ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Analysis of Heat Transfer in a Cylinder Comprises Composite Spheres over the Flow

Suresh Kumar Khatroth
Department of Mechanical Engineering,
Malla Reddy College of Engineering and Technology,
Secunderabad, Telangana-500100, India.

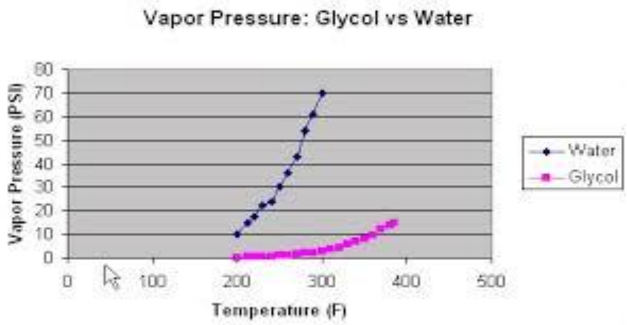
ABSTRACT:

The aim of the contribution is an analysis of transient heat transfer in a cylinder with an inner heat source. The object of the consideration is a multilayered composite sphere. The heat conduction in the cylinder comprising multilayered composite spheres is governed by finite volume method. Mathematical (classical) or physical formulations of the boundary condition and the perfect contact of the composite sphere and spherical layer is assumed. The boundary condition and the heat flux continuity condition at the interface are expressed by the Riemann-Lowville derivative. An exact solution of the problem under mathematical conditions is determined.

Thus in this project, "Analysis of Heat transfer In a Cylinder Comprises Composite Spheres over The Flow" the present composite spheres (graphite, steel, graphite) has been studied along with its thermal analysis in CFD and an enhancement in its flow resistance is made to decrease the flow resistance occurred to the composite spheres due to this heat is generated between the flow and composite spheres as we are having inside the composite spheres it will absorb. The heat generated due to the flow as the steel member absorbing more heat it will get burst if it goes on like that, so that brine solution is using to cool the steel member.

INTRODUCTION

In this model the conduction and convection takes place between the layers and coolant. The coolant used is generally a mixture of water and glycol, and depending on the application different glycol concentrations can be used. Fig shows the saturation properties (saturation Pressure and saturation temperature) of coolant for different glycol concentrations [1], [5].



Saturation properties of Water-Glycol coolant for different glycol concentrations. Information gathered from coolant properties tables.

The analysis was carried out throughout the domain and plots the temperature, velocity and pressure drop contours. The fundamental of the classical heat transfer theory is the Fourier law which leads to the parabolic partial differential equation of the heat conduction. A consequence of the Fourier's law is unrealistic speed of heat flow in the medium. This inconvenience can be avoided by a generalization of the Fourier law which leads to a fractional heat conduction equation. The heat conduction is governed by the fractional differential equation, If the heat transfer in a bounded medium is considered then the heat equation is complemented by boundary conditions[3]. The Dirichlet, Neumann and Robin boundary conditions are often used in describing the heat transfer between the body and the surroundings.

Cite this article as: Suresh Kumar Khatroth, "Analysis of Heat Transfer in a Cylinder Comprises Composite Spheres over the Flow", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 6 Issue 8, 2019, Page 65-71.

In the classical heat theory, the Neumann and Robin boundary conditions include the normal derivative at the boundary of the considered region. Introducing the time-fractional derivative in the Neumann and Robin boundary conditions, the physical formulation of these conditions is obtained.

In this paper, we present the solution of the fractional heat conduction problem in a cylinder consisting of composite spheres and a spherical layer. The mathematical and physical formulation of the Robin boundary conditions is considered. The perfect thermal contact of the inner sphere and the spherical layer is assumed. The effect of the fractional order on the temperature distribution in the sphere has been numerically investigated [7].

