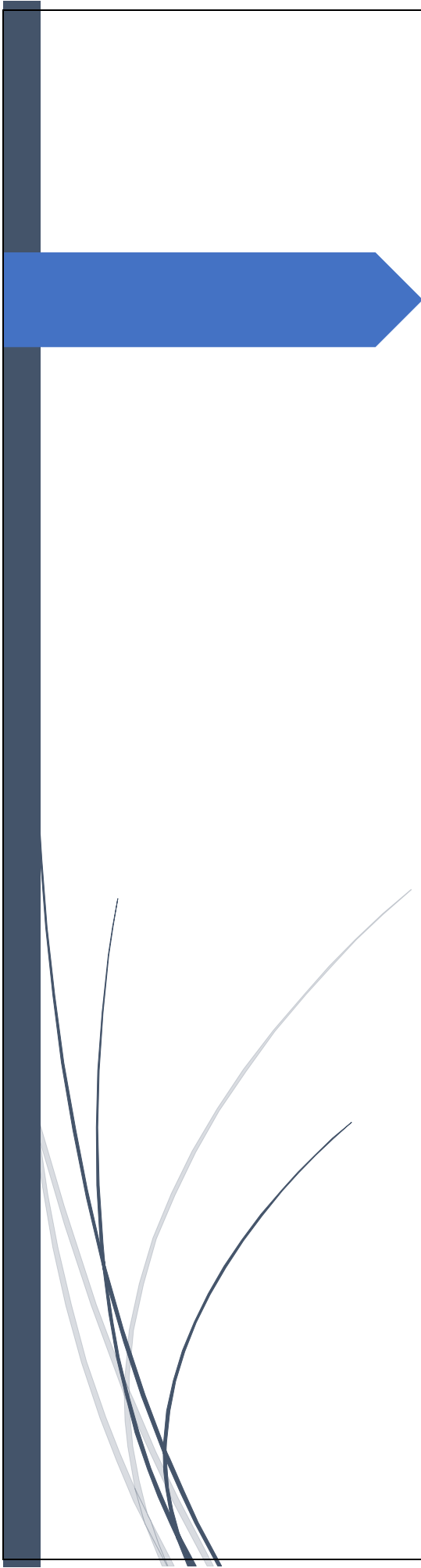




Simulation of Solar Water Desalination

By

Tina Bahador

*A final year project report
submitted in partial fulfillment
of the requirements for the
Bachelor of Science Degree in
Chemical Engineering*

at

*Bahai Institute for Higher
Education (BIHE)*

Supervised by

Dr. Farhang Sefidvash

August 2019

Abstract:

The demand for fresh water is increasing because of industrial development, intensified agriculture, increasing of world population and improvement of standards of life. Today, access to the fresh water has become a serious challenge in many parts of the world. One of the most affordable solution for this problem is to use solar still system to provide the drinking water from brackish water sources. One of the most important subjects of attention for many researchers has been to find appropriate regions for economical use of this system. Because of time-consuming and costly processes of solar still fabrication many researchers prefer to do computational fluid dynamics (CFD) simulation of solar still to assess the productivity of the system.

In this project we want to develop a 3D CFD model of simple solar still (single slope solar still) and we concentrate on the influence of the most important factor of the system productivity that is solar radiation intensity. At the end of the project we can conclude that if solar radiation intensity increases by 50%, the productivity of the system will be increased by 37.4%. We hope that by investigating the other important factors in productivity of the system in many other projects we can find the best regions for establishment of this system.

- **Contents:**

• Abstract:	2
• Acknowledgements:	5
• Introduction:	5
• Analytical method for get the amount of evaporated water and the physical properties of the system:	6
• Design:	13
• General:	13
○ Mesh:.....	13
○ Solver:	15
• Models:	17
○ Multiphase:	17
➤ Phases	22
➤ Phase Interactions	23
○ Viscous:.....	25
▪ Model:.....	30
▪ Near-wall treatment:	33
○ Energy:	34
➤ Heat transfer modeling	34
➤ Natural convection and buoyancy flows	36
➤ Natural convection-Boussinesq model	36
○ Radiation:	38
▪ Model:.....	39
▪ Solar load model:	43
• Boundary Conditions:	45
❖ In this project:	48
• Solution:	52
○ Methods:.....	52
▪ Pressure-Velocity Coupling:	52
▪ Gradient:	54
▪ Interpolation:	54

○ Controls:	57
○ Initialization:.....	58
• Solution conditions:	59
• Results and Discussion:	59
✓ Solar radiation= 1000W/m^2	59
✓ Solar radiation= 1500W/m^2	68
○ The Essay Results:	77
✓ Validation:	78
• Conclusion:	86
• Bibliography:	87

• Acknowledgements:

I wish to offer my best regards and thanks to dear Dr. Sefidvash for all of his sincere helps during this research paper and the department of chemical engineering especially dear Mr. Dolat for all of his kind supports and tolerances during 5 years of my studies. Also, with heartfelt thanks of my kind mother for all her helpful encouragements and with the memory of my father.

• Introduction:

Water is the essence of life and without it we can live only 3 or 4 days. Global human population growth demands the need of more fresh water, but water is a scarce resource. Almost 97 percent of the water on Earth is seawater with a high salt content. Water demand is increasing because of the industrial development, agricultural needs, home water usage, recreational and environmental activities (وہب اکاٹی- امین بھزادمہر- سعید فرساد, 1396).

Due to the expanding human population, competition for water is growing. According to the reports of UNICEF and the World health organization today 2.2 billion people don't have safely managed drinking water services, 4.2 billion people don't have safely managed sanitation services and 3 billion lack basic handwashing amenities (UNICEF, 2019).

Today, solar energy can be used as an important source of purifying water. Solar still is a low-technology cost solution that captures heat and energy from the sun, and with no adverse impact on the environment makes water cleaner and healthier for human use and consumption. This technique needs no fuel so, causes no pollution, but the distillate rate is a little low (Vaibhav Rai Kharea, Abhay Pratap Singhb, Hemant Kumarc, Rahul Khatri, 2016).

The solar still consists of a glass cover which allows the solar radiation to pass through it and the blackened base of the still absorbs most of the heat. Consequently, the water begins to warm and evaporates. So, the content of the air trapped between the water surface and the glass cover increases. At last the water vapor condenses on the inside of the glass cover. The salts and microbes of the original water are left behind and the condensed water can be saved to a storage tank to be used (شہرام دلفانی- ہادی پاسدارشہری- مریم کرمی, 1389).

The productivity of the solar still system depends on the design, operational and climate parameters. The designs parameters include water depth, insulation, cover material, its thickness and inclination and evaporation area. The operational parameters are feed water preheating and water salinity. The

climate parameters consist of solar radiation intensity, wind velocity, ambient temperature and relative humidity (Prachi K.Ithape, Anand R Nadgire, Dr.Shivprakash Bhagwatrao Barve, 2017).

The objective of this research is to develop a 3D CFD model of simple Solar still to investigate the radiation and evaporation phenomena in the solar still system. This model has been developed with the help of ANSYS Workbench and then simulated with Fluent. The concentration of this research is on the effects of the most important climate parameters i.e. solar intensity. To investigate the effect of solar intensity on the mass flow of saturated water vapor production which leads to water production after condensation, we assume that Direct Solar Irradiation (w/m^2) is always a fixed number and by this way we calculate the temperature of water ($^\circ\text{C}$) and the mass flow rate of vapor (kg/h) for two quantities of solar intensity 1500 w/m^2 and 1000 w/m^2 and then we compare them.

- **Analytical method for get the amount of evaporated water and the physical properties of the system:**

By the use of thermodynamic and heat transfer relations we can get the general equations for different parts of the system. At first, we must analyses the conditions of the inlet air. By knowing the wet and dry bulb temperature of inlet air and the psychrometric table, we can get the relative humidity (ϕ) of the inlet air. The ratio of amount of moisture the air holds (m_v) to the maximum amount of moisture the air can hold at the same temperature (m_g) is called relative humidity.

$$\phi = \frac{P_v}{P_g} \quad (1)$$

Also, we can define the relative humidity according to the density and specific volume.

$$\phi = \frac{P_v}{P_g} = \frac{\rho_v}{\rho_g} = \frac{V_v}{V_g} \quad (2)$$

The mass of water vapor which is present in a unit mass of dry air is called absolute or specific humidity:

$$\omega = \frac{m_v}{m_a} \quad (\text{kg water vapor/ kg dry air}) \quad (3)$$

By considering ideal gas we can define these relations:

$$M_v = \frac{P_v V}{R_v T} \quad (4)$$

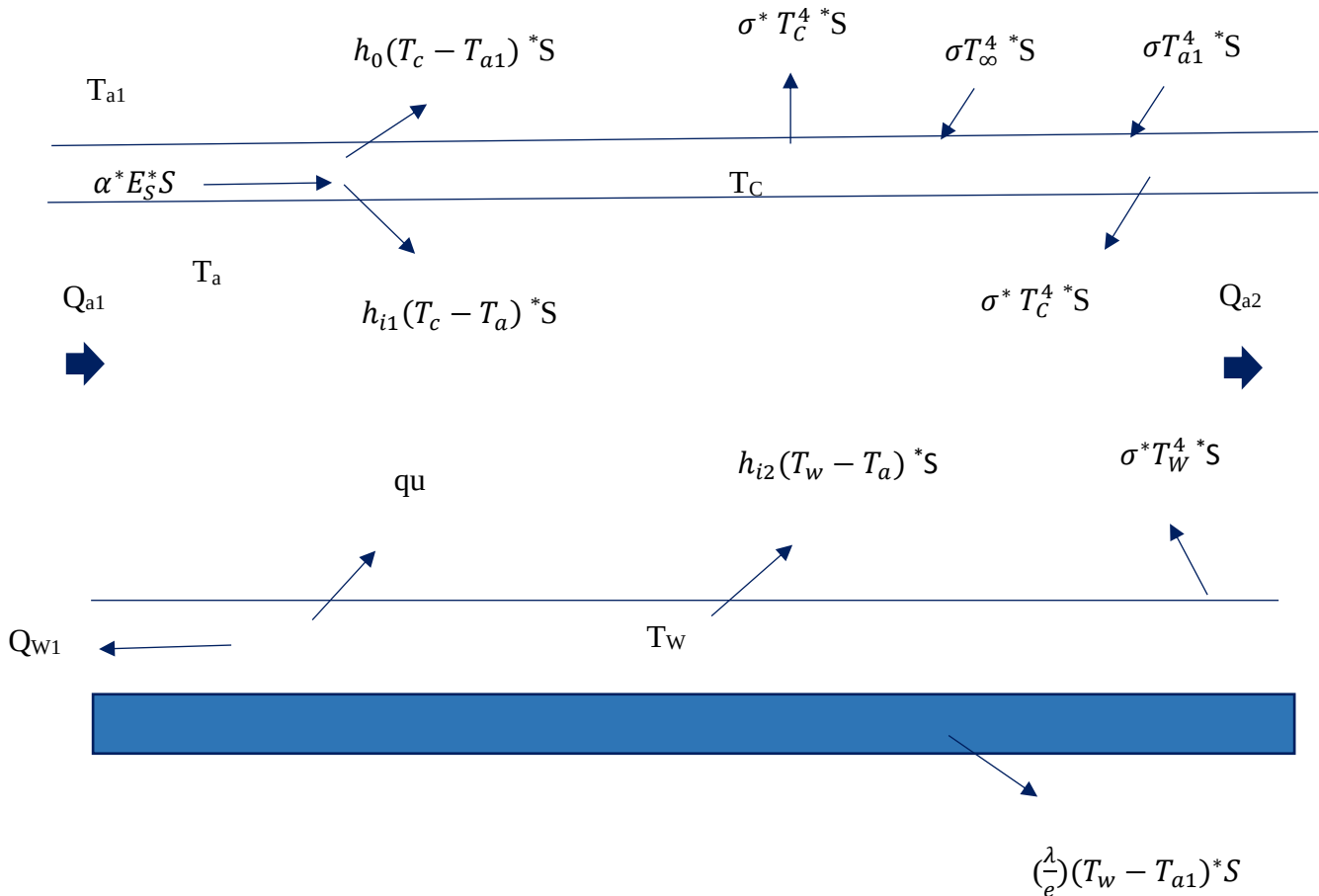
$$M_a = \frac{P_a V}{R_a T}$$

Also, we have:

$$\omega = 0.622 \frac{P_v}{P_a} \quad (\text{kg water vapor/ kg dry air}) \quad (5)$$

We can get partial pressure by having specific humidity from psychrometric tables and vapor pressure. Also, we can get the mass of the air and the mass of inlet vapor by knowing the volume of the system and the temperature of the environment.

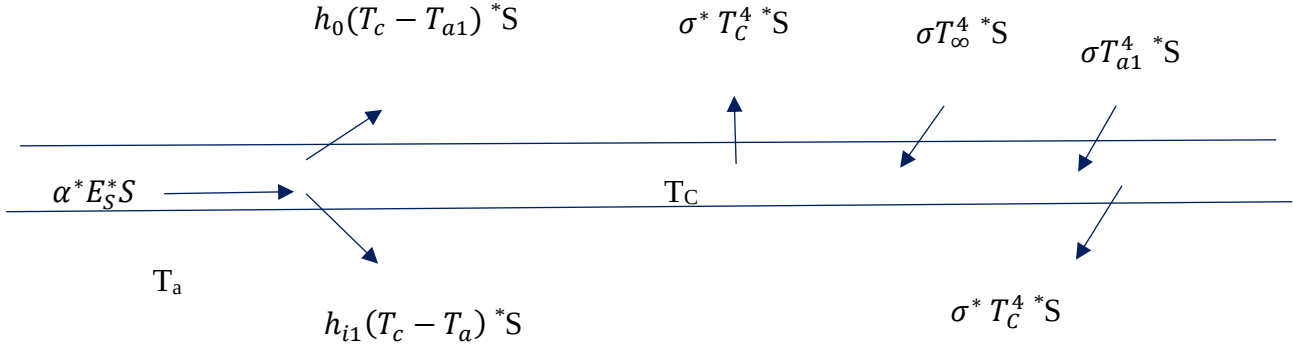
According to the picture heat transfer between the glass and the outlet air in the form of convection and radiation, between the glass and the air and the inlet vapor in the form of radiation and convection, between water and air and vapor in the form of radiation and convection and evaporation, between basin of the system and water in the form of radiation and convection and between the side wall of the system and outside air in the form of conduction can be done.



Now we can study all the processes:

- Heat transfer on the glass:

According to the picture we have this equation:



$E_s \left(\frac{W}{m^2} \right)$: The amount of solar radiation

$S(m^2)$: Area of surface

α : Glass absorption coefficient

$T_{a1}(k)$: Air temperature

$\sigma \left(\frac{W}{m^2 k^4} \right)$: Stefan – Boltzmann constant

$T_c(k)$: Glass temperature

$T_a(k)$: Averaged inlet and outlet air temperature

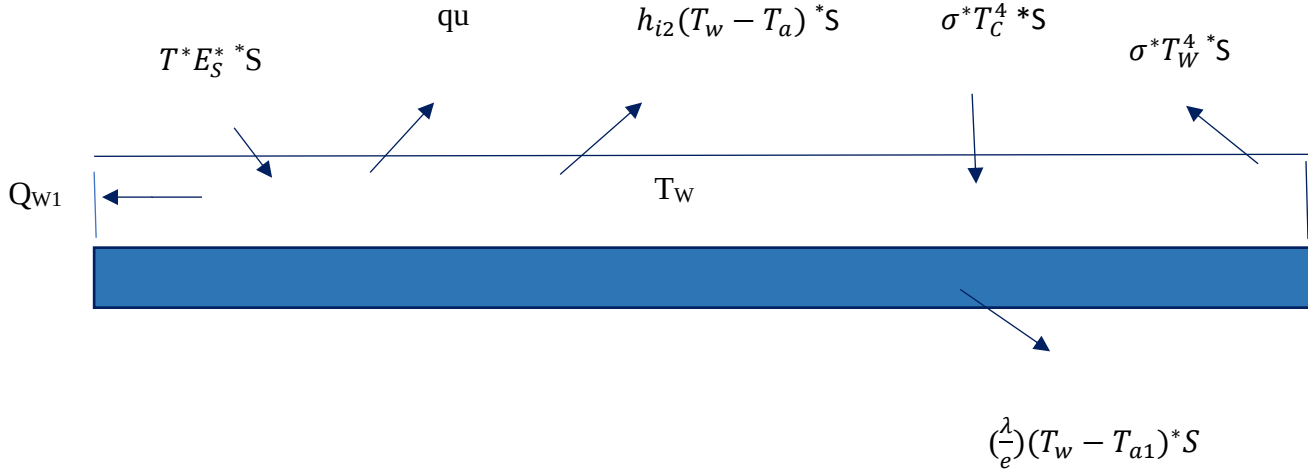
$h_o \left(\frac{W}{m^2 k} \right)$: Outlet convection heat transfer coefficient

$h_{i1} \left(\frac{W}{m^2 k} \right)$: Inlet convection heat transfer coefficient

$$\alpha E_s S + \sigma T_{a1}^4 S + \sigma T_{\infty}^4 S = 2\sigma T_c^4 S + h_o (T_c - T_{a1}) S + h_{i1} (T_a - T_c) S \quad (6)$$

- Heat transfer between basin of the system and water:

According to the picture we have these equations:



cond: The proportion of insulation thermal conductivity coefficient to insulation thickness

$T_w(k)$: The basin water temperature

$T_{w1}(k)$: The inlet water temperature

T : The glass transmittancy

$h_{fg} \left(\frac{j}{kg} \right)$: Latent heat of vaporization

$m_{evap} \left(\frac{kg}{s} \right)$: The mas of evaporated water

$CP_w \left(\frac{j}{kg-k} \right)$: Specific heat capacity of water

qu : The quantity of energy for water evaporation

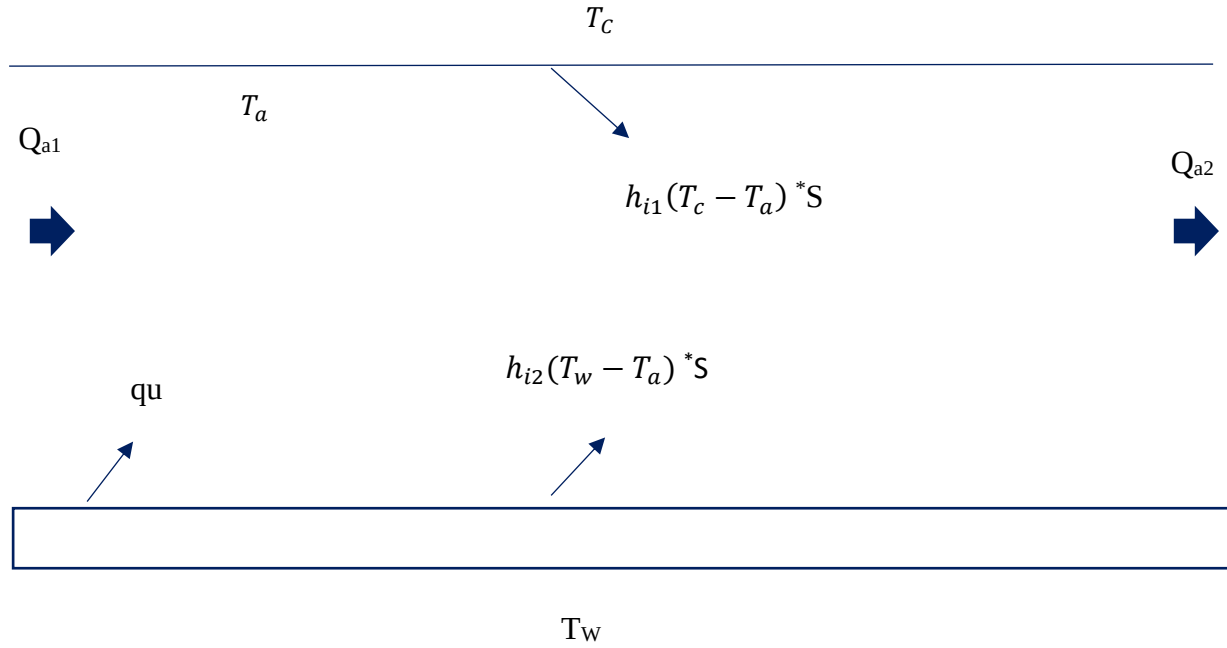
$$-Q_{w1} + \sigma T_c^4 S + T E_s S = h_{i2} (T_w - T_a) S + cond (T_w - T_{a1}) S + \sigma T_w^4 S + qu \quad (7)$$

$$Q_{w1} = m_{evap} CP_w (T_w - T_{w1}) \quad (8)$$

$$qu = m_{evap} h_{fg} \quad (9)$$

- Heat transfer on the mixture of water vapor and air:

In this picture all the processes of heat transfer have been shown:



$Q_{a1} \left(\frac{j}{s} \right)$: The heat energy of inlet air

$Q_{a2} \left(\frac{j}{s} \right)$: The heat energy of outlet air

$$Q_{a1} + h_{i1}(T_c - T_a)S + h_{i2}(T_w - T_a)S + qu = Q_{a2} \quad (10)$$

According to these equations:

$$Q_{a1} = m_{a1}h_{a1} + m_{v1}h_{v1} \quad h_{v1} = h_g \quad (11)$$

$$Q_{a2} = m_{a2}h_{a2} + m_{v2}h_{v2} \quad h_{v2} = h_g \quad (12)$$

$$m_{v2} = m_{evap} + m_{v1} \quad (13)$$

$$T_a = \frac{T_{a1} + T_{a2}}{2} \Rightarrow T_{a2} = 2T_a - T_{a1} \quad (14)$$

So, m_{evap} – T_C - T_W and T_a are the unknown of problem:

$$m_{\text{evap}} = h_{cc}(\rho_g - \rho_v)S \quad (15)$$

In which:

$h_{cc}(\frac{m}{s})$: Mass transfer coefficient

$\rho_g(\frac{kg}{m^3})$: Density of saturated vapor at specific temperature

$\rho_v(\frac{kg}{m^3})$: Density of vapor at specific temperature

$$PV = mRT \Rightarrow P = \rho RT = \rho \left(\frac{\bar{R}}{M} \right) T \Rightarrow PM = \rho \bar{R} T \Rightarrow \rho = \frac{PM}{\bar{R} T} \quad (16)$$

$$\bar{R} = 8314 \frac{J}{kg} \quad M = 18 \quad (17)$$

$$\rho = \frac{P}{462T} \quad (18)$$

By replacing the relation 18 in the relation 15, we have:

$$m_{\text{evap}} = \frac{h_{cc}}{462} \left[\frac{P_g}{T_w} - \frac{P_v}{T_a} \right] S \quad (19)$$

$P_g(Pa)$: The saturated vapor pressure at T_w

$P_v(Pa)$: The partial vapor pressure at T_a

$T_a(k)$: The average temperature of inlet and outlet air

We can rewrite the relation 5:

$$\omega = 0.622 \frac{P_v}{(P_{atm} - P_v)} \quad (20)$$

By replacing the relation (1) in the relation (20), we have:

$$\omega = 0.622 \frac{\varphi P_g}{(P_{atm} - \varphi P_g)} \quad (21)$$

Also, we can rewrite the relation (20):

$$P_v = \frac{\omega P_{atm}}{(0.622 + \omega)} \quad (22)$$

By considering the relation (3), we have:

$$\omega_1 = \frac{m_{v1}}{m_a} \quad \omega_2 = \frac{m_{v2}}{m_a} = \frac{m_{v1} + m_{evap}}{m_a} \quad (23)$$

By averaging the specific humidity, we have:

$$\omega = \frac{\omega_1 + \omega_2}{2} = \frac{2m_{v1} + m_{evap}}{2m_a} \quad (24)$$

By replacing the relation (21) in the relation (19), we have:

$$m_{evap} = \frac{h_{cc} S}{462} \left[P_g T_w - P_{atm} \left[\frac{2m_v + m_{evap}}{1.24m_a + 2m_v + m_{evap}} \right] \right] \quad (25)$$

In which:

$$h_{cc} = \frac{1.11h}{\rho C_p} \quad (26)$$

$h \left(\frac{W}{m^2.k} \right)$: The convection heat transfer coefficient.

$\rho \left(\frac{kg}{m^3} \right)$: Vapor density at specific temperature.

$C_p \left(\frac{J}{kg.k} \right)$: Specific heat capacity of vapor at specific temperature.

Now, we can get the four unknown T_C , T_w , T_a and m_{evap} by solving the four relations 6, 7, 10 and 19 simultaneously (Yunus A.Cengel- Michael A.Boles, 2011), (Yunus A.Cengel- Afshin J.Ghajar, 2015).

- **Design:**

In this project we do our researches on the simplest kind of solar still. The 3D geometry of solar still is modeled using ANSYS Design Modeler.

