Numerical and experimental analysis of effect of working fluid amount on the thermal performance of thermo-syphon system

Ayoob Khalid Jasim\textsuperscript{a} Basim Freegah\textsuperscript{b} Mohammed Hamed Alhamdo\textsuperscript{b}

\textsuperscript{a}M.Sc. Student Mechanical Engineering Department, Mustansiriya University, Baghdad, Iraq
\textsuperscript{b} Mechanical Engineering Department, Mustansiriya University, Baghdad, Iraq
ayoobkhalid411@gmail.com

Abstract
The thermal performance enhancement of a thermo-syphon system has been studied by many researchers. However, according to the best knowledge of the authors, the impact of the amount of the working fluid on the performance of thermo-syphon has not hitherto been studied. In the present work the impact of inserting a closed tube inside the riser pipe and effect of increase and decrease the amount of the working fluid on the thermal performance have been investigated. Several cases have been considered and tested. To ensure the validity of the theoretical results, an experimental work has been conducted and results compared with numerical results. The results show good match between numerical and experimental results. In particular, the maximum difference between experimental and numerical results is found 14.52\% and 11.23\% for water temperature inside tank and working fluid at outlet of the riser pipe respectively. As well as, the numerical results have been demonstrated that the amount of the working fluid within the thermos-syphon loop has a noticeable impact on the thermal performance of the regarding system. Furthermore, it is found that the (case-A-) is the best case among all cases under consideration regarding system the thermal performance of thermos-syphon. Moreover, a correlation equation to predict water temperature within the storage tank has been established. The accuracy of this equation around 95.6\%.

Keywords: Natural Convection, Thermo-syphon, Thermal Conversion Computational Fluid Dynamics (CFD), Amount of Working Fluid.

1. Introduction
Energy can be classifying according to the sources used in its production into traditional and renewable energy. Conventional energy needs fossil fuels to produce and cause pollution when
used. These reasons motivate many researchers to harness and use renewable energy. There are several sources of renewable energy, however solar energy is the most significant of these sources. One of the devices that used to heat water utilizing solar energy is a thermo-syphon. To improve the thermal performance of such devices, the impact of various parameters was studied by previous researchers. Some of these researchers have focused to study the effect of the riser pipe configuration K. S. Shashisheker [1] used triangular absorber pipe rather than circular tube to increase the heat transfer capabilities of the solar collector. The results showed that the thermal efficiency of the triangular pipe is higher than that of the circular tube. Mangesh Thakare and M.V.Khot [2] conducted experimentally investigation for several configurations of the riser pipe namely: circular, triangular, square and elliptical on the efficiency of solar collector. They were find that the triangular tube give maximum efficiency as compared to other tubes consideration under study. Pankaj N. Shrirao at el. [3] studied two configurations of the riser pipe namely: a circular and airfoil shape on the thermal efficiency of solar collector. They have been recorded that there are improved by 10 to 12% when using airfoil-shaped absorption tube as compared to the circular section-absorbent tube. Freegah and Al-Tabbakh [4] performed an experimental and numerical study to analyses the effect of riser pipe number and the ratio between length and diameter of riser on the thermal performance of thermo-syphon. They reported that the performance of the system improvement with increase both parameters consideration under study. However, length to diameter ratio of the riser pipe have more effect as compared to number of risers. Marroquín-De Jesus Angel at el. [5] carry out a numerical study to investigate the heat transfer of solar collector for rectangular and cylindrical cross section of the riser pipe. They reported there is no significant difference between the two models in terms of thermal performance.

Some researchers, were used another way to improve the thermal efficiency of the thermo-syphon system. This method boils down to inserting devices inside the riser pipes to make the flow into more turbulent and thus increase the heat transfer process. The effect of used twist tape insert with variable initial temperature on the heat transfer process of solar collector has been studied numerically by Keguang Yao at el [6]. The results shown that the twist tape insert led to more vortexes near the surface and this makes better heat transfer. R. Herrero Martin at el. [7] have studied the effect of inserting wire coils inside the absorption tubes to improve heat transfer. They reported that the thermal performance increase when use wire coils insert as
compared to standard collector. Rohit Khargotra at. el. [8] carried out experimental study to analysis the effect of use coil-spring and twist tape inserts inside the absorber tube on the thermal performance of flat plate collector. The results shown that the twisted tape has highest efficiency as compared to the other type consideration under study. Braa Khalid Ameen at. el. [9] conducted a comparative study of variable kinds of insertion namely: single, double and mixed twisted tape inside the riser pipe. The double twist tape type has been reported to have more effect on solar collector thermal performance compared to other types under study. The effect of insert devices namely wires mesh, twisted-tape and wire coil on the thermal performance of solar collector has been study experimentally by Sandhu et al. [10]. They reported that as compared to the traditional model, all insert devices consideration under study have best thermal performance.