COMPUTATIONAL FLUID DYNAMICS

CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of computer-based simulation. It consists of solving the discretized governing equations of fluid flow that describes the conservation laws of physics. There are various mathematical models that describe the movement of fluids and various engineering correlations that can be used for special cases. However, the most complete and accurate description comes from partial differential equations (PDEs) [9]. For instance, a flow field is characterized by balance in mass, momentum, and total energy described by the continuity equation, the Navier-Stokes equations, and the total energy equation:

$$\begin{aligned}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) &= -\nabla p + \nabla \cdot \tau + \mathbf{F} \\
\frac{\partial}{\partial t}\left[\rho\left(e + \frac{1}{2}u^2\right)\right] + \nabla \cdot \left[\rho \mathbf{u}\left(e + \frac{1}{2}u^2\right)\right] &= \nabla \cdot (k \nabla T) + \nabla \cdot (-p\mathbf{u} + \tau \cdot \mathbf{u}) + \mathbf{u} \cdot \mathbf{F} + Q
\end{aligned}$$

The solution to the mathematical model equations gives the velocity field, ; pressure, p; and temperature, T; of the fluid in the modeled domain.

CFD Methodology

The methodology to solve CFD problems consists of three main steps:

- pre-processing,
- processing and simulations, and
- post-processing of the results

Pre-processing:

Preprocessor consists of input of a flow problem by means of an operator friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The user activities at the Pre-processing stage involve:

1) Definition of the geometry of the region: The computational domain. Grid generation is the subdivision of the domain into a number of smaller, no overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modeled).

2) Definition of fluid properties: Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability [10]. Ultimately such programs will automatically refine the grid in areas of rapid variation.

Processing and simulation:

These are three distinct streams of numerical solutions techniques: finite difference, finite volume& finite element methods. In outline the numerical methods that form the basis of solver performs the following steps:

1) The approximation of unknown flow variables are by means of simple functions

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

2) Discretization by substitution of the approximation into the governing flow equations & subsequent mathematical manipulations.

Post-processing:

As in the pre-processing huge amount of development work has recently has taken place in the post processing field. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools.

These include:

1) Domain geometry & Grid display

2) Vector plots

3) Line & shaded contour plots

4) 2D & 3D surface plots

5) Particle tracking

6) View manipulation (translation, rotation, scaling etc.)

Flow Setup

STEP 1: creating setup for the solution

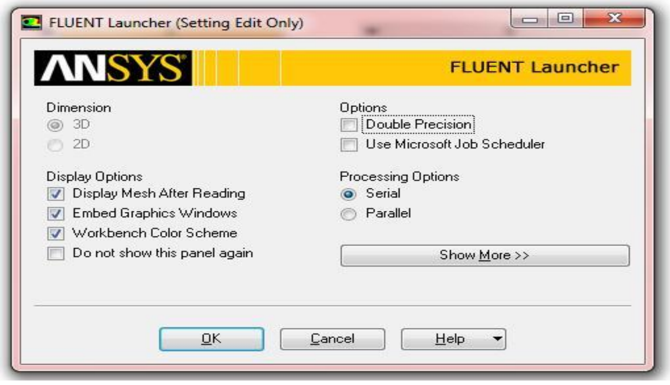


Figure 1: Showing Fluent launcher

- Click on set up FLUENT launcher window opens- ok as shown in above figure 5.1
- In FLUENT under problem setup check is performed to check the geometry and scale is changed to mm.
- In solver select the type as pressure-based, the velocity formulation as absolute and st the time as transient.

STEP2: In model under model setup select the energy equation and viscous as standard k-epsilon method. The standard k- ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart [8].

- In the derivation of the k- ε model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k- ε model [6] is therefore valid only for fully turbulent flows.

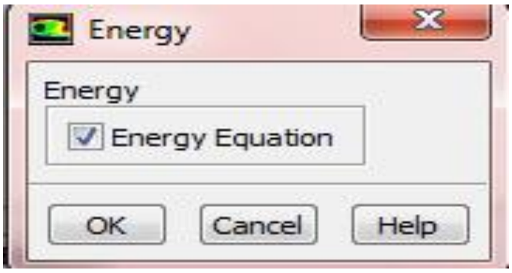


Figure 2: Represents the selection of energy equation in fluent.