Constituents:

- Solar still is made with an aluminum with a base area of 1m^2 .
- Sea water level of 5cm.
- Glass cover inclination of 15 degrees.

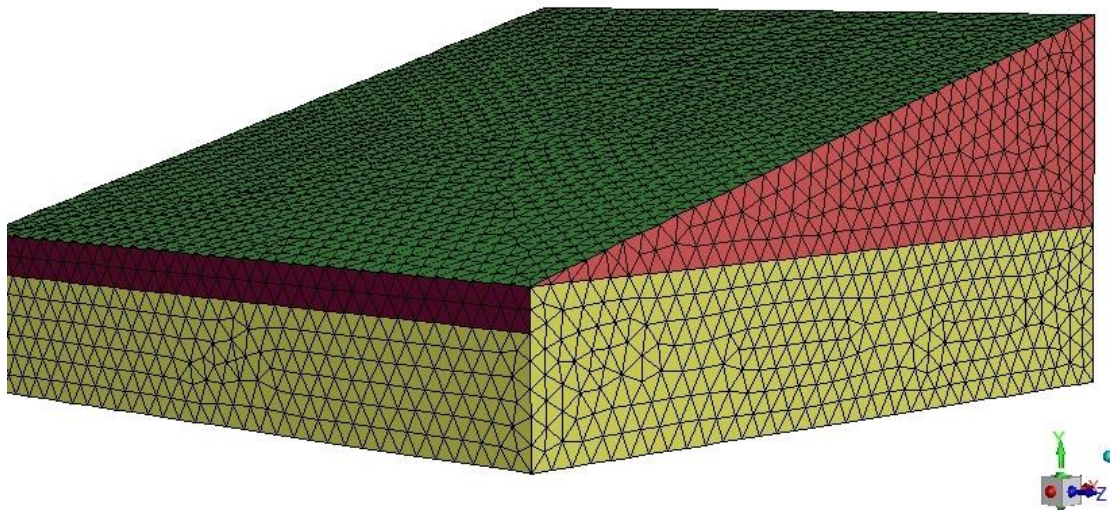


Figure 1, Single slope solar still.

- **General:**

- **Mesh:**

The quality of meshes:

The quality of meshes has an important role in stability and precision of numerical calculations.

Maximum Aspect ratio: The aspect of length to the width which is used to measure cells elongation is called aspect ratio. The ratio must be five or less in the main region of mesh and far from the boundary condition. However, the number much higher is also common in boundary conditions layer.

Minimum Orthogonal quality: besides minimum aspect ratio the quantity of minimum orthogonal quality is also reported. Number one refers to best quality and number zero shows the worst quality.

According to the supposed mesh for this problem this quantity is $1.78527\text{e-}1$ which is the result of extended cells near the boundary conditions.

According to the picture most of the cells have good quality and the quantity of orthogonal quality is equal to 0.929 but in boundary layers this number is equal to 1.78527×10^{-1} (Munoz, 2015).

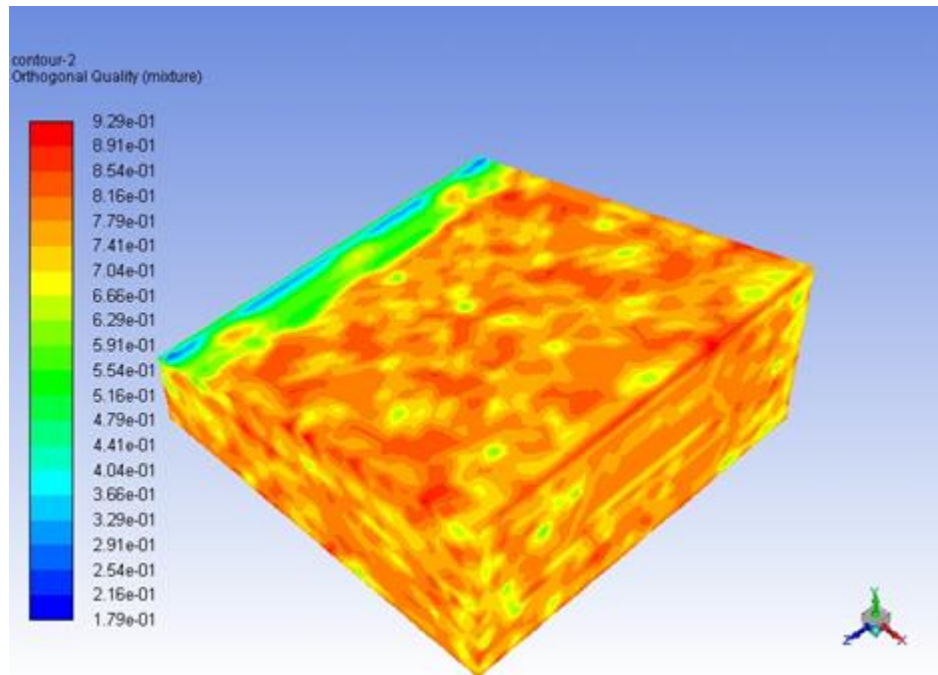


Figure 2, Orthogonal Quality of Meshes.



Figure 3, Mesh Properties.

Mesh Independence:

It's important to be sure that mesh presence causes correct answers. If we want to be sure about the answers, we must check that to solve the problem is independence from mesh which means that answers will not change with varying the mesh. For this purpose, we create smaller meshes and start the calculations till we reached convergence then we compared the results of two different size of meshes. Since the results were appropriately the same, we use the greater meshes for calculations because it needed less calculation times (LEAP CFD Team, 2012).

○ Solver:

Type:

- Pressure-based solver
- Density-based solver

We can use those two solvers in a vast limit of flows, but in some cases one solver is better than the other one. We use pressure-based solver most in incompressible and semi- compressible flows, but we use density- based solver in high velocity compressible flows.

There are two Implicit and Explicit methods for density- based solvers. The Implicit method in steady-state problems is more rapidly converged than the explicit method but it needs more memories.

For pressure- based solvers we have two segregated and coupled methods. In segregated algorithm the equations are solved separately but in coupled algorithm momentum and continuity equation are solved simultaneously. Coupled algorithm has higher converged velocity but it needs more memories. Coupled algorithm shows a better operation in most single-phased flows, but for multi-phased flows it cannot be used and it needs 1.5 twice more memories.

The models which are present in pressure- based solvers and not in density-based solvers are: Cavitation model, VOF model, Multiphase mixture model, multiphase Eulerian model, Non-premixed combustion model, Premixed combustion model, partially premixed combustion model, Rosseland radiation model, Solidification and melting model, Buoyancy operating pressure, ...

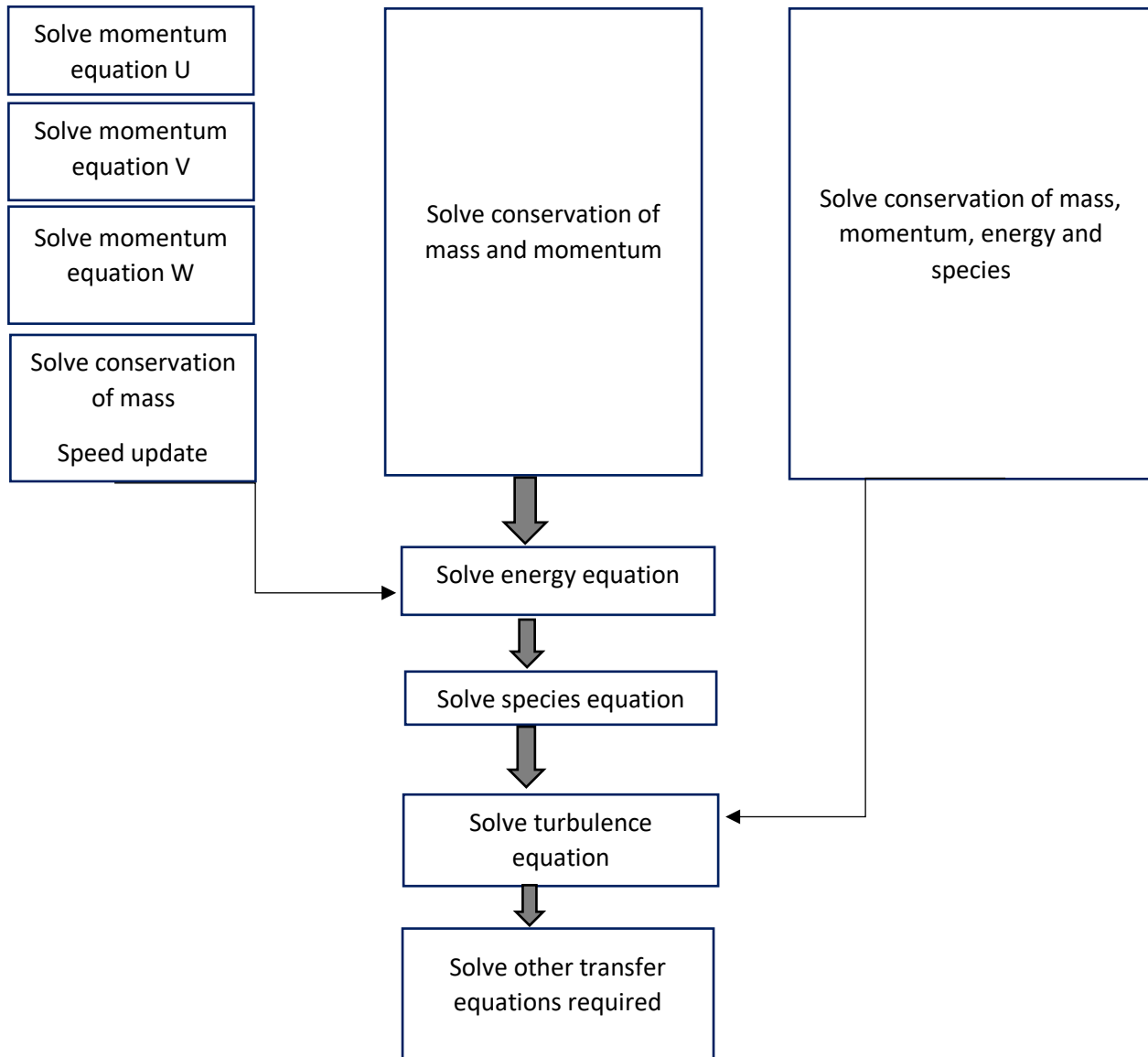
The models which are present in density- based solvers and not in pressure- based solvers:

Ideal gas models, Non- reflecting boundary conditions, water vapor multiphase model, ...

Pressure-based solver
(Segregated)

Pressure-based solver
PBCS(Coupled)

Density-based solver
DBCS(Coupled)



Graph 1

(دکتر امیرتوحیدی- مهندس هادی غفاری قهرودی, 1397)

General

Mesh

Scale... Check Report Quality

Display...

Solver

Type: ☒ Pressure-Based ☐ Density-Based

Velocity Formulation: ☒ Absolute ☐ Relative

Time: ☐ Steady ☒ Transient

☒ Gravity Units...

Gravitational Acceleration

X (m/s²) 0 P

Y (m/s²) -9.81 P

Z (m/s²) 0 P

Help

Figure 4

- **Models:**

- Multiphase:

Modeling of multiphase flow:

Phase is a type of material with specific boundary which shows a special dynamic answer to the potential field of environmental flows. Generally, the phases are existed in three forms: solids, liquids and gases, but also it refers to materials in the same phase (such as liquid-liquid) which have different chemical properties.

Multiphase flows are existed in two types like mixture of different materials such as water and oil or different phases of material such as water and vapor.

In multiphase flows, fluid is defined with only a primary phase and some secondary phases. One of the primary phases is considered continuous and the secondary phases are scattered in it.

The multiphase flows can be classified according to the type of the phases which mixed together:

1- The multiphase flows Liquid/Liquid or Liquid/Gas

- Bubbly flow: Discrete gaseous or fluid bubbles in a continuous fluid such as evaporators, absorbers, aeration, flotation and ...
- Droplet flow: Discrete fluid droplets in a continuous gas such as absorbers, atomizers, combustors, dryers and ...
- Slug flow: Large bubbles in a continuous fluid such as large bubble motion in pipes or tanks.
- Stratified/ free-surface flow: Immiscible fluids which are separated by special interface such as free surface flow such as sloshing in offshore separator devices and boiling and condensation in nuclear reactors.

2- The multiphase flows Solid/Gas

- Particle-laden flow: Discrete solid particles in continuous gas such as cyclone separators, air classifiers, dust collectors and...
- Fluidized beds: A vertical cylinder with particles in which the gas rising through the bed and suspend them such as fluidized bed reactors and circulating fluidized beds.
- Pneumatic transport: The flow algorithm is dependent on factors like Reynolds number and the properties of particles such as transport of cement, grains and metal powders.

3- The multiphase flows Solid-Liquid

- Slurry flow: Transport of particles in liquids such as slurry transport and mineral processing.
- Hydro transport: Densely-distributed of solid particles in a continuous liquid such as mineral processing and biomedical and physiochemical fluid systems.
- Sedimentation: It refers to a high column which contains a uniform mixture of particles, but the particles come down slowly and sediment in the form of sludge layers such as mineral processing.

Generally, there are two methods for studying multiphase flows:

- 1- Euler- Lagrange method
- 2- Euler-Euler method

The choice of the best method for modeling multiphase flows is very important and depends on whether the flow is stratified or disperse.

1- Euler-Lagrange method

In discrete phase model Euler- Lagrange method is used. In this state discrete phase able to change the momentum, mass and energy of fluid phase. In this case the fluid phase is solved by Navier-Stokes equation, and the discrete phase is calculated by tracing a large number of particles, bubbles or droplets. This method is suitable for spray- liquid fuel combustion, coal and the flows containing particles.

However, this model isn't suitable for liquid-liquid mixture modeling, fluidized bed and states in which the second phase volume proportion isn't negligible.

2- Euler- Euler method

Since the volume of one phase cannot be occupied by another phase, the concept of phase volume fraction is given. These volume fractions are assumed to be continuous functions of space and time that their sum is equal to one.

There are three different models for multiphase Euler-Euler which are named:

- 1- Volume of fluid model (VOF)
- 2- Mixture model
- 3- Eulerian model

1- Volume of Fluid (VOF)

This model is designed for detection of position and movement of free surface of two or more immiscible fluids. In this model a separate collection of momentum equations is shared by the fluids and volume fraction of fluid is detected in calculated cells.

Applications:

- ✓ The movement of great bubbles in a liquid.
- ✓ Stratified flows.
- ✓ Free- surface flows.

- ✓ Filling of tanks.
- ✓ The movement of sludge and stodge.
- ✓ The movement of fluids after breaking of dam.

Limitations:

- ✓ This model should not be used in empty regions without any fluids. In fact, all the controlled volumes must be filled with a single fluid phase or a combination of phases.
- ✓ Only one of the phases can be considered as the ideal compressible gas.
- ✓ The use of explicit second-order time steps formulation is not possible.

2- Mixture model

In this model the simple assumption of small Stocks numbers is considered. In this model the momentum equations are solved and relative velocity for determination of diffusion of phases are specified.

Applications:

- ✓ The flows of suspended particles with low load
- ✓ Bubbly flow
- ✓ Sedimentation
- ✓ Cyclone separation

Limitations:

- ✓ Only one of the phases can be considered as a compressible ideal gas.
- ✓ Impossibility of melting and solidification modeling.
- ✓ The impossibility of using in non-viscous flows.
- ✓ The impossibility of using shell conduction model for walls.

3- Eulerian model

This is the most complex model of multiphase and in which the n collection of momentum and continuity equation for each phase is solved.

Applications:

- ✓ Bubbly flow.
- ✓ Full particles flow.

- ✓ Droplet flow.
- ✓ Fluidized beds.

Limitations:

- ✓ The Reynolds Stress turbulence model is not available on a per phase basis.
- ✓ Impossibility of melting and solidification modeling.
- ✓ Inviscid flow is not allowed.

The calculation is largely depended on the number of transition equations and the degree of coupling. So, before solving the Eulerian model the problem must be simplified as far as possible (16.Multiphase Flows, 2009).

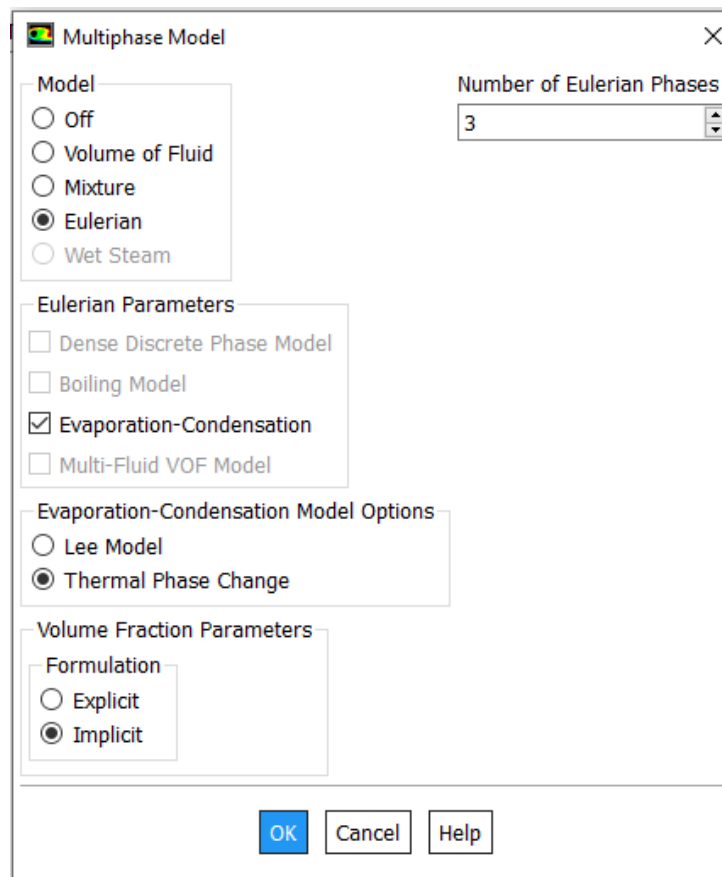


Figure 5

➤ Phases

The figure shows three stacked dialog boxes for defining phases in a simulation software. Each dialog box has a title bar with a small icon and a close button (X).

- Primary Phase:**
 - Name: phase-1
 - Phase Material: air (dropdown menu)
 - Edit... button
- Secondary Phase (phase-2):**
 - Name: phase-2
 - Phase Material: water-liquid (dropdown menu)
 - Edit... button
 - ☐ Granular
 - ☐ Interfacial Area Concentration
 - Properties section:
 - Diameter (m): constant (dropdown menu)
 - Edit... button
 - 1e-05 (text input)
- Secondary Phase (phase-3):**
 - Name: phase-3
 - Phase Material: water-vapor (dropdown menu)
 - Edit... button
 - ☐ Granular
 - ☐ Interfacial Area Concentration
 - Properties section:
 - Diameter (m): constant (dropdown menu)
 - Edit... button
 - 1e-05 (text input)

At the bottom of the stack, there are four buttons: OK (blue), Delete, Cancel, and Help.

Figure 6

Granular:

The granular flows refer to the flows that the concentration of solid particles is very high that leads to the high frequency of collision between them.

Molecular cloud theory is considered for particles phases which leads to the creation of extra tension in momentum equations.

➤ Phase Interactions

▪ Drag

For two granular and non-granular flows, the drag function must be determined to calculate coefficient of momentum variation.

The choice of appropriate drag function for a pair of phases:

- Schiller- naumann

This is a presupposed model of solvers which is generally appropriate for calculating multiphase fluid-fluid.

- Morsi- alexander

This is the most complete model but with less stability and is appropriate for fluid-fluid drag function in which the defined function is regulated on a vast domain of Reynold number.

- Symmetric

This model is appropriate for fluid-fluid drag function and is used in flows in which the separate phase is turned to a continued phase in the solving domain.

- Universal-drag

This model is used for bubble-liquid or droplet-gas flows when the characters of the flow range length are larger than the average size of the particles.

- Wen-yu

This model is used for fluid-solid function and is appropriate for dilution flows.

- Gidaspow

This model is used for fluid-solid function and is appropriate for concentrated flow in fluidize bed.

- Syamlal-obrien

This model is used for solid-solid function.

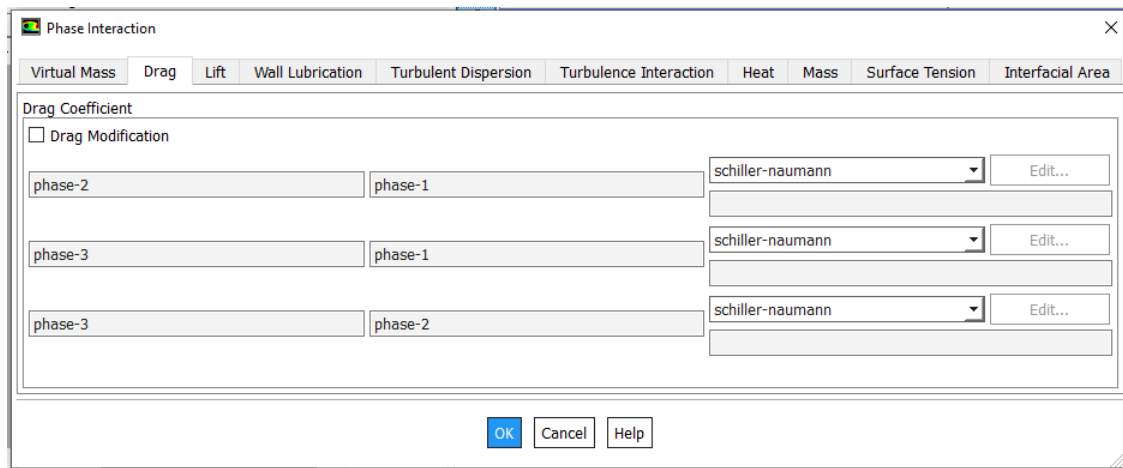


Figure 7

- Lift

Usually the lift force is very trivial and is negligible compared to drag force.

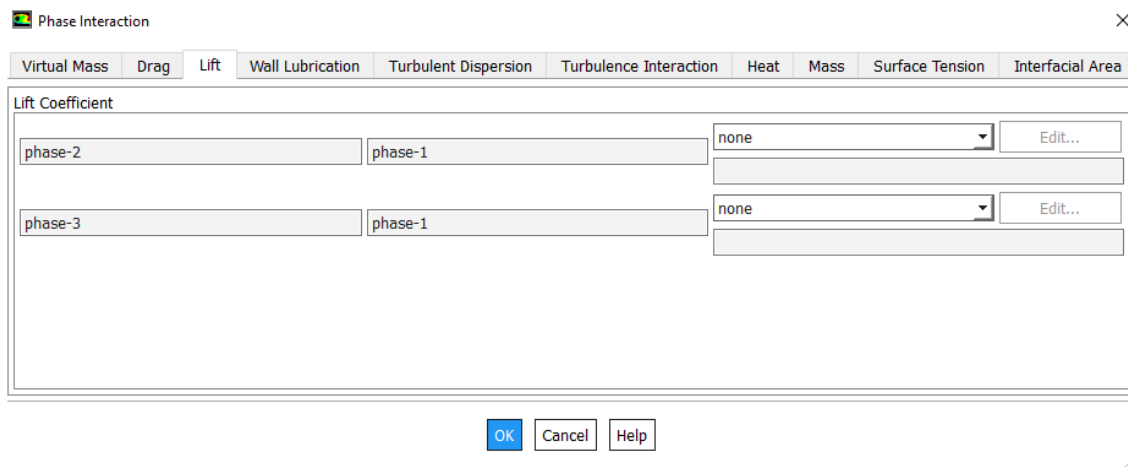


Figure 8

- Virtual mass

It is only considered when the density of secondary phase is very smaller than the density of primary phase.

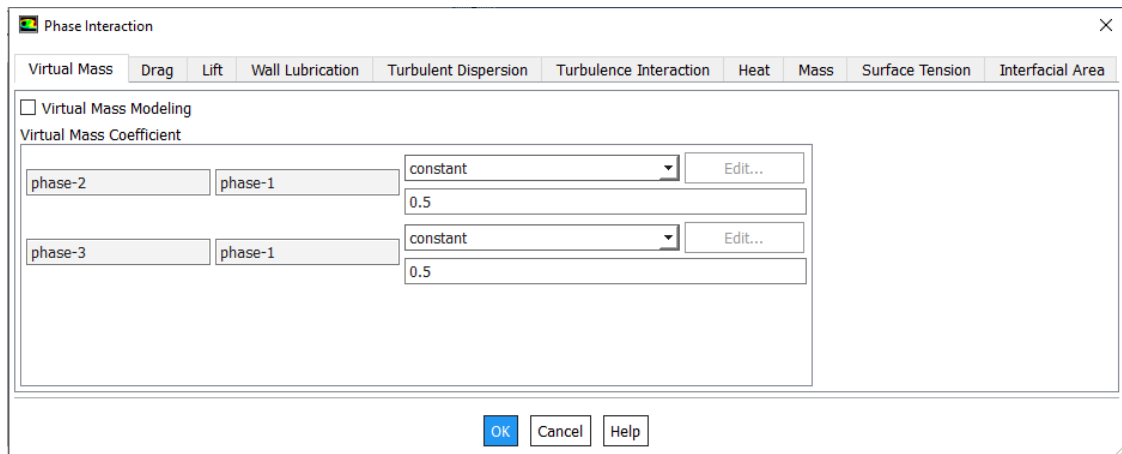


Figure 9

- Heat

In considering heat transfer the appropriate relation for heat transfer coefficient must be chosen.

