JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

DESIGN OF PARABOLIC TROUGH COLLECTOR AND COMPARISON WITH NUMERICAL STUDIES TO EXAMINE THE IMPACT OF NANOFLUIDS ON THE THERMAL PERFORMANCE

Ch Raghavendra Siva Kumar[1], B Raghu Kumar[2]

[1*]P.G Student, Department of Mechanical Engineering, Prasad V Potluri Siddhartha Institute of Technology, Kanuru, Vijayawada, Andhra Pradesh, India

[2] Head & Professor, Department of Mechanical Engineering, Prasad V Potluri Siddhartha Institute of Technology, Kanuru, Vijayawada, Andhra Pradesh, India

*Corresponding Author

Email raghavendrasivakumarc@gmail.com

Abstract

The most developed device for using solar energy in high-temperature applications is the parabolic trough collector. Utilizing the sterling cycle and water as a heat transfer fluid, the PTC system is employed in novel domains including the production of steam and hot water as well as electricity. This work's goal is to provide a thorough study of the literature on the subject and to do a CFD analysis of a PTC utilizing ANSYS and nanofluids. The working fluid in parabolic trough collectors is nanofluid. The evaluated research use ANSYS Fluent software to simulate computational fluid dynamics on a parabolic trough collector.

The primary objectives were to evaluate the impact of nanofluids on the thermal performance of the process and examine the temperature changes that resulted from the use of various concentrations of nanoparticles, such as Al3O2, SiO2, Cuo, and TiO2, with base fluid water at a volumetric concentration of approximately 0.3%, which was taken from a literature survey and also compared with empirical solutions by varying volumetric concentration. It was found that, for the most part, nanofluids increase energy efficiency, convection heat transfer coefficient, and thermal efficiency.

Keywords *Parabolic Trough Collector (PTC), Nano fluids, thermal conductivity, density, viscosity*

1. INTRODUCTION

1.1 Introduction to Parabolic Trough Collector

This is the most sophisticated CST technology, now in use globally, accounting for over 90% of the STE capacity. Large mirrors shaped like parabolas are used in solar fields with trough systems to gather sunlight, as seen in Fig. They are connected in long lines that can stretch up to 300 meters, and they track the sun's path on a single axis, usually from East to West, throughout the day [2, 3].

The sunbeam is directed by the parabolic reflectors onto a receiver pipe that is filled with a specific heat-transfer fluid and situated near the parabola's focal line. A unique coating on these receivers maximizes energy absorption while reducing infrared re-irradiation. The pipes function inside an evacuated glass shell to prevent convection heat losses. To create the superheated steam that powers the turbine, the thermal energy is extracted by the heat transfer fluid (such as molten salt or synthetic oil) moving through the heat-absorbing pipe [3]. The fluid is recycled back into the system once it has transferred its heat.

1.2 Nano Fluids

A fluid containing nanoparticles—particles smaller than a nanometer—is referred to as a nanofluid. These fluids are specifically designed for colloidal nanoparticle suspensions in a base fluid. Usually, metals, oxides, carbides, or carbon nanotubes are employed to create the nanoparticles that are used in nanofluids. Water, ethylene glycol, and oil are examples of common base fluids. Because of their unique characteristics, nanofluids are used in a wide range of heat transfer applications, such as grinding, machining, engine cooling/vehicle thermal management, home refrigerators, chillers, heat exchangers, fuel cells, pharmaceuticals, and hybrid-powered engines.[4]

Compared to the fluid in which they are produced, they exhibit superior thermal conductivity and convective heat transfer coefficient. Understanding a nanofluid's rheological behavior is crucial to determining whether it is suitable for convective heat transfer applications.

1.3 Computational fluid dynamics (CFD)

It is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions.[6]

This CFD relies on the Navier-Stokes formulas. These equations explain the relationship between a moving fluid's velocity, pressure, temperature, and density. They originate from the application of Newton's second law to fluid motion and the presumption that the stress in the fluid is the sum of a diffusing viscous component and a pressure term.

Software that increases the speed and accuracy of complicated simulation scenarios, including transonic or turbulent flows, results from ongoing research. An analytical or empirical examination of a specific problem might be utilized for comparison in the first validation process of such software. A comprehensive verification is frequently carried out by extensive testing, such as flying tests.[7]

2. MATERIAL AND METHODS

2.1 Project specifications

The dimensions and other parameters of components of the parabolic trough collector are shown in the table below

Parabolic trough

Length	2 m
Width	1.5 m
Rim angle (ϕ)	90^{o}
Focus	0.375 m
Thickness	5mm
Reflectivity	0.9

Table 2.1 Specifications of collector

Absorbing tube

Inner diameter	0.038 m
Outer diameter	0.042 m
Length	2m
Solar absorptivity	0.94
Thermal emittance	0.08

Table 2.2 Specifications of absorbing tube

Design parameters obtained from # [2] The present work focuses on simulating the process of the parabolic trough collector in ANSYS fluent software to perform the CFD flow analysis on working fluid and study the temperature change in the outlet temperature of working fluid with different concentrations of water-based Sio_2, Tio_2, Al2O3, and CUO working fluid. This section involves copying the required materials that are to be assigned for the components of the model, from the ANSYS FLUENT material database and defining the materials that are not available in the database. The required materials are Silicon, Titanium, Aluminum, Copper, water, water-based Al2O3 Nanofluid, and water-based CuO Nanofluid.

