CFD Analysis of counter flow pipe heat exchanger for various absorbent combination of vapors absorption refrigeration system

1st Hitesh Kumar Sahu, 2nd Devnarayan Dewangan, 3rd Dr.Shailesh Kumar Verma

Assistant Professor, Department of Mechanical Engineering, Shri Shankracharya Institute of Professional Management and Technology, Raipur , Raipur, India

hiteshsahu72@gmail.com

Assistant Professor, Department of Mechanical Engineering, Shri Shankracharya Institute of Professional Management and Technology, Raipur , Raipur, India

dndewangan9@gmail.com

Assistant Professor, Department of Mechanical Engineering, Shri Shankracharya Institute of Professional Management and Technology, Raipur, Raipur, India

dr.skverma16@gmail.com

ABSTRACT— Technology has changed the way we live. With recent advancement in the field of science, better living condition along with efficient problem-solving methods are available. In the areas like energy resources, there is a varied of advancement based on diverse factors like availability, location, geographical condition, etc. However, the greenhouse effect's growing is concerned and the conventional energy sector's depletion are forcing engineers and scientists to shift toward nonconventional sources of energy. Solar power is the best option among renewable energy sources for refrigeration and air conditioning since it coincides with the greatest cooling load and the period of maximum solar radiation input.

By changing fluid velocity, an outlet temperature in the heat exchanger within the generator is examined using a computational fluid dynamics model. A binary solution composed from absorbent and refrigerant acts as the working fluid in an absorption system. This project illustrates the different conditions which arise by used of three different refrigerant absorbent pairs like aqua ammonia, lithium bromide-water and sodium Thiocyanate-ammonia Design modeler is employed to finish the model. The temperature variation at the heat exchanger's outlet is investigated with ANSYS, and the difference between the starting point and final temperatures is investigated. For different fluid velocities for different refrigerant absorbent pairs are noted. The efficiency of heat exchanger is calculated. Thus, the best effective pair is selected based on the result for the efficiency of heat exchangers have an important effect on the solar absorption refrigeration system's COP. This project helps in determining the correct pair of refrigerant absorbent to be used in the city during specific month period for preservation of food and vaccines.

Keywords—CFD, Refrigerant, COP.

I. INTRODUCTION

The way we study and employ fluid dynamics and heat transfer has evolved significantly as an outcome of the development of reliable numerical methods for solving physical problems and the swift development of digital computers. This method, referred to as computational fluid dynamics, or CFD in short, has made it possible to evaluate complex flow geometries with the same ease as if using traditional approaches for resolving idealized problems. Therefore, CFD may be viewed as a field of study which incorporates numerical analysis and fluid dynamics. Historically, the aerospace industries' requirements drove the initial growth of CFD in the years between the 1960s and 1970s. However, there are actually applications of modern CFD in all areas of engineering, particularly biomedical, electrical, mechanical, electronics, chemical, aerospace, and ocean. Testing, playing around, and the general amount of time invested on research and development are replaced by CFD.

By altering fluid velocities, a computational fluid dynamic, or CFD, model is utilized for investigating the outlet temperatures in a heat exchanger within the generator in addition to the outlet temperatures in the evaporator and the heat fluxrate at the evaporator's walls. The evaluation is carried out using ANSYS 14. The software that is used to model issues with fluid flow is known as FLUENT.

II. CFDMODELLING

2.1CFDPrograms:

Commercial CFD software have been developed as a result of the availability of user-friendly interfaces and the emergence of inexpensive high-performance computing hardware. For carrying out a CFD analysis, one had tocreate individual code prior to the widespread adoption of these CFD applications. Although certain elements of the code from one program could be used in another, the programs were usually different from one another. The programs underwent inadequate evaluation, and the reliability of the findings was frequently raised. Thoroughly researched commercial CFD programs have not only enabled research engineers concentrate more effectively on the physical system, but have also established CFD analysis as an established design method in industry. Each renowned CFD software includes three parts.