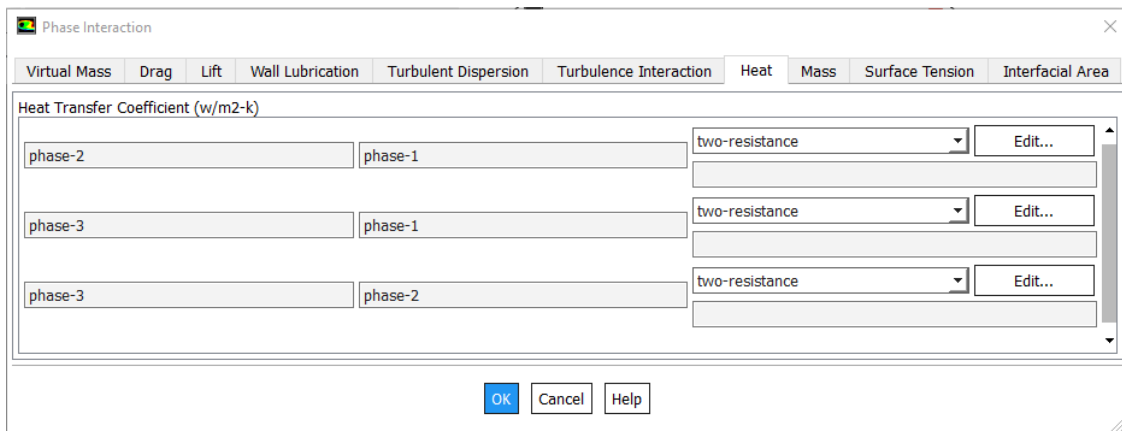


Figure 10

(35.5.3 Phase Interaction Dialog Box, 2009), (دکتر مهدی دادخواه تهرانی- مهندس مائده نعمتی, 1397).

- Viscous:

The methods of turbulence modeling:

- 1- Direct Numerical Simulation (DNS)
- 2- Reynolds-averaged Navier-Stokes (RANS)
- 3- Large Eddy Simulation (LES)
- 1- Direct Numerical Simulation (DNS)

In this method the transient three dimensional Navier-Stokes equations are solved for all the scales (space-time), so there is a large spectrum of frequencies and wavelengths which causes the direct numerical solutions of Navier-stokes equations in turbulence and time dependent position which requires many points, and as a result the calculation systems must be very costly and strong.

Also, we know that the turbulence flows are largely influenced by boundary conditions so in direct solving of equations all the details of the flow like wall roughness and fluctuations of time and position must be precisely considered. This matter also needs very complicated calculations which is exceeded the velocity and memory of the most powerful today computers.

Although this method is very valuable for many researches, it is only suitable for simple geometry and average Reynolds numbers. Unfortunately, this is a costly method for engineering and experimental goals and is in need of very powerful computers.

2- Reynolds-averaged Navier-Stokes (RANS)

In this method the equations for average time quantity and turbulent oscillations flows are solved. For computing the average time period, we must use this formula:

$$\bar{f} = \lim_{T \rightarrow \infty} \frac{1}{T} \int_0^T f(x_i, t) dt$$

Instantaneous field is defined by the sum of average and oscillation components:

$$p = \bar{p} + p'$$

$$u_i = \bar{u}_i + u_i'$$

When we average the Navier-Stokes equations, we can get the Reynolds averaged Navier-Stokes equation:

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0$$

$$\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_m}{\partial x_m} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{u_i' u_j'})$$



Reynolds stress tensor (R_{ij})

$$R_{ij} = -\rho \left[\frac{\overline{u'u'}}{\overline{v'u'}} \quad \frac{\overline{u'v'}}{\overline{w'v'}} \quad \frac{\overline{u'w'}}{\overline{v'w'}} \right]$$

The only difference between the equations of laminar flow with the averaged equations of turbulence flow is the Reynolds stress presence. This is not physically a stress but is the representative of the exchange of increased inertia by the influence of turbulence flow.

R_{ij} is also Reynolds stress tensor and is gained by the averaging of convection acceleration in momentum equations and indicates the combination of the mixture as a result of turbulence and smoothing of averaging.

By using appropriate statistical methods, we don't need to use very small and dense element and determining precise boundary and elementary condition is not needed.

The RANS model can be divided according to Reynolds stress calculation into two groups:

1- Eddy Viscosity Model (EVM)

In this model it is assumed that strain is proportional to stress. The only unknown quantity is effective turbulent viscosity.

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} (\rho k + \mu_t \frac{\partial \bar{u}_m}{\partial x_m})$$

Eddy viscosity

Eddy viscosity is not a fluid property but it is a character of turbulent flow and is changed in flow field according to its position. This modeling is only used for isotropic flows and cannot be used for excessive circular flows, excessive curved lines flows and ...

Dimensional analysis shows that if we have the needed scales like velocity and length, it will be easy to determine eddy viscosity.

$$\frac{\mu_t}{\rho} = v_t \left[\frac{L^2}{t^2} \right] = \left[\frac{L^2}{t^2} \right] \times [t] = \left[\frac{L}{t} \right] \times [L]$$

For example, in simple flows by having the length and velocity scales we can determine turbulent viscosity. Also, when we have time and velocity scales eddy viscosity can be determined.

2- Reynolds Stress Model (RSM)

Transition equations for Reynolds stress terms are gotten. This is a more complicated model than the EVM and tries to remove its defects because there are more unknown variables. So, we need more equations. This is a more costly and it is difficult to converge proportional to two equation models. This model is more appropriate for three dimensional complex flows with curved lines and circular flows.

$$\frac{\partial \rho \overline{u_i' u_j'}}{\partial t} + \frac{\partial}{\partial x_k} (\overline{u_k u_i' u_j'}) = P_{ij} + F_{ij} + D_{ij}^T + \varphi_{ij} - \varepsilon_{ij}$$

The product of tension

Turbulence
diffusion

Pressure
strain

Depreciation

❖ ANSYS FLUENT TURBULENCE MODELS

Increasing accuracy
and efficiency



Single equation models

Spalart-Allmaras

Increase the cost and
time of calculations



Two equation models

Standard k- ϵ

RNG k- ϵ

Realizable k- ϵ

Standard k- ω

SST k- ω

4-Equation v2f

Reynolds Stress Model

k-kl- ω Transition Model

SST Transition Model

Detached Eddy Simulation (DES)

Large Eddy Simulation (LES)

Graph 2

- Model:
- ✓ One equation model:
- Spalart-Allmaras

This is a low-cost model of RANS and solves the transition equation for reformed eddy viscosity. This is a simple one equation model and uses only one equation to get distribution turbulence viscosity (μ_t).

It is more used for low Reynolds numbers. If we move from low Reynolds number to high ones, the capability and accuracy of this model is reduced.

This model is appropriate for aerodynamic applications, turbomachine turbulence flow such as supersonic/transonic flows near the airfoils and boundary layer flows.

- ✓ Two equation model:

These equations have high capability with simple forms and by solving two transition equations we can get two quantity of turbulence velocity and length scales independently.

The beginning point of all two equation models is the use of Boussinesq approximation and presentation of a transition equation for turbulence kinetic energy(k). The choice of the second variable is optional.

Generally, all the two equation models are very useful for industrial simulation because of their high predictability, more economical and high precision for a vast range of turbulence flows.

$$\rho \left(\frac{\partial u_i}{\partial t} + u_i u_{i,j} \right) = -P_i + B_i + [\mu(u_{i,j} + u_{j,i}) - \rho \overline{u'_i u'_j}]$$



Uses Eddy-Viscosity equations for calculating
Reynolds stress such as $-\rho \overline{u'_i u'_j} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$

Uses Eddy-Viscosity models such as k-ε and k-ω for determining turbulent viscosity μ_t .

- k-epsilon Model:
- 1- Standard k-ε model

SKE is the most common turbulence model for industrial applications. It has experimental coefficients that their quantity is gotten on the bases of experimental results. We can change the calculated

coefficients to get the best results according to experimental consequences. But this model is very sensitive to any changes of the coefficients. This model can give good answers to many vast areas of complicated engineering problems if it comes with the Boussinesq relation. This model is only valid for turbulent flows and have common methods of calculation for viscous heating, compressibility, and buoyancy with other models of $k-\epsilon$. This model has a weak performance for complex flows with high pressure gradient, dissociation and curved line flow but it is appropriate for primary repetition and parametric studies.

2- RNG $k-\epsilon$ model (Renormalized Group)

In this method, the $k-\epsilon$ coefficient model is not gotten by experimental results. They are gotten by analytical method according to the Renormalization Group theory. The results of this model for simple flows in balance turbulence states are like the standard $k-\epsilon$ model but for imbalanced flows specially the flows with recirculation has the results with lower diffusion compared to the standard model. Addition options of this model can help to predict the circular flows and low Reynolds number. This model is appropriate for complex shear flow with rapid strain, average circular movement, eddies and transient local flows.

3- Realizable $k-\epsilon$ model (RKE)

This model has an optimum operation for circular flows, boundary layer flows under intense reverse pressure gradient and dissociation flows.

- k -omega Model:

1- Standard $k-\omega$ model (SKW)

These models are very popular and it has a lot of power when integration equations near the walls. Therefore, there is less need or even no need to use empirical wall functions in them.

In this model two transition equations are solved to calculate k and ω . Special loss rates. (ϵ/k) will be calculated according to the Wilcox relation and it has a good operation in simulation in flows limited to the wall, free shear flows and low Reynolds number flows. This model is appropriate for complex boundary layer flows, flows under reverse pressure gradient and compressible flows. This model has a vast application for aerospace and turbomachine subjects.

2- Shear Stress Transport k- ω model (SST K-W)

This is a type of Standard k- ω model which uses a combined function. Main model Wilcox is used near the walls and Standard k- ϵ model is combined with it for regions far from the walls. Transition and shear options is gotten from Standard k- ω model and no options contained compressibility. This model is more secure and precise for a vast group of engineering flows like reverse pressure gradient flows, airfoils, and transonic shock waves than the standard model.

✓ Reynolds Stress Model (RSM):

The two equations models have very precise answers in a vast group of engineering applications but, have many defects in some cases, so RSM models are used in such cases. In this model the Reynolds Stress is directly solved by transition equations and the isotropic viscosity assumption of the rest models is avoided. It is used for very circular flows and is in need of more time processing and memories and also very hard to converge. This model is appropriate for three dimensional complex flows, strong curved line flows and circular flows.

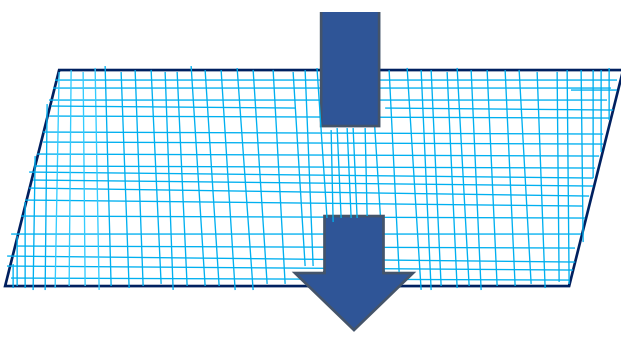
✓ Large Eddy Simulation (LES):

$$u_i(x, t) = \bar{u}_i(x, t) + u'_i(x, t)$$

$\downarrow$
 Instantaneous
component

$\downarrow$
 Resolved
Scale

$\downarrow$
 Sub grid
Scale



Filter, Δ

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial u_i}{\partial x_j} \right)$$

Navier Stokes filtered equations:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{\partial \tau_{ij}}{\partial x_j}$$

$$\tau_{ij} = \rho (\overline{u_i u_j} - \bar{u}_i \bar{u}_j)$$

(Sub grid scale Turbulent stress)

In this method spectrum of turbulent eddies in Navier-Stoke equations is filtered and this filter is a function of grid measure. So, the eddies smaller than grid measure are omitted and are modeled by a

sub grid scale (SGS) and larger eddies are numerically solved by using filtered Navier-Stoke equations. This model is very successful in simulation of combustion- mixing and external aerodynamics.

- Near-wall treatment:

Solving gradients near the walls are very high but, accurate calculations near the wall causes more convergence simulation. For calculating near the wall only one of these options can be chosen:

1- Standard Wall Function

If this model is using, a high Reynolds number of turbulence models such as RKE, SKE or RNG will be applied. Generally, if mixing calculations in the middle of the range of solutions are more important than the wall forces this model is used.

2- Enhanced Wall Treatment

This is a mixture of Law-of-the wall and Two-layer zonal model and is appropriate for low Reynolds number flows or flows with complex phenomenon near the wall. It needs tiny mesh near the wall and is able to solve viscos sub layer.

3- Non- Equilibrium Wall Function

It is very responsive in high pressure gradients and dissociation flows (حیدرزاده, 1394) - (Modeling Turbulent Flows, 2006).

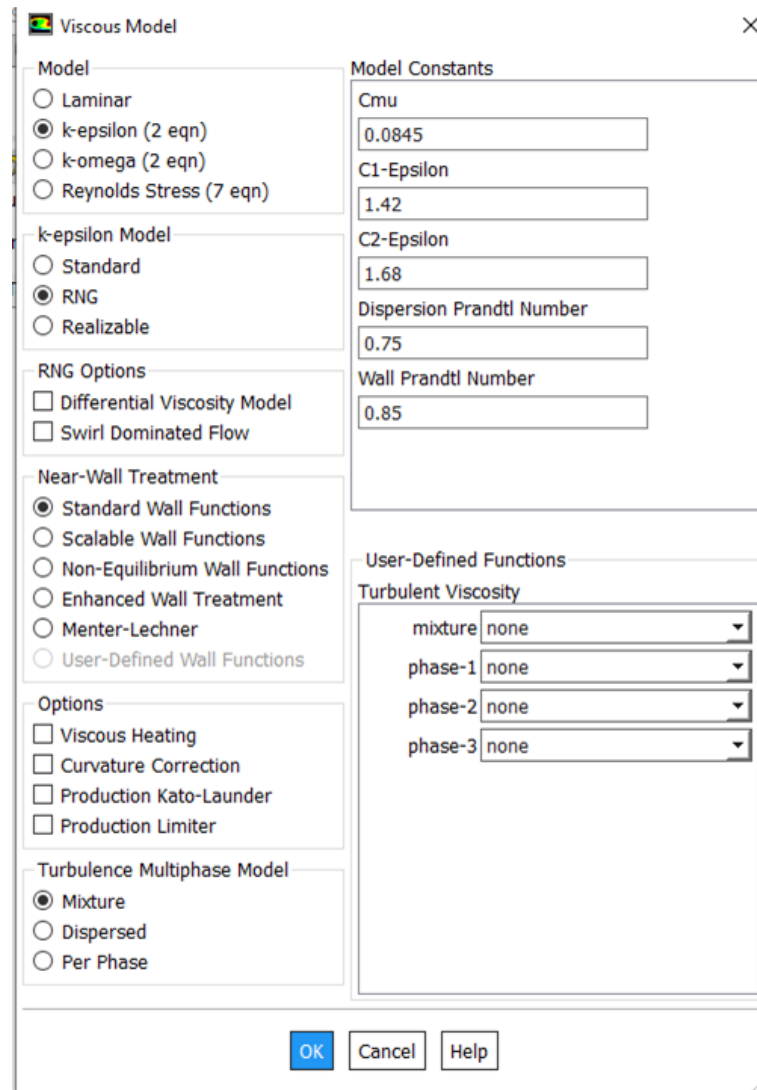


Figure 11

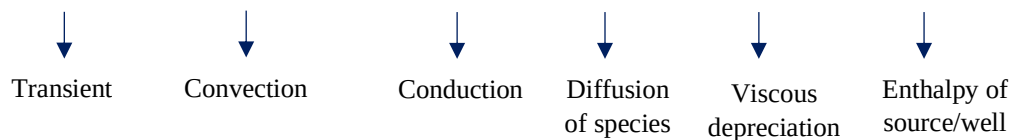
○ Energy:

➤ Heat transfer modeling

Energy equation:

The general energy equation is expressed as follow:

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot [V(\rho E + p)] = \nabla \cdot [k_{eff} \nabla T - \sum_j h_j J_j + \tau_{eff} \cdot V] + S_h$$



1- The energy E per unit mass is determined as follow:

$$E = h - \frac{P}{\rho} + \frac{v^2}{2}$$

In compressible flow or when we choose density-based solver, pressure work and kinetic energy are considered but ,in a pressure-based solver the pressure work and kinetic energy are forgotten. So, when we need them to solve compressible flows, we must add by text command.

Define/models/energy?

Include pressure work in energy equation? Yes

Include kinetic energy in energy equation? Yes

2- The viscous depreciation energy source or viscous heating is expressed as follow:

$$\nabla \cdot \tau_{eff} \cdot V$$

This expression is important when the viscous shear is large in fluid or in high speed compressible flows, but in a pressure-based solver it is neglected and should be performed as needed:

Viscous model → Viscous heating → on

3- The energy source derived from the diffusion of the components, which indicates the effect of enthalpy transfer due to the diffusion of components, is expressed as follow:

$$\nabla \cdot \sum_j h_j J_j$$

It is always considered in a density-based solver, but in a pressure-based solver, if necessary, should be active as follow:

Species Model → Diffusion Energy Source

4- Only on the pressure-based solver, non-isotropic thermal conductivity in the solid state is determined as follow:

$$\nabla \cdot (K_{ij} \nabla T)$$

➤ Natural convection and buoyancy flows

If the added heat to the fluid causes its density change, then natural convection will occur. In natural convection the proportion of Grashof number to Reynolds number will show the importance of the buoyancy force.

If this number is greater than one, buoyancy force must be considered.

$$\frac{Gr}{Re^2} = \frac{g\beta\Delta TL}{\nu^2}$$

In natural convection the buoyancy force will be evaluated by Rayleigh number. In Rayleigh number smaller than 10^8 the flow is laminar and in $10^8 < Ra < 10^{10}$ the flow is transient.

$$Ra = \frac{g\beta\Delta TL^3 \rho}{\mu \alpha}$$

By considering gravity term, pressure gradient and volume force term in momentum equation, we will have:

$$-\frac{\partial p}{\partial x} + \rho g \Rightarrow -\frac{\partial p'}{\partial x} + (\rho - \rho_0)g$$

This pressure transfer will prevent the display of rounding error numbers after activating gravity:

$$p' = p - \rho_0 g x$$

➤ Natural convection-Boussinesq model

In most engineering problems for approximation of density changes by temperature we can use the Boussinesq model which has a high convergence speed. In this the density behavior in all solving equations except in boundary term of momentum equation is fixed.

This model assumes that the fluid density except for volume forces in momentum equation at the direction of gravity, is uniform.

$$(\rho - \rho_0)g = -\rho_0\beta(T - T_0)g$$

In which we have: ρ_0 = Flow density T_0 = Operating temperature β =Thermal expansion coefficient

The estimation of Boussinesq is accurate in this equation:

$$\beta(T - T_0) \ll 1$$

Boussinesq estimation for many natural convection flows in contrast to the use of fluid density as the temperature function gives a better convergence because constant density assumptions reduce non-linearity (دکتر امیر توحیدی - مهندس هادی غفاری قهرودی, 1397).

✓ User inputs for natural convection:

The process of defining buoyancy forces in the simulation of natural convection is as follows:

1: Active heat transfer calculations:

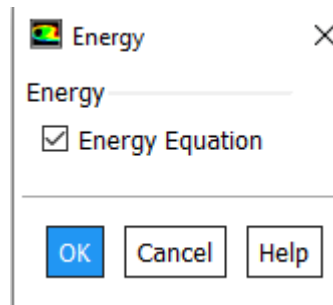


Figure 12

2: Enter operating conditions:

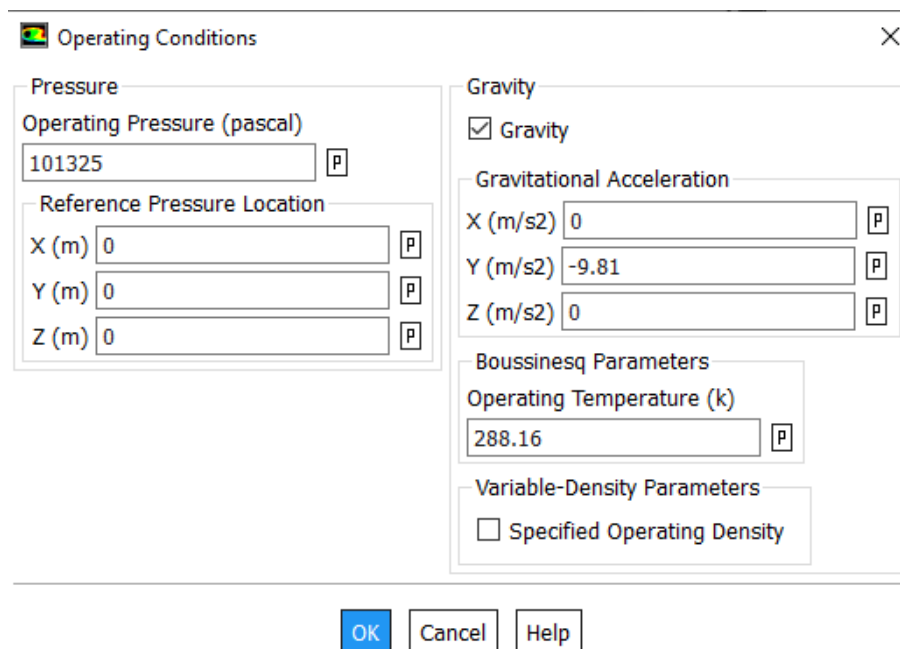


Figure 13

○ Radiation:

For modelling heat transfer by radiation getting the equation of radiative transfer is necessary.

Every input radiation loses some of its intensity because of absorption and scattering in the medium phenomenon and causes that we have a formula for attenuation.

$$(dI)_{loss} = -(a + \mu_s)I ds$$

a is the absorption coefficient [m^{-1}] and μ_s the scattering coefficient [m^{-1}].

Since every material has the property of scattering and emission, a part of input radiation will be augmented in its way, so we have formulas for augmentation in two parts.

It receives the emission of the medium: $(dI)_{em} = aI_b ds$

I_b is the black body intensity.

It also gains energy by scattering: $(dI)_{sca}(s \rightarrow) = \frac{\mu_s}{4\pi} \int_0^{4\pi} I(s \rightarrow') \phi(s \rightarrow', s \rightarrow) d\Omega'$

$s \rightarrow'$ is the scattering direction vector, ϕ the scattering phase function [sr^{-1}] and Ω' a solid angle [sr].

We get Total radiative transfer equation by joining the above three equations:

$$\frac{dI(\vec{r}, s \rightarrow)}{ds} + (a + \mu_s)I(\vec{r}, s \rightarrow) = aI_b(\vec{r}) + \frac{\mu_s}{4\pi} \int_0^{4\pi} I(\vec{r}, s \rightarrow') \phi(s \rightarrow', s \rightarrow) d\Omega'$$

Also, Energy balance equation for thermal radiation propagating in a direction s is:

$$I_\lambda(s + ds, s^\wedge, t + dt) - I_\lambda(s, s^\wedge, t) = \alpha_\lambda I_{b\lambda}(s, t) ds - \alpha_\lambda I_\lambda(s, s^\wedge, t) ds - \mu_{s\lambda} I_{b\lambda}(s, s^\wedge, t) ds + \frac{\mu_{s\lambda}}{4\pi} \int_{4\pi} I_\lambda(s_i^\wedge) \phi_\lambda(s_i^\wedge, s) d\phi_i ds$$

By introducing $c = ds/dt$ one can find

$$\frac{1}{c} \frac{\partial I_\lambda}{\partial t} + \frac{\partial I_\lambda}{\partial S} = \frac{1}{c} \frac{\partial I_\lambda}{\partial t} + s^\wedge \cdot \nabla I_\lambda = \alpha_\lambda I_{b\lambda} - \beta_\lambda I_\lambda + \frac{\mu_{s\lambda}}{4\pi} \int_{4\pi} I_\lambda(s_i^\wedge) \phi_\lambda(s_i^\wedge, s^\wedge) d\phi_i$$

▪ Model:

For modelling radiation in participating media, we couple General energy equation and Total radiation transfer equation (RTE).