The properties of the materials aluminum, copper, and water are available in the fluent database. The physical properties of Nanofluids with different concentrations are defined in the materials model to create the material. The thermo physical properties of these materials are shown in the below table.
The thermo physical properties of water-based Al2O3 Nanofluid, and water-based CuO Nanofluid with different volumetric concentrations are given in the project description chapter.

Nanofluid	Density (kg/m³)	Specific Heat (J/kg·K)	Thermal Conductivity (W/m·K)	Viscosity (Pa·s)
Cuo	2650.9	3088	1.297	0.00156
Sio2	1038	3806.2	0.521	0.0091
Tio2	1053.6	3526.2	0.431	0.00160
Al2o3	1060.3	3512.2	0.445	0.00175

Table 2.3 nanoparticles and water thermo physical properties at 0.3% vol. concentration, Values obtained from # [2]

Nanofluid	Density (kg/m³)	Specific Heat (J/kg-K)	Thermal Conductivity (W/m·K)	Viscosity (Pa s)
Cuo	1022.765	4052.29	0.5926	0.001575
Sio2	1003.415	4129.76	0.6472	0.00920
Tio2	1012.64	4092.94	0.5913	0.00160
Al2o3	1011.38	4097.46	0.5905	0.001769

Table 2.4 nanoparticles and water thermo physical properties at 0.45% vol. concentration

2.2 Properties of Nano Fluids

The properties of Nanofluids will be based on the type of Nanoparticle employed and the base fluid used. It also depends on the volumetric concentration of Nanoparticles. Volumetric concentration means the total percentage of Nanoparticles present in 100 % of Nanofluid. The Nano fluid that we are using is a water-based fluid where a certain volume of Nanoparticles of CUO and Al2O3 is added to water to create Nanofluid.

The physical and thermal properties of Nanofluid with different concentrations are calculated using the correlations described below. Subjects & selection method

1. Density of Nanofluid
   rnf =rnpØ + (1-Ø)rbf .
   where on = density of nanofluid
   RNP = density of Nanoparticle
   Ø = volumetric concentration

2. Specific heat of Nanofluid
   Cnf = (ØrnpCpnp+(1-Ø)rbfCpbf) /ref
   WhereCnf =specific heat of Nanofluid

3. Cpnp = specific heat of Nanoparticle
   Cpbf= specific heat of the base fluid
   Thermal conductivity (K)

Knf = (Knp+2Kbf+2Ø(Knp–Kbf) /Knp+2Kbf-Ø(Knp–Kbf)) *Kbf

WhereKnf = thermal conductivity of Nanofluid in W/m-K
Knp = thermal conductivity of Nanoparticle in W/m-K
Kbf = thermal conductivity of base fluid in W/m-K

4. Viscosity of Nanofluid
 mnf = mbsf(1-Ø)-2.5
 where = viscosity of Nanofluid
 mbf = viscosity of base fluid

These are the correlations used to calculate thermo physical properties of nanofluid with water as the base fluid. The calculated properties are shown in the table below. Using these properties a Nanofluid material, for each concentration is defined separately in the ansys fluent materials tab and assigned to the working fluid domain to study the temperature characteristics at the outlet of the absorber tube.
Ansys fluent
In this CFD analysis of parabolic trough collector simulation working fluid is made to flow inside the absorber tube is carried out using ANSYS FLUENT 2021 R1 software. Solar flux effects are simulated using the solar load model available in the software.
The following governing equations are solved by the software to obtain the desired results.

1. Continuity equation
 (jρ/jt)+ (j(ρuj)/jxi) = 0

2. Momentum equation
 (j(ρui)/jt)+ (j(ρujρui)/jxi)=(j/jxj)[ρδij+mt((j(jui)/jxj)+(j(juj)/jxi))]+ρgi

3. Energy equation
 ((j(ρCpT)/jt)+(j(ρuiCpT)/jxi)-(j[l(jT/jxj)]/jxj=ST

Where, U_I is the time velocity vector, ρ is the density of fluid, Cp is the specific heat of fluid, T is temperature, δij is the Skronecker delta function, l is the bulk viscosity coefficient, xi, and xj are spatial

2.3 Procedure Methodology

The design of the parabolic trough collector is done in CATIA V5 R20 software. Part design workbench is used to create the solid components of the parabolic trough collector. The designed part is then analyzed in the fluent workbench of ANSYS software.

