China Petroleum Processing and Petrochemical Technology

Catalyst Research Volume 23, Issue 2, December 2023 Pp. 4242-4254

NUMERICAL INVESTIGATION OF HEAT TRANSFER IMPROVEMENT FORCED CONVECTION APPLICATIONS USING PCM.

S Bala Mohan

Research Scholar, Department Of Mechanical Engineering, Jntua College Of Engineering And Technology & Anantapuramu : 515002

Dr B Omprakash

Assistant Professor, Department Of Mechanical Engineering, Jntua College Of Engineering And Technology & Anantapuramu : 515002

Abstract:

The Heat transfer is the important criteria in many industrial applications in the Present study a solid copper sphere and a hollow copper sphere filled with Paraffin Wax is Placed in side a tube and a constant heat flux of 100 Watts is to the Sphere where Air with different Reynolds number ranging from 4000 to 15000 is considered to investigate the Forced Convection Heat transfer between Spheres and Air the design of the setup is done in solid works and the simulation is done using Computational Fluid dynamics the comparison of heat transfer between Solid sphere and Sphere filled with Paraffin Wax is Done to investigate which gives best res result.

Key words: Phase Change Materials , Paraffin Wax , CFD , Heat transfer , Forced Convection.

I. Introduction

The heat transfer and flow structure passing around multiple heated bluff bodies are widely investigated problems due to their importance in several engineerings such as high temperature gas-cooled nuclear reactor - HTGR, bio-film reactor and solar receivers, etc. applications such as nuclear power plants, food and chemical processing, and so on. The flow separation complexity and the vortex shedding mechanisms, generated by the sphere arrays have attracted large attention. In the literature, numerical and experimental investigations have been conducted to analyze and understand the wake flow structure around a single sphere and the various arrangements of two spheres. In the early studies,

Advanced nuclear power plants are equipped with passive systems for emergency decay heat removal from reactor equipment (PEHRSs) to the final heat absorber (ambient air) in case of development of accidents accompanied with primary cooling circuit leakage and for transferring heat to the final heat absorber (ambient air) (Mousavian et al. 2004; Maio Vitale et al. 2012; Zhang et al. 2012; Dmitriev et al. 2013). For nuclear power plants designed, passive safety systems were firstly used in the AES-92 Project followed by all subsequent design projects (AES-2006, VVER-1200, VVER-TOI). Design of passive safety system is based on the use of special loop transferring heat to air heat exchangers (Zvirin 1981; Andrushechko et al. 2010; Safety assessment report 2013). PEHRS in its present form consists of four independent loops incorporating heat exchange equipment with air ducts (Galiev et al. 2017). Removal of heat from outer surfaces of the heat exchanger is achieved using the processes based on the action of natural forces, such as unforced natural circulation.

Hassanzadeh et al. (2013) used the LES method to predict the flow structures around two side-by-side spheres for gap ratios of G/D = 1.5, 2.0, 3.0 at Re = 5000. They showed that the wake interactions were strongly affected by the separation distance with little nozzle effect beyond G/D = 3.00.

Using dye visualisation and PIV measurements, Pinar et al. (2013) experimentally examined the flow characteristics of both a single and two side-by-side spheres at Re = 5000. Their qualitative and visual data provide insightful information for evaluating numerical models. Even little research has been done on the heat transfer for streams around two adjacent spheres at Reynolds numbers of roughly 5000.

Seyed M (2014) were investigated wake structures and turbulent flow fields of a wall-mounted spherical obstacle placed in a thin laminar boundary layer, of thickness 14 % of the sphere diameter, were investigated at a Reynolds number of 17,800

Sercan Yagmur [2015] explored into during time, there has been improvement in our understanding of and ability to anticipate quantitatively changes in velocity and pressure in turbulent flows surrounding such bluff bodies. The primary goal of this work is to investigate the fluid flows in the wake region of various bluff bodies, which including circular, square, and triangle cross section cylinders positioned horizontally perpendicular to the initial condition.

Takayuki Nagata also looked at direct numerical simulation at low Re around a sphere on subsonic to supersonic flow (2016).

The flow and heat transport across two adjacent spheres were quantitatively examined by Shiyang (2017) at Re = 5000 using the large eddy simulation (LES) approach. Between the two adjacent spheres, there were gap ratios (G/D) that varied from 1.25 to 3.0. As the gap ratio falls, the cross-stream mixing strengthens. The wake zone length and the drag coefficients on the spheres are affected by a biassed gap flow that happens when G/D = 1.25.

The non-slipping adhesion contact across two elastic spheres with differing beginning temperatures is examined byPeng X and Huang (2017). Under the typically applied force is predicted.

Using direct numerical simulation (DNS) of the three-dimensional compressible Navier-Stokes equations, Takayuki Nagata(2018).numerically examined the flow parameters around an isolated sphere under isothermal circumstances for flows with high Mach numbers and low Reynolds numbers discovered at M = 0.3–2.0, Re = 100–300, with TR = 0.5–2.0, the flow around with a sphere in isolation. As a result, the impact of TR on Nu, the drag coefficient, and the flow geometry was investigated. Under high-TR conditions, the flow field stabilises in the subsonic regime, and as TR rises, the separation point shifts toward the downstream.

A H Abed looked into this in 2018. This project's goal is to assess how vortices affect the flow pattern around a heated sphere and the behaviour of heat transfer. Using a stationary copper sphere inside of a cylindrical channel with a constant channel-to-sphere diameter ratio, numerical simulation and experimental verification are carried out. ANSYS-FLUENT is used to do numerical simulation for three-dimensional steady-state flow by solving the Reynolds-averaged Navier Stokes (RANS) equations.

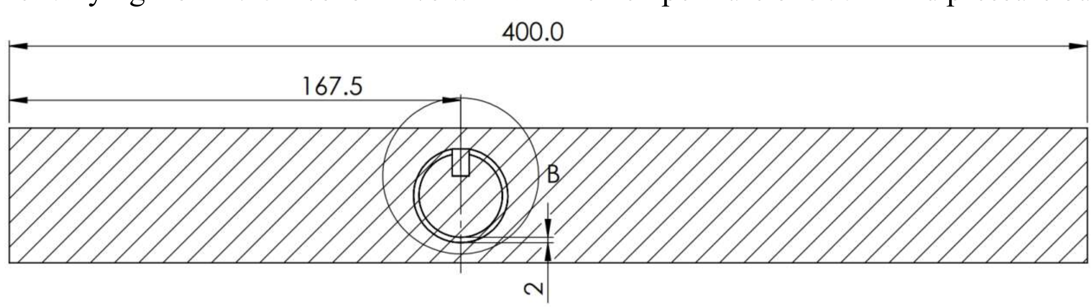
Zhang et al. (2019) quantitatively examined the impact of the distance between two vertically stacked spheres on natural heat transfer .They observed that the flow recirculation zone has a significant impact on the heat transfer coefficient, and as the distance between two separate spheres increases by about 24 times the diameter of the sphere, the flow recirculation zone occurs over the upper sphere with a circumferential angle of 142.5° to 180°. The data from a single sphere cannot accurately forecast the mass transfer process in the sphere swarms for the majority of technical and practical application.