(1) Apre-processor.

(2) Themainsolver.

(3) Apost-processor.

(1) The pre-processor:

The initial phase of CFD analysis is known as pre- processing, in which the user

(a) Defines modeling targets,

(b) Determines the computational domain,and

(c) Develops and constructs the mesh system.

Understanding the true problem and identifying the computational domain are the initial steps in the CFD modeling process. The most important component of the pre-processor activity occurs next: generations of the mesh structure. It is thought that mesh generation consumes up more than half of a CFD analysts operate. The mesh structure influences estimation time and solution accuracy. In general, the optional mesh is coarser in parts where relatively little variation in the variables is expected and finer in areas where significant variation is expected. All of the major CFD structures offer the capacity to import shape and geometry data from CFD packages like AutoCAD, that reduces technical difficulties and improves effectiveness.

(2) The main solver:

The basis of CFD software is the solver. For the purpose of to compute the flow field, it sets up the equations and resolves them employing the grid points generated by the pre-processor and the options selected by the analyst. The following activities belong to in the process: when the model is completely set up, the solution is initialized, which allows computation start and keeping track of temporary results at every iteration's time step. The following tasks are involved in the process: once the model is completely set up, the solution is initialized, allowing for the computation start and the monitoring of interim ways results at every iteration's time step.

(3) The post-processor:

The final part of CFD software is the post-processor. It helps in the user's evaluation of the results while offering helpful data. The outcomes could be shown as contour plots of scalar variables or vector plots of vector quantities like velocity. Examples include temperature and pressure, streamlines, and animation in the occurrence indicate that the simulation is unpredictable. With the use of appropriate formulas, global parameters such as the Nusselt number, skin friction coefficient, and coefficient can be determined. In addition, these CFD post-processor data can be transmitted to visualization and graph-plotting uses to enhance display and plotting. In the past ten years, numerous versions of general-purpose computational fluid dynamics (CFD) programs have been developed. Among these, PHOENIS and FLUENT are important. These packages are all usually constructed via the finite volume method. The vast majority of CFD software applications come with their own post processors and mesh generators. TECPLOT and FIELDVIEW are two prominent visualization programs that are used with CFD applications.

Pre-processing	Geometry creation
	Mesh Generation
	Named selection
Main solver	Material selection
	Boundary condition
	Solving the equation
Post processing	Contour plotting
	Animation
	Result

III. ANALYSISOF HEATEXCHANGER

3.1 GEOMETRYCREATION

The geometry has been produced using ANSYS

18.2's design modeler. The double pipe counter flow heat exchanger has the following measurements

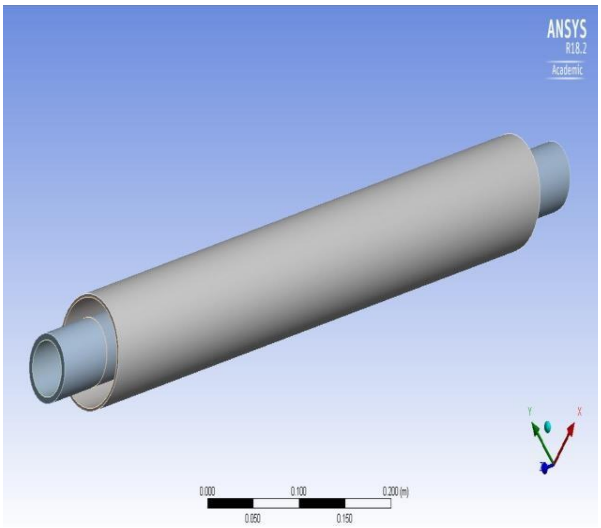

Fig.1 Double-pipe heat exchanger design

TABLE.1 DIMENSIONS OF HEAT EXCHANGER

External pipe diameter	66mm
Thickness of external pipe	4mm
Length of outer pipe	1250mm
Internal pipe diameter	34mm
Internal pipe thickness	8mm
Length of internal pipe	1500mm

