International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

Design and Analysis of Multipurpose Relief Vehicle

Ayush Madhapure[1], Prajwal Deshmukh[2], Sumit Taywade[3], Ankush Nikalje[4], Swaraj Dhote[5], Rishikesh Deshmukh[6], S. Yeole[7]

[1, 2, 3, 4, 5, 6, 7]*P.R. Pote Patil College of Engineering and Management, Amravati, India*

Abstract: ***The rocker bogie suspension system has all the tough capabilities so that it can work in rough terrain because of its weight distributing property on all its 12 wheels. The only disadvantage of the mechanism is that its speed is slow. In this project during research we have deeply focused on the 12 wheels mechanism & its design which has certain advantages of linear bogie motion in defending the whole system or assembly from getting disengaged and rollover during working in high speed operations. This has magnificently increased the chances of dependability of structure on uneven surface & also its high speed examination with hurdle height capacity as double the diameter of the wheels .The requirement to develop a highly steady suspension system capable of operating in multi terrain surfaces while keeping all the wheels in contact with the ground. To design a machine that can navigate terrains where the left and right rockers independently climb different obstacles.***

Keyword: Rocker bogie mechanism, Rover, Multipurpose

I. INTRODUCTION

Recently NASA started a motivated study on a very sophisticated device called Mars Pathfinder. This is a primary rover explorer during the program. The future rovers will have to be made in such a way that they would travel several kilometer over a long period of months and handle uneven samples. They must are going to be somewhat independent. They are being made likely for the missions that are dynamic and moderately difficult. The designing of the rover must be methodical and should relate to environment that is vital. Manufacturing prototype are very important for rover development. Modest flexible of rocker bogie device are developed and are used as valuation.

Cultivating the acts of an easier four wheel rover has also been discovered .In this effort, actuator idleness and also the location of the Centre of mass of a vehicle is demoralized to develop power. The strategy rely on real-time dimensions of wheel contact forces, which are hard to live in preparation. Traction can even be upgraded by checking the sliding of the rover. This research defines a physical model of a rocker-bogie rover, the Lightweight Survivable Rover. A proficient method of cracking its opposite kinematics and its quasi-static force analysis is drawn. The methods contain the properties of the rover's manipulator, actuator saturation and tire-slip reflections. A graphical interface that improves the sympathetic of the physics of the model is additionally defined.

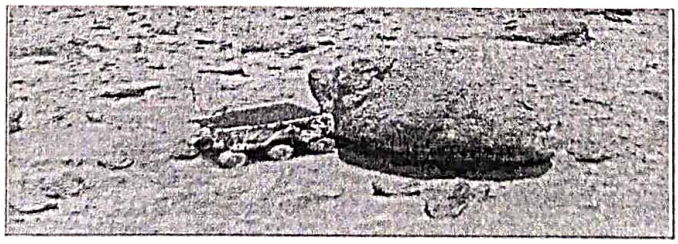

Figure 1: Sojourner examining the rock

An orange colored big ball softly bounced on the surface of Mars on July 4, 1997, with an uncommon robotic vehicle inside. This was the primary planetary mission which has been extensive public interest after first man on the moon. Small rover "Sojourner" conducted scientific experiments for 83 Sols and took many photographs. Travelling on another planet came from vision to real by the assistance of science and chronic motivated research. The operative mission cheered the scientists and NASA to continue the exploration with newly developed rover. Many rovers developed with different structures and scientific purposes after Sojourner. The 2 rovers were landed on Jan 2004 on different location on mars named Spirit and Opportunity. The study and the scientific effect were more powerful that the larger physical dimension of the vehicle. The space agency are now going to be continuing the robotic geologist in future all the three robots are the proof of their success.

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

CFD Analysis of Heat Transfer Using Twisted Tape with Uniform Alternate Length in Double Pipe U-Bend Heat Exchanger

[1]Vijaya Dabhade, [2]Prof. P. B. Ingle

Abstract- Heat exchanger is device that use continuously transfers the heat from one medium to another medium. This research mainly deals with use of linear twisted tape turbulator in U-bend pipe heat exchanger. This heat exchanger widely used in pasteurization process, digester heating, heat recovery, pre-heating. In present study the four types of twisted tape are used, typical twisted tape, perforated twisted tape, half-length twisted taped in copper tube. The perforated twisted tape decreases the pressure drop but the heat transfer same as the typical twisted tape .and the linear twisted tape create turbulence at the wall surface of inner pipe. In this research the combination of two twisted tape perforated twisted tape and linear twisted tape modification will be use and analysis of heat transfer and pressure drop.

Keywords: Heat Exchanger, Perforated Twisted Tape, Linear Twisted Tape, Pressure Drop, Heat Transfer

I. INTRODUCTION

Heat exchangers are the devices that are used to transfer heat from one fluid to other fluid. Condenser and evaporator that use in air conditioning units and refrigerants that type of heat exchanger have been used on our daily life. Large industrial heat exchangers in thermal power plants include boilers and condensers. In cars, heat exchangers take the shape of radiators and oil coolers. These heat exchangers are most abundant in all process industries and chemical industries. There are wide varieties of heat exchangers that are used for diverse purposes where the construction also differs widely. The heat exchangers can be classified into different types based on few fundamental concepts.