The problem is investigated both experimentally and analytically using computational fluid dynamics software for a Reynolds number ranging from A H Abed in 2020 investigated the Convective heat transfer and flow characteristics of the heated spheres in a tandem arrangement with constant pitch and diameter located in a cylindrical tube (2500 to 5500) The experimental data and the findings of the simulation show good agreement.

A detailed explanation of flow around a spherical body and its impact on numerous engineering applications under high Reynolds numbers is provided in Md. Mamunur Rashid's (2020) study, which is based on the sphere's diameter. In this work, we use a suitable numerical method to study the flow across a sphere at Reynold's number Re 30000. The sphere experiences turbulent flow, which is represented by the standard k- and standard k-turbulence models. In-depth discussion is provided along with appropriate contours and diagrams for certain fundamental fluid features of the flow phenomenon. For both turbulence models, it is possible to view the object's complex downstream flow phenomena. The research shown that the current numerical technique and these turbulent models have a greater agreement in predicting the behaviour of flows.

II. Geometry and computational methods.

The Single Sphere is placed at 167.5 mm in an 50 mm diameter pipe with a spherical diameter of 35 mm and shell thickness of 2 mm copper sphere where a 6.25 mm heating element at 100 W is used for the simulation as shown in figure 1 and 2. Air is used as a convection fluid with which the material of the sphere used is copper and the thickness of the coper sphere is 2 mm with which paraffin wax is filled in side the hallow sphere with the properties as shown in table-1 the inlet boundary conditions are defined as velocity inlet varying from 1.15 m/s to 4 m/s with an inlet temperature of 300 K and pressure outlet



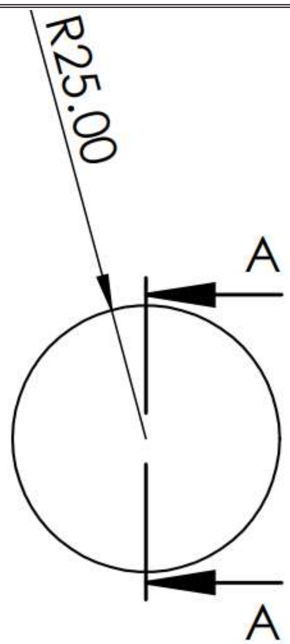
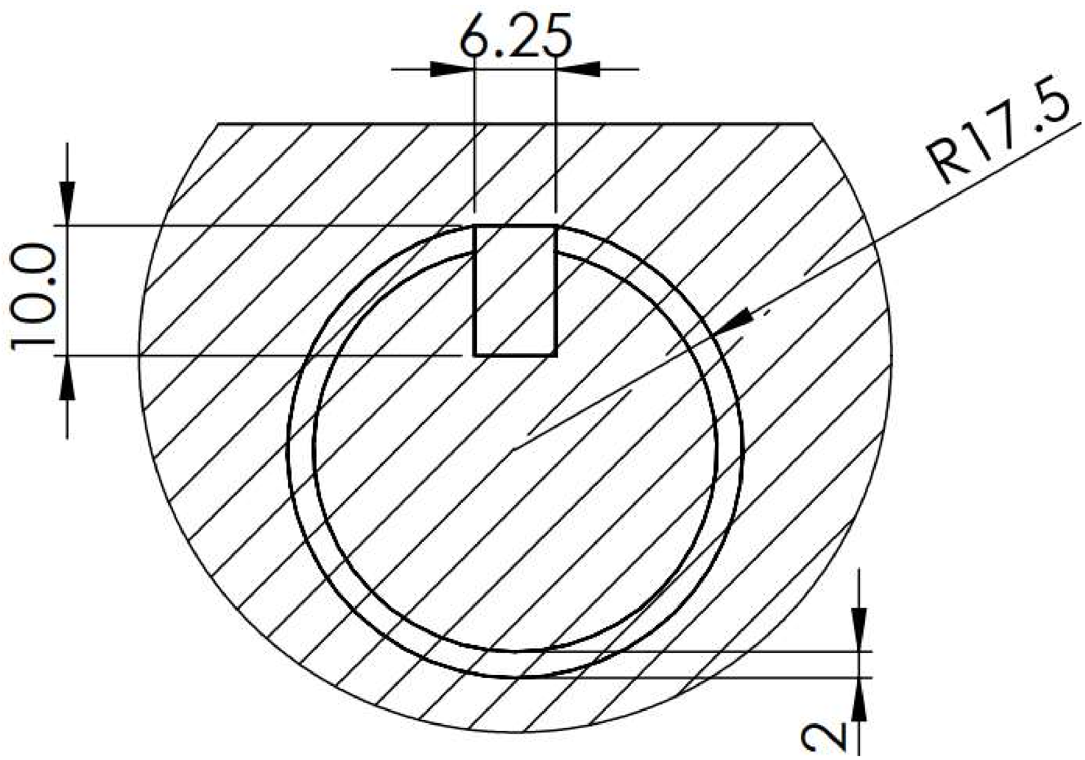


Fig 1 CFD setup used in the simulation.