3.2 MESHGENERATION

After the creation of geometry, the meshing of geometry is done with following data:

TABLE 2. MESH INFORMATION FOR DOUBLE PIPE HEAT EXCHANGER

Domain	Nodes	Elements
Cold_fluid	45400	33900
Hot-fluid	129890	123291
Inner_pipe	31000	15450
Outer_pipe	22700	11300
All Domains	228990	183941

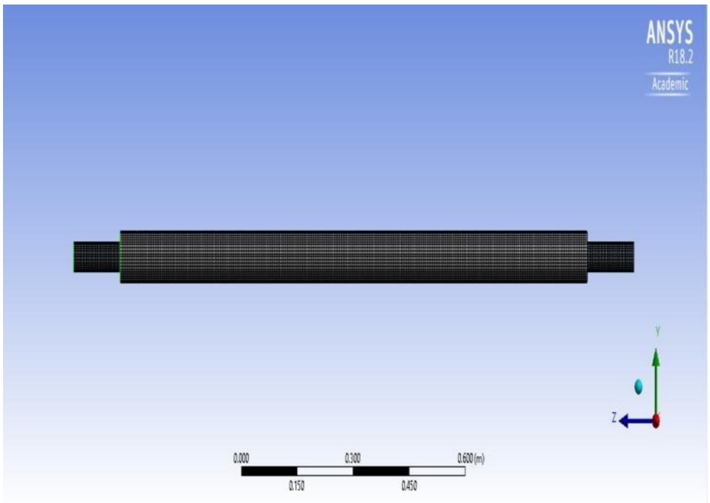

Fig.2 Design of Double Pipe Heat Exchanger

3.3 SOLUTIONSETUP

Certainly! Here is the text with spaces added after each word:

The following setup has been done for carrying out the analysis:

(a) The energy equation is kept on.

(b) The type of flow selected disk-epsilon, realizable, enhanced wall treatment.

(c) Following materials are selected:

Cold fluid: ammonia liquid, lithium bromide-water, sodium thiocynate - ammonia

Hot fluid: brine solution Outer pipe: copper Inner pipe: copper

(d) Boundary conditions for analysis are:

Brine:

Velocity (in m/s) = 0.4

Temperature at the intake (in K) = 360 K

Aqua-ammonia:

Velocity (in m/s) = 0.4, 0.6

Temperature at the intake (in K) = 312 K

Lithium bromide-water:

Velocity (in m/s) = 0.4, 0.6

Temperature at the entrance (in K) = 312 K

Sodium Thiocyanate–ammonia:

Velocity (in m/s) = 0.4, 0.6

Temperature at the entry point (in K) = 312 K

(e) Solution using CFD

(f) Type of initialization – Standard

(g) No. of iterations for the calculation – 100

3.2 ANSYS FLUENT

The program utilized for modeling fluid problems with flow is known as FLUENT. It solves the fluid's governing equations using the finite volume method. The mass and momentum conservation equation is solved for all flows using ANSYS Fluent. An extra equation for energy conservation is calculated for flows combining compressibility and heat transfer. The ANSYS FLUENT managing formulas are shown as follows:

1. Energy equation;
2. Continuity equation; and
3. Navier Stokes equation.

IV. RESULT & DISCUSSIONS

4.1ANALYSISRECORDINGS

The findings of the analysis performed with ANSYS for differentrefrigerant-absorbent pairs are as follows:

(I) FORAMMONIA-WATERPAIR:

(a) ANALYSIS1:Brinevelocity=0.4m/s; Ammonia-water velocity= 0.4 m/s

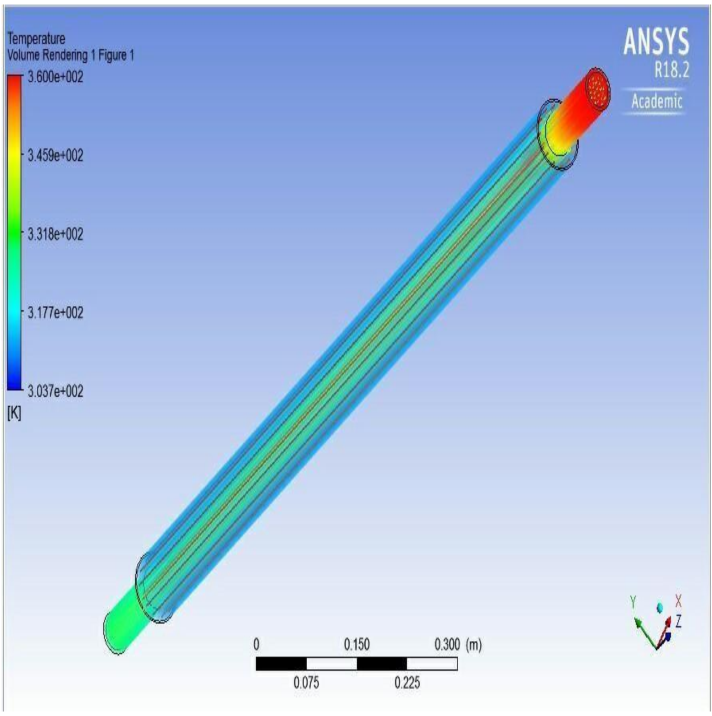

FIG.3 Temperature contour for analysis1

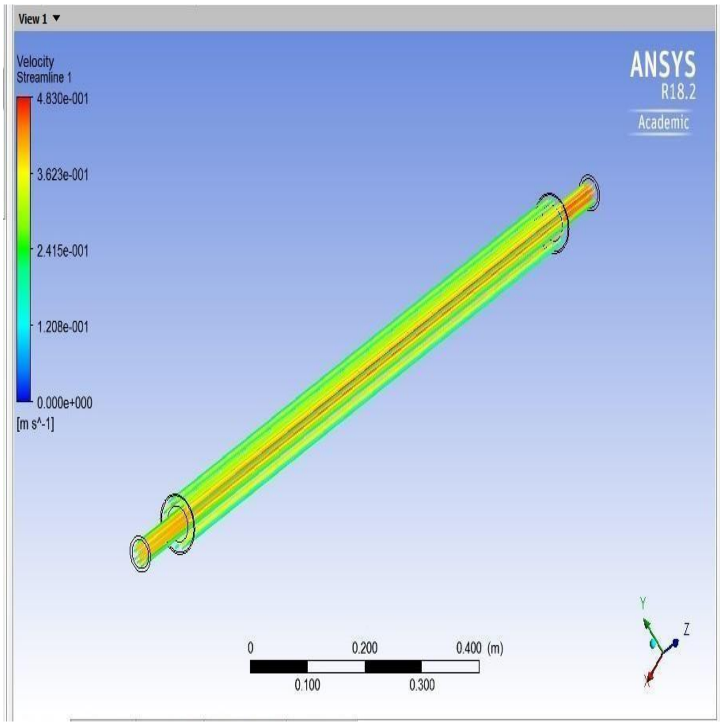

FIG.4 velocity contour for analysis1

TABLE3 RESULT OF ANALYSIS OF TEMPERATURE

S.NO.	Name solution	Inlet temperature	Outlet temperature	Inlet velocity	Outlet velocity
01.	Brine solution	360K	317.42K	0.4m/s	0.397 m/s
02.	Ammonia -water	312K	313.32K	0.4m/s	0.413 m/s