The objective of this study is to heat transfer and pressure drop characteristics of the U bend double pipe heat exchanger with and without half-length twisted tape insert. The effects of various relevant parameters on heat transfer characteristics and pressure drop are also investigated. The effect of heat transfer and pressure drop characteristics is studied using CFD (computational fluid dynamics) in ANSYS Fluent with and without inserted tape inside the pipe (tube). The comparative analysis of the experimental setup and software related results are taken. New data gathered during this work for heat transfer and pressure drop characteristics for the U bend double pipe heat exchanger with half-length twisted taped insert are proposed for practical applications.

Manuscript received April 5, 2021; revised 25 April, 2021 and published on May 30, 2021

Vijaya Dabhade, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. P. B. Ingle, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.

II. RELATED WORK

Mehdi Ghalambaz et. al. (2020) In a counterflow double-pipe heat exchanger with overlapped twisted tape inserts in both inner and outer tubes, the laminar convective heat transfer and fluid flow of Al2O3 nanofluid were explored. For the stationary twisted tapes, two models of the same (co-swirling twisted tapes) and opposite (counter-swirling twisted tapes) angular directions were examined. The design parameters for the computational fluid dynamic simulations were varied, including the angular orientation of twisted tape inserts, nanofluid volume concentration, and Reynolds number. When compared to a plain heat exchanger, it was discovered that inserting overlapped twisted tapes in the heat exchanger greatly improves thermal performance as well as the friction factor. The results show that co-swirling twisted tapes and counter-swirling twisted tapes models enhance the average Nusselt number by almost 35.2-66.2 percent and 42.1-68.7 percent, respectively, for Reynolds numbers ranging from 250 to 1000. The dimensionless number of performance assessment criterion was generated for all the captured configurations to examine the interplay between heat transfer enhancement and pressure loss penalty. In the case of counter-swirling twisted tapes model, the greatest value of performance assessment criterion is equivalent to 1.40 and 1.26 at inner and outer tubes at Reynolds number 1000 and volume portion of 3%.

Dr. Chandresh Sharma et. al. (2020), Provided an overview of various geometries and configurations of twisted tapes, as well as their impact on flow mechanisms,

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2020): 7.803

Heat Transfer Enhancement in Solar Air Heater: A Review

Alok Bharti[1], Prof. P. B. Ingle[2]

[1]Student Dept of Mechanical Engineering, P.R Pote Patil College of Engineering and Technology Amravati Maharashtra, India
Email: abharti98[at]gmail.com
[2]Assistant Professor Dept of Mechanical Engineering, P.R Pote Patil College of Engineering and Technology Amravati, Maharashtra, India
Email: prakashingle2018[at]gmail.com

Abstract: ***Investigators have shown that by providing the obstruction in the flow of air through the duct enhances the heat transfer of solar air heater. This enhancement is accompanied by considerable rise in pumping power. In view of the fact the investigator needs to carefully examine the method of providing obstruction in the flow. In this paper some methods of providing obstruction in the flow have been formulated on the basis of heat transfer enhancement and thermo hydraulic performance. The objective of this paper is to review the various methods used to enhance the heat transfer rate with little penalty of friction. Correlations developed by various researchers with the help of experimental results are discussed in this paper. These correlations are used to predict the thermo hydraulic performance of solar air heaters.***

Keywords: Solar air heater, heat transfer enhancement, thermo hydraulic performance

1. Introduction

Energy plays key role for economic and social development. Demand for energy has been rising rapidly with growing population, transportation and industrialization. Due to continuous use of fossil fuels, not only the energy starvation is felt at global level but another serious problem of environment degradation has also been resulted. The rapid depletion of conventional energy sources has necessitated search for alternative energy sources to meet the energy demand of immediate future and for generations to come. Of the many alternatives, solar energy stands out as the brightest long range promise towards meeting the continually increasing demand for energy. Solar energy is available freely, omnipresent and an indigenous source of energy provides a clean and pollution free atmosphere. The simplest and the most efficient way to utilize solar energy are to convert it into thermal energy for heating applications by using solar collectors. Solar air heaters, because of their inherent simplicity are cheap and most widely used collector devices. Solar air heaters are being used for many applications at low and moderate temperatures. Some of these are crop drying, timber seasoning, space heating, cooking etc. The thermal efficiency of solar air heater has been found to be low due low thermal capacity of air and because of low convective heat transfer coefficient between absorber plate and flowing air in the duct. Attempts has been made to enhance the heat transfer rate by use of extending surface in form of fins, by providing artificial roughness in the duct, , by providing transverse rib in the direction of flow and by introducing baffled duct.