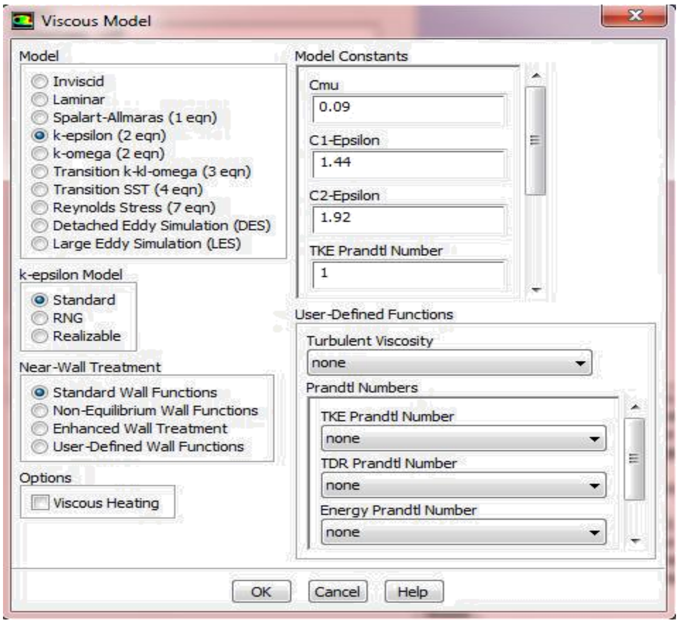


Figure 3: Selection of k-epsilon method under viscous model

- In materials select fluid – density – ideal gas click on change/create – close.

STEP 3: Selecting the cell-zone conditions as fluid.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

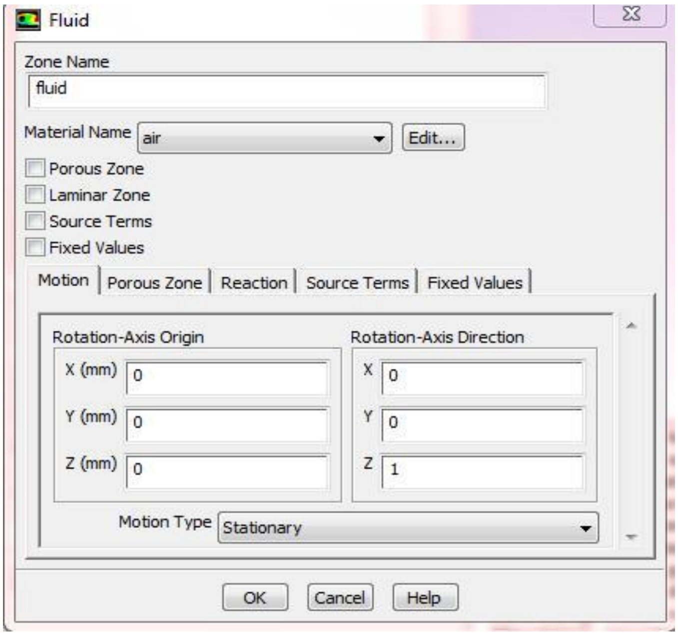


Figure 4: Specifying the cell zone conditions

STEP 4: In boundary conditions specify bottom and top as interface, inlet as velocity-inlet, outlet as pressure-outlet, wing tip and wing root as wall condition. The Boundary Conditions task page allows you to set the type of a boundary and display other dialog boxes to set the boundary condition parameters for each boundary.

