) respectively.

5. **Numerical Method:**

To achieve the present work, the four differential equations (5) to (8) were solved numerically by ANSYS 15.0 package using finite volume approach based on SIMPLE algorithm. Also, the second order upwind scheme was employed to solve the momentum and energy equations. ANSYS Workbench was used to carry out the computational domains and the mesh generation. The under relaxation factors are also considered to be 0.3 and 0.7 for pressure and momentum, respectively. Also, for all simulations the convergence criteria was of $10^{-3}$ for continuity and velocity components and $10^{-6}$ for energy. In order to have best numerical simulation which covers the problem features, good quality mesh generation is necessary. In addition, fine meshes were needed for near wall regions. Converged solutions for all cases were accomplished by considering the iterative process resulting residuals for all governing equations did not change with the iteration progress also the computational error can be ignored, then the iteration manually stopped. For most of the cases, the iterations are around (620-710).

**RESULTS AND DISCUSSIONS**

6.1 **Thermal and Hydrodynamic Effect of Nanofluid:**

Distributions of (Nu) for different types of nanofluids with $\phi = 4\%$ for $Re = 450$ and $Q = 60 \text{ W/m}^2$ is shown in figure (3). Because of the constant heat flux boundary condition, the maximum value of (Nu) is proportional inversely to the minimum wall temperature. The results of maximum Nusselt number showed a developing near the side-wall, and it moves further downstream. From this figure we can see clearly that the nanofluids have larger Nusselt number than distilled water due to the high thermal conductivity of nanofluids. Figure (4) demonstrate the variation of Nusselt number versus Reynolds number at different values. The heat transfer rate
tends to increase with increasing in Reynolds number value due to the decreasing in thermal boundary layer thickness.

The friction coefficient distributions for different nanofluid concentrations is shown in figure (5). The maximum temperature degree records near the stepped wall and backward-facing step corner, where the heat exchanging rate is weak, therefore the friction factor at this region decreases and then increases along the test section. From figure (5), we can see clearly that the friction factor increases as nanofluid concentration increase, but the coefficient of friction factor decreases as Reynolds number increases as shown in figure (6) below. This is because the skin friction coefficient is proportional to the velocity inversely.

6.2 Velocity Distribution:

The distributions of velocity at different nanoparticles concentrations for Re = 350, and Q = 60 W/m² and different values of x/H sections alongside the down-stream wall are shown in figure (7). The results show that with increasing of volume concentration leads to augment the thermophysical of nanoparticles. In this case, the velocity increases as nanoparticles concentration increases because of the density decrement.

The velocity distributions of different types of nanofluids (φ = 4%) for Re = 350 and Q = 60 W/m² along the down-stream wall is shown in figures (8), (9). We notice clearly from the figures that the low density nanofluids such as SiO₂ have higher velocity distribution than Al₂O₃ at constant (Re) for the case at x/H = 1 and x/H = 30. Behind the step wall the opposite flow is performed because of the region of recirculation that attached to the step and the vortex and because of the buoyancy force, the flow changes its direction downstream the channel.

6.3 Velocity vector and stream lines:

The velocity vector change during the flow over a backward-step is shown in figure (10). The back flow occurs at the beginning and the parabolic shape appears another time at the duct center. The different locations of stream lines contour along the duct are shown clearly in figure (11).

CONCLUSIONS:

THE NUMERICAL SIMULATION OF LAMINAR MIXED CONVECTION HEAT TRANSFER WITH NANOFLUIDS FLOW OVER BACKWARD FACING STEP PLACED IN A HORIZONTAL DUCT WAS CARRIED OUT. THE FOLLOWING CONCLUSIONS CAN BE REMARKED AS FOLLOWS:

1- THE FIGURES REVEALED THAT SiO₂ NANOFLUID GIVES THE LARGEST VALUE OF NUSSELT NUMBER AND VELOCITY DISTRIBUTION AND THEN Al₂O₃, WHILE THE PURE WATER IS THE SMALLEST.

2- The Nusselt number value increased with volume concentration of nanofluids increasing.

3- The velocity distribution and Nusselt number increased with increasing in Reynolds number values and vice versa for the skin friction.

Nomenclature:

\( C_p \): Specific heat (kJ kg\(^{-1}\)\( ^\circ\)C\(^{-1}\))

\( D_1, D_2 \): Tube diameters (m)

\( k \): Thermal conductivity (Wm\(^{-1}\)\( ^\circ\)C\(^{-1}\))

\( Q \): Heat flux (W/m\(^2\))

\( Re \): Reynolds number (dimensionless)

\( T \): Temperature (\(^\circ\)C)

\( V \): Main velocity (m s\(^{-1}\))

Greek Symbols:

\( \Delta P \): Pressure drop, Pa

\( \rho \): Density (kgm\(^{-3}\))

\( \phi \): Nanoparticle volume concentration (dimensionless).

Subscripts:

\( nf \): Nanofluid

\( f \): Friction factor

\( bf \): Base fluid

\( p \): Particles
Fig. 3: The distribution of local Nusselt number along the duct center with $\varphi = 4\%$ for SiO$_2$ at Re = 450

Fig. 4: The distribution of local Nusselt number along the duct centerline with $\varphi = 4\%$ for SiO$_2$ at various Reynolds number

Fig. 5: The skin coefficient of friction distribution along the duct center with for Re = 450 and Q = 60W/m$^2$ at different concentration (SiO$_2$)
Fig. 6: Skin friction coefficient of $\text{SiO}_2$ for $\phi = 4\%$ and $Q = 60\text{W/m}^2$

Fig. 7: Velocity distribution of different nanoparticles volume fractions for $\text{Re}=350$ and $Q=60\text{W/m}^2$ at $x/H = 30$

Fig. 8: Velocity distribution of different nanofluids with $\phi = 4\%$ at $x/H = 30$ and $\text{Re} = 350$. 
Fig. 9: Velocity distribution of different nanofluids with $\varphi = 4\%$ at $x/H = 1$ and $Re = 350$.

Fig. 10: Velocity vectors of different location during nanofluid flow in the duct.

Fig. 11: Stream lines contour of different location during nanofluid flow in the duct.
REFERENCES