Turbulence is created by obstruction in the viscous sub layer to obtain heat transfer enhancement. Several methods have been tested so far to enhance heat transfer with consumption of pumping power. This paper is the review of work done by various researchers on solar air heater to enhance its performance.

2. Methods of Performance Enhancement

A. By using artificial Roughness

CFD Based Performance Analysis of Artificially Roughened Solar Air Heater by Arun Kumar Yadav. In this paper author has performed a CFD analysis of a solar air heater with an artificial friction which increases the turbulence and heat Reynolds number increases because of it heat transfer rate enhances.

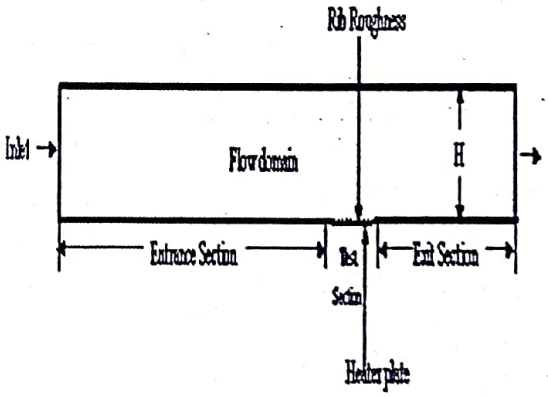

Figure: 2D representation of experimental setup for SAH with artificial roughness

After doing CFD analysis Results are presented in form of graphs, representing the average Nusselt number at different Reynolds numbers, and in form of temperature and velocity contours at particular sections for a fixed Reynolds number.

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 05 | May 2021 www.irjet.net p-ISSN: 2395-0072

Thermal Analysis of Second Stage Aero Gas Turbine Blade

Miss. T. A. Sawwalakhe[1], Prof. P. B. Ingle[2]

P.G. student, Department Of Thermal Engineering, PRPCET, Amravati, Maharashtra, India[1]

Asst. Professor, Department Of Mechanical Engineering, PRPCET, Amravati, Maharashtra, India[2]

Abstract - *In this experiment here we are detecting the hot zone the profile of second stage gas turbine blade. On the gas turbine blade various flue gases are affected on blade profile, so here we studied the effect of flue gas. We analyze and detected the heat zones on second stage gas turbine blade. For detecting the heat zones the thermal paint are applied on the surface of second stage gas turbine blade. Non-contact type temperature sensors are used to sense the temperature. By thermal analysis the direction of the temperature flow is known on turbine blade. A structural analysis has been carried out to investigate the stresses, shear stress and displacements of the turbine blade.*

Key Words: Thermal Paint, Turbine Blade, Non-contact type temperature sensor, Effect of Flue Gas, Hot Zone Detection

1. INTRODUCTION

Aerodynamic design techniques of gas turbine compressors have dramatically changed in the last few years. While traditional 2D design procedures are consolidated for preliminary calculations, emerging techniques have been developed and are being used almost routinely within industries and academia. The compressor design still remains a very complex and multidisciplinary task, where aerothermodynamics issues, traditionally considered prevalent, now become part of a more general design approach, where aeromechanical, technological, structural, noise-related concerns and many other matters have to be taken into account simultaneously, thus leading to a very challenging problem for designers. The interesting and alternative options are in and casing treatments for enhanced stall margin and fact available for compressor 3D design, such as new blade shapes for improved on-off design efficiency, end wall contouring many others. For this reason, while experimental activity remains decisive for ultimate assessment of design choices, numerical design optimization techniques, along with Computational Fluid Dynamics (CFD) are assuming more and more importance for the detailed design and concrete evaluation of options.

1.1 History

If we throw the light on the history of gas turbine we can observe that, for more than a half-century gas turbine engines pioneered for military jet fighters have hung under the wings of commercial airliners. For nearly as long, manufacturers have built industrial gas turbines to drive electricity generators and pump oil and gas. Many gas turbines, large and small, are designed specifically for these industrial applications. Aero derivative gas turbines used for these industrial applications are adapted directly from existing aircraft engines. Aero derivative gas turbines emerged in the late 1960s with unique performance attributes in comparison to the existing industrial gas turbines. Aero derivative units could startup more quickly for peak and emergency electricity generation. Also, aero derivative turbines offered lower weight in a smaller footprint, which was ideal for offshore platforms. Furthermore, their higher efficiency, coupled with simplified installation and maintenance, saved money for pipeline operators.

The performance of aero derivative gas turbines is largely enabled by the engineering successes sustained in the lucrative aircraft engine market. The role and impact of industry-government partner ships in advancing aircraft engine technology is widely appreciated and has directly impacted the evolution of aero derivative gas turbines.

2. LITERATURE SURVEY

After the study of second stage aero gas turbine, the few things are comes in front that the aero gas turbine operates above the temperature 2000- 3500 degree Celsius. at this much high temperature the gas turbine blade material will not sustain anymore and this is the main problem we have been observe. At this high temperature there may chances of bending of material due to various stresses act on the profile of second stage gas turbine blades. There is also chances of creeping and overheating of blade material. Because of this entire problem there may chances of decreasing life of blade.