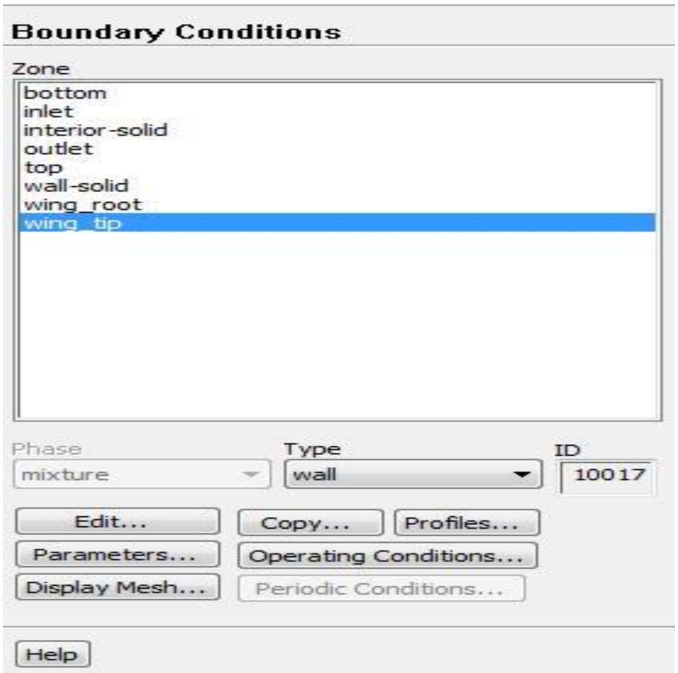


Figure 5: defining the boundary conditions.

- Giving Wall-solid as pressure-far-field condition with mach 0.9 and velocity 297 m/s.

STEP 5:

- Specifying mesh interfaces as top and bottom.
- Under reference value in problem setup mention wall-solid to compute the solution and select the reference zone as fluid as specified in cell-zone conditions [4].

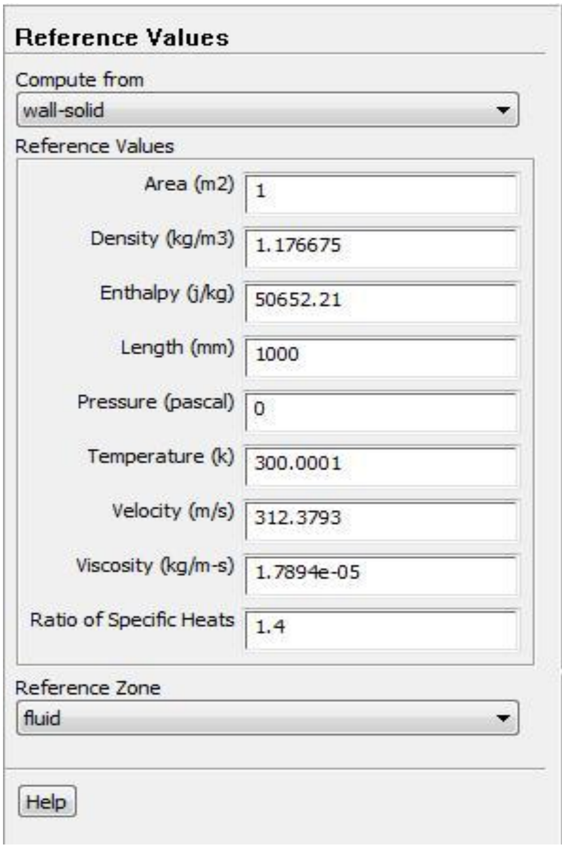


Figure 6: shows specifying the reference values

In solution method select the scheme as coupled.

The coupled algorithm solves the momentum and pressure-based continuity equations together. The full implicit coupling is achieved through an implicit discretization of pressure gradient terms in the momentum equations, and an implicit discretization of the face mass flux, including the Rhie-Chow pressure dissipation terms [2].

In solution initialization enter wall-solid in compute from tab and then click on initialize.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