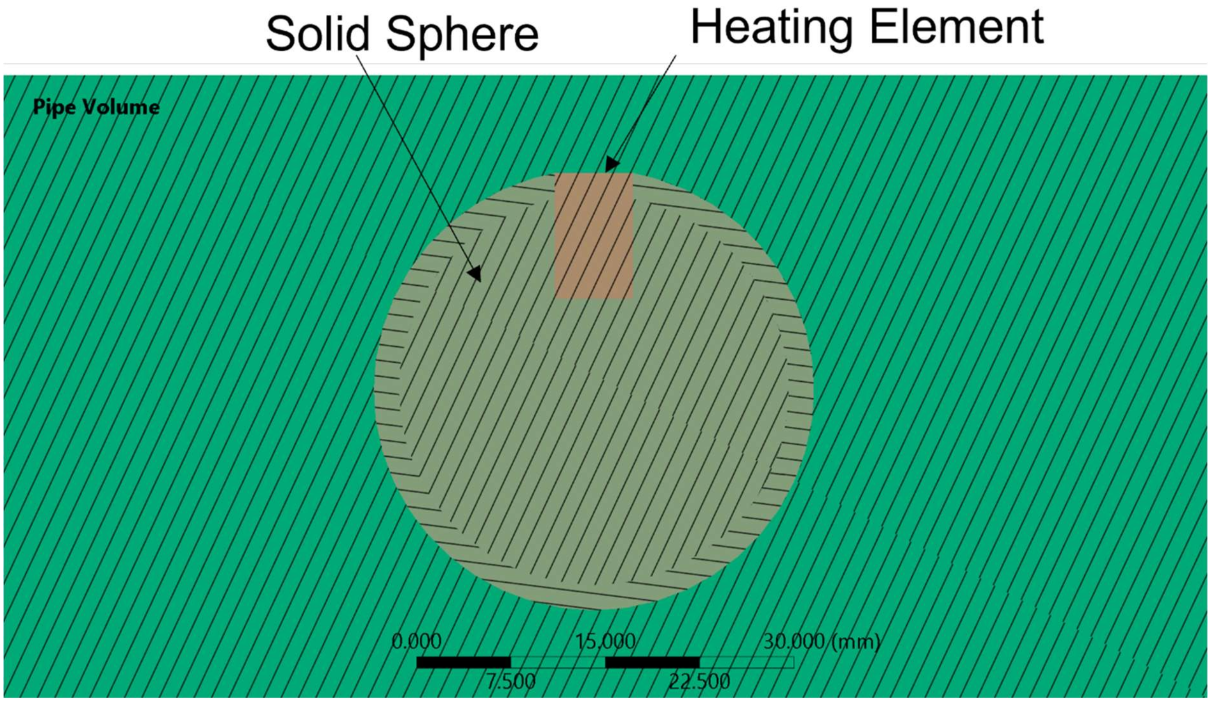


Fig 2 Sphere Configuration used in the CFD analysis.

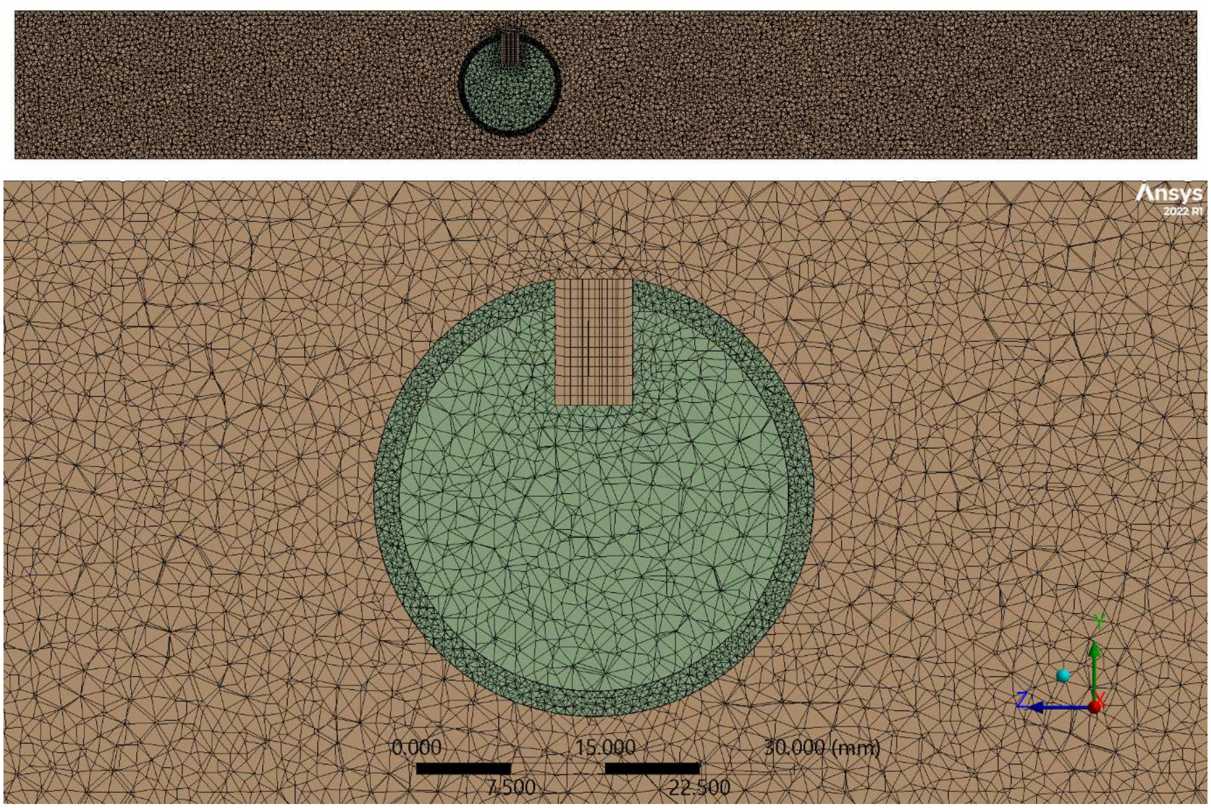


Fig 3 Meshing Done For CFD Setup

Catalyst Research Volume 23, Issue 2, December 2023 Pp. 4242-4254

The Above Fig Shows the Illustration of Meshing done for CFD test setup where the Solid sphere and Pipe Volume is created with Tetrahedron elements and the heating element is comprised with Hexahedron elements with 196769 Nodes and 1046243 elements. The CFD simulations were performed using the commercial software ANSYS FLUENT v.22. Three dimensional Reynolds-averaged Navier–Stokes equations were employed for the simulation. Both laminar and turbulent flow regimes were carried out, the Re number was in the range 4000 < Re < 15000. Laminar flow simulation was performed for Reynolds number Figure 1. Schematic view of the geometrical for CFD simulations. Figure 3. Meshes for the solid and in a region between a sphere and the wall channel (Re = 2500) and the turbulent simulations for Reynolds numbers range 4000 < Re < 15000. The K-epsilon turbulence model which blended the advantages of the k-ε and k-ω models were applied in this simulation to obtain an optimal model formulation that accurately predicts the turbulent effect near the surface and improve the surface shear stress and heat transfer predictions. The modelled equations for the SST k-epsilon model can be written as

Table 1 PCM properties

S.No	Parameter	Value	units
1	Density	8980	Kg/m3
2	Specific heat	385	J/ Kg-K
3	Thermal conductivity	381	W/m K
4	Viscosity	9.6e-6	Kg/ m-s
5	Pure solvent melting Heat	206000	J/kg
6	Solidus temperature	1350	K
7	Liquidous	1350	K

III. Results and discussion

In order to validate our model and compare reliability of the CFD simulation the initial results in steady state is compared with those presented in Abed (2019) with average surface temperature and heat transfer coefficient is shown in figure 4 with an error of 4 % at the Reynolds number of 5000 with this we can say that the method used to simulate is inline with the expectations.