On the other hand, and in order to make evaluate the thermal performance of the thermos-syphon solar water system easier, equations to predicate the thermal performance of such system have been developed by a few researchers. A numerical study the effect of placing a metallic mesh inside a riser pipe has been conducted by Iordanou et al. [11]. They have established two equations to predict Nusselt number, these equations are as function of Rayleigh number only. Razavi et al. [12] conducted an experimental study for investigation of using polypropylene pipes in solar collector system on the heat transfer rate. They have developed an equation to calculate rate of heat transfer of such system, this equation is as function of Reynolds number and Prandtl number. Basim and Aouf [4] developed an equation to predict the amount of the working fluid which flow inside the thermo-syphon loop.

According to best knowledge that preview above, it is observed that that the effect of the amount of the working fluid on the thermo-syphon system has not hitherto been studied. Therefore, in the present study, the impact of the amount of the working fluid on the thermal performance of thermo-syphon system has been studied. As well as, due to the temperature of water inside the storage tank is one of significant parameters that give clearly indicate on the thermal performance of thermo-syphon. So, an equation to predict the thermal performance of a thermos-syphon through calculate temperature of water within the storage tank has been developed.

2. Experimental work
Setup of the present experimental has been tested under winter condition of Baghdad city. Figures (1-2) demonstrate the components of the rig used in the current study. The rig consists of the riser pipe to collect the heat, the down-comer that return the working fluid from the heat exchanger to the riser pipe, storage tank, the heat exchanger which placed within the storage tank, thermocouples to measure the working fluid’s temperature at the inlet and outlet of the riser pipe, and water temperature at the storage tank, data logger, solar power meter (TES1333R) and laptop computer to recorder this data. All specifications of rig system consideration under the current study are show in table (1).

Figure (1) Schematic view of the thermo-syphon closed loop under study.
Table 1. Specifications of the system under present study (dimensions of the system are measured in units of meters).

| Item                                      | Dimension or material |
|-------------------------------------------|-----------------------|
| Height of thermo-syphon system            | 1.2                   |
| Width of thermo-syphon system             | 0.6                   |
| The collector incline angle               | 33º                   |
| Number of riser pipes                     | 1                     |
| The riser pipe length                     | 1                     |
| Inner diameter of the riser pipe and down-comer | 0.01446              |
| Outer diameter of the riser pipe and down-comer | 0.01588               |
| Material of the riser pipe                | Copper                |
| The tank diameter                         | 0.2                   |
| The tank high                             | 0.3                   |
| The tank material                         | Plastic               |
| Insulation thickness of rubber plastic sponge insulation | 0.01 |

3. **Numerical computation**

To analyze the natural convection inside the thermo-syphon system and the storage tank, CFD model has been developed. In general, CFD analysis include three main steps namely: (a) pre-processing, (b) solver execution, and (c) post-processing. The first step includes building the
required model and the mesh generation of the domain. The second step involves provide the model with boundary conditions. While, the last step is intended to display results.

3.1. Geometry

In order to increase the heat transfer amount and thus enhance the thermal performance of the thermo-syphon, there are several ways to achieve that. One of the most important these ways is increase the surface area of the riser pipe. In the present study, the riser pipe surface area was increased by insert a closed tube within the riser pipes. The amount of this increase is around 13.8% as compared to traditional model. Furthermore, three riser pipe configurations were developed regarding amount the working fluid within the thermo-syphon: (Case -A-) have same amount of the working fluid as compared to amount of working fluid of traditional model, (Case -B-) have less amount of the working fluid by 10%, and (Case -C-) have more amount of the working fluid by 10% as shown in figure (3). The cases above have been a chested by increase and decrease the volume of the inserting tube. As well as, the figure (3) shows all specifications for models under consideration.
3.2. Time steps independence