Geometric modeling of parabolic trough collector using ANSYS
First, a plane has to be selected on which a 2D sketch of the geometry will be designed. To align with the coordinate axis present in the ANSYS software XY plane is selected as the sketching plane.
Parabolic trough Parabola command in the profile tool bar is utilized in sketching the model of the parabola according to the geometric specifications. A width of 1.5m with the focus point of the parabola as the origin is given to the sketch using the constraint tool. The focus of the parabola is given as 0.375mm. Then to create a solid part pad command in part design is used with length as two meter. A thickness of 5mm is given to the parabolic trough.
Absorber tube Again, XY plane is selected to sketch the geometry. Using circle command in profile tool bar two concentric circles are drawn with center point as origin and diameters as specified. Then a solid part is created in part design used pad same as the parabolic trough.
The design specifications of the components of parabolic trough collector are specified in the chapter before. The designed part geometry of the parabolic trough collector is shown in the Fig 5.1
The design file should be saved with .stp extension so that the geometry file can be imported in to the ANSYS.

2.4 Analysis Procedure In Ansys Fluent
The procedure of solving a problem in ansys fluent is a step by step procedure which includes importing or designing geometry, generating mesh, setting up the physics and solver conditions, solving the problem for solution and observing the results.

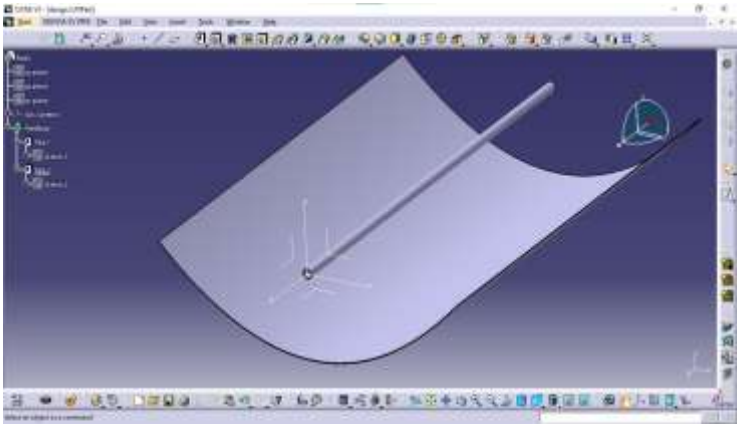


Fig 2.1 Designed model of PTC in CATIA V5

Fluent contains following main elements

1. **Pre-processing**
   a. Geometry
   b. Mesh
   c. Physics and solver configuration
2. **Solution**
   a. Compute solutions (solving the solution)
3. **Post processing**
   a. Examine results

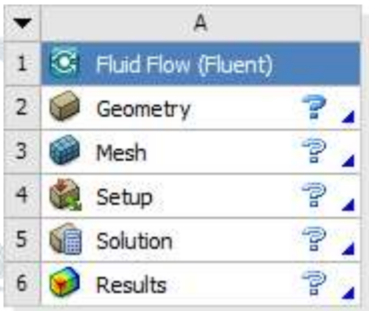


Fig 2.2 Fluent solver steps

2.5 Geometry

To import the geometry file, right click on the geometry will prompt us to import command using which we can browse for the .stp file created and imported in to the ANSYS.

Fluid domain needs to be created to simulate the flow of working fluid. Ansys is used to create the fluid domain. A solid cylinder of 38mm diameter and 2m length is modeled in Ansys to fit inside the created absorber tube geometry. This cylinder will act as a working fluid that will flow inside the absorber tube. It is shown in the Fig below.

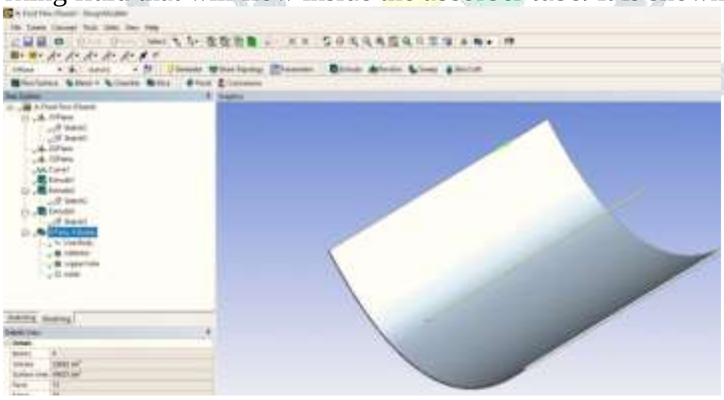


Fig 2.3 Fluid domain model designed in Ansys

2.6 Mesh

Discretization of the given geometry in to smaller number of cells is called meshing. The purpose of meshing is to actually make the problem solvable using finite element. By meshing we can break down the domain in to number of pieces, each piece representing an element or cell. We need these elements to apply finite element since, finite element is about solving the elements to get a local solution and combining all these solutions of elements to build the global solution for the problem. Another aspect of meshing is the accuracy of the solution. A refined mesh will generate an accurate global solution.