1- Surface to Surface (S2S)

The simplest model in Fluent for radiation heat transfer is S2S model which only considers surface to surface radiation without any absorption, emission or scattering. It is only suitable for radiative transfer problems without participating media such as CO₂, H₂O steam and ...

In this model the optical thickness is zero and the solving method is based on View factor calculation.

Advantages:

- ✓ Low time per iteration because of view factor method.
- ✓ It is more accurate in cases of localized heat sources than DO or any ray tracing method.
- ✓ It has smaller memory usage.

Disadvantages:

- ✓ In the S2S model all the surfaces are assumed to be diffused.
- ✓ It only assumes gray radiation.
- ✓ As the number of surfaces increase, the required storage increases very rapidly.
- ✓ We cannot use it in model participation radiation problems.
- ✓ It cannot be used in symmetry or periodicity boundary conditions.
- ✓ It is not a conservative model.

2- Discrete Transfer (DTRM)

Another simple model is Discrete Transfer Radiation Model (DTRM) which considers only absorption and emission phenomena without designation of scattering.

Main assumption: This model considers all the radiations over a certain range of solid angles only as a single ray.

$$\frac{dI}{ds} = -\alpha I + \frac{\alpha \sigma T^4}{\pi}$$

$$I(s) = \frac{\sigma T^4}{\pi} (1 - e^{-\alpha s}) + I_0 e^{-\alpha s}$$

Advantages:

- ✓ It is a simple model.
- ✓ It is applicable over a wide range of optical thicknesses.
- ✓ Increasing the number of rays can lead to more accuracy.

Disadvantages:

- ✓ Without particle/radiation interaction.
- ✓ Without considering scattering.
- ✓ Only applicable for diffuse surfaces.
- ✓ It only assumes gray radiation.
- ✓ It is not appropriate for dynamic mesh.
- ✓ Accountable for optically thin media.
- ✓ It is not a conservative model.

3- P1

The next model is P-1 which is the simplification of P-N model. In this model the intensity in a participating medium can be shown as a rapidly converging series whose terms are based on orthogonal spherical harmonics. The P-1 model in Fluent is a four-term abbreviation of P-N model.

Main assumption: We have a diffusion equation for incident radiation which is the result of the integration of the directional dependence in RTE.

This model solves a simple diffusion equation for the incident radiation which is the sum of all radiative intensity in all directions.

$$\frac{\partial}{\partial x_i} \left(\Gamma \frac{\partial G}{\partial x_i} \right) + 4\epsilon\sigma T^4 = aG$$

Also, by altering the diffusivity we can model the scattering effects:

$$\Gamma = - \frac{1}{3(a + \sigma_s) - C\sigma_s}$$

C: Linear-anisotropic phase function coefficient ($-1 < C < 1$).

Radiation flux, q_i , is then:

$$q_i = -\Gamma \frac{\partial G}{\partial x_i}$$

Advantages:

- ✓ This is a simple, cheap model.
- ✓ This conservative model allows particulate scattering.
- ✓ Assume the effect of particles, droplets and soot.
- ✓ It can use in complex geometry.

Disadvantages:

- ✓ It is appropriate for optically thick media and it has less accuracy if the optical thickness is small.
- ✓ This model is not accurate for localized heat sources.
- ✓ It is applicable for diffuse wall surfaces.
- ✓ The implementation assumes gray radiation.

4- Rosseland

The next model which is simplest than P-1 is the ROSSELAND model. In this model we assume a theoretical value assigned by the blackbody expression without solving an equation for G (the irradiance). The other properties of it is just like the P-1 model, but our equation is simpler than P-1.

Radiation intensity = the black body intensity at the gas temperature.

$$G = 4n^2\sigma T^4$$

The radiative heat flux diffuses due to high optical thickness:

$$q_i = -\Gamma \frac{\partial G}{\partial x_i}$$

By joining these equations, we get a simple equation for the local radiative heat flux related to local temperature:

$$q_r = -16n^2\sigma T^3 \frac{\partial T}{\partial x_i}$$

Advantages of this model over the P-1 model:

- ✓ ROSSELAND model is faster than the P-1 and needs less memory.

Disadvantages over the P-1 model:

- ✓ It is appropriate for media with very large optical thickness. ($aL > 3$).

5- Discrete Ordinates (DO)

The most accurate model is the Discrete ordinates (DO) model. This model solves the equation of radiative transfer for a set of discrete directions. This method is a finite differencing of directional dependence radiative transfer equation. This model converts the equation of radiative transfer into a transport equation for radiation intensity.

$$\nabla \cdot [I(\vec{r}, \vec{s})\vec{s}] + (a + \mu_s)I(\vec{r}, \vec{s}) = aI_b(\vec{r}) + \frac{\mu_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}')\phi(\vec{s}', \vec{s})d\Omega'$$

For the DO model in Fluent, we have 2 accomplishments. In the Uncoupled one, the equation for energy and radiation intensities can be solved one by one but the Coupled approach solves the discrete energy and intensity equation simultaneously at each cell.

Advantages:

- ✓ It is usable to all optical thicknesses.
- ✓ It is appropriate for particulate and anisotropic scattering.
- ✓ It is suitable for modeling radiation in semi-transparent media.
- ✓ It is a model of non-gray banded radiation.
- ✓ It is assumed both diffuse and specular reflection.
- ✓ This is the most conservative model.
- ✓ We can increase accuracy by using a finer discretization.

Disadvantages:

- ✓ This is a computationally expensive model.

- Solar load model:

For considering solar radiation effects in the limit of calculations we can use solar load model. There are two options for modeling of this:

- 1- Solar Ray Tracing
- 2- DO Irradiation

Solar Ray Tracing model has high efficiency and in which the solar load will be considered as a heat source in energy equation. This algorithm for transferring solar radiation energy is compatible with all radiation models and for determining its specification, the Sun Direction Vector and Illumination Parameters which contain Direct Solar Irradiation(W/m^2) and Diffuse Solar Irradiation (W/m^2) must be specified.

As in simulation we used P1 model so in the part of Create/Edit materials, the quantities Absorption coefficient ($1/\text{m}$) and Scattering coefficient ($1/\text{m}$) and also Refractive Index of the fluid must be given.

Also, since in modeling we used radiation so in the problem, extra boundary condition must be regulated in the inlet and outlet of the walls. In inlet and outlet boundaries emissivity must be refined proportional to the boundary and the presupposed number one has been considered for it (BELEY, 2016)- (Modeling Radiation, 2013).

Radiation Model [X]

Model

☐ Off

☐ Rosseland

☒ P1

☐ Discrete Transfer (DTRM)

☐ Surface to Surface (S2S)

☐ Discrete Ordinates (DO)

Non-Gray Model

Number of Bands

Solar Load

Model

☐ Off

☒ Solar Ray Tracing

☐ DO Irradiation

Update Parameters

Time Steps per Solar Load Update

Sun Direction Vector

X Y Z

☐ Use Direction Computed from Solar Calculator

Illumination Parameters

Direct Solar Irradiation (w/m2)

Diffuse Solar Irradiation (w/m2)

Spectral Fraction $[V/(V+IR)]$

Figure 14

Create/Edit Materials

Name: water-vapor
 Chemical Formula: h2o

Material Type: fluid
 Fluent Fluid Materials: water-vapor (h2o)
 Mixture: none

Order Materials by:
☒ Name
☐ Chemical Formula

Fluent Database...
 User-Defined Database...

Properties

Density (kg/m3)	boussinesq	Edit...
	0.5544	
Cp (Specific Heat) (J/kg-K)	constant	Edit...
	2014	
Thermal Conductivity (W/m-K)	constant	Edit...
	0.0261	
Viscosity (kg/m-s)	constant	Edit...
	1.34e-05	
Molecular Weight (kg/kmol)	constant	Edit...
	18.01534	
Standard State Enthalpy (J/kgmol)	constant	Edit...
	-2.418379e+08	
Reference Temperature (K)	constant	Edit...
	300	
Thermal Expansion Coefficient (1/K)	constant	Edit...
	0.0033	

Change/Create Delete Close Help

Figure 15

- **Boundary Conditions:**

- 1- Velocity inlet:

This boundary condition is recommended for incompressible flows with Mach numbers lower than 1 and the only inlet of it is the velocity quantity.

- 2- Pressure inlet:

This boundary condition is recommended for both compressible and incompressible flows in which the inlet pressure is specified. Solver for stability and precision of solutions receives the presupposed total temperature and pressure.

In simulation of the flows within the channels, the total pressure is calculated by this formula:

Incompressible flows:

$$\rho_{Total} = \rho_{Static} + \frac{\rho V^2}{2}$$

Compressible flows:

$$\rho_{Total,abs} = \rho_{Static,abs} \left(1 + \frac{k-1}{2} M^2\right)^{\frac{k}{k-1}}$$

$$T_{Total,abs} = T_{Static,abs} \left(1 + \frac{k-1}{2} M^2\right)$$

3- Mass flow inlet:

This boundary condition is recommended more for compressible flows, but the use of it for incompressible flows is acceptable, and in which the mass flow rate is regulated according to the optimum quantity in the physical problem.

Supersonic/ Initial pressure quantity is applied in two cases:

- ✓ When the flow is momentary supersonic.
- ✓ The regulated pressure is considered as the initialization of the static pressure in all the solution range.

4- Pressure outlet:

This boundary is applied for the simulation of both compressible and incompressible flows. In which the regulated quantity of relative pressure is equal to the static pressure of the environment that the flow is evacuated.

Generally, in incompressible flows that the inlet velocity is regulated in them and the goal is to calculate velocity fields and pressure gradient, the quantity of outlet pressure is not important and we can regulate it arbitrary. However, if the properties of the fluid are a function of pressure like the ideal gas and the depended reaction to the pressure like cavitation then the outlet pressure quantity must be regulated precisely. In compressible flows density is a function of pressure so the regulated outlet pressure is very important. By adjustment of outlet flow rate in Target Mass flow rate the solver automatically change the outlet pressure quantity to adjust the ideal flow rate. The Radial Equilibrium Distribution option is applied for simulation of turbomachine.

5- Outflow:

It's only applied in incompressible simulations and in conditions that the inlet flow quantity is not a function of pressure, and in which all the quantities like the outlet pressure in the solution range is extrapolated so, it is only applied in fully developed flows. Generally, this boundary is the least desirable outlet boundary of the solver and the use of it is not recommended in any simulation.

6- Outlet-Vent:

This boundary is completely similar to the pressure outlet boundary condition, but in which a loss of pressure is considered on the outlet boundary. It is applied in simulation of the flows in which an equipment of pressure loss like a filter is installed on their outlet.

7- Exhaust-Fan:

This boundary is completely similar to the pressure outlet boundary condition, but in which a boosting pressure is considered on the outlet boundary. It is applied in simulation of the flows in which a boosting pressure equipment like a fan or a compressor is installed on their outlet.

8- Intake-Fan:

This boundary is completely similar to the pressure inlet boundary condition, but in which a boosting pressure is considered on the inlet boundary.

9- Inlet-Vent:

This boundary is completely similar to the pressure inlet boundary condition, but in which a loss of pressure is considered on the inlet boundary.

10- Wall:

The wall boundary condition is applied in two cases:

- ✓ The wall with movement which is used as a replacement for symmetry boundary condition.
- ✓ Wall without movement which is applied in different conditions.

This boundary hydrodynamically don't need any specific settings and the presuppositions are acceptable in them and only usable hydrodynamical adjustment is related to the simulation of the moving boundaries and roughness height in turbulence flows. The wall can be simulated thermal in

five ways and actually these applied boundary conditions on the wall can be done out of the solution range and at the place where simulation is not done.

- Heat flux

In this condition we can do the supposed heat flux on the wall.

- Temperature

In this condition the temperature of the wall is supposed fixed during the simulation.

- Convection

In this condition it is supposed that an outlet flow can do convection with the wall boundary.

- Radiation

In this condition it is supposed that an exterior radiation with specific temperature is radiated to the wall. The necessary quantities for this boundary are External Radiation Temperature and External Emissivity. If the radiation models in simulation are active then Internal Emissivity is also specified.

- Mixed

In this condition it is supposed to do a combination of radiation and convection on the wall.

- Via System Coupling

In this condition we can use the wall thermal boundary condition via simulation in another ANSYS solver and transfer the calculated boundary condition to another solver.

11- Symmetry:

In conditions that the physical simulation has geometric and flow symmetry we can simulate half of it and consider the effect of the other half by this condition (محمدجیدی)- (Boundary Conditions , 2006).

- ❖ In this project:

- Boundary condition for inlet water:

In this section water enter the solar still proportional to the surface evaporation to fix the water level. We use the mass flow inlet boundary condition for this and we must specify the mass flow inlet for water phase. We must also specify the Inlet Emissivity.

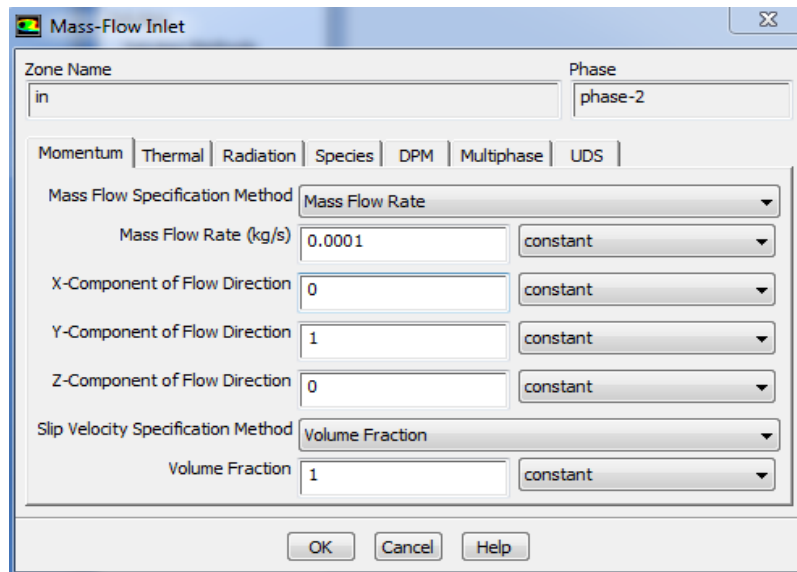


Figure 16

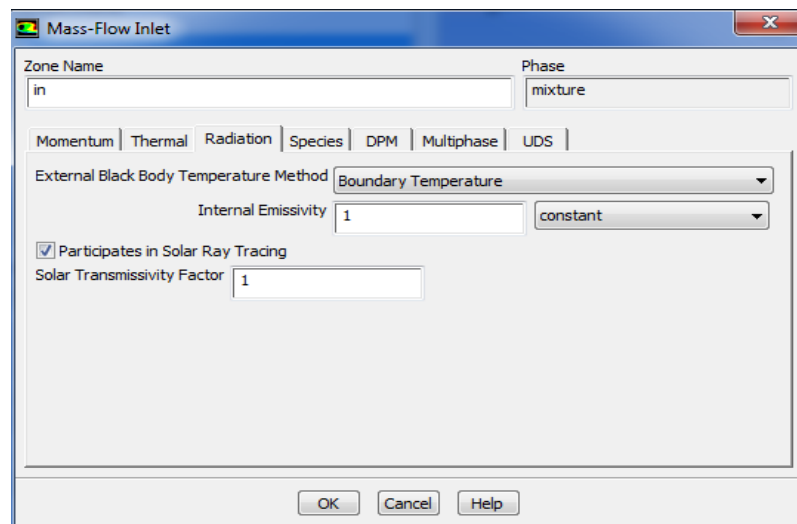
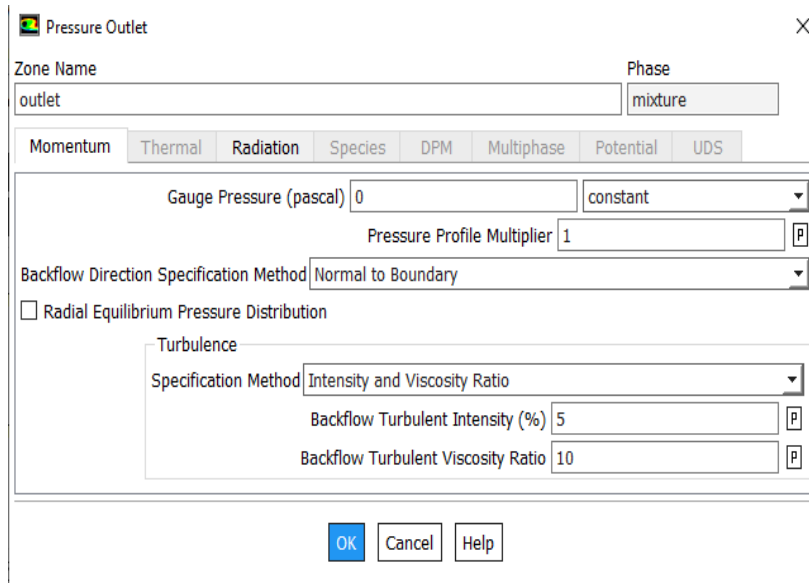


Figure 17

- Outlet boundary condition:

In this section we use pressure outlet boundary condition because at the output of the problem no physical characteristic except pressure is specified.



Pressure Outlet

Zone Name: outlet Phase: mixture

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Gauge Pressure (pascal): 0 constant

Pressure Profile Multiplier: 1

Backflow Direction Specification Method: Normal to Boundary

☐ Radial Equilibrium Pressure Distribution

Turbulence

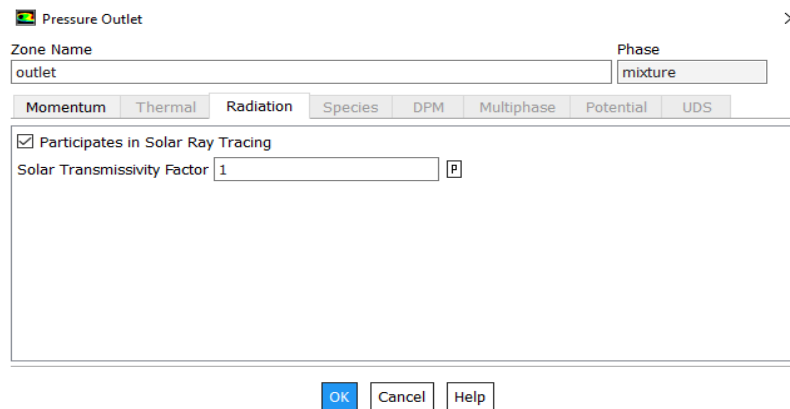
Specification Method: Intensity and Viscosity Ratio

Backflow Turbulent Intensity (%): 5

Backflow Turbulent Viscosity Ratio: 10

OK Cancel Help

Figure 18



Pressure Outlet

Zone Name: outlet Phase: mixture

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

☒ Participates in Solar Ray Tracing

Solar Transmissivity Factor: 1

OK Cancel Help

Figure 19

- Boundary condition for absorber:

When solar radiation is radiated to the absorber plate, a part of it is absorbed which must be entered by considering solar absorption coefficient. Also, the part of participates in solar ray tracing is activated to consider solar radiation on the plate.

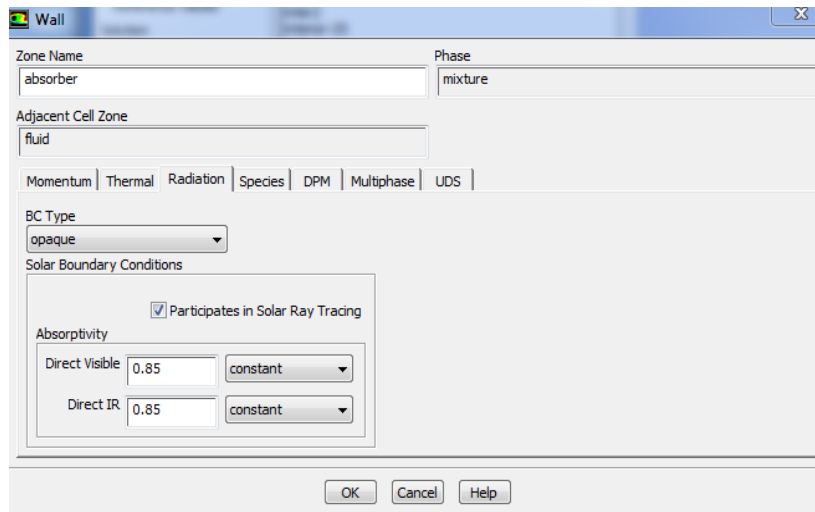


Figure 20

- Boundary condition for reflective plates

These plates are placed across the solar still to reflect solar radiation to the water of the still to increase the temperature of water. This reflection is produced by activating participates in solar ray tracing and considering low solar absorption coefficient. When the plates are opaque and absorption coefficient is low, the solver calculates the reflective coefficient by subtracting the absorption coefficient quantity from number one.

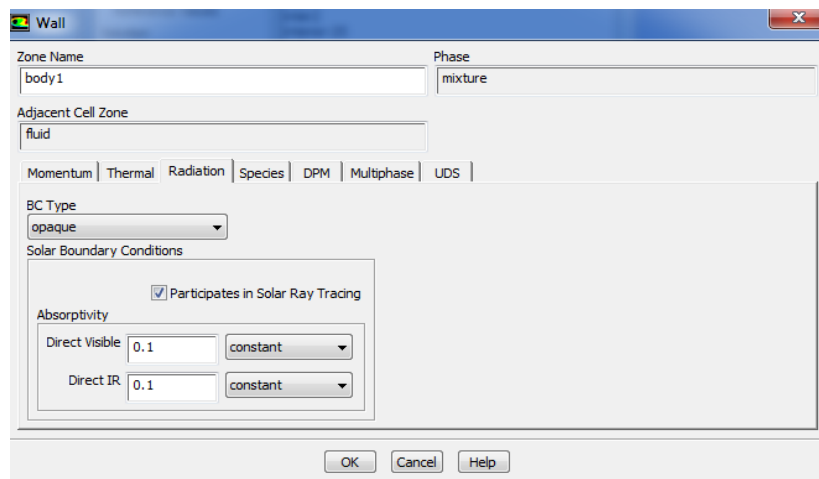


Figure 21

- Boundary condition for glass

Glass acts as a semi-transparent media and transfers most of the solar radiation and only absorbs and reflects a small part. For this condition we must activate the participates in solar ray tracing and then use the semi-transparent option. We must enter the absorption and transmission coefficients.

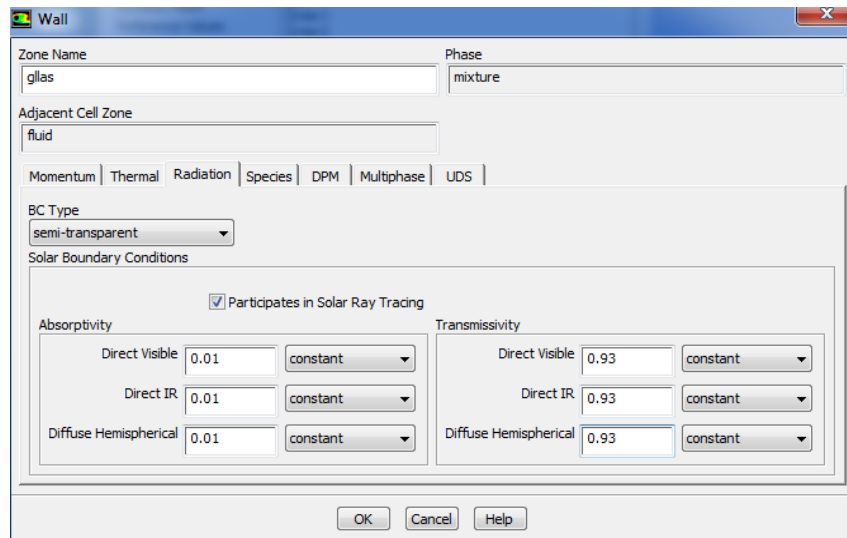


Figure 22

- Solution:
 - Methods:
 - Pressure-Velocity Coupling:

The purpose of pressure- velocity coupling is a numerical algorithm for getting an equation for pressure, using combining of continuity and momentum equation.

1- Semi-Implicit Method for Pressure-Linked Equations (SIMPLE)

In this method the relation between velocity and pressure is corrected to verify the conservation of mass and the pressure field is calculated.

The momentum equation is solved by guessing pressure field p^* . So, the plane flux J_f^* is equal to:

$$J_f^* = \hat{J}_f + d_f(p_{c0}^* - p_{c1}^*)$$

The continuity equation is not solved so we must add the correction factor J'_f to the face flux J_f^* . So, we have:

$$J_f = J_f^* + J'_f$$

J'_f in SIMPLE algorithm is written:

$$J'_f = d_f(p'_{c0} - p'_{c1})$$

p' is the cell pressure correction.