In order to accurate the results and save the numerical simulation time, time step independence has been used. In the current study, three different time simulation steps have been selected as shown in table (2). As shown in this table, time step of 12 second was chosen for analysis purpose of all cases under current study.

| No | Time step (second) | Difference water temperature (°C) | Percentage difference (%) |
|----|--------------------|-----------------------------------|---------------------------|
| 1  | 3                  | 34.89                             |                           |
| 2  | 6                  | 35.01                             | 0.34                      |
| 3  | 12                 | 35.05                             | 0.11                      |

3.3. Mesh generation and independence

In the current study an unstructured hexagonal meshing is chosen to produce mesh for the domain of the thermo-syphon system as shown in figure (4). The hexagonal mesh was considering one of the most common techniques compared to other types because it has several
benefits, such as ability to link complex engineering shapes, fewer memory requirements, offers more exact results due to lower diffusion [13].

![Thermos-syphon mesh domain](image)

Figure (4) Thermos-syphon mesh domain

Grid independency tests are conducted in order to verify the selected of the suitable number of mesh utilized in the numerical simulation. The results of these tests are depict in table (3).

| Case     | Element number, (million) | Water temperature (°C) | Percentage difference (%) |
|----------|---------------------------|------------------------|---------------------------|
| Traditional | 1.9                       | 12.05                  |                           |
|           | 3.9                       | 12.028                 | 0.1825                    |
| Case -A-  | 2.3                       | 14.21                  |                           |
|           | 4.7                       | 14.23                  | 0.1405                    |
| Case -B-  | 3.1                       | 12.86                  |                           |
|           | 6.3                       | 12.84                  | 0.1555                    |
| Case -C-  | 3.3                       | 13.4                   |                           |
|           | 6.8                       | 13.43                  | 0.2233                    |

3.4. Boundary Conditions

The boundary conditions that used in numerical simulation for all cases under consideration are the initial temperature of water and working fluid is 23°C, the volume of water inside storage tank is (9.4 L) and volume of the working fluid within traditional thermo-syphon is (0.59 L).
Furthermore, two difference volume of the working fluid has been used namely: 10% less and 10% more as compared to the working fluid volume of traditional model. In additional, transient heat flux for seven hours has been applied on the riser pipe, that calculated by using Eq. (1) [14]. Furthermore, user-defined function (UDF) has been used as input of heat flux values into Ansys Fluent (CFD). As well as, the water was used as working fluid within the thermos-syphon loop. The property’s value of water that used under this study are density, viscosity, specific heat, thermal conductivity and thermal expansion rate are 998.2 kg/m$^3$, 0.001003 kg/(m.s), 4182 J/kg-K, 0.6 W/m-K and 0.000149K$^{-1}$.

\[
q = I_f \varepsilon \tau [\sin \delta \sin(\theta-\alpha) + \cos \delta \cos(\theta-\alpha) \cos \omega]
\]

where $I_f$, $\delta$, $\theta$, $\alpha$, $\tau$ and $\omega$ denote intensity of solar radiation, the incline angle of thermosyphon, local latitude, the transmittance of atmospheric and the sun hour angle respectively. while, $\varepsilon$ represent the earth’s orbit correction factor that can be estimated by using Eq. (2) [15].

\[
\varepsilon = \left[ 1 + 0.033 \cos \left( \frac{360N_d}{365} \right) \right]
\]

where $N_d$ and represent the day number of the year

### 3.5. Governing Equations of Fluid Flow

There are several equations such as, continuity, energy and Navier-Stokes are used to simulate the natural convection heat transfer process for the thermo-syphon system. Below some presumptions are considered under the present study.

The working fluid that circulating inside the thermo-syphon loop is water.

The flow is single phase, laminar, steady state and compressible.

Based on film temperature, all properties of water are determined.

The continuity equation of incompressible flow liquids can be described as:

\[
\frac{\partial}{\partial x} (\rho u_i) = 0
\]

Regarding the heat conduction, the different materials, the energy equation can give by [16] as:

\[
K_m \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) = 0
\]

While, the equation of energy can be represented as [17]:
∇.(ρhU) = -p∇U + ∇(k∇T) + φ + s_h

Finally, Navier-Stokes equations of the three dimensional can write as set of equations as shown in Eq. (6) [18]:

\nabla (\rho U u) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \\
\nabla (\rho U v) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \\
\nabla (\rho U w) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}

4. Results and discussions

There are two main aims of the present section are conduct comparison the current results of (CFD) with the experimental results in order to show reliability of the numerical results. As well as, study an impact of amount of the working fluid within the thermo-syphon loop on the thermal performance of this such system. The section concludes with a focus on develop correlation equation of water temperature inside the storage tank according to the results of numerical simulations.