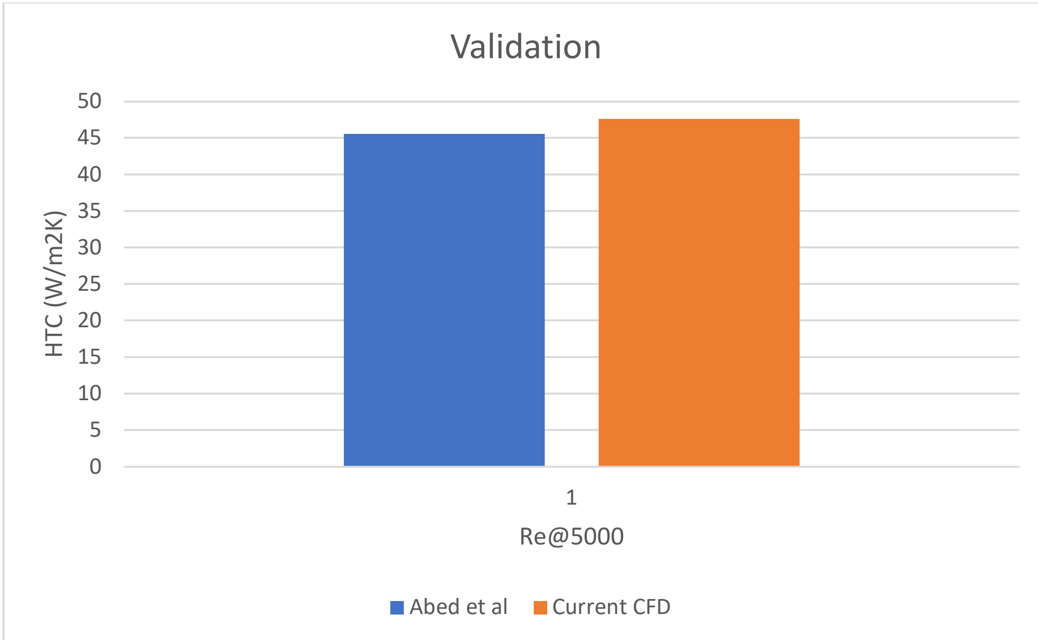


Fig 4 Validation with References

The fig 4 shows the validation of cfd model considered in this research which shows a very appreciative agreement between results with an error % of below 5%

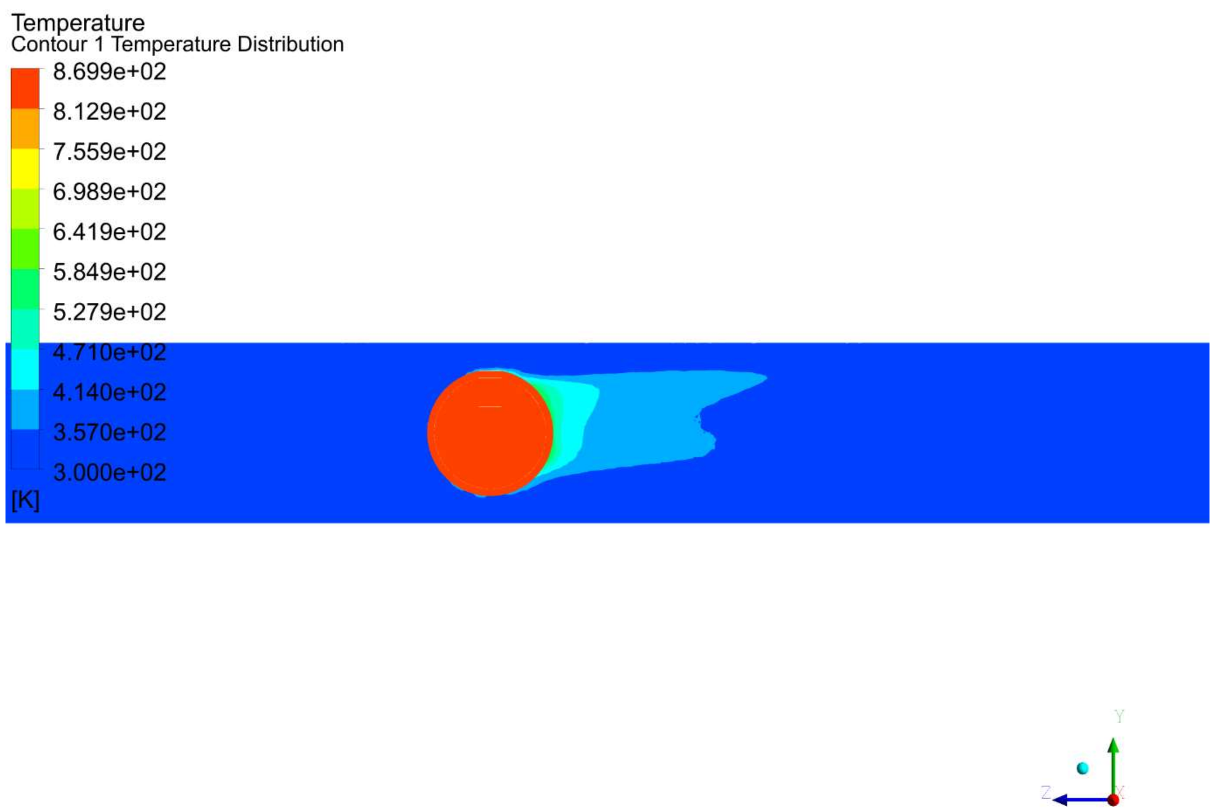


Fig 5 Temperature distribution of the flow over hot solid copper sphere.