SIMPLE algorithm replaces the flux correction equations in continuity equations to get the pressure correction p' in the cell:

$$a_p p' = \sum_{nb} a_{nb} p'_{nb} + b$$

b is the inlet mass flow rate to cell.

$$b = \sum_f^{N_{faces}} J_f^* A_f$$

The pressure correction equation may be solved by the algebraic multigrid (AMG) method. The cell pressure and the face flux can be corrected by this equation:

$$p = p^* + \alpha_p p'$$

$$J'_f = J_f^* + d_f(p'_{c0} - p'_{c1})$$

α_p is the under – relaxation factor for pressure.

2- SIMPLE-Consistent (SIMPLEC)

In simple problems like laminar flow without specific physical model it can provide faster convergence.

3- Pressure-Implicit with Splitting of Operators (PISO)

This method has a predictor step and two corrector steps. It is suitable for transient compressible flows. This algorithm uses a more CPU time for every iteration, but it can decrease the number of iterations required for convergence. This algorithm is recommended in transient problems specially when the step time is large.

4- Coupled

This method is appropriate for steady-state single phase flows but is not recommended for Eulerian multiphase flows, the none-iterative time advancement solver (NITA) and periodic mass flow boundary conditions (Pressure-Velocity Coupling, 2009), (Deshpande, 2012).

- Gradient:

The gradient of solver variables is needed for calculating diffusion terms, velocity derivatives and higher order discretization.

$$V_{\text{cell}} \frac{\partial(\rho\varphi)}{\partial t} + \sum_f^{N_{\text{faces}}} \rho_f V_f \varphi_f \cdot A_f = \sum_f^{N_{\text{faces}}} \alpha_f \nabla \varphi_f \cdot A_f + S_\varphi V_{\text{cell}}$$

We can calculate the gradient of solver variables by three methods:

1- Green-Gauss Cell-Based

This is an ANSYS FLUENT presupposition method, but can cause wrong diffusion in flows and disturb flow fields.

2- Green-Gauss Node-Based

This is a more accurate method and can minimize the wrong diffusions.

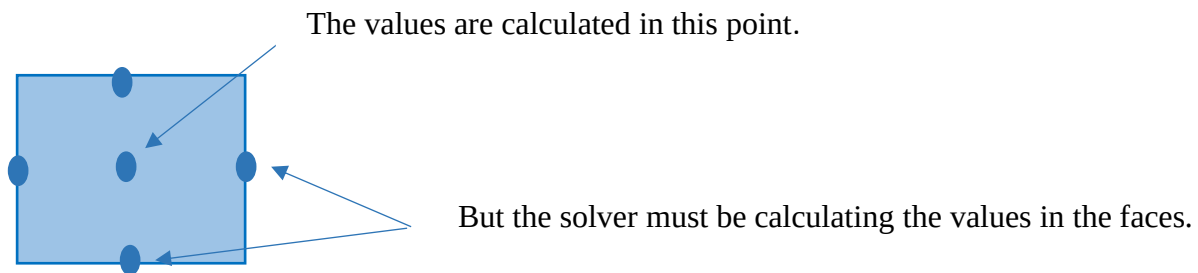
3- Least-Squares Cell Based

The precision of this method is the same as the Green-Gauss Node-Based method and its qualities are the same as the Node-Base Gradient but it needs less calculation time, so it's a more rapid method (Evaluation of Gradients and Derivatives, 2006).

- Interpolation:

In ANSYS FLUENT the solver variables will be solved at the center of controlled volumes. According to presupposition the separated values of scalar φ will be saved in the centers of cells but for getting transition terms in Navier-Stokes equation, we need that the φ_f values be on the planes and this is done by upwind interpolation method. Actually, field variables should be interpolation according to controlled volumes faces.

$$V_{\text{cell}} \frac{\partial(\rho\varphi)}{\partial t} + \sum_f^{N_{\text{faces}}} \rho_f V_f \varphi_f \cdot A_f = \sum_f^{N_{\text{faces}}} \alpha_f \nabla \varphi_f \cdot A_f + S_\varphi V_{\text{cell}}$$



The interpolation methods for transition terms in transfer equation are:

1- First-Order Upwind

In this method the values of φ_f at the face is equal to cell centered value in the cell upstream of the face. It has a first order accuracy and in relation to other interpolation methods, it converged more simply.

2- Power Law

For getting the quantity values in planes, it uses exact solution. For flows with Reynolds number down five, it is more accurate than the first order model.

3- Second- Order Upwind

For getting the quantity values in planes, it uses multidimensional linear method. It has the second order accuracy and more appropriate for tri/tet mesh or when flow is not aligned with grid.

4- First-to-Higher Order Blending

For getting better accuracy it uses separation blending factor method. A number between zero to one must be selected. The zero number will proceed to the first-order and the one number to the higher order discretion. When we use higher order methods, we increase the accuracy of the system, but the convergent will be harder.

5- Central- Differencing Scheme

In this method the values of φ_f at the face is determined by linear interpolation between the cell centered values. It has the second order accuracy and will be used in LES turbulence models.

6- Monotone Upstream-centered Schemes for Conservation Laws (MUSCL)

It has the third-order accuracy and is appropriate for unstructured meshes and also for secondary and eddy flows.

7- Quadratic Upwind Interpolation (QUICK)

It is appropriate for quad/hex and hybrid meshes and also for rotating/swirling flows (Bakker), (Introduction to ANSYS FLUENT , 2010).

Solution Methods

Pressure-Velocity Coupling

Scheme
Phase Coupled SIMPLE

☐ Solve N-Phase Volume Fraction Equations

Spatial Discretization

Gradient
Least Squares Cell Based

Momentum
Second Order Upwind

Volume Fraction
First Order Upwind

Turbulent Kinetic Energy
Second Order Upwind

Turbulent Dissipation Rate
Second Order Upwind

Transient Formulation
Second Order Implicit

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation Options...

Figure 23

- Controls:

Relaxation Factor:

In pressure-based solver a relaxation factor is used for stability of repetitive process. At first, we begin the calculations with the presupposition of relaxation factors by the solver but if we saw that the residuals are increased after 4 or 5 repetitions, then we must decrease the relaxation factors. Usually the increase of relaxation factors can cause a little increase in residuals and this action has no influence on solver process. However, when the residuals are increased abruptly, we must, stop the process and decrease relaxation factors (Introduction to ANSYS FLUENT , 2010).

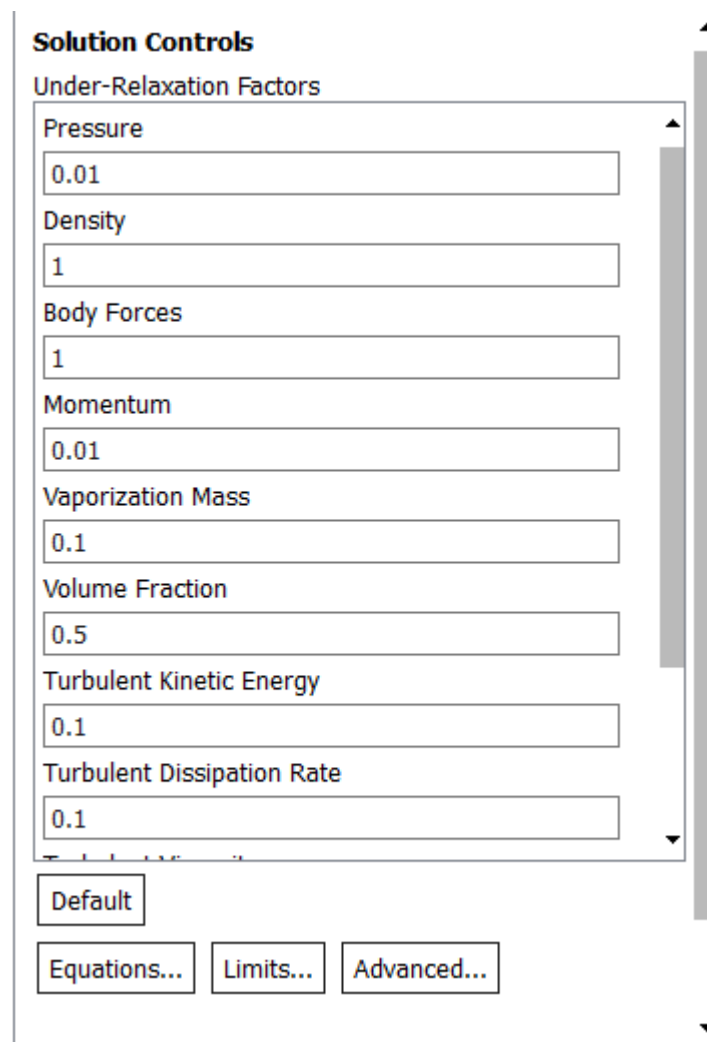


Figure 24

- Initialization:

In ANSYS FLUENT the solver acts in an iterative manner, so at first a given value must exist for every quantity in every grid cell and this amount is named solution initialization.

This initialization must be actual and reasonable to have a better convergence.

Two methods for initialization:

1- Hybrid Initialization

This method can give a rapid approximation of the flow field. In this method the Laplace equation can be solved for determination of pressure and velocity fields. All the other variables like temperature, turbulence, mass fraction, volume fraction, ... can be joined together automatically according to average quantities of solver region or by the special interpolation method.

2- Standard initialization

This method is a presupposition of ANSYS FLUENT solver and according to it a separate quantity is allotted to every variable in the domain (دکتر امیر توحیدی - مهندس هادی غفاری قهرودی, 1397).

Gauge pressure(pascal) :0

Turbulent Kinetic Energy(m²/s²):1

Turbulent Dissipation Rate(m²/s³):1

Phase-1 X Velocity(m/s):0

Phase-2 X Velocity(m/s):0

Phase-1 Y Velocity(m/s):0

Phase-2 Y Velocity(m/s):0

Phase-1 Z Velocity(m/s):0

Phase-2 Z Velocity(m/s):0

Phase-1 Temperature(K):300

Pase-2 Volume Fraction:0

Phase-3 X Velocity(m/s):0

Phase-2 Temperature(K):300

Phase-3 Y Velocity(m/s):0

Phase-3 Z Velocity(m/s):0

Pase-3 Volume Fraction:0

Phase-3 Temperature(K):300

Solution conditions:

Solver	Pressure based
Formulation	Implicit
Turbulent model	k-e (RNG)
Model	Elurean
Eulerian parameter	Evaporation- Condensation
Time	Unsteady
Pressure- velocity coupling	Simple
Velocity formulation	Absolute
Wall function	Standard

Table 1

• **Results and Discussion:**

Simulation considered from 8:00h until 16:00h.

- ✓ Sea water level:5 cm
- ✓ Solar radiation= 1000W/m^2
- Graphics: Contours of volume fraction

Volume fraction of water:

8:00

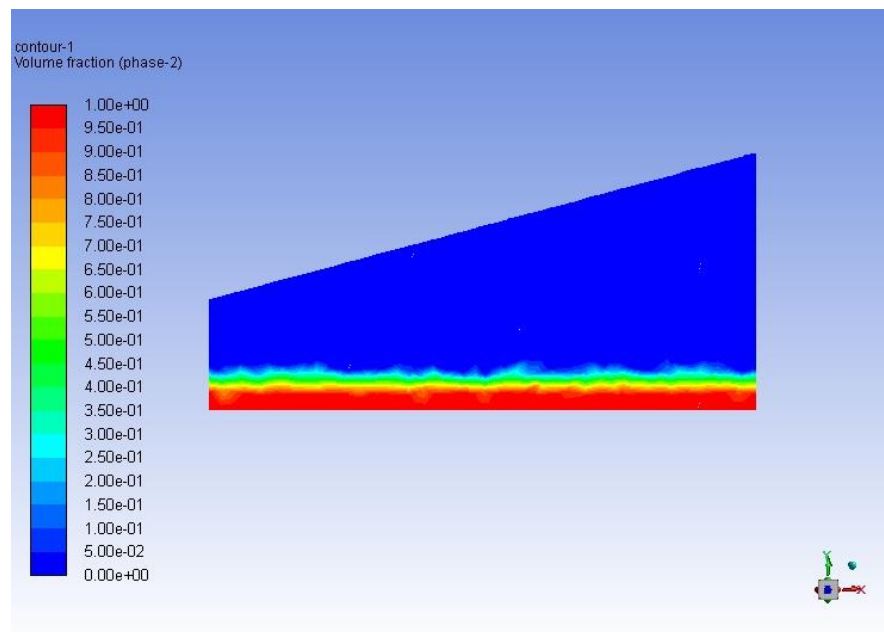


Figure 25

10:00

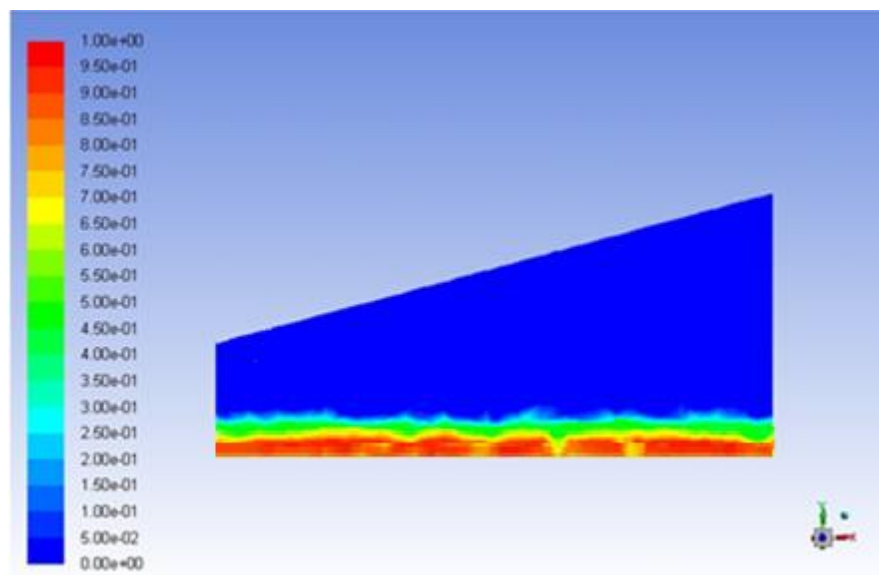


Figure 26

12:00

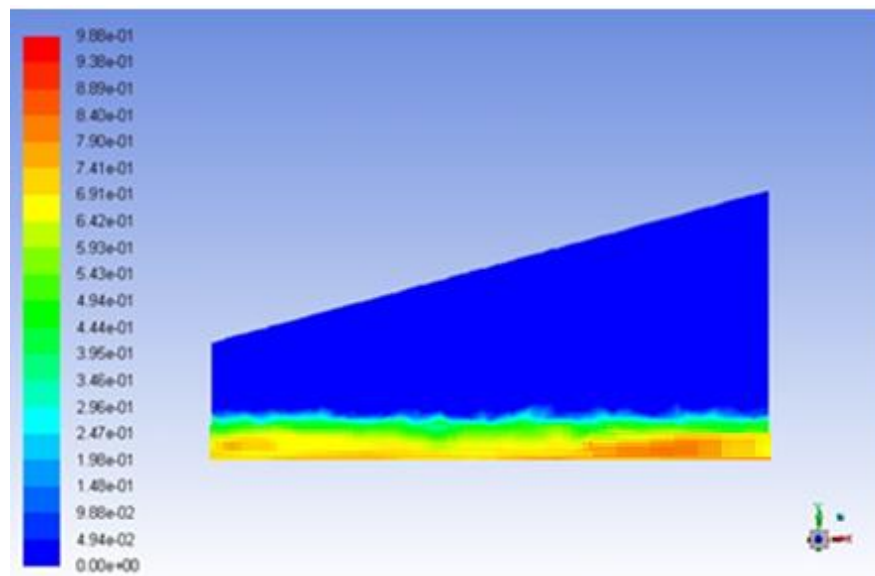


Figure 27

14:00

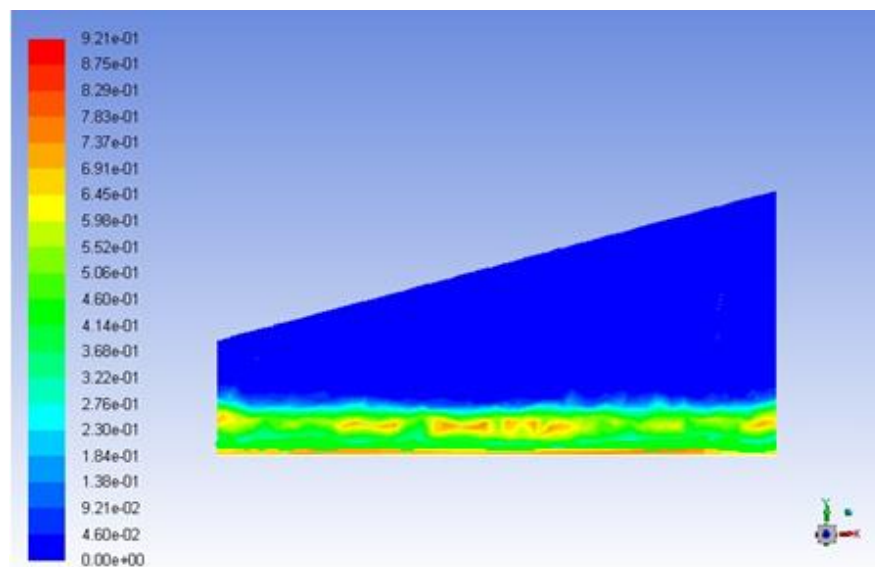


Figure 28

16:00

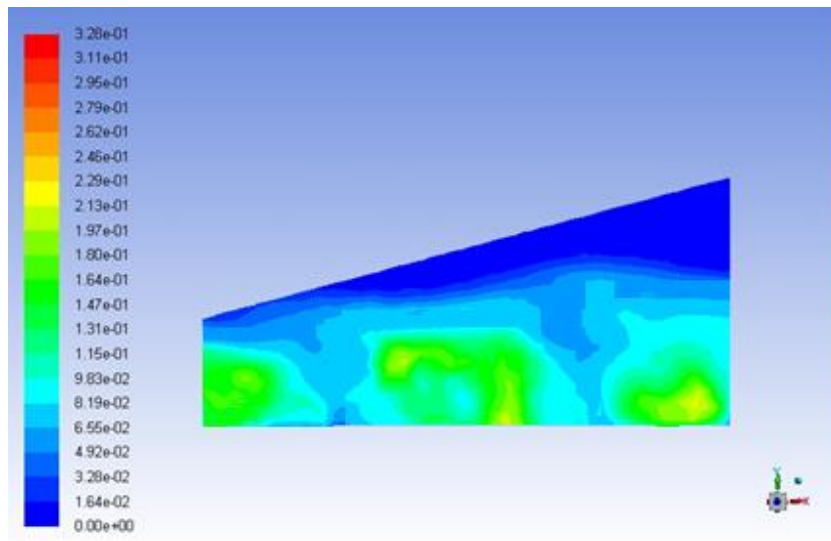


Figure 29

Volume fraction of vapor:

8:00

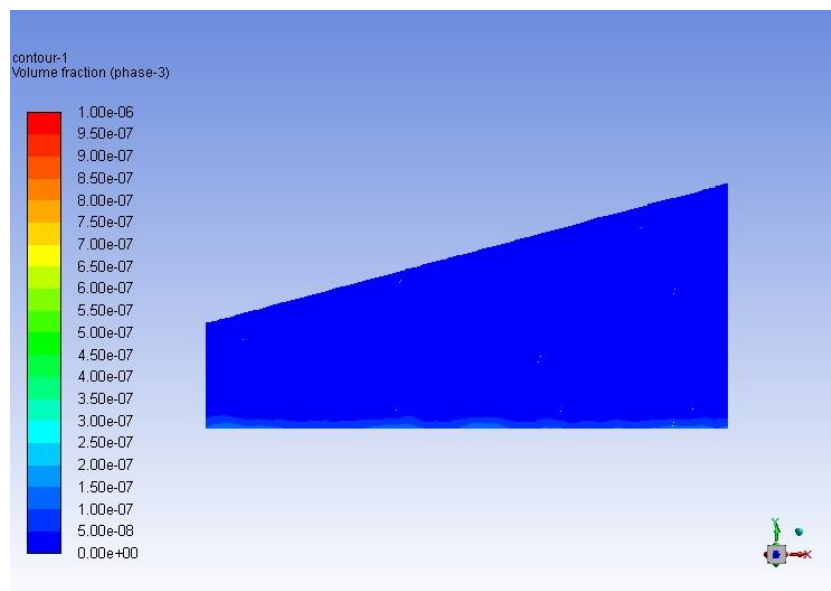


Figure 30

10:00

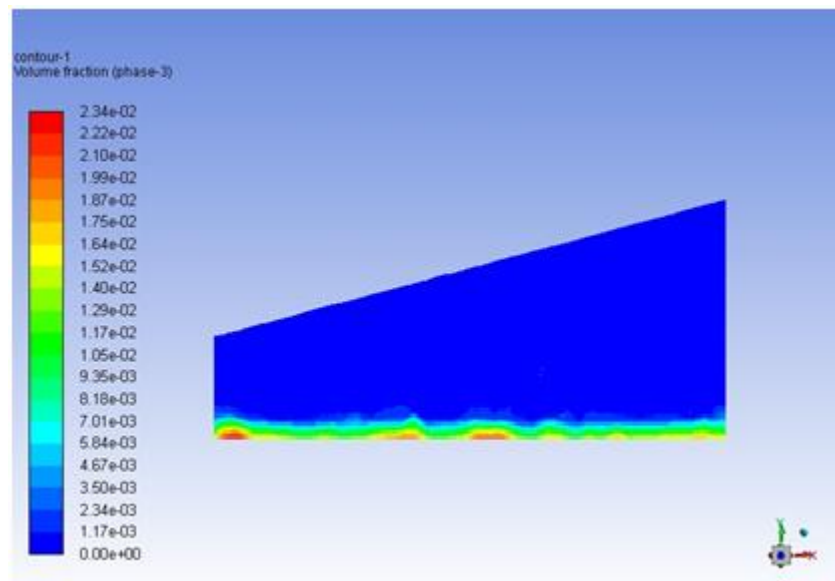


Figure 31

12:00

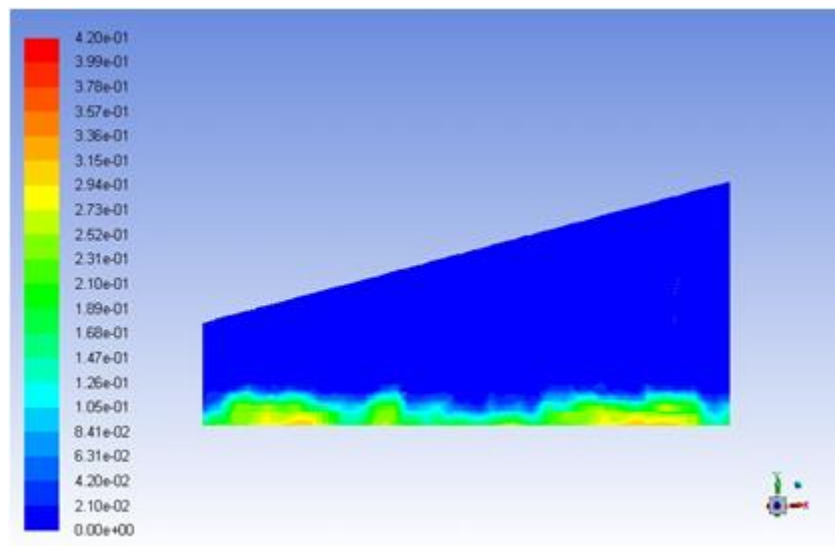


Figure 32

14:00

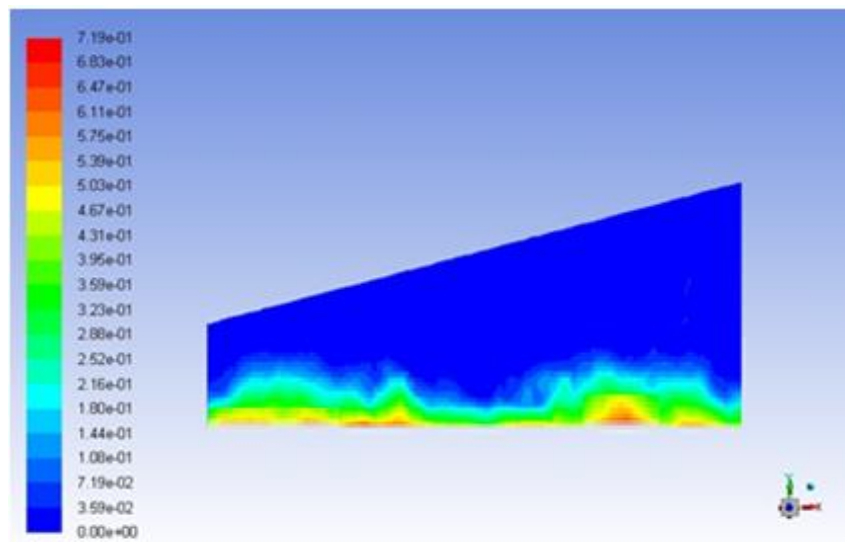


Figure 33

16:00

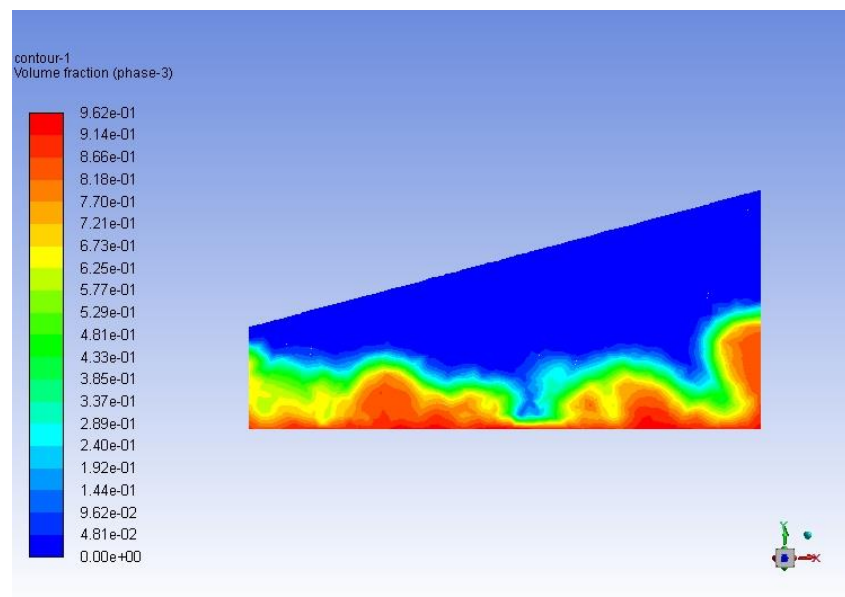


Figure 34

Temperature of water and glass:

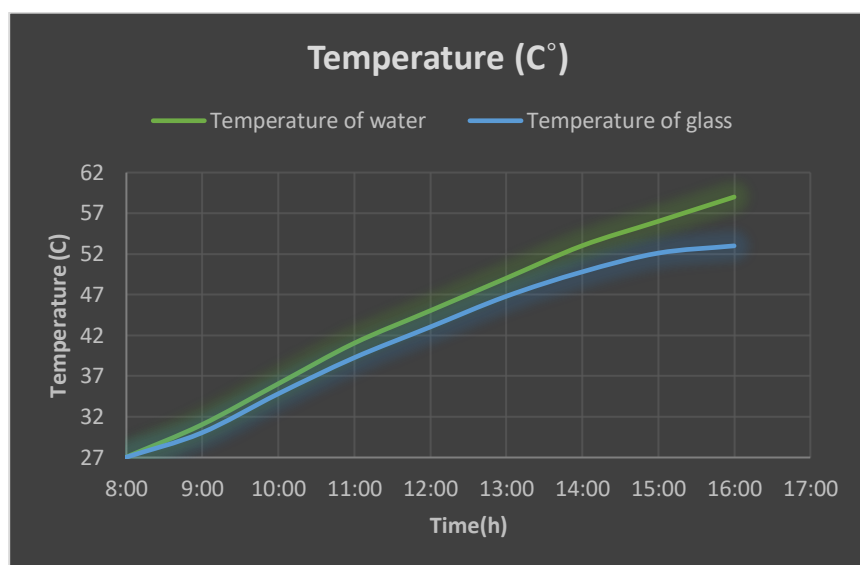


Figure 35

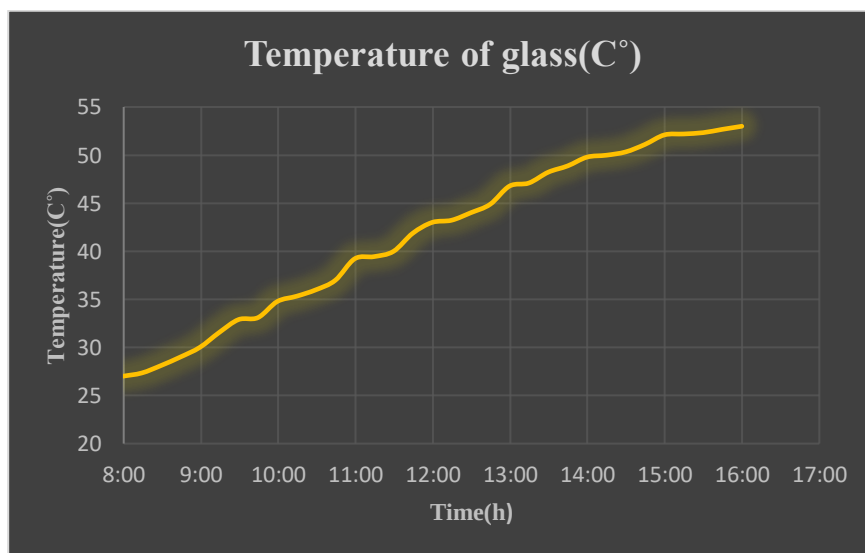


Figure 36

Evaporative heat transfer coefficient between the water mass and the glass cover:

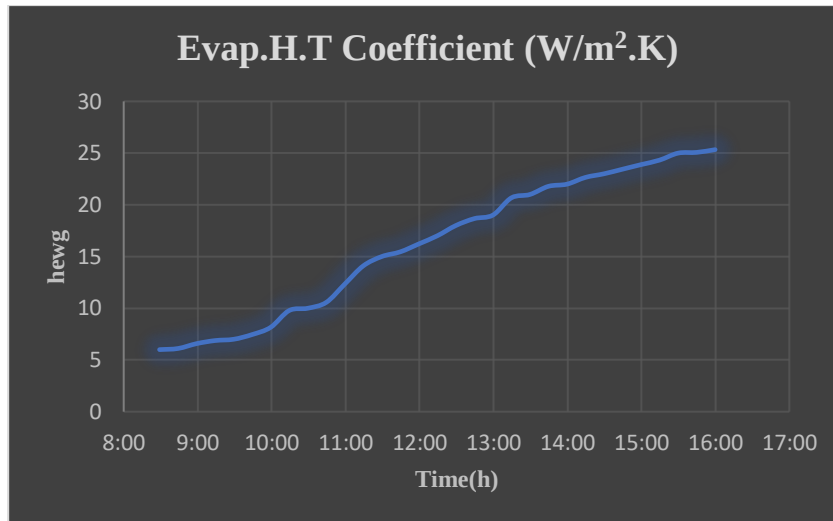


Figure 37

- Report Definitions: Flux Report → Mass Flow Rate

Productivity of solar desalination system:

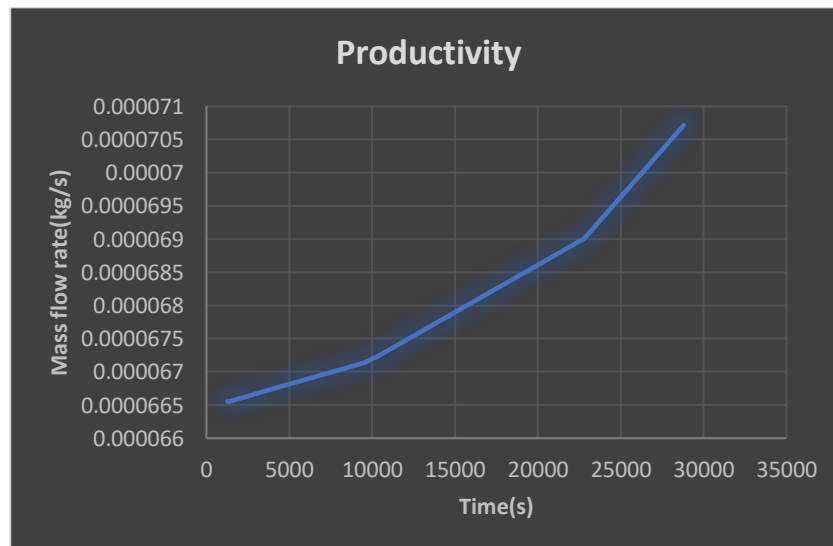


Figure 38

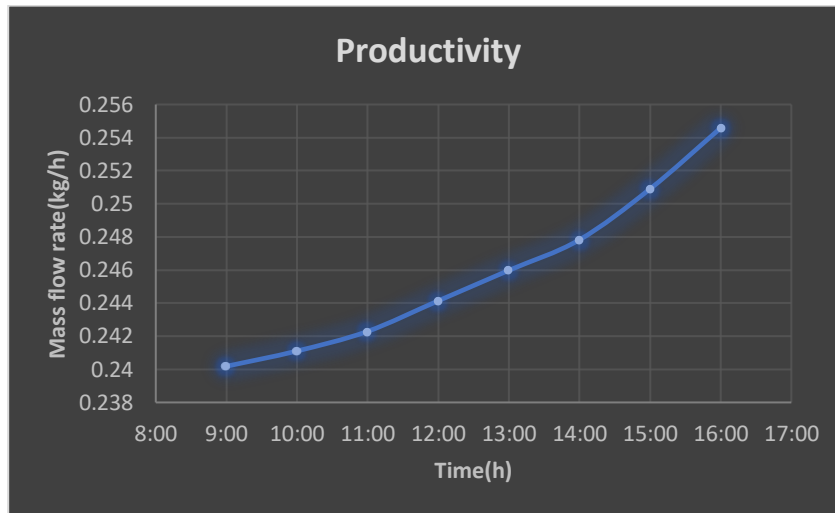


Figure 39

According to the below graph if the solar radiation intensity is 1000W/m^2 , the productivity of the system will be about $0.278\text{ kg/m}^2\cdot\text{hr}$. This simulation is for the water height of 4cm. However, in our simulation the height of water is 5cm. On the other side we know that if we reduce the height of water 60%, the productivity of the system will be increased by 26%. (OO, 2009)

In the essay the height of water is about $\frac{|4-5|}{5} \times 100\% = 20\%$ lower than the height of water in our simulation. So, we expect that the productivity of that system is about 9% higher than our simulated system.

$$\checkmark \frac{0.278-0.2546}{0.2546} \times 100\% = 9.19\%$$

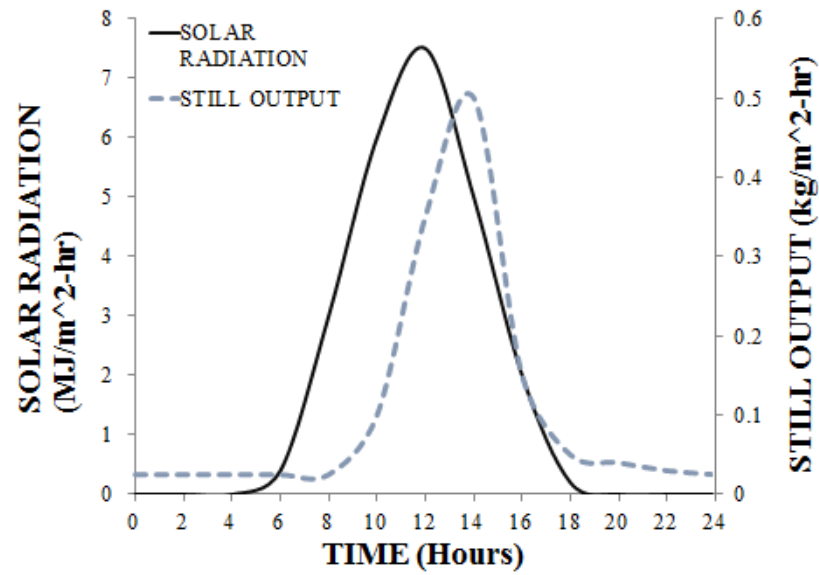


Figure 40

(Barve, 2017)

- ✓ Solar radiation= 1500 W/m^2
- Graphics: Contours of volume fraction

Volume fraction of water:

8:00

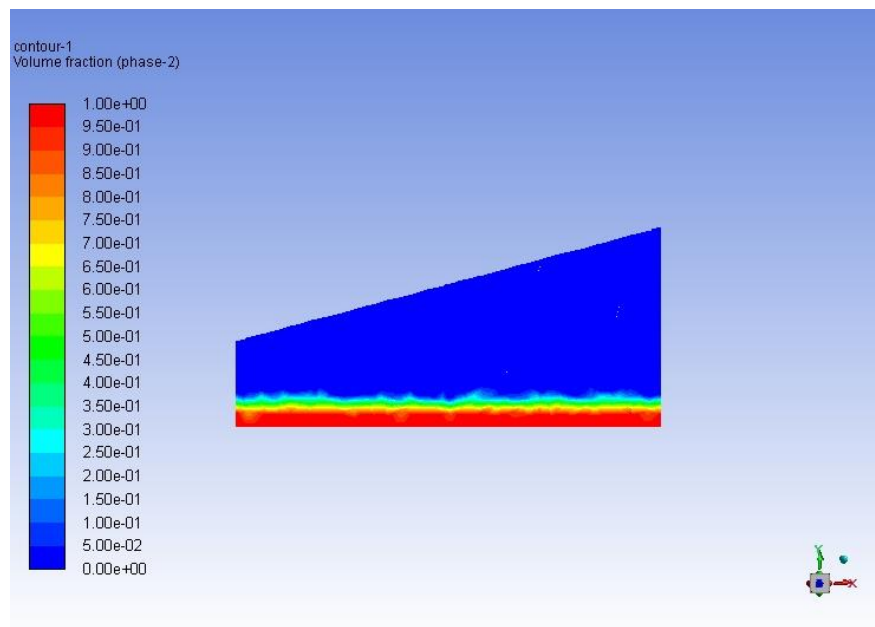


Figure 41

10:00

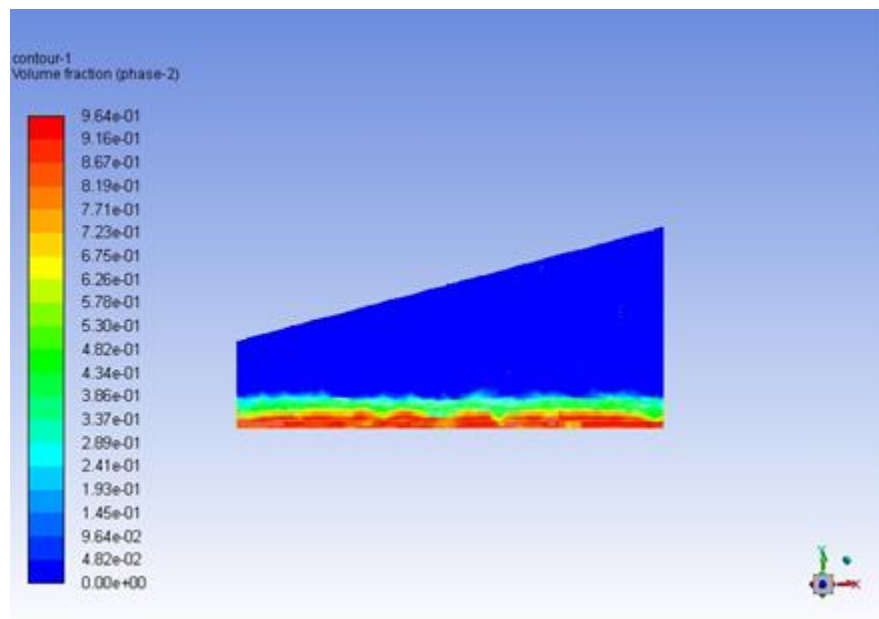


Figure 42

12:00

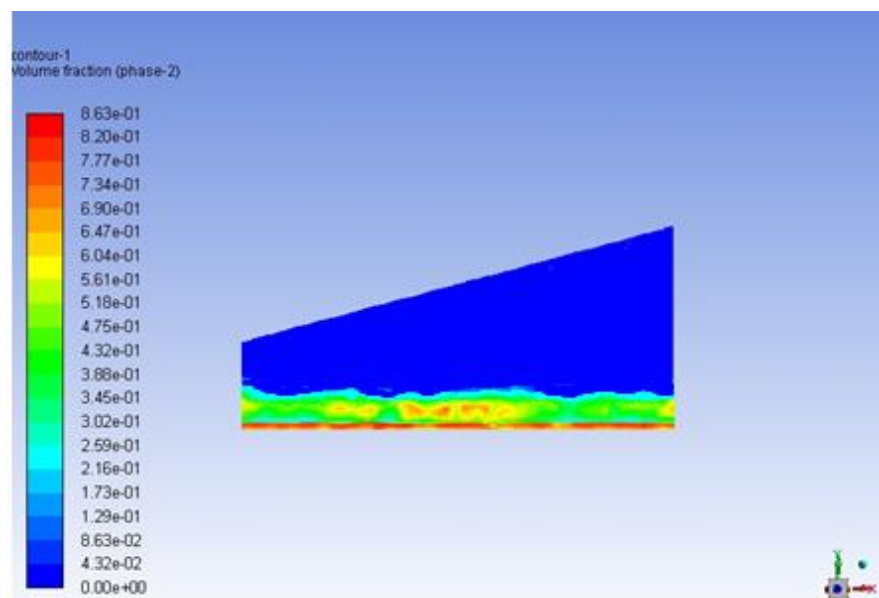


Figure 43

14:00

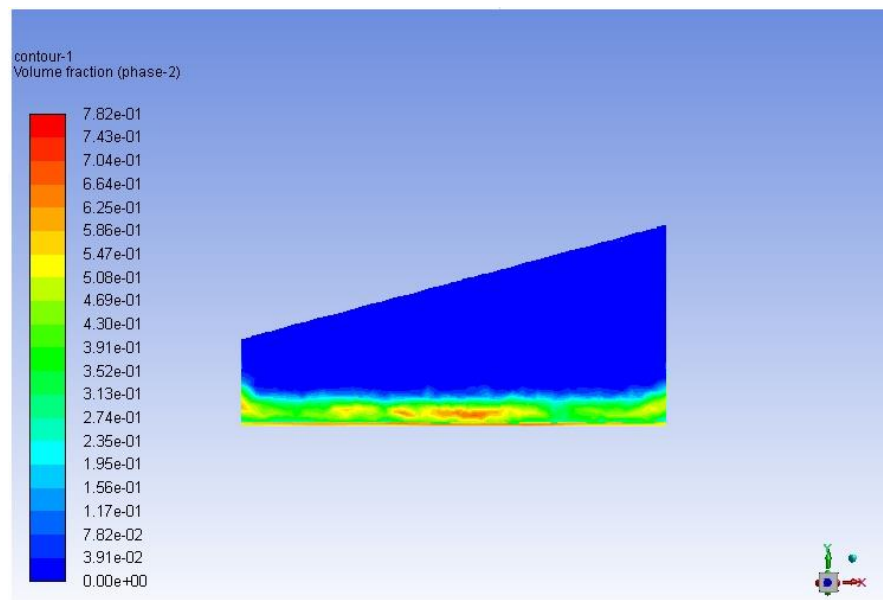


Figure 44

16:00

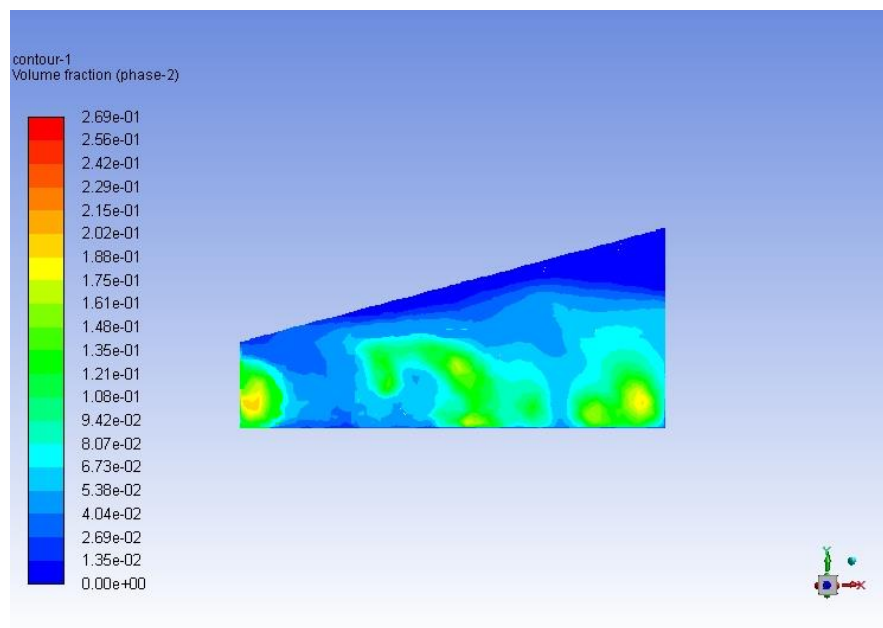


Figure 45

Volume fraction of vapor:

8:00

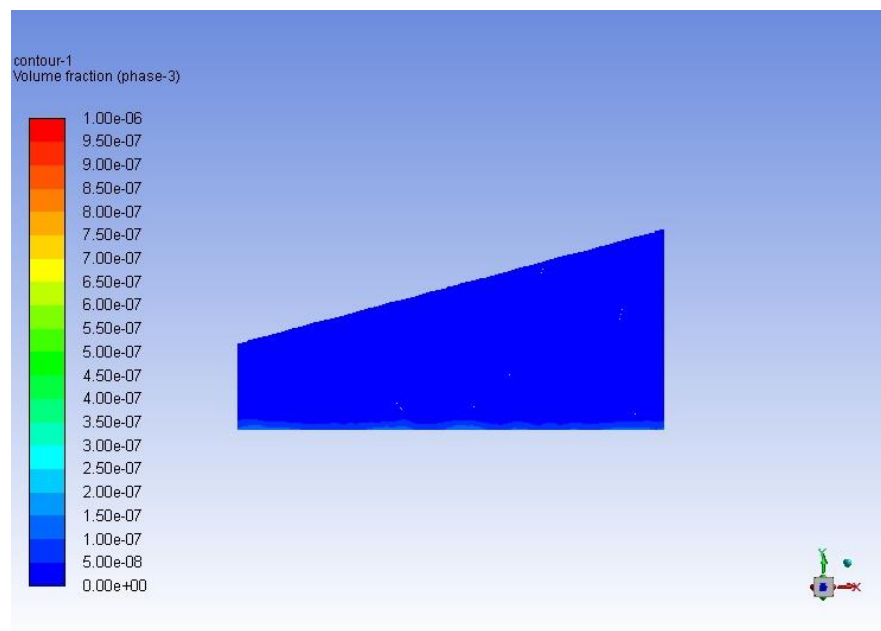


Figure 46

10:00

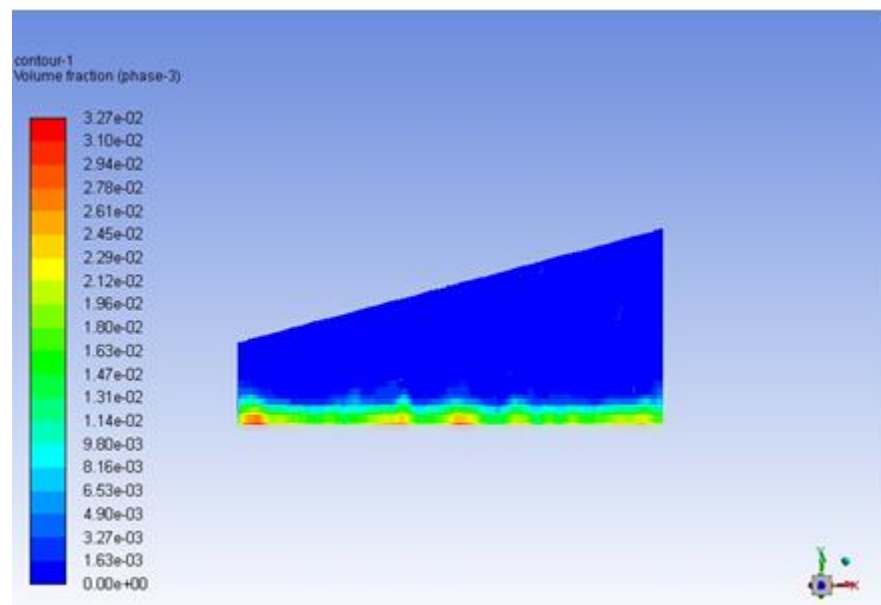


Figure 47

12:00

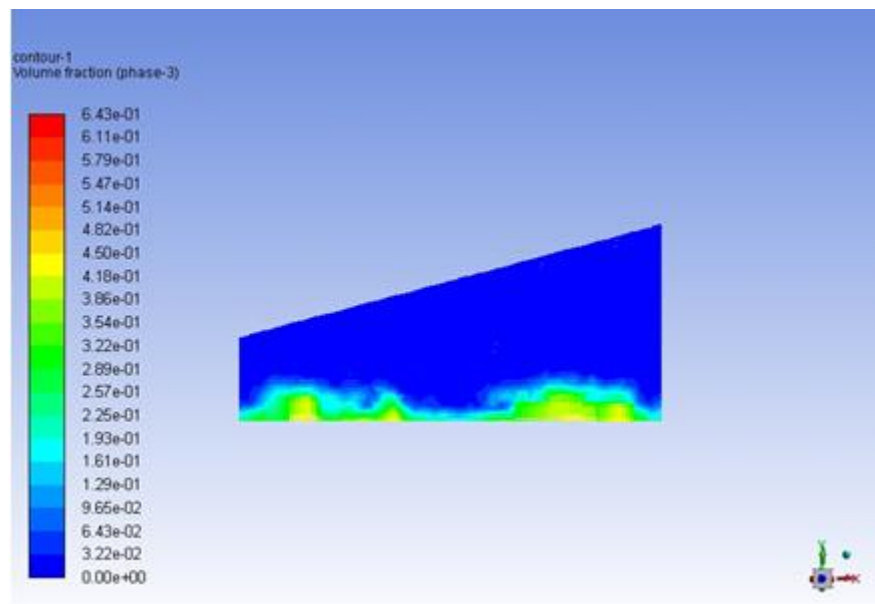


Figure 48

14:00

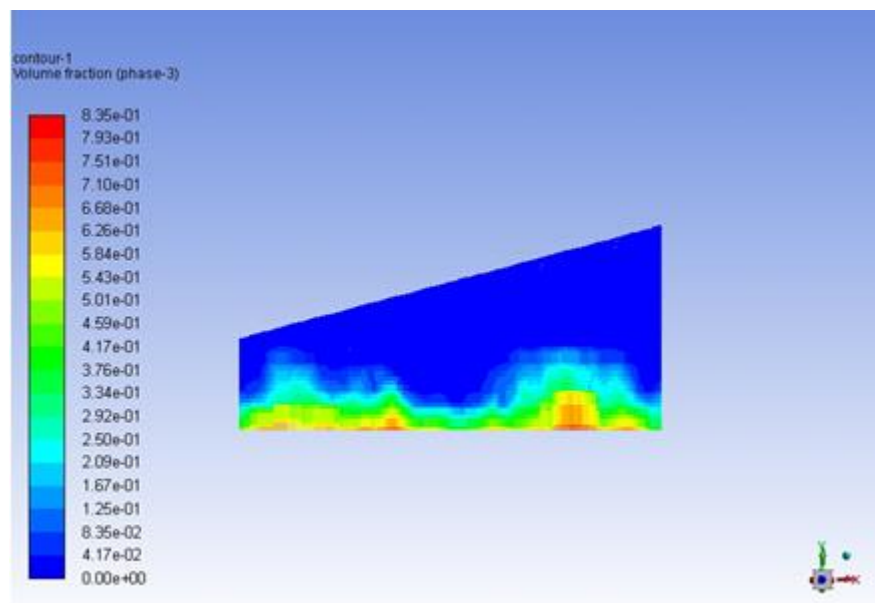


Figure 49

16:00

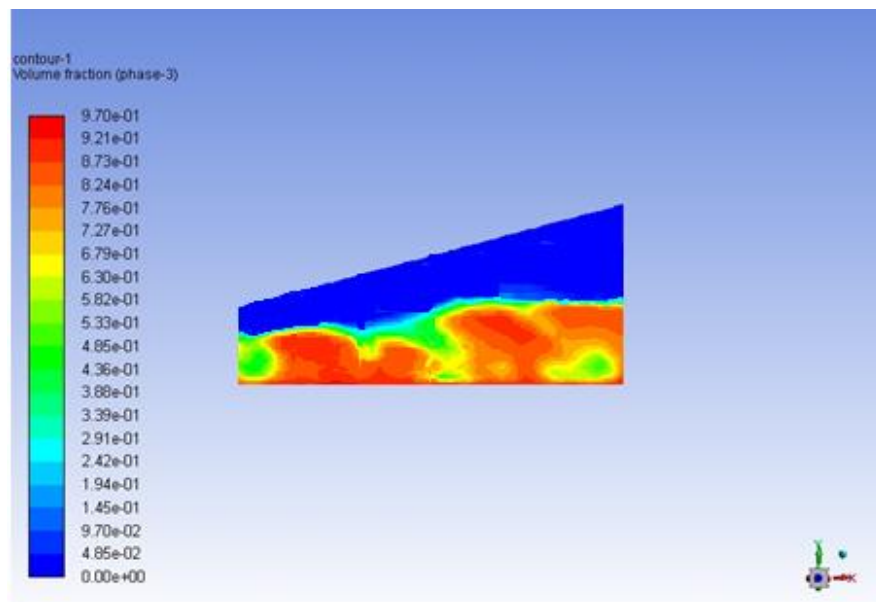


Figure 50

Temperature of water and glass:

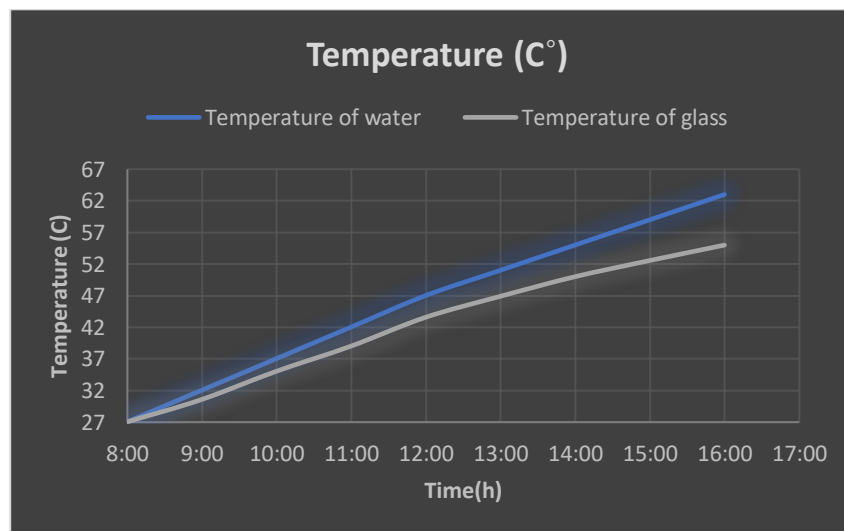


Figure 51

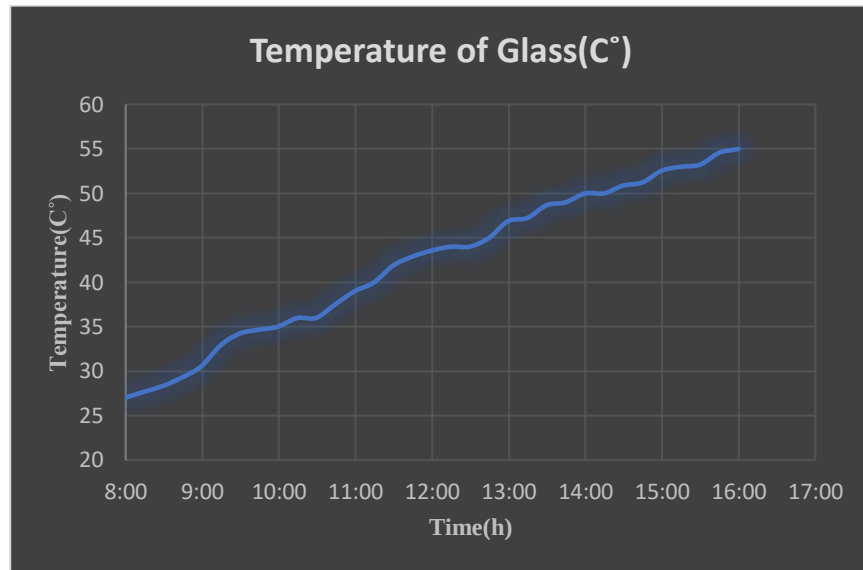


Figure 52

Evaporative heat transfer coefficient between the water mass and the glass cover:

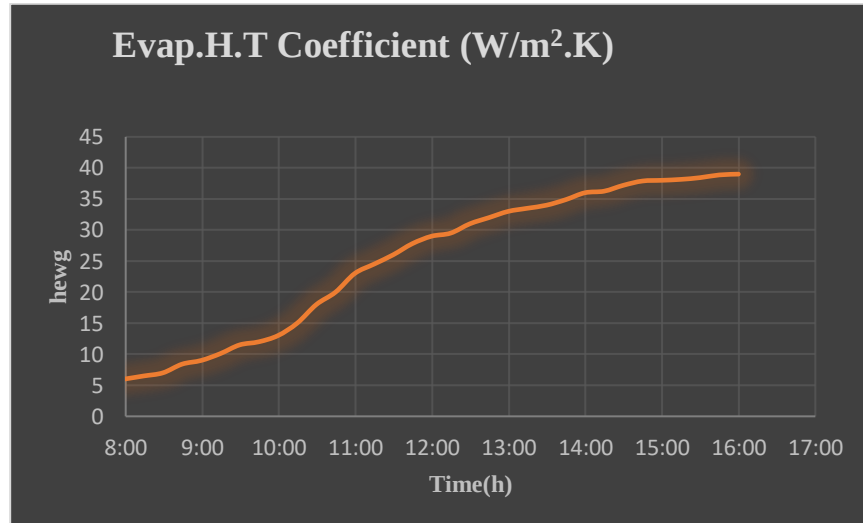


Figure 53

- Report Definitions: Flux Report → Mass Flow Rate

Productivity of solar desalination system:

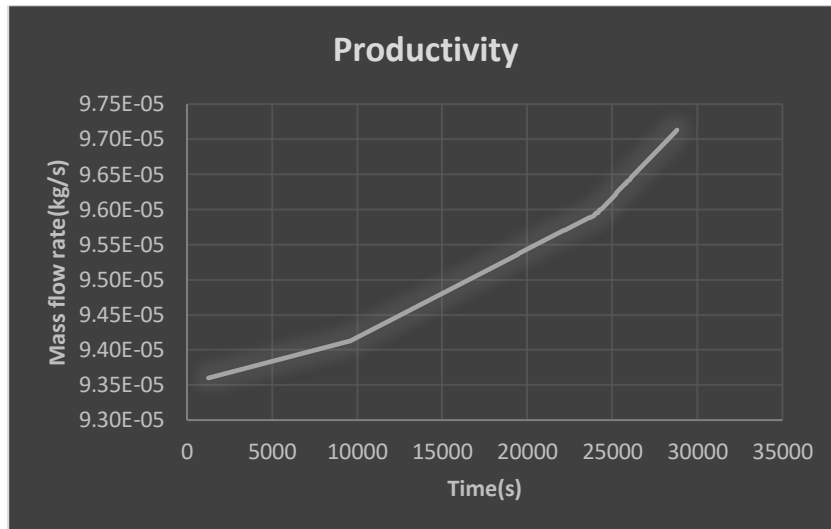


Figure 54

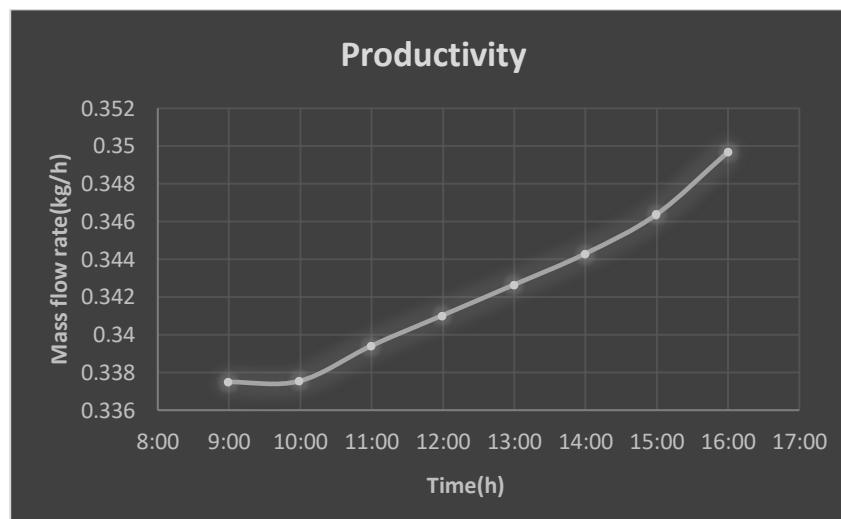


Figure 55

According to the below graph if the solar radiation intensity is 1500 W/m^2 , the productivity of the system will be about $0.390 \text{ kg/m}^2 \cdot \text{hr}$. This simulation is for the water height of 4 cm . However, in our simulation the height of water is 5 cm . On the other side we know that if we reduce the height of water 60% , the productivity of the system will be increased by 26% . (OO, 2009)

In the essay the height of water is about $\frac{|4-5|}{5} \times 100\% = 20\%$ lower than the height of water in our simulation. So, we expect that the productivity of that system is about 9% higher than our simulated system.

$$\checkmark \frac{0.390 - 0.3497}{0.3497} \times 100\% = 11.5\%$$

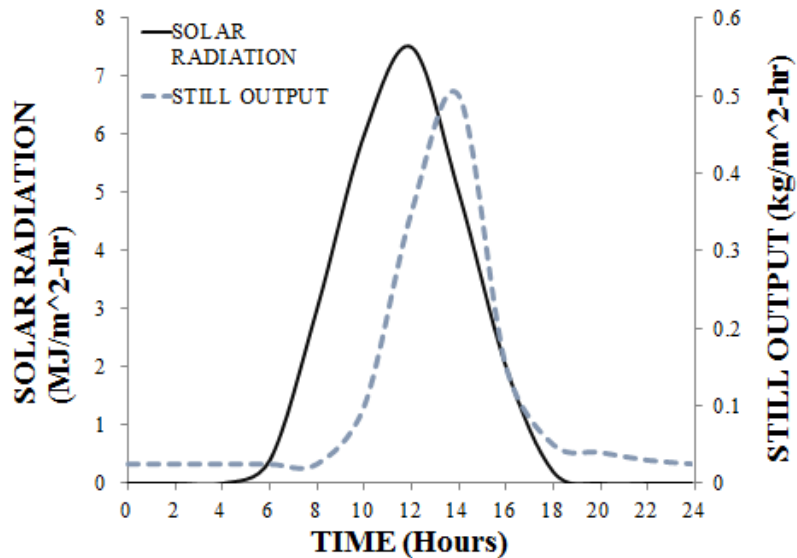


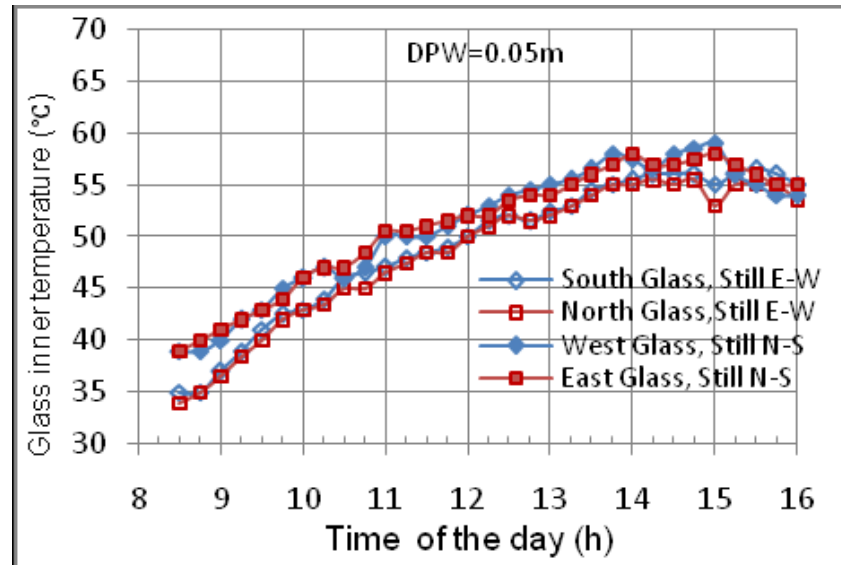
Figure 56

(Barve, 2017)

On the other essay we see that a double slope solar still is fabricated and investigations are carried out under the open environmental conditions of Aligarh, India that the range of solar intensity is taken between $0-1000 \text{ W/m}^2$ (Siddiqui, 2012). This kind of solar still with east-west direction has a more receipt of solar radiation versus to the single slope solar still system (شهرام دلفانی- هادی پاسدار شهری- مریم (کرمی, 1389).

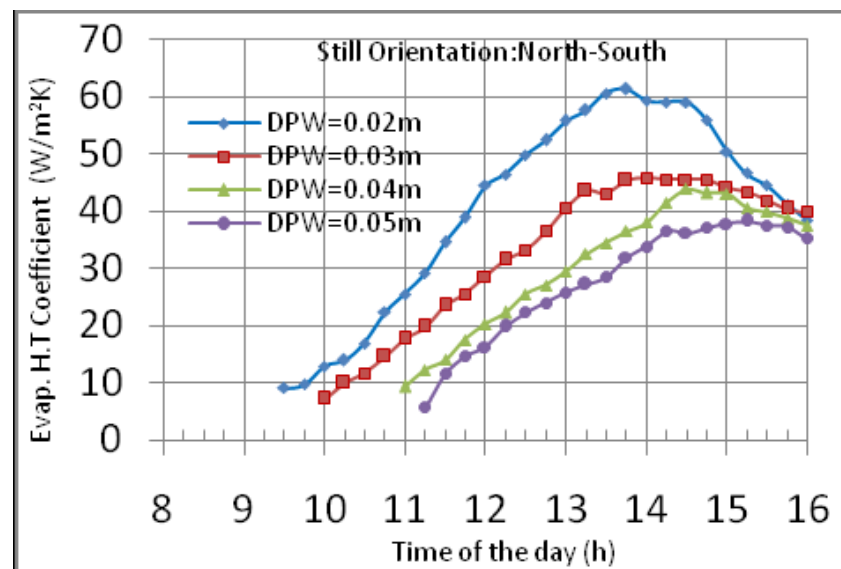
Temperature of Glass($^{\circ}\text{C}$):

- The Essay Results:



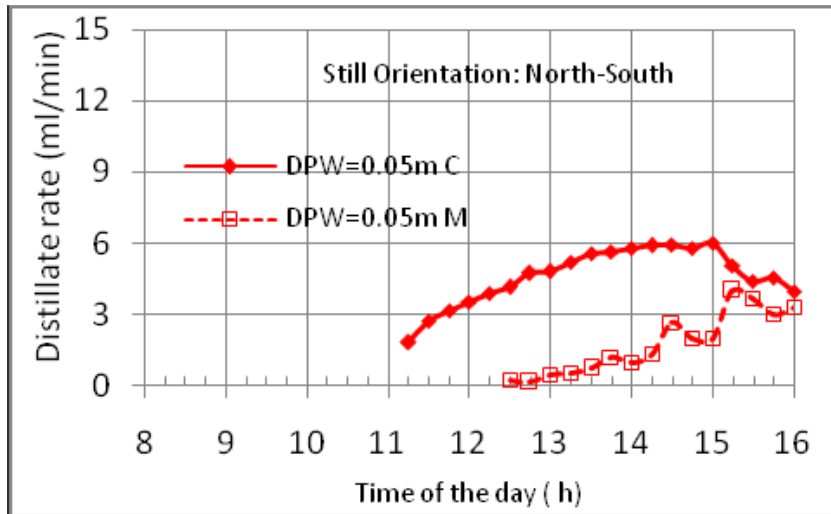
Essay Graph 1

Evaporative heat transfer coefficient between the water mass and glass cover:



Essay Graph 2

Productivity:



Essay Graph 3

$$\text{Minimum productivity: } 1.9 \frac{\text{ml}}{\text{min}} \times \frac{60\text{min}}{1\text{hr}} \times \frac{1\text{m}^3}{10^6\text{ml}} \times \frac{997\text{kg}}{1\text{m}^3} = 0.113658 \frac{\text{kg}}{\text{h}}$$

$$\text{Maximum productivity: } 6 \frac{\text{ml}}{\text{min}} \times \frac{60\text{min}}{1\text{hr}} \times \frac{1\text{m}^3}{10^6\text{ml}} \times \frac{997\text{kg}}{1\text{m}^3} = 0.35892 \frac{\text{kg}}{\text{h}}$$

✓ Validation:

As we know the solar intensity in our studied essay is between 0 to 1000 W/m² and as we approach to the noon time the intensity of radiation increases and after 13:30 it decreases. But, in our simulation we assume the solar intensity is constant at all the day times and at the first case it is about 1000W/m² and at the second case it is about 1500W/m².

- Temperature of Glass:

In the studied essay the primary temperature of the glass is higher than the assumed temperature in our simulation case. According to our explanations we conclude that the intensity of the glass temperature increase in our simulation case is higher than its increase in our studied essay. So, it is logic to accept that by approaching to the noon time the difference between the temperatures of the glasses in two cases decrease.

$$I=1000\text{W/m}^2$$

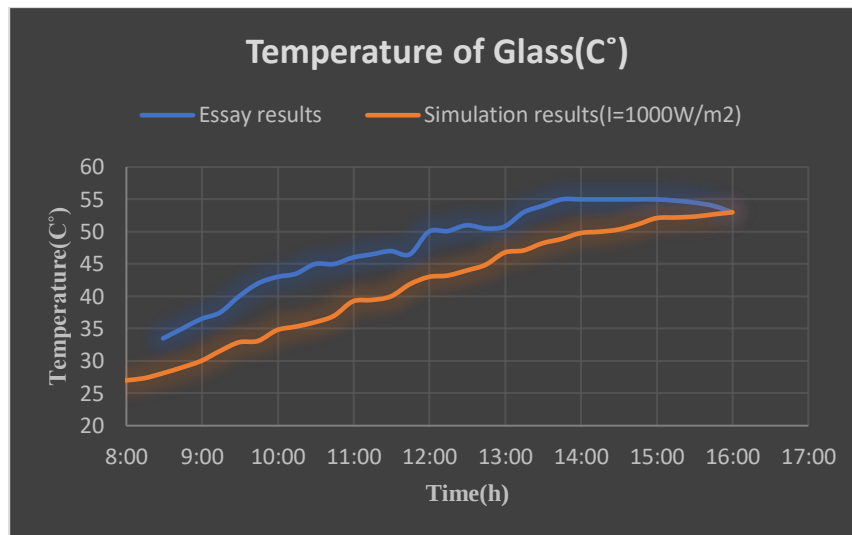


Figure 57

$$I=1500\text{W/m}^2$$

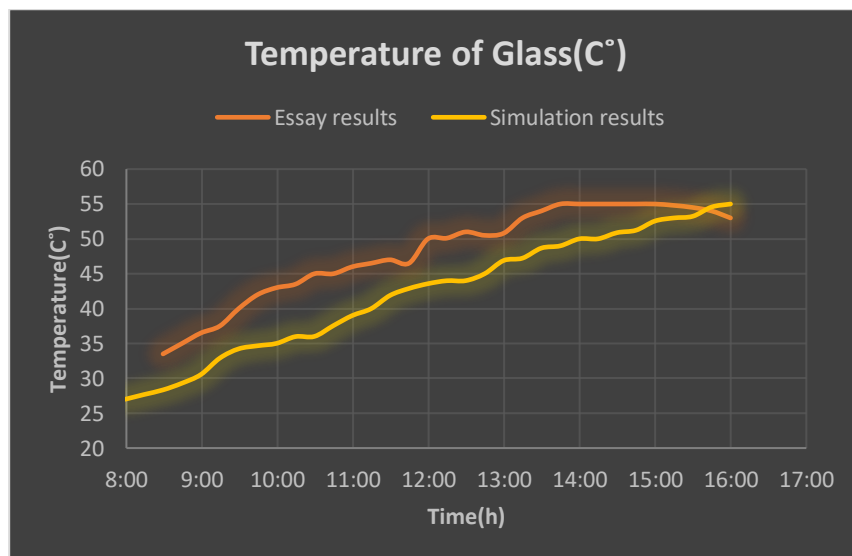


Figure 58

- Evaporative heat transfer coefficient between the water mass and glass cover and Productivity of the system:

According to the above explanations it is logic to accept that at the primary hours of the day that the intensity of radiation in our simulation case is more than this quantity in the studied essay and as the result the evaporative heat transfer coefficient and the productivity is also more than these quantities

in the essay. As we approach the noon time in the first case of our simulation the received intensity of solar radiation in double slope solar still is more than the single slope solar still and as the result the quantities of evaporative heat transfer coefficient and the productivity in our studied essay is more than these quantities in our simulation case.