Average temperature = 327.28K

(b) ANALYSIS2:Brinevelocity=0.4m/s; Ammonia-water velocity = 0.6 m/s

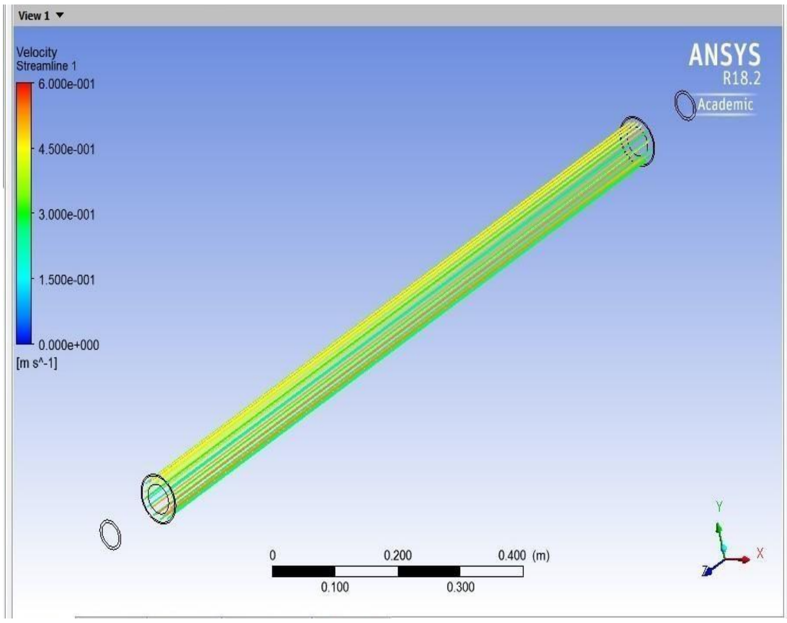

FIG.5 Analysis2's temperature contour

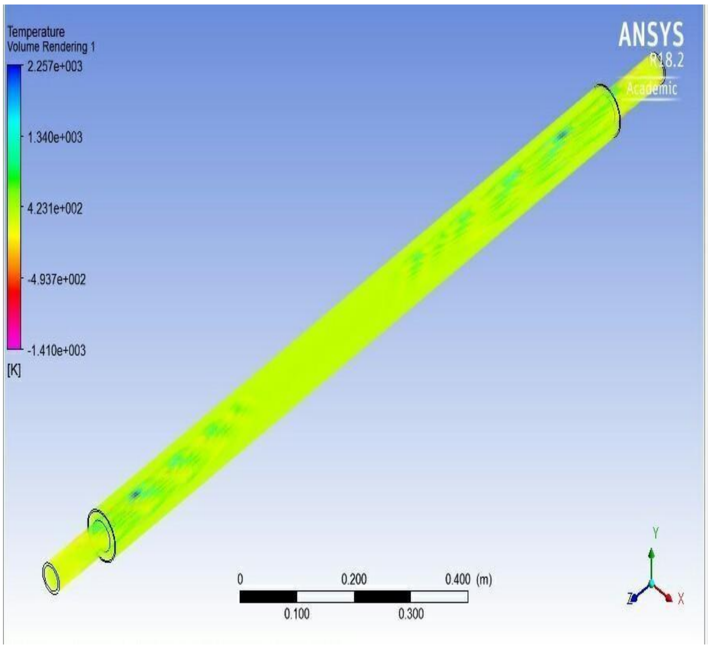

FIG.6 Velocity contour for analysis

2 TABLE4 .RESULT OF ANALYSIS OF TEMPERATURE

S.NO.	Nameof solution	Inlet temperature	Outlet temperature	Inlet velocity	Outlet velocity
01.	Brine solution	360K	309.76K	0.4m/s	0.409m/s
02.	Ammonia-water solution	312K	313.09K	0.6m/s	0.595 m/s

Averagetemperature=324.73K

(II) LITHIUM BROMIDE- WATER (LiBr-water) PAIR:

(a) ANALYSIS3:Brinevelocity=0.4m/s; LiBr- water velocity= 0.4 m/s

TABLE5 RESULT OF ANALYSIS OF TEMPERATURE

S.NO.	Name of solution	Inlet temperature	Outlet temperature	Inlet velocity	Outlet Velocity
01.	Brine solution	360K	318.44K	0.4m/s	0.398 m/s
02.	Libr-water	312K	313.428K	0.4m/s	0.408 m/s

Average temperature=327.59K

(b) ANALYSIS4:Brinevelocity=0.4m/s;Libr- water velocity= 0.6 m/s

TABLE6 RESULT OF ANALYSIS OF TEMPERATURE

S.NO.	Name ofthe solution	Inlet temperat ure	Outlet temperature	Inlet Velocity	Outlet Velocity
01.	Brine solution	360K	341.84K	0.4m/s	0.408 m/s
02.	Libr- water	312K	312.552K	0.6m/s	0.597 m/s

Averagetemperature=333.97K

(III) SODIUM THIOCYANATE-AMMONIA (NaSCN-ammonia) PAIR:

(a) ANALYSIS5:Brinevelocity=0.4m/s; NaSCN-ammonia= 0.4 m/s

TABLE7 RESULT OF ANALYSIS OF TEMPERATURE

S.NO.	Nameof solution	Inlet temperature	Outlet temperature	Inlet velocity	Outlet Velocity
01.	Brine solution	360K	339.97K	0.4m/s	0.397 m/s
02.	NaSCN- ammonia	312K	349.20K	0.4m/s	0.407 m/s

Averagetemperature=341.17K

(b) ANALYSIS6:Brinevelocity=0.4m/s; NaSCN-ammonia velocity= 0.6 m/s

V. CONCLUSION

Following conclusions are obtained:

(i) Based on the temperatures obtained at the outlet of the heat exchanger, the biggest output temperature can be observed for a refrigerant-absorbent pair is obtained in the case of sodium thiocyanate-ammonia refrigerant absorbent pair. i.e. 349.20 K (for 0.4 m/s) and 337.31 K (for 0.6 m/s) of velocity.

(ii) Therefore, an ideal result is achieved in the case of sodium thiocyanate ammonia for velocity of 0.4 m/s. This explains the fact that out of all these refrigerant absorbent pairs, sodium thiocyanate ammonia with velocity of 0.4 m/s is perfect under the circumstances.

(iii) Also, when compared among the velocities, it is found that the maximum temperature is in the case of lower velocity, which is true as slower the fluid moves, more will be the heat exchanging taking place.

(iv) Thus, we have found a refrigerant absorbent pair which is a possible option for solar vapour absorption refrigeration system, as more heated will be the refrigerant-absorbent pair, less heat would be required in the generator for evaporation and even at low solar intensity, better COP can be obtained.

VI. SCOPEOFFUTUREWORK

This project may go on by using other refrigerant absorbent pairs crawling their way forward in the area of solar vapour absorption refrigeration system like lithium nitrate-ammonia. Analysis at different velocities can be carried out for more pairs and at different conditions, the best operating pair can be found. For analysis, other types of heat exchangers, including shell and tube, helical coil, and tubes in tube heat exchangers, may also be used.

REFERENCES

[1] H. Arun kumar, Sunny Benson P, George Arun, Antony Jesbin, CFD Analysis of 24 hour operating solar refrigeration Technology,IJIRST,Vol.1, Issue 11, April2015.

[2] Y.Fan, L.Luo, B.Souyri (2007), Review of solar absorption refrigeration technologies: Development and applications.ELSEVIER,1758-1775.

[3] Vazhapilly Christy V, Modelling and experimental analysis of generator in Vapour absorption refrigeration system al.IJERA, Vol.3,Issue-5, Sept-Oct 2013.

[4] Jibin Johnson, Abdul Anzar V M, Abith Shani, Harif Rahiman P,Hashmi Hameed T S, Nithin V S, CFD Analysis of Double pipe heat exchanger.