After opening the mechanical APDL application the geometry is displayed. First default mesh is generated using the mesh option. The default mesh is not refined. Edge sizing is applied by inserting sizing option in the mesh. Edge sizing with number of divisions as 40 is applied to the fluid domain.

Again edge sizing with number of divisions 40 is applied to the absorbing tube to create a uniform cell boundary and conformal mesh at the contact region. Edge Sizing is used to smoothen the edges and to get a good mesh. Body sizing is of 20 mm is applied to the parabolic trough body. As the value of the Body Sizing element decreases the skewness value of the design decreases. The Body Sizing of the design has to be done in order to create a good mesh and meshes the whole body at once. After meshing is done quality of the mesh has to be checked. The skewness value of the mesh has to be as minimum as possible, and the orthogonal quality of the mesh should be between 0.7-0.95 for a good quality mesh.

Inlet and outlet named selections are given to the faces of fluid domain one on each face. The upper face of absorber tube is given a named selection as absorbing surface, and the upper face of parabolic trough is given a named selection as reflecting surface.

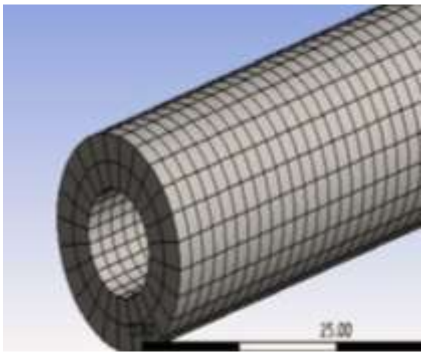


Fig 2.4 Mesh of fluid domain

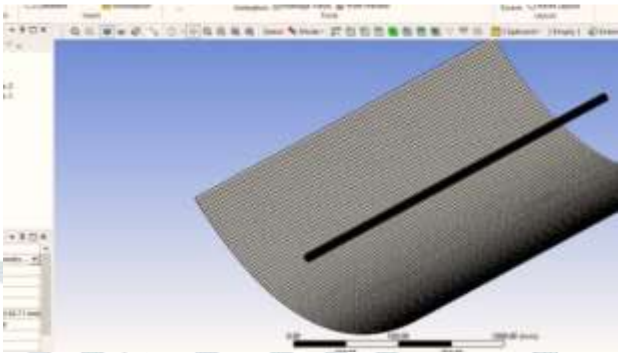


Fig 2.5 Meshed domain

2.7 Set up

The analysis has to be done using fluent solver. The set up will launch a fluent solver which is used to apply physics and solver configuration of the analysis. In the schematic double-clicking the set up will launch the ANSYS Fluent. When the ANSYS Fluent is first started fluent launcher is displayed, allowing to view and set certain ANSYS FLUENT start-up options. The mesh is automatically loaded and displayed in the graphics window by default when fluent setup is started. Double precision and parallel processing option has been selected for a faster and more accurate solution.

General properties
The steady model has been selected as the analysis being done is not a time dependent analysis, and gravity i.e. acceleration due to gravity of 9.81 m2/s is applied in negative Y direction.

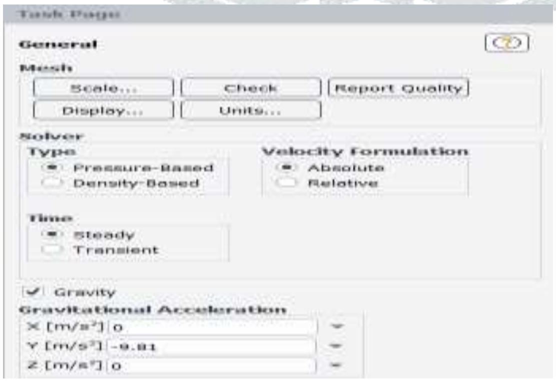


Fig 2.6 Task page

Model set up
Energy in the model energy is turned on. It will enable the transfer of energy between components of the geometry. This condition will solve the energy equation.
Viscous model laminar model is selected as the viscous model for this analysis.

Radiation
Ansys fluent provides five radiation models which allow you to include radiation, with or without participating medium, in heat transfer simulation. In addition to these models ANSYS fluent also provides a solar load model that allows us to include the effects of solar radiation in simulation.
Surface to surface (s2s) model surface to surface radiation model is applied to solve the radiation equation between the geometry. It is used to simulate the radiation heat transfer, which is arising in the closed set of the diffuse surfaces. The surface-to-surface radiation model can be used to account for the radiation exchange in an enclosure of grey-diffuse

surfaces. In this radiation model assumes that the surfaces are grey and diffuse i.e. There is no dependency of the wave length of the incoming radiation on the surface. The energy exchange between two surfaces depends in part on their size, separation distance and orientation.
These parameters are accounted for, by a geometric function called a “view factor”. In s2s radiation model, the view factor of the participating zone are calculated. Outer surface of the absorber tube is taken into consideration during the calculation of shape factor.