Catalyst Research Volume 23, Issue 2, December 2023 Pp. 4242-4254

The above fig shows the distribution of the temperature inside the sphere and to the surrounding flow region the fig represents the flow is at Re=15000 red coloured region shows high temperature area and blue coloured region shows low temperature area.

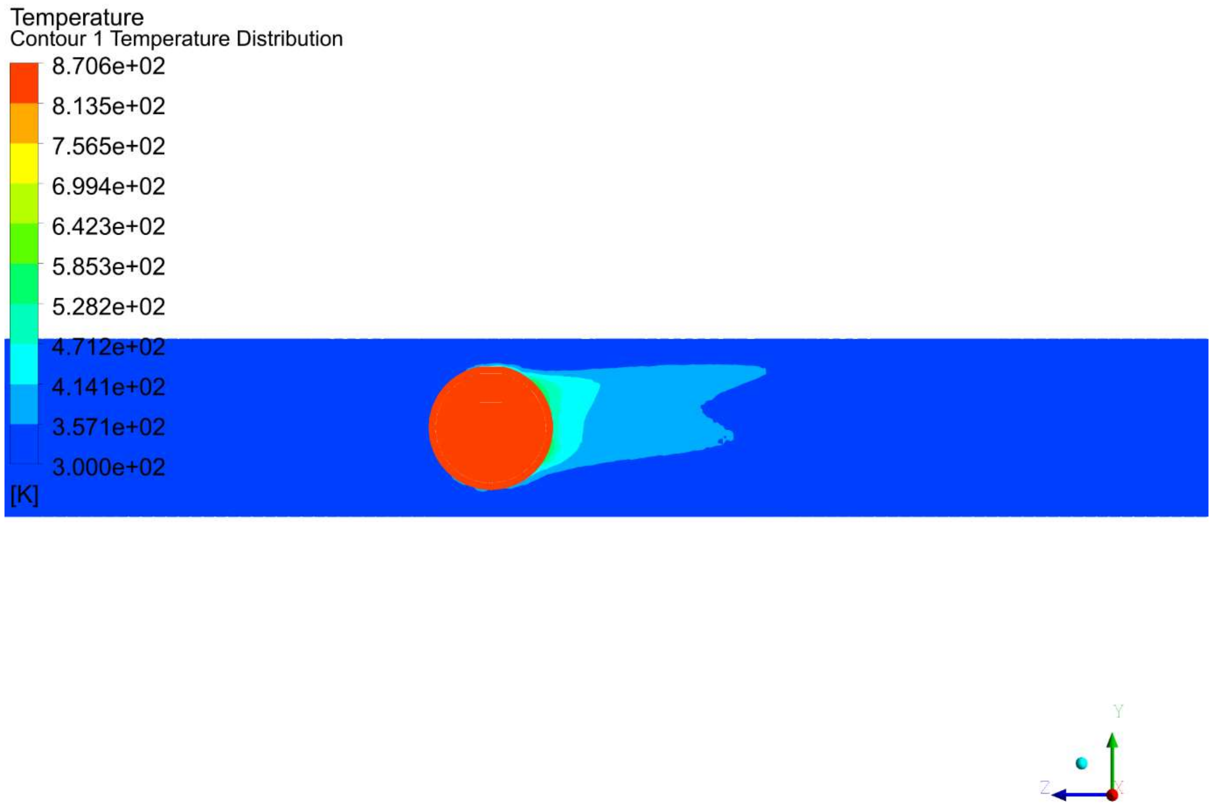


Fig 6 Temperature distribution of the flow over hot sphere filled with PCM.

The above fig shows the distribution of the temperature inside the sphere and to the surrounding flow region the fig represents the flow is at Re=15000 red coloured region shows high temperature area and blue coloured region shows low temperature area.