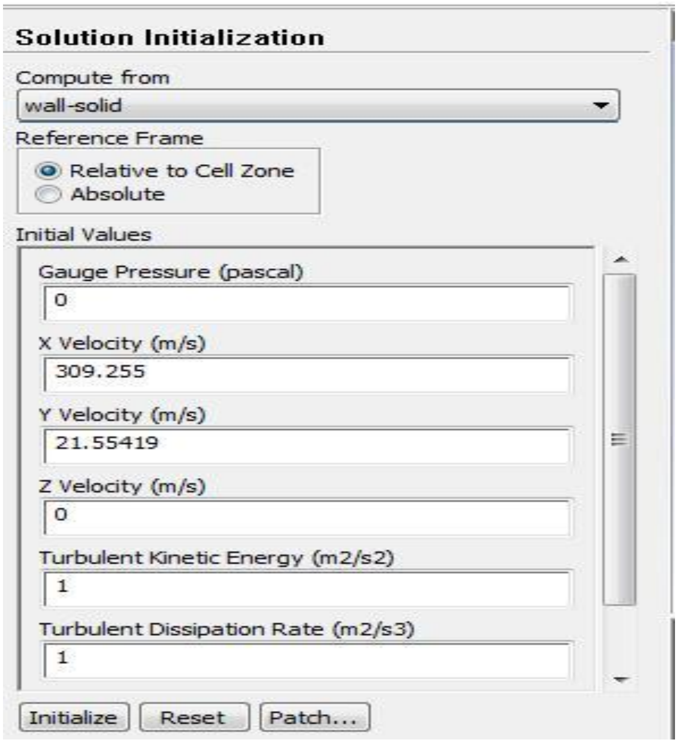


Figure 7: Shows solution initialization to compute from wall-solid

In Run calculation enter the no. of iterations as 20 and then check case, then click on calculate to start the iterations. Specifying the time step and number of time steps as shown in the figure below 8.

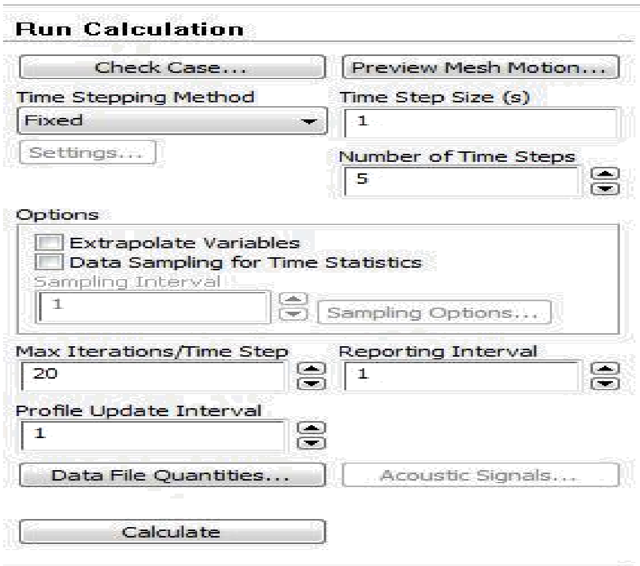


Figure 8: Specifying the iterations and time step in run calculations.

After completion of the iterations the lift, drag and moment graphs are obtained with respect to the iteration, as shown in below figures 9.

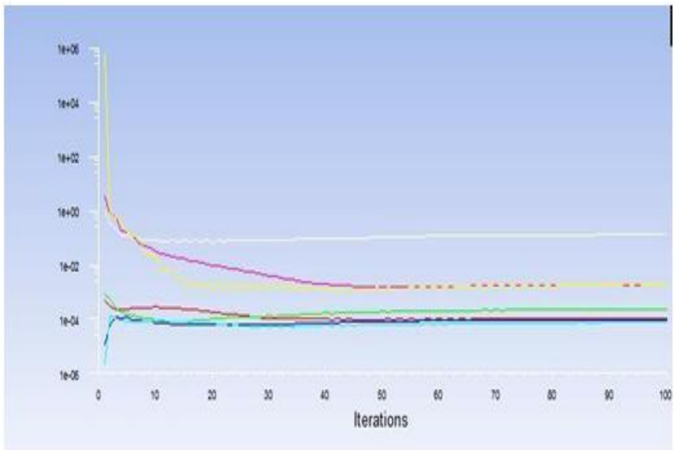


Figure 9: Scaled residuals graph

RESULTS AND DISCUSSIUON

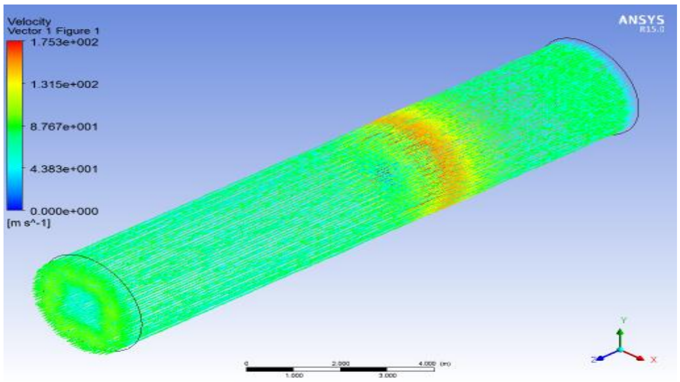


Figure 1: Velocity vectors (showing flow direction), at velocity 44 m/s.