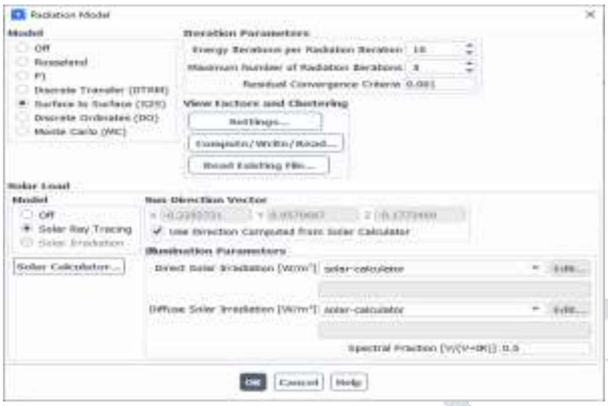


Fig 2.7 Radiation model set up for analysis

Fig 2.8 Parameters given to solar calculator

Solar load model Ansys fluent provides a solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain The solar load model includes a solar calculator utility that can be used to construct the sun's location in the sky for a given time-of-day, date, and position. Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.The solar load model's ray tracing algorithm can be used to predict the direct illumination energy source that results from incident solar radiation. It takes a beam that is modelled using the sun position vector and illumination parameters, applies it to any or all wall or inlet/outlet boundary zones that you specify, performs a face-by-face shading analysis to determine well-defined shadows on all boundary faces and interior walls, and computes the heat flux on the boundary faces that results from the incident radiation.Solar load cell is used for modelling the solar fluxes.

In the solar load cell, value of latitude, longitude of the location and date & time of the experiment specified. Values of the mesh orientation as desired like negative z axis for the north and while for east positive x-axis is also specified in the solar calculator for the present analysis. On substitution of all the required inputs in the solar calculator, values for direct normal solar radiation on the ground, diffuse solar radiation for both vertical and horizontal surface, ground reflected solar radiation for vertical surface and vector for sun direction are obtained. The solar calculator is set at the 79.41 E longitude and 13.62 N latitude which corresponds to the location Tirupati at 13 00 (IST).

The solar radiation values calculated from solar calculator are

Direct normal solar irradiation	878.223 [W/m^2]
Diffuse solar irradiation-vertical surface	82.7263 [W/m^2]
Diffuse solar irradiation-horizontal surface	117.682 [W/m^2]
Ground reflected solar irradiation	95.8203 [W/m^2]

Table 2.5 Radiation outcomes of solar calculator

2.8 Boundary conditions

Various boundary conditions are applied for the wall boundaries created for solving the governing equations. They are

Inlet mass flow rate inlet with 0.5 kg/s and fluid inlet temperature of 300K.

Fig 2.9 Cell zone condition for PTC

Fig 2.10 Inlet boundary condition

Outlet it is assigned as outflow condition
The walls that are generated by the mesh are shown in the given below.