2.1 Objectives of Study

In the present work CFD analysis is used to examine the heat zone analysis of second stage gas turbine blade. The temperature indicating Paint is a kind of technology which is widely used to measure the surface temperature of high temperature components of gas turbine engine. Analysis of hot zones occupied in the profile of second stage gas turbine blade. The portion where the hot zones are obtained should be replaced by another toughest material for increasing the efficiency of the gas turbine blade.

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

Study on Solar Dryers for drying Agricultural Products

[1]Amar Tolmare, [2]Prof. P. B. Ingle

Abstract— Drying is the process of removing liquid from a solid material by evaporation. In drying technique, the concept of environment is of importance as it refers to the properties of air-vapor mixture that controls the rate of drying. The temperature and pace at which the liquid vaporises depends on the vapour concentration in the surrounding atmosphere when a suitable amount of heat is available for drying. Solar drying refers to techniques for drying that make use of the sun's energy. A sun dryer is a sealed container that protects food from damage, birds, insects, and unexpected rain. Solar dryers are divided into two categories: active and passive. Forced convection dryers are active, while natural convection dryers are passive. Active dryers are more expensive than passive dryers. Solar dryers must be made more efficient and less expensive. We evaluated the numerous studies on passive solar dryers for drying agricultural products in this paper.

Keywords—Solar Dryer, Passive Solar Dryer, Natural Circulation dryer

I. Introduction

Agricultural products must be preserved in order to be kept for an extended period of time without deteriorating in quality. Several industrial-scale process technologies have been used to preserve food goods, among them. The most efficient and dependable procedure is drying. It provides a highly effective and practical method of preservation for reducing postharvest losses and compensating for supply constraints. Drying is a simple method of removing moisture from a product to get the appropriate moisture content. It is a high-energy operation. Apart from extending storage life, the primary goal of drying is to improve quality, ease of handling, further processing, and sanitation, and it is likely the oldest method of food preservation used by humans.

Drying is a dehydration procedure for food goods that involves lowering the moisture content of the item to extend its shelf life by limiting bacterial growth [1]. The drying process occurs in two stages: the first occurs at the surface of the drying material at a constant drying rate, analogous to water vaporisation into the ambient, and the second occurs according to the features of the drying product with a decreasing drying rate [2].

Manuscript received April 5, 2021; revised 25 April, 2021 and published on May 03, 2021
Amar Tolmare, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. V. G. Gore, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Dr. P. R. Wadnerkar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India

II. Classification Of Solar Dryer

Solar dryers are available in different types and different ranges for various applications.

A. Active Dryers

External mechanisms, such as fans or pumps, are used to move solar energy in the form of warm air from the collector area to the drying beds in active solar drying systems. As a result, all active solar dryers are forced convection dryers due to their application. A typical active solar dryer employs motorised fans or ventilators for air circulation and only uses solar energy for heat. These dryers are widely used in large-scale commercial drying operations, where they are used in conjunction with traditional fossil-fuel dryers to improve drying control by reducing the effect of solar insolation changes on drying air temperature. [4]

B. Passive Dryers

Air is heated and circulated naturally in a passive solar dryer by buoyancy force, wind pressure, or a combination of both. The passive mode is used by the normal and reverse absorber cabinet dryers, as well as the greenhouse dryer. Passive crop drying is still used in many Mediterranean, tropical, and subtropical climates, particularly in Africa and Asia, as well as in small agricultural communities. [5]

C. Integral Type Dryer

Moisture is extracted from the top of Integral type dryers; air enters the cabinet from below and exits from the top. The only difference between this and a sun-drying type of drier is that the food product is protected by a glass cover. When sunlight strikes a glass surface, three things happen: some light is absorbed, some light is reflected back to the glass, and some light is transmitted. As portion of the radiation is absorbed by the surface of the crop, the temperature rises. The glass cover avoids direct convective losses to the environment, which helps to raise the temperature of agricultural products and the temperature of the cabinet.

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

Critical Review and Analysis of Pressure Vessel Structures

[1]Aniket R. Aware, [2]Prof. V. G. Gore, [3]Prof. P. B. Ingle

Abstract— This paper provides an overview of the pressure vessel, starting with its background and a brief history. Then, the geometry, main components, classification, applications, materials and fabrication process of the pressure vessel are also discussed. When designing or performing optimization on the vessel, it is crucial for the designers to familiar with the types of failures and loadings, to select appropriate analytical methods to analyse the vessel. As well as the design parameters such as thickness, design pressure or allowable stresses, which can alter the performance, efficiency and safety of the vessel. Since the design of the pressure vessel is governed by the codes and standards, some of the commonly used codes are presented, with more details included for the ASME pressure vessel code.

Keywords— Pressure Vessel, Cylindrical Shape, Vertical and Horizontal Vessel.