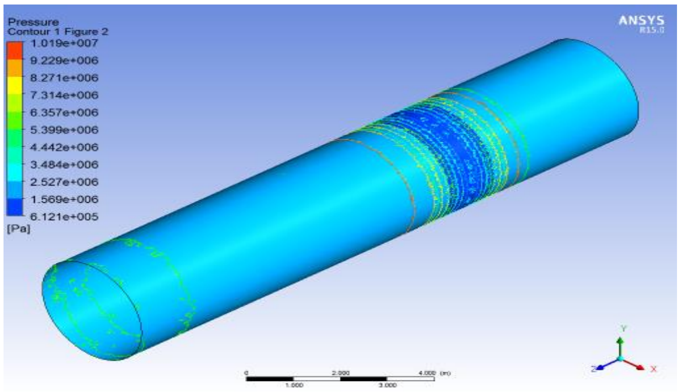


Figure 2: Pressure contours on the cylinder wall aroused at velocity 44 m/s.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

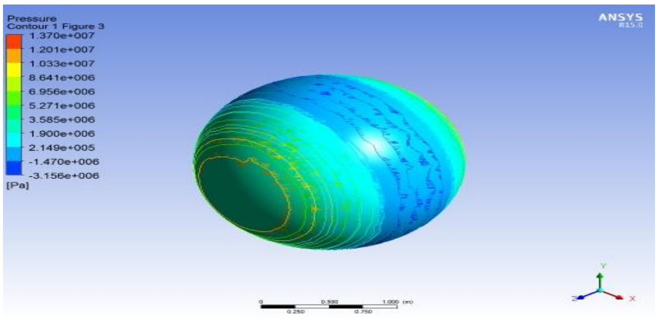


Figure 3: Pressure contours on the composite sphere, for the velocity 44 m/s.

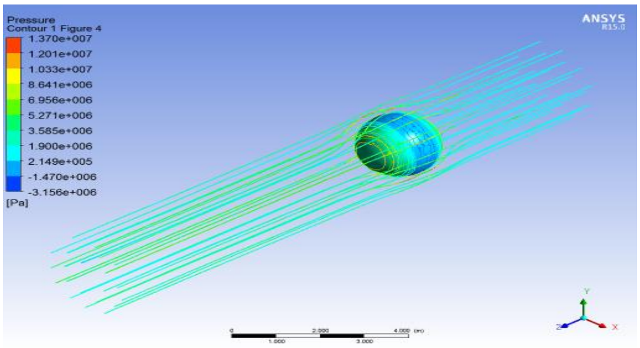


Figure 4: Stream line and pressure contours on the composite sphere, flowing over the cylinder.

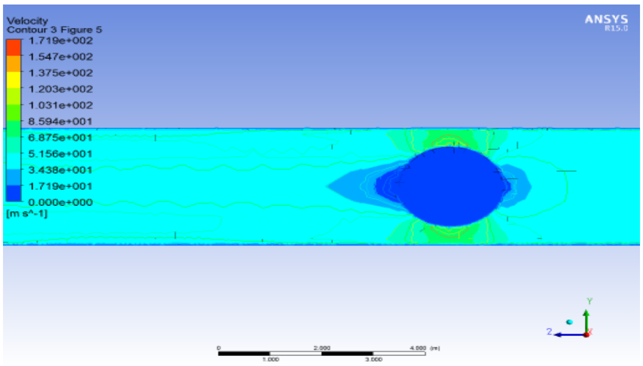


Figure 5: Velocity contours of Cylinder-Sphere tube at velocity 44 m/s.