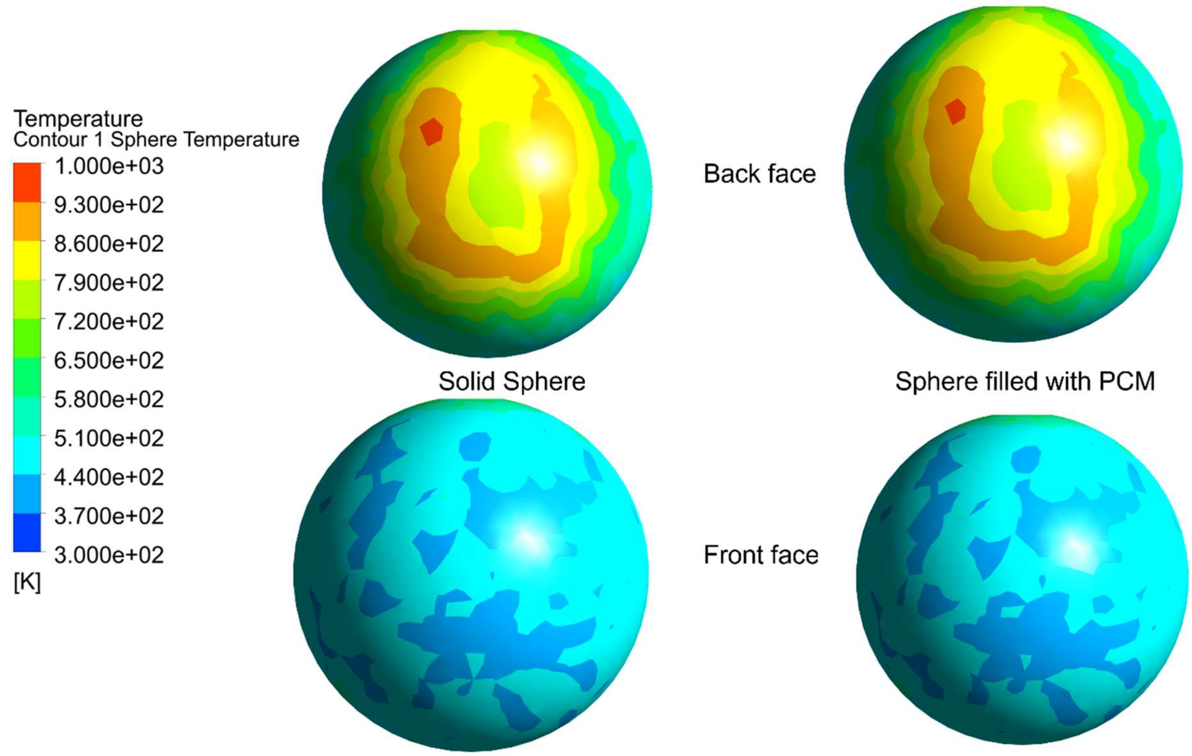


Fig 7 Temperature distribution on sphere @ Re=4000

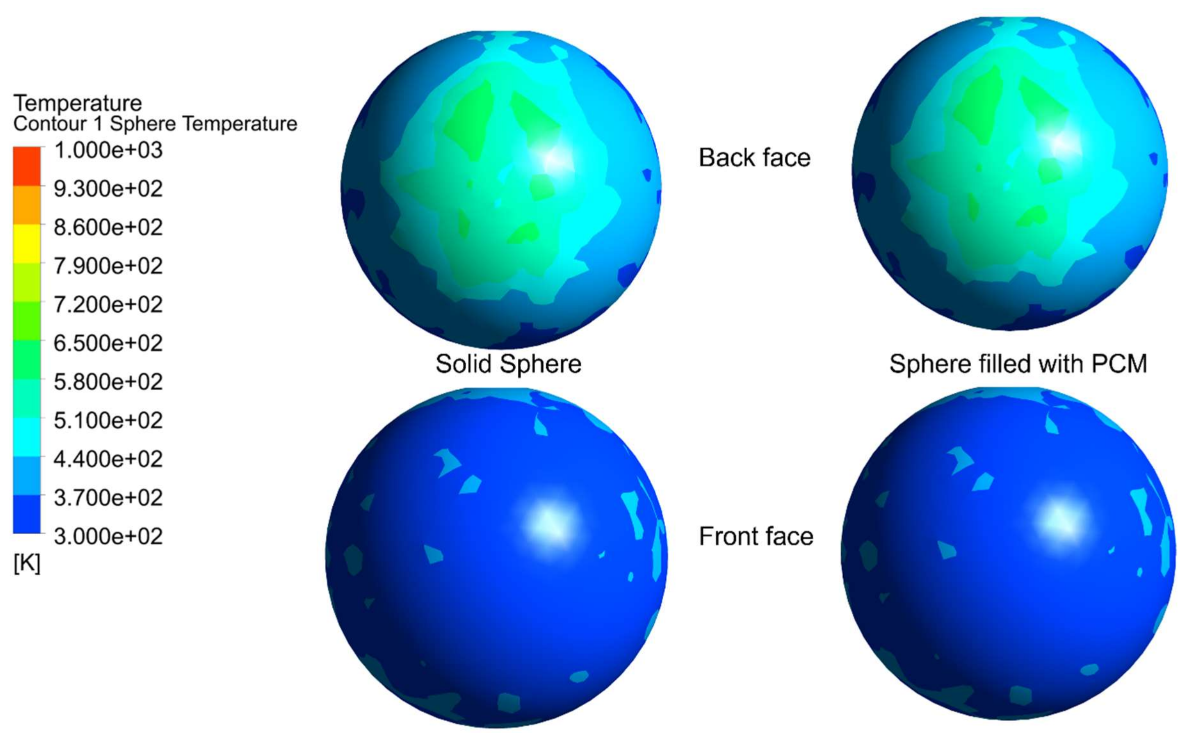


Fig 8 Temperature distribution on sphere @ Re=15000

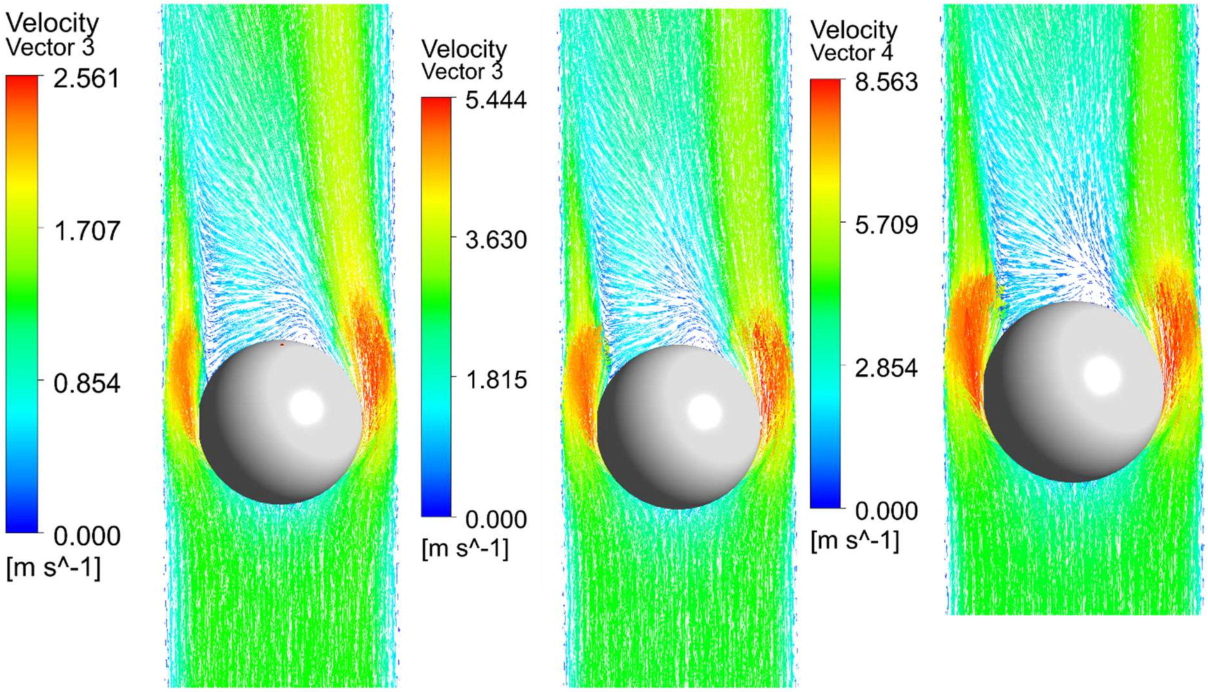


Fig 9 Velocity Vector profiles for spheres @ Re=4000,9000,15000

Table 2 Heat transfer Values Generated from CFD

Heat transfer coefficient					
Solid Sphere				Sphere filled with PCM	
S.No	Re	Heat transfer Coefficient (W/m2K)	Nusselt Number	Heat transfer Coefficient (W/m2K)	Nusselt Number
1	4131.231672	45.8348	94.6999	46.8108	96.6918
2	5388.56305	47.6115	98.3708	48.643	100.4772
3	7184.750733	49.8016	102.896	50.934	105.211
4	8980.938416	51.3507	106.096	52.2729	107.977
5	10777.1261	52.024	107.488	52.7342	108.93
6	12573.31378	50.9237	105.214	52.0804	107.579
7	14369.50147	53.2352	109.99	53.4749	110.461