I. Introduction

In the year 1495, Leonardo da Vinci has documented his pressure vessels design in the book, Codex Madrid I with the concept of pressurized air containers lifting the heavy weights underwater. Without the initiative to generate steam in boilers that spur an industrial revolution in the 1800s, vessels which resemble what can be seen today will never come true [5]. Pressure vessels are designed to carry, store and receive process fluids, gases, and liquids under required temperature and pressure limit [3]. They are often subjected to constant or cyclic internal/external pressure loading, with the difference between the operating pressure and ambient pressure. Due to the difference in operating pressure, the state of the fluid present in the vessel will undergo changes [2]. A ruptured pressure vessel can be hazardous, possibly leading to poison gas leaks, fires or explosions which may cause significant losses of human lives and property. To counteract this problem, the local providences and some states began enacting rules, which made it tedious for the manufacturers as the rules lack uniformity and differ from one location to another. Thus, American Society of Mechanical Engineers (ASME) took the initiative and established the standard specifications and design formulation for the pressure vessels. In the year 1911, the first edition of pressure vessel code was developed and then released in 1914, which is now known as ASME Boiler and Pressure Vessel Code (BPVC).

The code eventually developed over time and is presented in eleven sections, with multiple subdivisions, parts, subsections, mandatory and non-mandatory appendices. Many countries follow the BPVC as their official code or even developed their own [6]. It is impossible to eliminate the accidents completely, however, by studying the behavior of the pressure vessels, the likelihood of accidents can be reduced or prevented [12]. With that, the design of a vessel needs to achieve a balance between the safety as well as economics. To accomplish this task, it requires the understanding of parameters affecting the pressure vessel due to varying loads, pressure and thickness. Fortunately, the established engineering standards have resulted in the recent advancements in the pressure vessels engineering such as material breakthrough with increased strength, durability, corrosion resistance and new joining methods (explosion welding, friction stir welding, etc.) that allows the design of vessels to be safer and more reliable.

On the other hand, pressure vessel can be made of any shape, but in the industry, spherical, cylindrical and conical pressure vessels are often employed. Ideally, the spherical shape can hold the internal pressure with evenly distributed stresses on the surface both internally and externally. It is advantageous in term of structural strength when compared to a cylindrical pressure vessel made of same wall thickness. However, the spherical shape may present manufacturability and costs concern. Thus, the cylindrical shape is more commonly used due to lower manufacturing costs and the ability to use the space efficiently. To overcome the structural weakness, few types of rounded or hemispherical ends will be fitted. The geometry of the vessel ultimately depends on the type of applications. Nonetheless, regardless of the types of the pressure vessel, a vessel should consist of closure heads, shell, openings, some functional attachments, a combination of nozzles and supports [9], as illustrated in Figure 1 which shows the vertical and horizontal vessel arrangement with the components labeled. The functions for each of the components are summarized as below:

April 5, 2021; revised 25 April, 2021 and published on May 03, 2021

Aniket R. Aware, PG Student, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. V. G. Gore, Assistant Professor, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. P. B. Ingle, Assistant Professor, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India

Appendix-A

George Washington International Law Review
ISSN-1534-9977, E-ISSN-0748-4305

Article Received: 22th May, 2021; Article Revised: 20th June, 2021; Article Accepted: 29th June, 2021

REVIEW ON PERFORMANCE OF SOLAR CHIMNEY

P. P. Jamnik
G Scholar, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., prashantjamnik9@gmail.com

V. G. Gore
Assistant Professor, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., gorevijay1973@gmail.com

P. B. Ingale
Assistant Professor, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., prakashingle2018@gmail.com

bstract: The solar chimney utilizes solar energy to generate a flow of air which could be used for ventilation s well as power generation. The current research studies researches conducted by various scholars in mproving the performance of the solar chimney. The studies are conducted using both experimental and umerical methods. The outcome of numerous scheme & material parameters on the performance of solar himney is examined. The outcome of operating conditions & thermal intensity on solar chimney performance also studied.

eywords: Solar Chimney, Power, Ventilation

INTRODUCTION

entilation can be described as the supply & elimination of air to & from any place. To regulate air ontaminant, temperatures or moisture in this region, during the breathing process, air may or may not be nditioned. Ventilation has been introduced in many countries around the world, as ventilation played an mportant role in the health, convenience and productivity of their occupants the minimum fresh air volume equired to breathe is expected to be around 1.2 liters per second per human, except for security. Other than the inimum quantity should be supplied to satisfy the needs of occupants' oxygen, smell dilution or carbon oxide dilution and in the case there are very sensitive heat gains, so the rise in temperature extremes should be inimized. A significant architectural criterion is the ventilation system in a structure that helps increase ermal comfort and indoor air quality. Lastly, the following factors are important:

Climate parameters, like moisture, airspeed, temperature and air contaminant quantities.
Included parameters including such as humidity, carbon dioxide, fragrances and smoke.