In the second case of our simulation as we approach the noon time the received quantity of the solar radiation in our simulation and in our studied essay will be closer to each other and as the result the quantities of evaporative heat transfer coefficient and the productivity of the two systems will be also closer to each other.

$$I=1000\text{W/m}^2$$

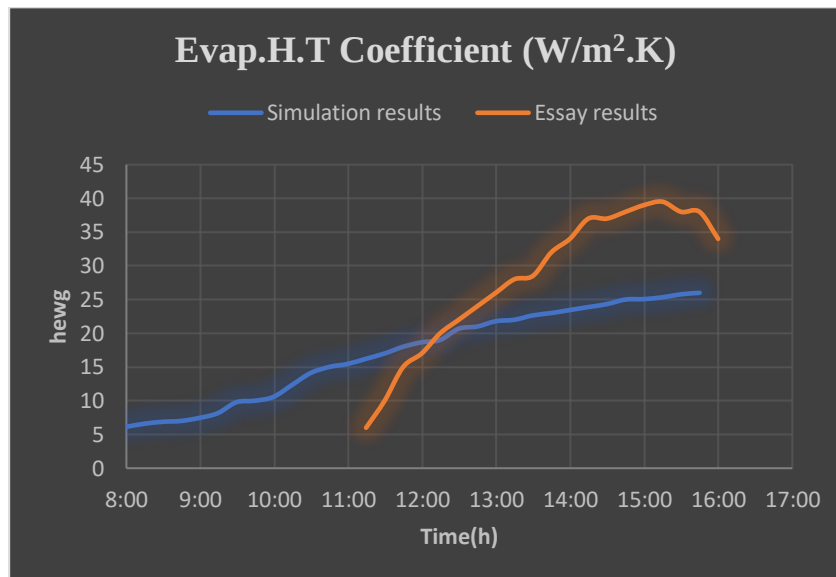


Figure 59

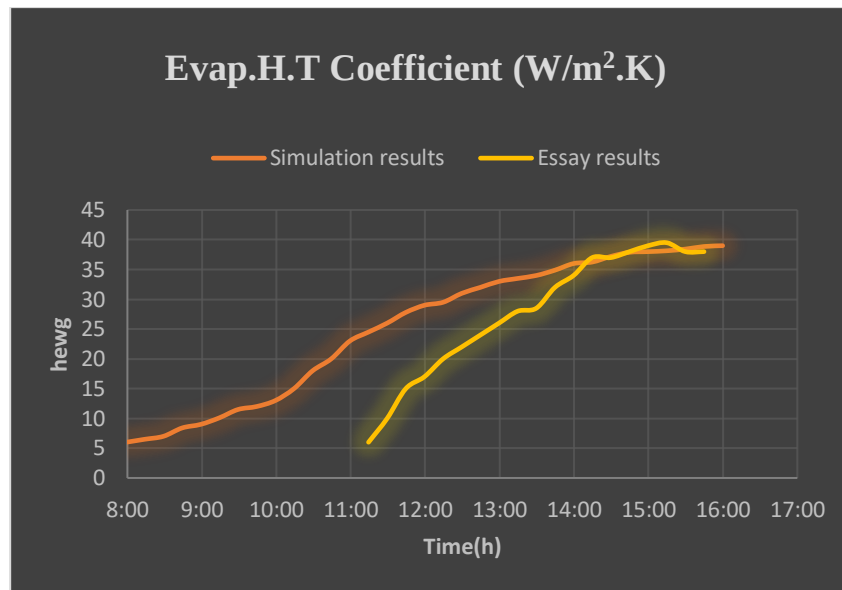
$I=1500\text{W/m}^2$ 

Figure 60

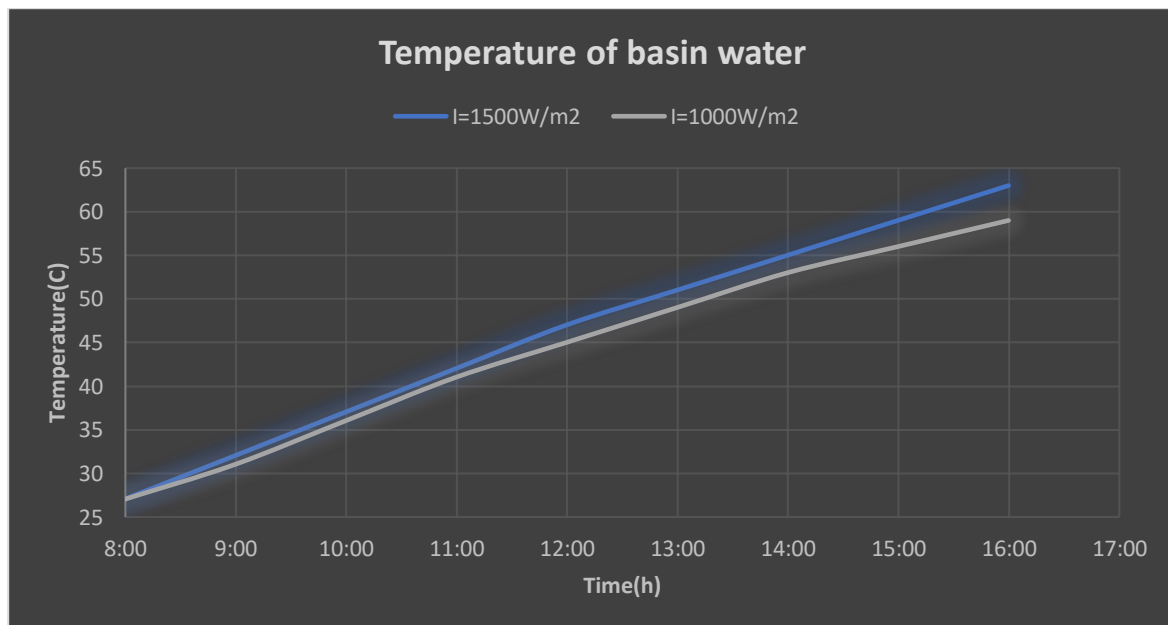


Figure 61

✓ **Hand calculation for validation:**

$$I = 1500 \frac{W}{m^2} \quad A = 1m^2 \quad \text{Height of water} = 5cm$$

$$V_{Water} = 0.05m^3 \quad m_{Water} = 0.05m^3 \times \frac{997kg}{1m^3} = 49.85kg$$

$$Q = mL_f + mc\Delta T \quad L_f = 2264 \frac{kJ}{kg}$$

$$Q = 0.35kg \times 2264 \times 10^3 \frac{J}{kg} + 49.85kg \times 4200 \frac{J}{kg.K} \times 1K = 1001770J$$

$$1s \quad 1500J$$

$$t \quad 1001770J \Rightarrow t = 11.13min$$

It takes 11 minutes for the water temperature to rise one degree.

$$\Delta T = (63 - 27)^\circ C = 36^\circ C$$

$$\Delta t = 36 \times 11.13 = 400.68min \approx 7hr$$

$$\text{Percentage of error} = \left(\frac{8-7}{7} \times 100\% \right) = 14.28\%$$

$$I = 1000 \frac{W}{m^2}$$

$$Q = mL_f + mc\Delta T \quad L_f = 2264 \frac{kJ}{kg}$$

$$Q = 0.2546kg \times 2264 \times 10^3 \frac{J}{kg} + 49.85kg \times 4200 \frac{J}{kg.K} \times 1K = 785784.4J$$

$$1s \quad 1500J$$

$$t \quad 785784.4J \Rightarrow t = 13.095 min$$

It takes 13.095 minutes for the water temperature to rise one degree.

$$\Delta T = (59 - 27)^{\circ}\text{C} = 32^{\circ}\text{C}$$

$$\Delta t = 32 \times 13.095 = 419.04 \text{ min} \approx 7 \text{ hr}$$

$$\text{Percentage of error} = \left(\frac{8-7}{7} \times 100\% \right) = 14.28\%$$

As a result, theoretically it takes 7hr to raise the temperature of water at the first state about 36°C and at the second state about 32°C but empirically it takes 8hr and the simulation has about 14% errors and it maybe as a result of systemically heat losses and etc.

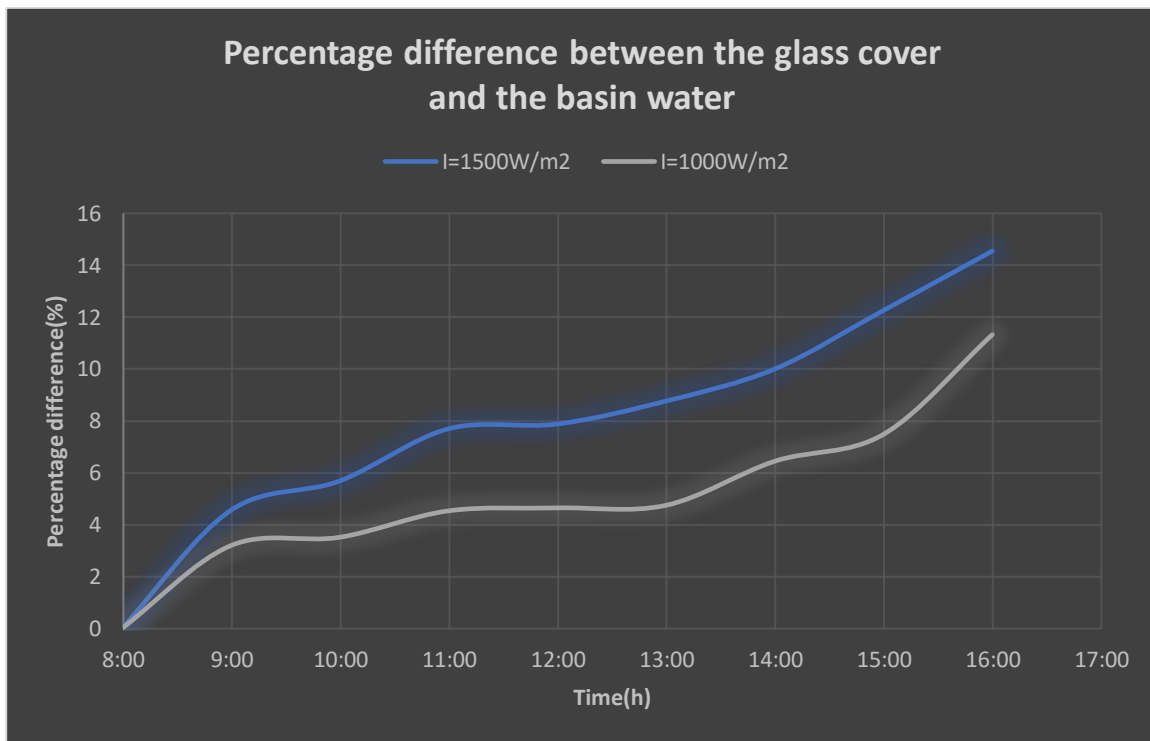


Figure 62

If the difference in temperature between the glass cover and the basin water increases, the natural circulation of the air mass inside the solar still will also increase. As the result both evaporative and convective heat transfer from basin water to glass cover will increase (Yousef H. Zurigat, Mousa K. Abu-Arabi, 2003). As we see in the above graph the percent of the difference between the glass cover and the basin water temperature in solar intensity of 1500W/m^2 to the solar intensity of 1000W/m^2 is higher, so it is logic that the evaporative heat transfer from basin water to glass cover for this solar

intensity is higher and the productivity of the system for this case is more than the productivity for solar intensity of 1000W/m^2 .

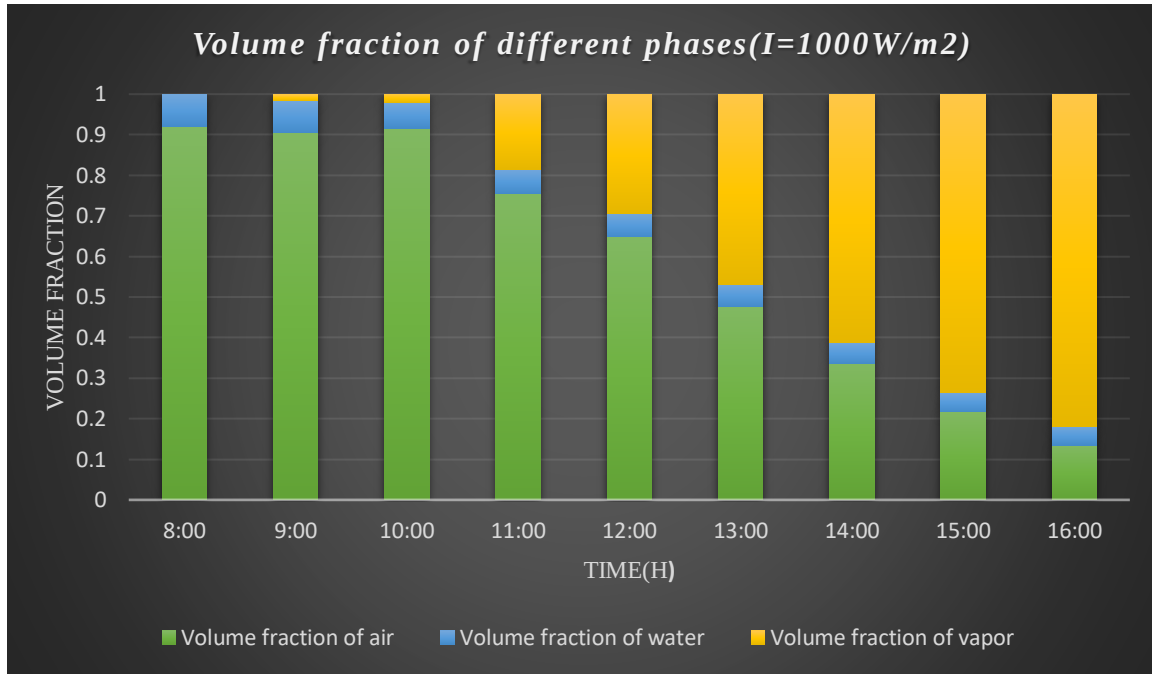


Figure 63

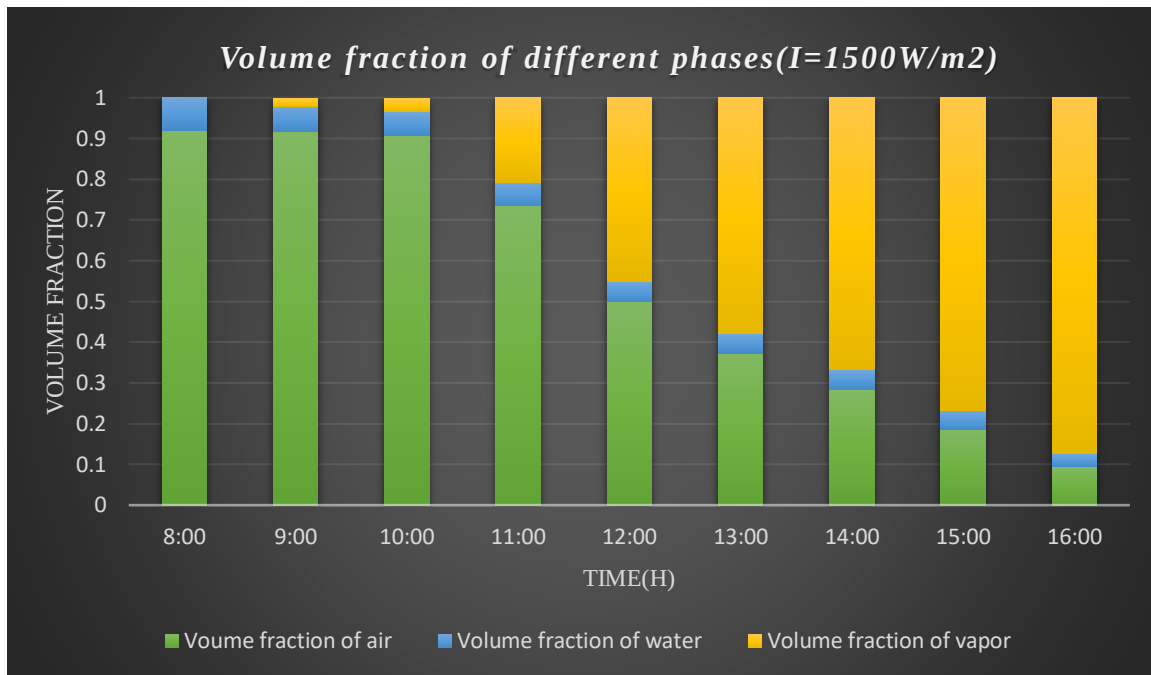


Figure 64

We know that at the free surface of every liquid, the evaporation happens in every temperature. Kinetic energy of the molecules of the liquid surface changes by the collision with other molecules. In a specific moment some of the molecules have a very high kinetic energy and some of them have a low energy. When a molecule near the liquid surface gets enough energy to overcome the vapor pressure, then it escapes the liquid surface and changes to a gas phase. One of the most important factors of the evaporation speed is the temperature of the liquid. When the temperature increases, the kinetic energy of the molecules also increases and the number of molecules which have enough energy to escape the liquid surface also increase (Fennell, 2018). For this reason, we see that at the above graph till 10:00 o'clock that the temperature of the water is low the volume fraction of vapor is also very low but after 10:00 o'clock it will increase more rapidly.

- **Conclusion:**

In a solar still system with a base area of 1m^2 which is filled with sea water level of 5cm; if all operational and climate parameters except solar intensity are fixed, by increasing solar intensity to 50%, the productivity of solar still will be increased to 37.4%.

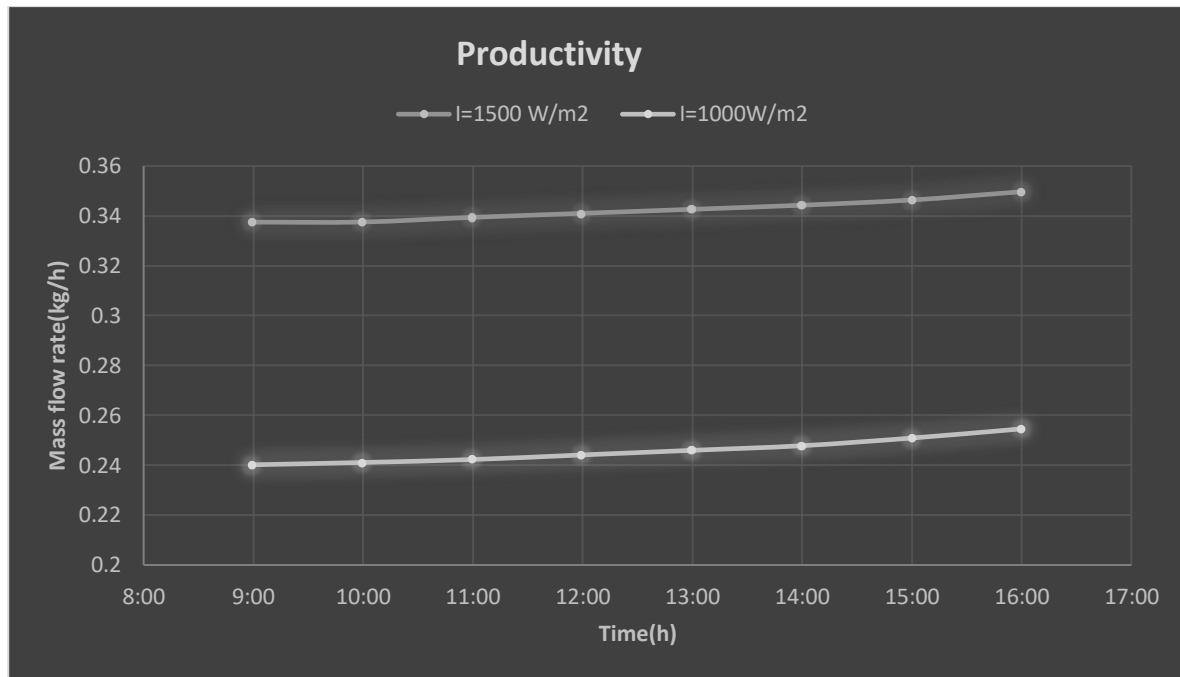


Figure 65

We conclude that the influence of increasing the solar radiation intensity on the productivity of the system is high so we should use the solar still system in the region where the solar intensity is relatively high.

It is also necessary to investigate the influence of other parameters on the productivity of the system to distinguish the appropriate regions to optimize the use of this system.

- **Bibliography:**

16. Multiphase Flows. (2009, January 23). Retrieved from ANSYS FLUENT 12.0/12.1 Documentation : <http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node288.htm>
- 35.5.3 Phase Interaction Dialog Box. (2009, January 29). Retrieved from ANSYS FLUENT 12.0/12.1 Documentation : <http://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node1101.htm>
- Bakker, A. (n.d.). Lecture 5- Solution Methods. Applied Computational Fluid Dynamics , 23-28.
- Barve, P. K.-A.-D. (2017). CLIMATIC AND DESIGN PARAMETERS EFFECTS ON THE PRODUCTIVITY OF. ResearchGate, 4.
- BELEY, B. (2016). Geometry of Tube Radiant Heaters. PRAGUE: CZECH TECHNICAL UNIVERSITY IN PRAGUE FACULTY OF MECHANICAL ENGINEERING.
- Boundary Conditions . (2006, December). Retrieved from Introductory FLUENT Training: www.fluentusers.com
- Deshpande, A. (2012, November 4). Pressure Velocity Coupling. Retrieved from in SlideShare: <https://www.slideshare.net/parabajinkya0070/pressure-velocity-coupling>
- Evaluation of Gradients and Derivatives. (2006, September 20). Retrieved from ANSYS FLUENT: <http://www.ipt.ntnu.no/manuals/Fluent.Inc/fluent6.3.26/help/html/ug/node994.htm>
- Fennell, J. (2018). What is Evaporation? Retrieved from Study.com: <https://study.com/academy/lesson/what-is-evaporation-definition-examples-quiz.html>
- Introduction to ANSYS FLUENT . (2010, December). Retrieved from ANSYS: https://imechanica.org/files/fluent_13.0_lecture05-solver-settings.pdf
- LEAP CFD Team. (2012, January 17). TIPS & TRICKS: CONVERGENCE AND MESH INDEPENDENCE STUDY. Retrieved from Leap australia: <https://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/>
- Modeling Radiation. (2013, November). Retrieved from ANSYS Fluent Theory Guide: <http://www.ansys.com>

- Modeling Turbulent Flows. (2006, December). Retrieved from Introductory FLUENT Training : http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf
- Munoz, G. (2015). MESH QUALITY AND ADVENCED TOPICS ANSYS WORKBENCH 16.0. Introduction to ansys meshing, 11-22.
- OO, B. (2009). Experimental study of the enhancement parameters on a single slope solar still productivity. *Desalination*, 136-143.
- Prachi K.Ithape, Anand R Nadgire, Dr.Shivprakash Bhagwatrao Barve. (2017). CLIMATE AND DESIGN PARAMETERS EFFECTS ON THE PRODUCTIVITY OF SOLAR STILLs. ResearchGate, 18.
- Pressure-Velocity Coupling. (2009, January 23). Retrieved from ANSYS: <http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node373.htm>
- Siddiqui, W. J.-P. (2012). Effect of water depth and still orientation on productivity for passive. *International Journal of Engineering Research and Applications (IJERA)*, 1662-1665.
- UNICEF, W. (2019, June 18). 1 in 3 people globally do not have access to safe drinking water. Retrieved from World Health Organization: <https://www.who.int/news-room/detail/18-06-2019-1-in-3-people-globally-do-not-have-access-to-safe-drinking-water-%E2%80%93-unicef-who>
- Vaibhav Rai Kharea, Abhay Pratap Singhb, Hemant Kumarc, Rahul Khatri. (2016). Modelling and Performance Enhancement of Single Slope Solar Still using CFD. *ScienceDirect*, 448.
- Yousef H. Zurigat, Mousa K. Abu-Arabi. (2003). Modelling and performance analysis of a regenerative solar desalination unit. *ELSEVIER*, 1061-1072.
- Yunus A.Cengel- Afshin J.Ghajar. (2015). *Heat and Mass Transfer Fundamentals & Applications*. New York: McGraw-Hill Education.
- Yunus A.Cengel- Michael A.Boles. (2011). *Thermodynamics*. New York: McGraw-Hill Education.
- وہب اکاتی- امین بھزادمہر- سعید فرساد. (1396). آب شیرین کن های خورشیدی. خراسان جنوبی: انتشارات آوای قلم.
- حیدرزادہ, م. (1394). دینامیک سیالات محاسباتی کاربردی با ANSYS Fluent. تهران: انتشارات دانشگاهی کیان.

دکتر امیر توحیدی - مهندس هادی غفاری قهرودی. (1397). راهنمای جامع ANSYS FLUENT مقدماتی. تهران: موسسه فرهنگی هنری دیباگران.

دکتر مهدی دادخواه تهرانی- مهندس مائده نعمتی. (1397). انسیس فلونت پیشرفته- شبیه سازی جریان های چندفازی. تهران: موسسه فرهنگی هنری دیباگران.

شهرام دلفانی- هادی پاسدارشهری- مریم کرمی. (1389). آب شیرین کن های خورشیدی. تهران: مرکز تحقیقات ساختمان و مسکن.

محمدجدیدی. شرائط مرزی و پارامترهای آشفستگی در نرم افزار FLUENT. گروه دینامیک سیالات محاسباتی دانشگاه صنعتی اصفهان .