4.1. Numerical computations validation

To demonstration the numerical results precision, both of the geometry and boundary conditions were utilized exactly similar to the experimental test rig. According to this basis, the water temperature within the storage tank and temperature of the working fluid at the outlet of the riser pipe for seven hours operation were compared as depicted in figures. (5-6) respectively. Based on these figures, can be clearly seen there are satisfy convergence between the results of numerical solution (CFD) and the experimental work. The maximum incongruity between numerical and experimental results is 14.52% and 11.23% for water temperature and temperature of the working fluid respectively. The reasons behind this difference between an experimental results and current results probably due to devices measurement errors that used in experimental, and the thermal losses of system.
Figure (5) Comparison between experimental and numerical results of water temperature within the storage tank.

Figure (6) Comparison between experimental and numerical results of the working fluid at outlet of the riser pipe.
As stated above, according to the great convergence between experimental and numerical results. The developed numerical approach can be considered to be sufficiently accurate and can be applied effectively to investigate the effect of parameters considered in the present work as detailed in the next section.

4.2. Effect of amount of working fluid

In this section, four sets of simulations will be taken up in depth to examine the impact of amount of working fluid inside the thermos-syphon system. First, traditional model, second (Case -A-) the amount of working fluid same as amount of the working fluid of traditional model. third, (Case -B-) the amount of working fluid is less than amount of the working fluid of traditional model by 10%. Last, (Case -C-) the amount of working fluid is larger than amount of the working fluid of traditional model by 10%.

Figure (7) shows the temperature variation respect to time of (a) water inside the storage tank and (b) the working fluid within the thermos-syphon at the outlet of the riser pipe for traditional model and (Case -A-). From figure (7, a) it can be clearly seen that the percentage rise in water temperature inside the storage tank was 15.14 percent compared to conventional model. While, figure (7, b) shows that the maximum working fluid temperature enhancement percentage at the riser pipe outlet is 19.89%. The reason behind this increase in temperature of both working fluid and water is an increase the riser pipe surface area.

![Graph showing temperature variation over time for traditional model and Case -A-](image_url)
Figure (7) temperature distribution against the time for a) water temperature inside the storage tank and b) the working fluid at riser pipe outlet. The effect of a decrease of 10% in the amount of working fluid inside the thermo-syphon loop on the temperature of each tank and the working fluid can be seen clearly in Figure 8 (a and b) respectively. This figure shows that the behavior for both temperature of water and working fluid in current case (Case-B-) is same behavior previous case (Case-A-). The maximum enhancement for new model (Case-B-) regarding water temperature within the storage tank is 10.07% as compared to traditional model as shown in figure (8, a). While, figure (8, b) show that the maximum improvement for the working fluid at riser pipe outlet is 22.5% as compared to traditional model.
Figure (8) temperature distribution against the time for a) water temperature inside the storage tank and b) the working fluid with 10% decrease of the working fluid.
Figure (9) shows the effect of increasing the amount of working fluid inside the thermo-syphon loop by 10% on the thermal efficiency of such device. Figure (9, a) shows the variation of water temperature inside the storage tank, while Figure (9, b) shows the variation of the working fluid temperature at outlet of the riser pipe. Figure (9) depicts that the general behavior of the water temperature and working fluid is the same as in the previous cases under consideration. It can be seen from this figure that there is enhancement for new model (Case- C-) as compared to traditional model by 10.14 percent and 6.3 percent for water temperature and the working fluid within the storage tank and outlet of the riser pipe.
Based on the above results, it is clear that the amount of working fluid has an impact on the thermal performance of the solar water system compared to the traditional model. Furthermore, the best case among the cases in this study can be clearly indicated in Table (4).