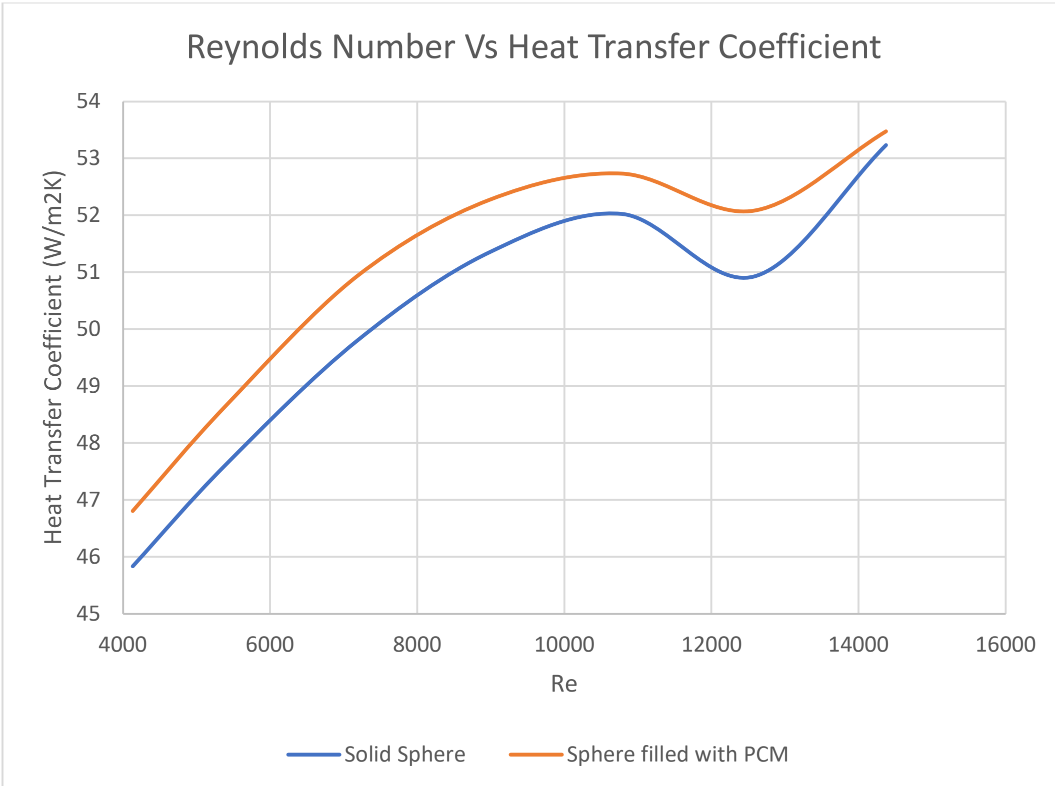


Fig 9 Re Vs Heat transfer coefficient for solid sphere and sphere filled with PCM