Appendix-C

George Washington International Law Review
ISSN-1534-9977, E-ISSN-0748-4305

Article Received: 22th May, 2021; Article Revised: 20th June, 2021; Article Accepted: 29th June, 2021

CFD ANALYSIS OF SOLAR CHIMNEY FOR PERFORMANCE IMPROVEMENT USING ANSYS CFX

P. P. Jamnik
G Scholar, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., prashantjamnik9@gmail.com

V. G. Gore
Assistant Professor, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., gorevijay1973@gmail.com

P. B. Ingale
Assistant Professor, Department of Mechanical Engineering, P R Pote (Patil) College of Engineering and Management, Amravati., prakashingle2018@gmail.com

bstract: Solar energy is abundantly available in most of the regions on earth. This solar energy can be used in lectricity generation using the solar chimney. The performance of the turbine placed inside the solar chimney an be enhanced by increasing pressure differences. The current research investigates the performance of solar himneys with flat type collectors and staggered design collectors using techniques of Computational Fluid ynamics. The CFD analysis is conducted using ANSYS software. For analysis, the model RNG k-epsilon is sed. The pressure distribution and velocity distribution plots are generated for both designs and comparative udies are conducted.

eywords: CFD, Pressure, Solar Chimney

. INTRODUCTION

nconventional energy sources may be used for electrical power generation, which attracts a considerable deal f attention, according to energy experts [1]. Green resources-based power generation technology will help everal countries to improve the balance of their payments. As a renewable energy form that is most abundant nd broadly dispersed, solar energy has great advantages for arid and semi-arid areas. The sun's energy is arvested through a number of solar devices worldwide. Researchers known as solar chimneys have shown an ncredible breakthrough in past years. Figure 1 below shows the schematic of the solar chimney.

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

Study of Various Nano Fluids: Coolants Technology

[1]Syed Bilaluddin Syed Jalaluddin, [2]Prof. V. G. Gore, [3]Dr. P. R. Wadnerkar

Abstract— Today, the demand of automobile vehicles is on peak. So, it is a great challenge for automotive industries to provide an efficient and economical engine. The performance of an engine affects by various systems like fuel supply system, lubrication system, transmission system, cooling system etc. So, it becomes essential to account them while designing an engine for improves the engines performance. Cooling system is one of the important systems amongst all. It is responsible to carry large amount of heat waste to surroundings for efficient working of an engine. It also enhances heat transfer and fuel economy which leads to maximize the performance of an engine. Most internal combustion engines are fluid cooled using either air or a liquid coolant run through a heat exchanger (radiator) cooled by air. The heat transfer through radiator can be improved by maximizing the heat transfer area and increasing the heat transfer coefficient. The heat transfer coefficient can be increased either by using more efficient heat transfer methods or by improving the thermo physical properties of the heat transfer material i.e., coolant. Earlier, Water was widely used in radiator as a coolant for its good ability to holding heat, transfer heat and can be readily obtained. Also, the mixture of water & ethylene glycol later introduced as a coolant. Both of them having certain merits & demerits. With the advancement of nanotechnology, the new generation of heat transfer fluids called, "Nanofluids" have been developed and researchers found that these fluids offer higher thermal conductivity compared to that of conventional coolants. Nanofluids which consist of a carrier liquid, such as water, ethylene glycol dispersed with tiny nano-scale particles known as nanoparticles. This comprehensive study on cooling system importance, coolant used in automobiles and its limitations and applications and challenges of Nanofluids as a coolant have been compiled and reviewed for automobile radiator.

Keywords— Cooling System, Heat Transfer, Nanofluids.

I. INTRODUCTION

Continuous technological development in automotive industries has increased the demand for high efficiency engines. A high efficiency engine is not only based on its performance but also for better fuel economy and less emission. There are many systems which influence the engine performance like fuel ignition system, emission system, cooling system, etc. one of the parameters which affects the performance of engine is the cooling rate of radiator in engine cooling system. Addition of fins is one of the approaches to increase the heat transfer rate of the radiator. It provides greater heat transfer area and enhances the air convective heat transfer coefficient. However, traditional approach of increasing the cooling rate by using fins has already reached to their limit [10]. As a result, there is a need of new and innovative heat transfer fluids for improving heat transfer rate in an automotive car radiator. In addition, heat transfer fluids at air and fluid side such as water and ethylene glycol exhibit very low thermal conductivity. With the advancement of nanotechnology, the new generation of heat transfer fluids called, "Nanofluids" have been developed and researchers found that these fluids offer higher thermal conductivity compared to that of conventional coolants. Nanofluids which consist of a carrier liquid, such as water, ethylene glycol dispersed with tiny nano-scale particles known as nanoparticles.