**Table 4 The effect of amount of the working fluid on the performance of system**

| Amount of the working fluid          | Temperature of the working fluid at outlet of the riser pipe (°C) | Water Temperature within the storage tank (°C) |
|--------------------------------------|-------------------------------------------------------------------|-----------------------------------------------|
| Same amount compared to conventional model (Case-A-) | 47.3 | 37.2 |
| 10% less as compared to traditional model (Case-B-) | 48.1 | 36.4 |
| 10% more as compared to traditional model (Case-C-) | 44.7 | 35.8 |

According to the table (3) it can be settled that maintaining the same amount of working fluid within the thermo-syphon loop in case of any improvement on such systems is very important. Despite, case -B- have higher temperature of the working fluid at the out the riser pipe
compared other cases. However, case -A- achieved highest water temperature inside the storage tank as compared to other cases. Due to, the main purpose of such systems is to heat the water inside the storage tank and hence it is one of the most important factors that give an impression of the thermal performance of such systems. Consequently, this case can be considered the best case among the cases under study.

4.3. Prediction of water temperature

From the numerical results, using multiple regression analysis, Eq. (7) has been expressed in order to predict water temperature within the storage tank as function of various parameters. These parameters are initial temperature, maximum heat flux during the day, the heat flux at any time, the volume of water within the storage tank, the volume of working fluid within the thermos-syphon loop and time.

\[
T_{w,t} = 1.275T_i \left( \frac{q_{\text{max}}}{q} \right)^{0.058} \left( \frac{D_{t,p}}{D_{t,t}} \right)^{0.042} \left( \frac{\text{Time}}{12} \right)^{0.964}
\]

Where, \( T_{w,t} \), \( T_i \), \( q_{\text{max}} \), \( q \), \( D_{t,p} \), \( D_{t,t} \) and 12 are represented the water temperature within the storage tank, initial temperature, maximum heat flux, heat flux at any time, diameter of riser pipe, tube diameter which placed inside the riser pipe and mid-day respectively.

To categorize the accuracy level of the Eq. (7) the determined data from the equation has been plotted against the numerical data from CFD as shown in figure (10). This figure shows that the maximum difference between them is 4.4%. Therefore, according to this observation, Eq. (7) can be used with considerable accuracy arrive around 95.6% to predict the temperature of water within the storage tank.
In order to validate the helpfulness of an equation which established using multiple regressions analysis, numerous statistical tests have been carrying out. Table 5 illustrates that the derived equation for water temperature within the storage tank has achieved the satisfactory criteria for all the tests. Hence, this an equation can be used with considerable confidence for further applications.

Table 5 Statistical tests and acceptance criteria for the water temperature within the storage tank

| Type of test             | Acceptance criteria                              | Eq. (7)   |
|-------------------------|--------------------------------------------------|-----------|
| F-value                 | If (F less than F critical) is accepted [19]     | F         |
|                         |                                                  | 1.012     |
|                         |                                                  | F critical| 2.032     |
| Durbin-Watson statistic | Accepted if less than 2 [20]                     |           |
|                         |                                                  | 1.135     |
| t-Test                  | Around zero [21]                                 |           |
|                         |                                                  | 0.0002    |
| Lilliefors test         | Accepted if equal zero                           |           |
|                         | Rejected if equal one [22]                       |           |
|                         |                                                  | zero      |
| Chi-square              | Accepted if less than 0.05 [23]                  |           |
|                         |                                                  | 0.0104    |

5. Conclusions
An experimental and numerical study has been carried out to investigate the effect of inserting a closed tube inside the riser pipe on the thermal performance of the thermo-syphon system. The effect the working fluid amount on system thermal performance has been studied. The
results have been shown that the maximum difference between numerical and experimental results were 14.52% and 11.23% for water temperature inside the storage tank and working fluid temperature at the riser pipe outlet respectively. Moreover, all developed models under investigation are fowed to have thermal performance higher than traditional model. This performance improvement can be clearly seen through the high temperature of water inside storage tank and temperature of the working fluid. Among all cases under investigation, (case-A-) is found as the best thermal response model. The temperature of water tank and working fluid temperature at the riser pipe outlet are approximately 15.14 and 19.89% higher for (case-A-) as compared to traditional model. In addition, a correlation equation to predict water temperature inside storage tank has been developed. This equation is capable of predicting the temperature of water storage tank with accuracy of 95.6%.