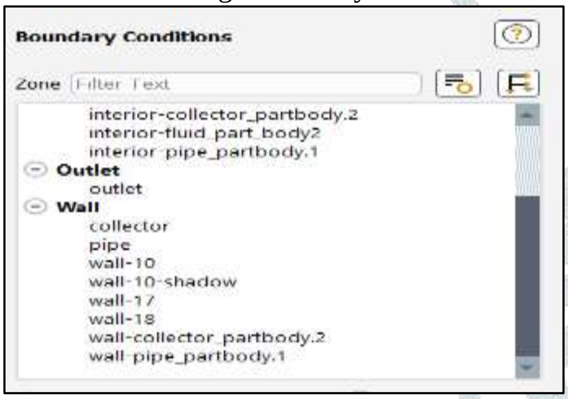


Fig 2.11 Walls generated

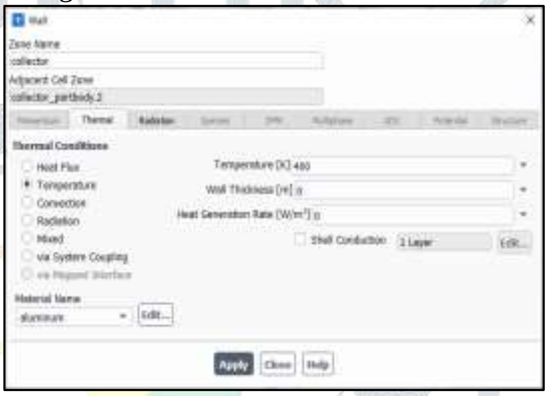


Fig 2.12 Boundary condition for collector

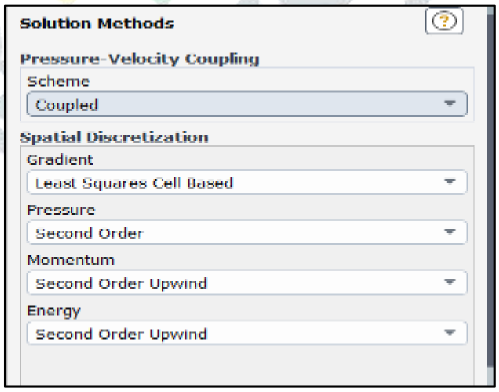


Fig 2.13 Boundary condition for absorbing surface wall Fig 2.14 Solution methods

For all other walls that are generated coupled thermal condition is applied with the respective radiation parameters and participation in solar ray tracing is enabled.
After that view factor file is computed in the radiation model for the calculation of view factors between the components involving in the radiation.

Numerical methodology different governing equations of mass, momentum and energy are solved through the finite volume method using pressure based segregated spatially implicit solver. Analysis is carried out for stead state condition to observe the temperature raise in working fluid due to absorption of energy.

2.9 Solution methods

In this section, methods to solve the governing equations of mass, momentum and energy is selected. Coupled scheme is selected in the pressure- velocity coupling. This scheme is used for achieving the coupling between momentum and continuity equation. It is used to solve the momentum and pressure-based continuity equations together. The coupled scheme obtains a robust and efficient single-phase implementation for steady state flows. In the spatial discretization least, square cell based method is selected. For solving the pressure, momentum and energy equations second-order upwind scheme is used.

3 RESULTS AND DISCUSSIONS

ANSYS software was used to perform a fluid study of a parabolic trough collector with varying volumetric concentrations of water-based Al2O3 and water-based CUO nanofluids. The findings are shown below.

3.1 Computational fluid dynamics contours

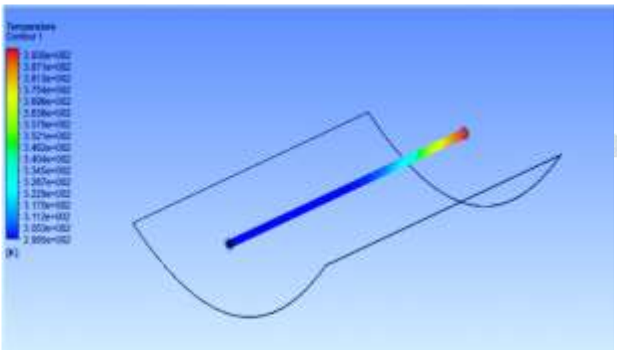


Fig 3.1 Cuo temperature contour at 0.3% volumetric concentration from # [2]

The maximum temperature of the outlet of the water observed is 385.3 K. The velocity contour is shown above. Because of the shear stress that the wall boundary imparts on the fluid, it is noticed that the fluid velocity is lowest close to the wall.
Because of the usage of the nanofluids as working fluid temperature, the outlet has improved.

3.2 Temperature contours of nano fluid

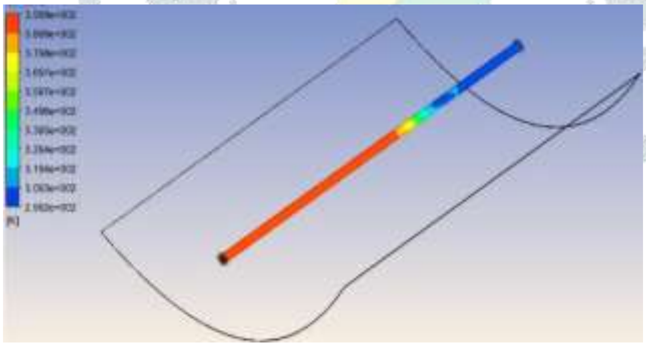


Fig 3.2 Cuo temperature contour obtained at 0.45 % volumetric concentration

Let us write the initial concentration as initial 0.3% and the new volumetric concentration as new 0.45%. Using a proportionate relationship, we can estimate the outlet temperatures at 0.45% concentration given the outlet temperatures at 0.3% concentration. This is how it works.

Proportionate Increase It is reasonable to infer that the increase in volumetric concentration will cause a proportionate change in thermal conductivity and, in turn, the heat transfer rate. Calculating the Outlet Temperature The outlet temperature at a concentration of 0.3% was previously determined. Using the ratio of the concentrations, we can now calculate the outlet temperature at 0.45% concentration.