Nanofluids seem to be potential replacement of conventional coolants in engine cooling system. Recently there has been considerable research findings reported which highlights superior heat transfer performances of Nanofluids. Nanofluids are potential heat transfer fluids with enhanced thermo physical properties and heat transfer performance. It can be applied in many devices for better performances (i.e. energy, heat transfer and other performances). Nanofluids are formed by suspending metallic or non-metallic oxide nanoparticles in traditional heat transfer fluids. This newly introduced category of cooling fluids containing ultrafine nanoparticles (1–100 nm) has displayed interesting behavior during experiments including increased thermal conductivity and improved heat transfer coefficient compared to a pure fluid. The use of nanofluid as coolants would allow for smaller size and better positioning of the radiators. It also increases the efficiency of the system with less amount of fluid. It results that coolant pumps could be shrunk and engines could be operated at higher temperatures. These novel and advanced concepts of coolants offer exciting heat transfer characteristics compared to conventional coolants. Yu et al., [11] reported that about 15-40% of heat transfer enhancement can be achieved by using various types of Nanofluids. This translates into a better aerodynamic feature for design of an automotive car frontal area. Coefficient of

April 5, 2021; revised 25 April, 2021 and published on May 03, 2021

Syed Bilaluddin Syed Jalaluddin, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. V. G. Gore, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Dr. P. R. Wadnerkar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

Experimental Study on Thermal Performance of a Closed Loop Pulsating Heat Pipe

[1]Dr. P. R. Wadnerkar, [2]Ku. Vaishnavi S. Sonar, [3]Prof. Vijay G. Gore

Abstract-*Pulsating heat pipes (PHP) have created a newer arena and promising possibilities as passive devices for heat transfer applications, especially suited for thermal management of electronics. A closed loop pulsating heat pipe (CLPHP) made of copper with 2.5 mm ID and 3 mm OD with fin in the condenser section is used in the present work to evaluate the heat transfer performances where the evaporation section is 50 mm, adiabatic section is 120 mm and condensation section is 80 mm. The attempt is to study on experimental basis the heat transfer performances of CLPHP for two working fluid conditions. Acetone and Ethanol are used as working fluids with 60% filling ratio to 100% filling ratio in CLPHP with 3 loops during the experiment. The experimental results indicate a strong influence of gravity and thermo physical properties of the working fluid on the performance of the CLPHP studied with different orientation and heat load. The results demonstrate the effect of heat by operated working fluid used in condenser section, the input heat flux, and physiochemical properties of the working fluid on the thermal performance of the device. The CLPHP at 0o inclination exhibit the considerable enhancement of heat transfer in condition of acetone as working fluid compared with that of ethanol as working fluid inside the CLPHP. The results indicate that the performance of CLPHP changes with different fill ratio and heat load. Better heat transfer performance was obtained from Acetone rather than ethanol for same heat load parameters during experimentation.*

Keywords- Pulsating Heat Pipe, Fluid, Heat Transfer Ratio, Condenser, Evaporator

I. INTRODUCTION

Thermal management of micro devices, modelling of small-scale heat transfer devices become an important criterion for electronics and miniaturization. Thermally speaking for such challenging problems, pulsating heat pipe (PHP) provides a hopeful solution. Among PHP's mainly a closed loop pulsating heat pipe will be aim proved solution than an open loop device. PHP is essentially a small pipe filled with working fluid. It comprises of a simple meandering tube of capillary dimensions by numerous U-turns and connected end to end. The working fluid is filled in the tube after partial evacuating of the tube. PHP is a two-phase heat transfer device consists of a heating zone at the bottom end, cold zone at the other end and an optional adiabatic zone. One end of this tube bundle receives heat from the electronic system. Between the vapour segments, the plugs or slugs are formed by the liquid in the pipe. The pressure difference which is caused due to the evaporation and absorption of part of the liquid upon encountering heat will be driving force for the movements of slugs and plugs in the pipe flow. Present fluid used in operating pressure maintained privileged the pulsating heat pipe be governed by the operating temperature of the heat pipe i.e., the amount of heat energy released by the electronic system. Even though PHP systems look simply, its working mechanism is relatively complex.

Manuscript received April 5, 2021; revised 25 April, 2021 and published on May 30, 2021

Dr. P. R. Wadnerkar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India
Ku. Vaishnavi S. Sonar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. V. G. Gore, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.

II. OBJECTIVES OF PROJECT

[1] To study the available literature on PHP application and performance.

[2] To find out the suitable working fluid and collect literature on the studies with similar working fluids.

[3] To develop experimental setup for the investigation.

[4] To conduct testing and sample run.

[5] To conduct final experimentation and compile.

[6] To analyse the results.

III. RELATED WORK

N SanthiSree et. al. [1] influence of different working fluids is experimentally investigated on a two loop CLPHP, varying the evaporator heat flux. Pure fluids, viz., water, acetone, benzene and binary mixture, viz., Acetone-water and Benzene-water are utilized on working fluids. The heat input considered at the evaporator is 32W, 48W and 60W. The filling ratio is kept as 50 %. The results show that among the working fluids considered for the study, acetone exhibits least thermal resistance among the pure fluids at all heat fluxes considered in the analysis, while Acetone-water mixture has exhibited least thermal resistance among the water-based mixtures.