References

[1] Basavanna, S., and K. S. Shashishekar. "CFD analysis of triangular absorber tube of a solar flat plate collector." Int. J. Mech. Eng. & Rob. Res 2.1 (2013): 19-24.
[2] Thakare, Mangesh, and M. V. Khot. "Performance investigation of formed tubes of different geometry for a Flat-plate solar collector." International Journal of current Engineering and technology, special issue-5 (2016)
[3] Shelke, V., and C. Patil. "Analyze the effect of variations in shape of tubes for flat plate solar water heater." International Journal of Scientific Engineering and Research (JSER) 3.4 (2015): 118-124.
[4] Freegah, Basim, and Aouf A. Al-tabbakh. "Experimental and numerical analysis of a thermosyphon solar water heater for domestic applications." U.P.B. Sci. Bull., Series D, Vol. 81, Iss. 1, 2019 ISSN 1454-2358
[5] Manuel, Olivares-Ramírez Juan, et al. "Analysis of flow and heat transfer in a flat solar collector with rectangular and cylindrical geometry using CFD." Ingeniería, Investigación y Tecnología 14.4 (2013): 553-561.
[6] Yao, Keguang, et al. "Performance evaluation of all-glass evacuated tube solar water heater with twist tape inserts using CFD." Energy Procedia 70.Suppl. C (2015): 332-339.
[7] Martín, R. Herrero, García A. Pinar, and J. Pérez García. "Experimental heat transfer research in enhanced flat-plate solar collectors." World Renewable Energy Congress-Sweden; 8-13 May; 2011; Linköping; Sweden. No. 057. Linköping University Electronic Press, 2011
[8] Khargotra, Rohit, et al. "Performance investigation and comparison of different turbulator shapes in solar water heating collector system." AIP Conference Proceedings. Vol. 1953. No. 1. AIP Publishing LLC, 2018.
[9] Ameen, Braa Khalid, Mustafa B. Al-Hadithi, and Obaid T. Fadhil. "Heat Transfer Enhancement of Flat Plate Solar Collectors for Water Heating in Iraq Climatic Conditions." Al-Nahrain Journal for Engineering Sciences 18.2 (2015): 259-272.
[10] Sandhu, G., K. Siddiqui, and A. Garcia, Experimental study on the combined effects of inclination angle and insert devices on the performance of a flat-plate solar collector. International Journal of Heat and Mass Transfer, 2014. 71: p. 251-263.
[11] TSIRIGOTIS, G. and G. Iordanou, Computational Fluid Dynamics (CFD) investigations of the effect of placing a metallic mesh in the channels of a passive solar collector model. Elektronika ir Elektrotechnika, 2012. 121(5): p. 69-74.
[12] Razavi, J., M. Riazi, and M. Mahmoodi, Rate of heat transfer in polypropylene tubes in solar water heaters. Solar Energy, 2003. 74(6): p. 441-445.
[13] Hu, Jiange, et al. "Overlay Grid Based Geometry Cleanup." IMR. 2002.
[14] Aung, N.Z. and S. Li, Numerical investigation on effect of riser diameter and inclination on system parameters in a two-phase closed loop thermosyphon solar water heater. Energy Conversion and Management, 2013. 75: p. 25-35.
[15] Handbook, A., 1985 Fundamentals. American Society of Heating, Refrigerating, and Air Conditioning Engineers, Inc., Atlanta, Georgia, 1985.
[16] J. Blazek, Computational Fluid Dynamics: Principles and Applications,
[17] K.A. Hoffmann, S.T. Chiang, Comput. Fluid Dyn. (2000).
[18] B.R. Munson, D.F. Young, T.H. Okiishi, Fundamentals of Fluid Mechanics, John Wiley & Sons. Inc., New York, 1994, p. 2.
[19] Excel Easy, F-Test, (2016). Available from: http://www.excel-easy.com/examples/f-test.html.
[20] Rawlings, J.O., Pantula, S.G. and Dickey, D.A. Applied regression analysis. a research tool: Springer Science & Business Media, (2001).
[21] Runkel, P. What Are T Values and P Values in Statistics? (2016).
[22] De Smith, M. STATSREF: Statistical Analysis Handbook-a web-based statistics, (2015).
[23] Preacher, K.J., Calculation for the chi-square test: An interactive calculation tool for chi-square tests of goodness of fit and independence [Computer software], (2001).