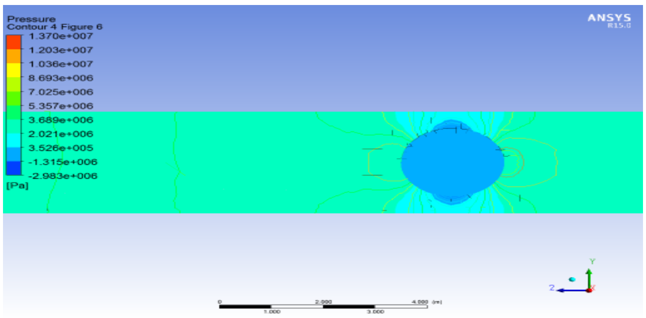


Figure 6: Pressure contours of Cylinder-Sphere tube at velocity 44 m/s.

CONCLUSION

Thus in this project, "Analysis of Heat Transfer In a Cylinder comprises Composite Spheres over The Flow" the present composite spheres (graphite, steel, graphite) has been studied along with its thermal analysis in CFD and an enhancement in its flow resistance is made to decrease the flow resistance occurred to the composite spheres due to this heat is generated between the flow and composite spheres as we are having inside the composite spheres it will absorb. The heat generated due to the flow as the steel member absorbing more heat it will get burst if it goes on like that, so that water-glycol solution is using to cool the steel member.

About this project finally I observed the analysis of the flow is having certain pressure, velocity and temperature drop occurs in the composite spheres.

FUTURE SCOPE

The influence of spherical particles on the radioactive properties of high-porosity fibrous composites is examined. The radioactive properties formulation is based on the independent scattering, which utilizes the solution of Maxwell's equations for the interaction of electromagnetic waves with the respective types of particles. The extinction, absorption and scattering coefficients, as well as the scattering phase function of the composites are formulated as the weighted average according to the fractional volume of the constituent particulates. The models are applied to predict the radioactive properties of thermal insulation composites containing fibers and spheres that are made of zirconium and silica. Numerical are results are presented to illustrate the influence of respective types of particulates on the radioactive properties of the composites.

REFERENCES

[1] Bhramara P." CFD Analysis of Two Phase Flow in a Horizontal Pipe – Prediction of Pressure Drop" International Journal of Aerospace and Mechanical Engineering 3:2 2009.

[2] Perumal Kumar," A CFD Study of Turbulent Convective Heat Transfer Enhancement in Circular Pipe flow" World Academy of Science, Engineering and Technology 68 2012.

[3] Jang J. Y., and Chen L. K., 1997, "Numerical analysis of heat transfer and fluid flow in a three-dimensional wavy-fin and tube heat exchanger," International Journal of Heat and Mass Transfer, 40(16), pp. 3981–3990

[4] Sahu M.," Developed Laminar Flow In Pipe Using Computational Fluid Dynamics",7th International R&D Conference on Development and Management of water and Energy Resources,4-6 Feb.2009,Bhubaneshwar,India.

[5] Bhore V.R (November 5 -11, 2005) "Performance Comparison of Flat Plate Collectors Fitted with Non Circular Risers with Integral Fins", ASME International Mechanical Engineering Congress And Exposition, Orlando, Florida.

[6] Leu J.-S., Wu Y.-H., and Jang J.-Y., 2004, "Heat transfer and fluid flow analysis in plate-fin and tube heat exchangers with a pair of block shape vortex generators," International Journal of Heat and Mass Transfer, 47(19-20), pp. 4327–4338.

[7] Zhang J.-F., He Y.-L., and Tao W.-Q., 2009, "3D numerical simulation on shell and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles – Part II: Simulation results of periodic model and comparison between continuous and non continuous helical baffles," International Journal of Heat and Mass Transfer, 52(23-24), pp. 5381–5389.

[8] Shih T. H., Liou W. W., Shabbir A., Yang Z., and Zhu J., 1995, "A new k-ϵ eddy viscosity model for high reynolds number turbulent flows," Computers and Fluids, 24(3), pp. 227–238.

[9] Žukauskas A., and Ulinskas R., 1988, Heat transfer in tube banks in crossflow, Hemisphere Pub. Corp., New York.

[10] Ravagnani M. A. S. S., Silva A. P., Biscaia E. C., and Caballero J. A., 2009, "Optimal Design of Shell-and-Tube Heat Exchangers Using Particle Swarm Optimization," Ind. Eng. Chem. Res., 48(6), pp. 2927–2935.