3.3 Numerical calculations

For 0.45% volumetric concentration of nanoparticles in water

Temperature of the Inlet (T inlet T inlet) 300 K The temperature at a concentration of 0.3%

CuO 399 K

Al2O3 387.36 K;

TiO3 387.01 K;

SiO_2 380.58 K

Concentration Ratio

Ratio = 0.45% /0.3% = 1.5

We may calculate the temperature rise at 0.45% concentration by multiplying the temperature rise at 0.3% concentration by this ratio, assuming a linear connection between concentration and temperature change. Calculations for Each Nanofluid

CuO

$\Delta T\ 0.3\% = 399K - 300K = 99K$

$\Delta T 0.45\% = 1.5 \times 99\ K = 148.5\ K$

Toutlet, CuO, $0.45\% = 300K + 148.5\ K = 448.5\ K$

SiO_2

$\Delta T 0.3\% = 380.58K - 300\ K = 80.58\ K$

$\Delta T 0.45\% = 1.5 \times 80.58\ K = 120.87K$

Toutlet, SiO_2, $0.45\% = 300K + 120.87K = 420.87K$

TiO_2

$\Delta T\ 0.3\% = 387.01K - 300K = 87.01K$

$\Delta T\ 0.45\% = 1.5 \times 87.01K = 130.515K$

Toutlet, TiO_2, $0.45\% = 300K + 130.515\ K = 430.515K$

Al_2O_3

$\Delta T\ 0.3\% = 387.36K - 300K = 87.36K$

$\Delta T\ 0.45\% = 1.5 \times 87.36K = 131.04K$

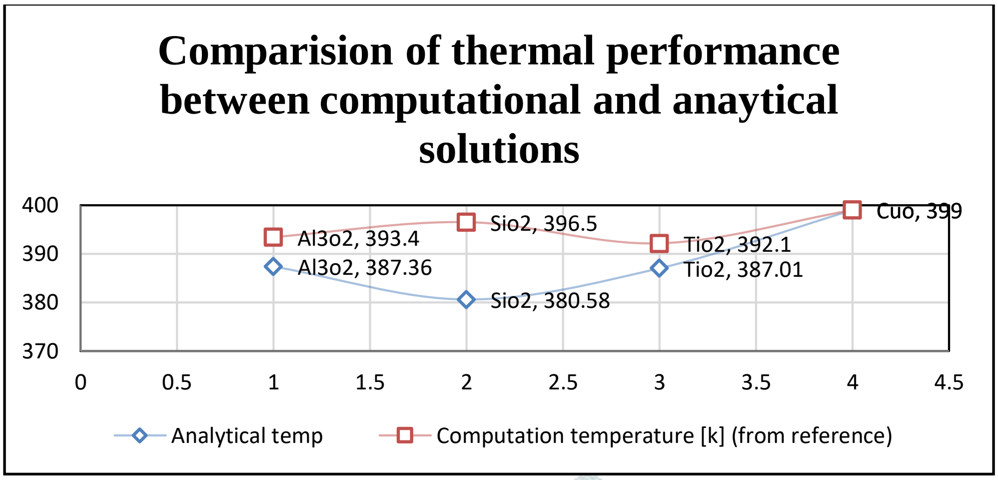
Toutlet, Al_2O_3, $0.45\% = 300K + 131.04K = 431.04K$

The temperature contours show that the use of nanofluids has raised the working fluid's output temperature. As a result, the temperature has risen in step with the volumetric concentration of the nanoparticles.

The temperature is raised via the employment of nanoparticles, which enhance the thermophysical properties of working fluids, particularly heat conductivity. The characteristics of the working fluid grow together with the volumetric concentration of nanoparticles, and this also affects the working fluid's outlet temperature.

Nanofluids	Maximum temperature, k (analytical)	Maximum Temperature , k (from reference)#[2]
Al3o2	387.36	393.4
Sio2	380.58	396.5
Tio2	387.01	392.1
Cuo	399	399.0

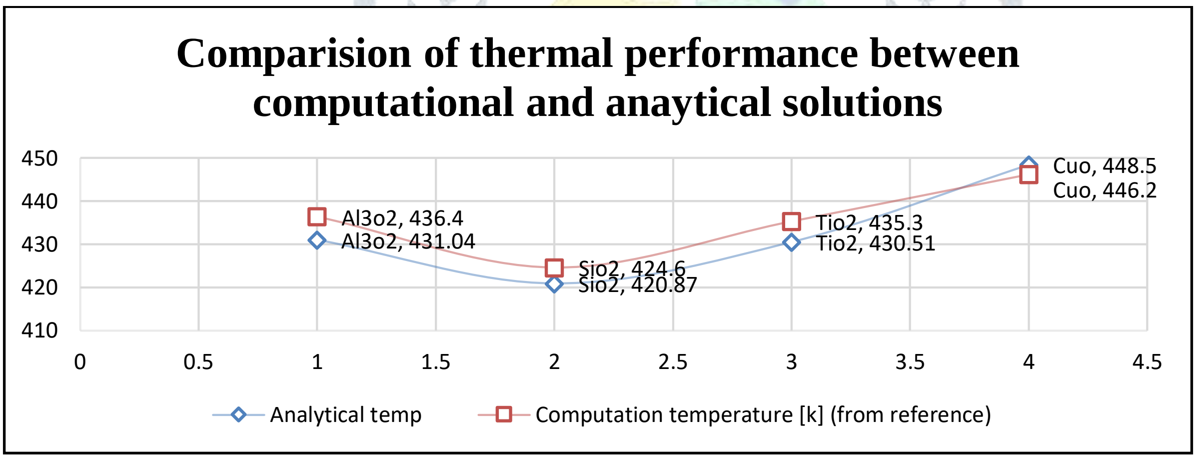
Table 3.1 Maximum outlet temperature for various nanofluids at 0.3% vol. concentration