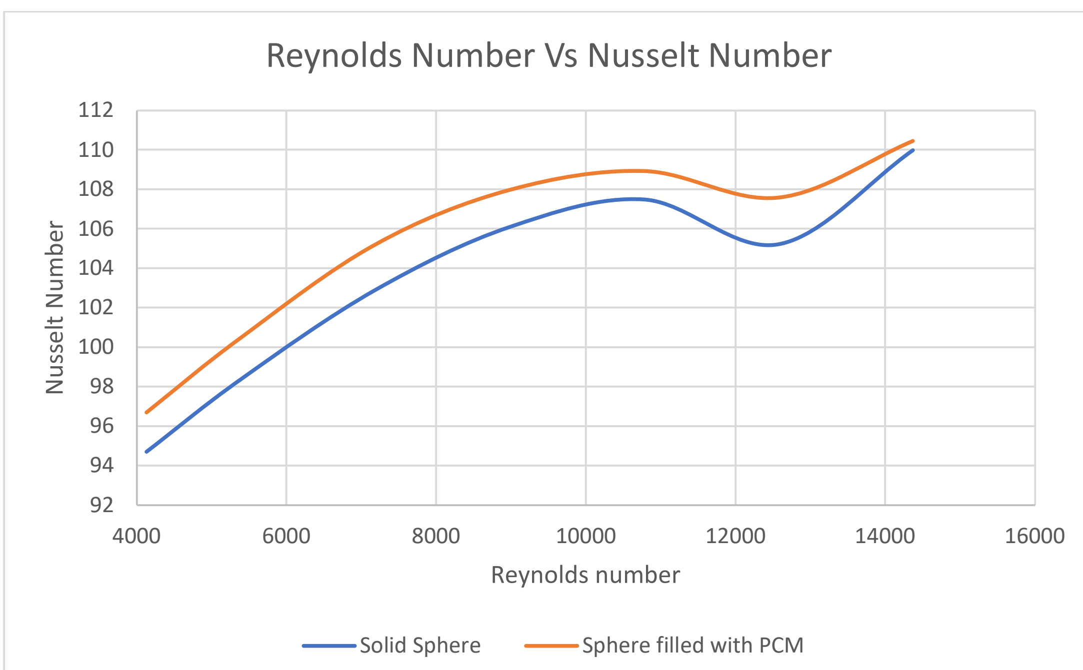


Fig 10 Re Vs Nusselt Number for solid sphere and sphere filled with PCM

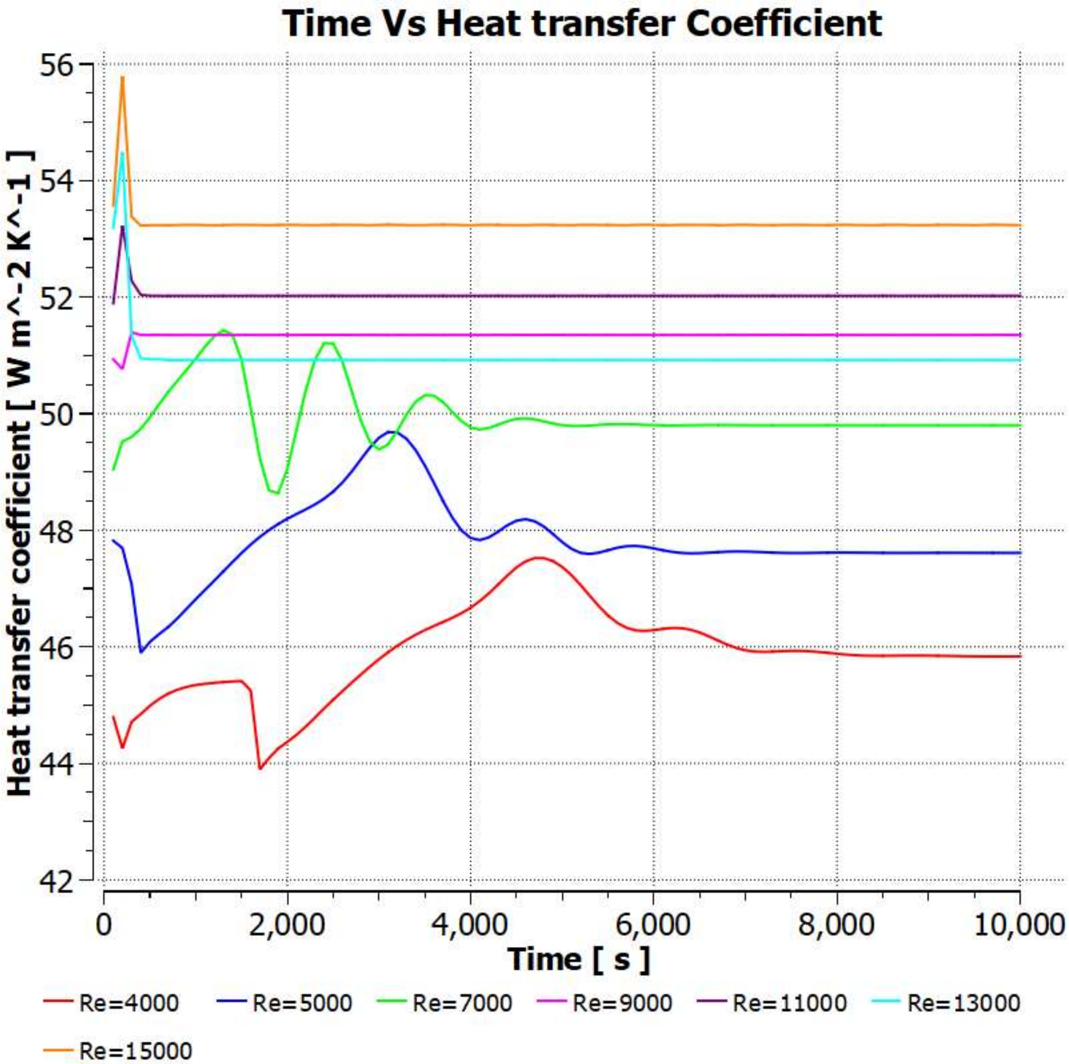


Fig 11 Time Vs heat transfer coefficient for Sphere Filled with PCM

IV. Conclusion.

Heat transfer and flow characteristics around the heated solid sphere and sphere filled with PCM were studied numerically in this paper impact of PCM on the heat transfer behaviour was studied of RE numbers and the constant heat flow conditions heat transfer coefficient and Nusselt number is calculated it is observed from the simulation done Sphere filled with PCM is showing higher heat transfer coefficient when compared to the solid sphere it is also observed that the frontal area of the Sphere is being cooled and the rear area where the wake region is forming is having lower heat transfer coefficient.

References

1. R. Hassanzadeh, B. Sahin, M. Ozgoren," Large eddy simulation of flow around two side-by-side spheres", J. Mech. Sci. Technol. 27 (2013) 1971–1979.
2. E. Pinar, B. Sahin, M. Ozgoren, H. Akilli," Experimental study of flow structures around side-by-side spheres", Ind. Eng. Chem. Res. 52 (2013) 14492–14503.
3. Seyed M. Hajimirzaie , Achilleas G. Tsakiris , James H. J. Buchholz,Athanasios N. Papanicolaou," Flow characteristics around a wall-mounted spherical obstacle in a thin boundary layer" Exp Fluids (2014) 55:1762.
4. Sercan Yagmur,Sercan Dogan , Muharrem H. Aksoy , Eyub Canli, Muammer Ozgoren "Experimental and Numerical Investigation of Flow Structures around Cylindrical Bluff Bodies" EPJ Web of Conferences ,92,02113, (2015)
5. Nagata T, Nonomura T, Takahashi S, Mizuno Y and Fukuda K. 2016. Investigation on subsonic to supersonic flow around a sphere at low Reynolds number of between 50 and 300 by direct numerical simulation Physics of fluids 28, 056101
6. Shiyang Li, Jian Yang, Qiuwang Wang "Large eddy simulation of flow and heat transfer past two side-by-side spheres" Applied Thermal Engineering ,Volume 121,Pages 810-819,2017.
7. Peng X and Huang G 2017," Effect of temperature difference on the adhesive contact between two spheres" Int. J. of Engineering Science 116 25-34.
8. Nagata T, Nonomura T, Takahashi S, Mizuno Y and Fukuda K 2018," Direct numerical simulation of flow around a heated/cooled isolated sphere up to a Reynolds number of 300 under subsonic to supersonic conditions" Int. J. of Heat and Mass Transfer 120 284-99.
9. A H Abed, S E Shcheklein 2018," Numerical simulation and experimental investigation of heat transfer and flow structures around heated spherical bluff bodies", Journal of Physics: Conference Series 1333 (2019) 032002 IOP Publishing
10. Zhang J, Zhen Q, Liu J and Lu W 2019 ,"Effect of spacing on laminar natural convection flow and heat transfer from two spheres in vertical arrangemen",t Int. J. of Heat and Mass Transfer 134 852-65.
11. A H Abed, S E Shcheklein 2020 Numerical and Experimental Investigation of heat transfer and flow structures around three heated spheres in tandem arrangement IV International Scientific and Technical Conference "Energy Systems 791 (2020) 012002.
12. Md. Mamunur Rashid , M.G.M. A Faruque," Numerical Investigation of Flow Over a Spherical Body at High Reynold's Number "International Journal of Scientific & Engineering Research Volume 11, Issue 3, March-2020 ISSN 2229-5518.