Graph 3.1 Comparison of thermal performance between computational and analytical solutions nanofluid at 0.3% vol. concentration

NANOFLUIDS	Analytical temp	Maximum temperature [k]
Al3o2	431.04	436.4
Sio2	420.87	424.6
Tio2	430.51	435.3
Cuo	448.5	446.2

Table 3.2 Maximum outlet temperature for various nanofluids at 0.45 vol. concentration



Graph 3.2 Comparison of thermal performance between computational and analytical solutions nanofluid at 0.45% vol. concentration

When water is used as the nanofluid and 0.45% volumetric concentration of nanoparticles, the highest exit temperature of the working fluid is recorded at 50 c. The maximum outlet temperature of CUO-water nanofluid is observed at 446.2 k and the analytical max outlet temperature of CUO-water nanofluid is observed at 448.50 k

4 CONCLUSIONS

The thermal performance of the parabolic trough collector is improved, according to the CFD simulation results, and is compared with the analytical solutions by varying volumetric concentrations such as 0.3% and 0.45% respectively, when nanofluid is used as the working fluid in place of water. Furthermore, an increase in volumetric concentration is linked to a corresponding enhancement in the parabolic trough collector's efficiency.

From the obtained results a maximum output temperature of 446.2K is observed by using 0.5 kg/s of intake mass flow rate, 300 K input temperature, and 0.45% volumetric concentration whereas the data obtained from reference shows a maximum temperature output of 399 k is observed at 0.3 Volumetric concentration

Hence on comparison, we concluded that the Copper-oxide-water nanofluid gives promisable values and better thermal performance for the same mass flow rate and input temperature.

5 REFERENCES

[1] Milind S. Patil, Sanjay P. Shekhawat, Performance Assessment of Parabolic Trough Collector Receiver with Al2O3 Nanofluid, J. Water Environ. Nanotechnol., 7(3) 332-343 Summer 2022.

[2] Ratnala Prasad, Kurini Veera Lokesh, Yetukuri Durga Ramesh, Paluri Bhaskar Satish Cfd Analysis Of Parabolic Trough Collector Using Nano Fluids, Industrial Engineering Journal Issn: 0970-2555 Volume : 53, Issue 4, No. 4, April : 2024

[3] Ashutosh Shirole a, Mahesh Wagha, Vivek Kulkarnib ,Thermal Performance Comparison of Parabolic TroughCollector (PTC) Using Various Nanofluids ,Int. Journal of Renewable Energy Development (IJRED)

[4] Syed Husnain Raza a, Adnan Qamar a, Fahad Noor a, Fahid Riaz b,Muhammad Usman a, Muhammad Farooq a Experimental analysis of thermal performance of direct absorption parabolic trough collector integrating water based nanofluids forsustainable environment applications, Case Studies in Thermal Engineering

[5]M Mahendran, G C Lee, K V Sharma, A Shahrani and R A Bakar(2012), Performance of evacuated tube solar collector using water-based titanium oxide nanofluid, "Journal of mechanical engineering and sciences(JMES)", vol 3, pp. 301-310.

[6] Y Raja Sekhar, K V Sharma, R ThundilKarupparaj, C Chiranjeevi(2013), Heat transfer enhancement with Al2O3nanofluids and twisted tapes in a pipe for solar thermal applications, "Procedia Engineering" 64.pp.1474-1484

[7] S. Kakaç, A. Pramuanjaroenkij, Review of convective heat transfer enhancement with nano-fluids,Int. J. Heat Mass transfer, vol. 52, pp. 3187-3196,2009.

[8] Moghadam A.J., Gord M.F., Sajadi M., Zadeh M.H., (2014), Effect of CuO/water nanofluid on the efficiency of a flat plate solar collector, "Ex-perimental Thermal Science", vol. 58, pp. 9-14.

[9] Han D., Meng Z., Wu D., Zhang C., Zhu H., (2011), Thermal properties of carbon black aqueous nanaofluids for solar absorption, " Nanoscale Re-search Letters", vol. 6, pp. 1-7.

[10] Sangotayo, Emmanuel Olayimika Ph.D., Peter, Benjamin Aina,Ph.D., Comparative Assessment of the Effect of Thermo-Physical Properties on the Performance of Parabolic Trough Solar Collector,JASIC Volume. 1, Issue 1 (2020)