M. A. Boda et. al. [2] Presented the wide-ranging review of the state of the applications, performance and materials of current heat pipe heat transferring devices. Heat pipes are becoming increasingly popular as passive heat transfer

International Journal of Advanced Innovative Technology in Engineering (IJAITE),
Vol.6, No.3, May 2021
E-ISSN: 2455-6491
Available online at www.ijaite.co.in

A Review on Mechanism of Fluid Flow and Transfer of Heat in Closed Loop Pulsating Heat Pipe

[1]Ku. Vaishnavi S. Sonar, [2]Prof. V. G. Gore, [3]Dr. P. R. Wadnerkar

Abstract— Now-a-days the researchers and people are moving towards such an innovation which meet the needs of the present without compromising the ability of future generations to meet their own needs. In such innovations pulsating heat pipe is one of the technologies which started in 1990 by Akachi. Pulsating heat pipe is a heat transfer device which has an effective heat transfer capability. The researchers had done many experimental and theoretical investigations, but they haven't got the complete knowledge about it because of its complex operational mechanism which consists of hydrodynamic and thermodynamic coupling effect. This paper gives a brief idea on the thermo-hydro dynamic characteristics of this device. The brief idea will be on internal diameter, cross-section of the tube and the amount of working fluid in system. In addition to all these the number of turns in device and thermo-physical properties of working fluid which leads to determine the thermal behavior. The motto of this paper is to make review paper on recent past years on papers which used refrigerants as working fluids and its fluid flow mechanism and finally lead upcoming researchers to have basic idea and future scope of device.

Keywords—Pulsating Heat Pipe, Thermo-Hydro Dynamic, Thermo-Physical, Working Fluid.

I. INTRODUCTION

Generally, the effective heat management device is Heat pipe. There are various heat management devices like thermosyphon, loop heat pipes etc. Now-a-days pulsating heat pipe is special device in scientific and engineering applications. It was started in 1990 by Akachi [1]. It is an unwicked passive heat pipe which has zigzag capillary tubes. It has huge applications in electronic industries [2]-[5], aerospace, Air conditioning [6], automobile technology [7]-[9]. Following are the advantages 1) wickless structure 2) high heat transfer capacity Shang et al [10] 3) simple and compact design which made the device to use in various places in engineering applications 4) costless which led to good number of experiments and made the researchers easy to have keen interest. The above advantages will give a special operational mechanism. After achieving a stable condition in PHP the pulsating motion of liquid and vapour slugs move because of surface tension and capillary properties. The main impetus for throbs is the vanishing and constriction of working liquid brought about by heat ingestion and dispersal. The operational instrument includes vanishing and buildup of slight fluid film and relies upon dynamic contact edge of working liquid and follows the development and streaming of Taylor bubble standard [11]. Even though the PHP is simple design, the thermo-hydrodynamic effects during the process lead to a complicated operational mechanism which is not yet revealed fully. The PHP are three types namely a) closed loop PHP b) PHP with check valve c) open loop PHP. These are shown in Fig1.

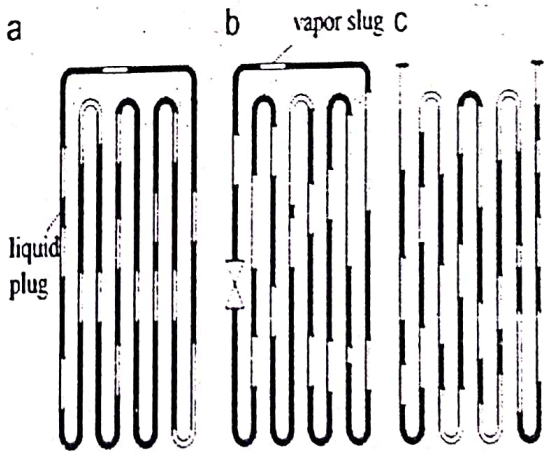

Figure 1: Classification of Pulsating heat pipe [29]

Research scholars are keenly focusing on pulsating heat pipe because of need in the today's fast emerging world to improve the heat transfer capacity. Even though we have good amount of research papers till now we need the latest

Manuscript received April 5, 2021; revised 25 April, 2021 and published on May 03, 2021
Ku. Vaishnavi S. Sonar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Prof. V. G. Gore, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India.
Dr. P. R. Wadnerkar, Department of Mechanical Engineering, P.R. Pote (Patil) College of Engineering and Management Amravati, India

Vidyabharati International Interdisciplinary Research Journal 11(2) **372-377** ISSN 2319-4979

DEVELOPMENT OF MODIFIED SOLAR WATER HEATING SYSTEM FOR EFFECTIVE UTILIZATION

S. Tondre*, R. Pokale, Y. Bansod, S. Yeole, G. Mahalle
P.R.Pote Patil College of Engineering and Management,Amravati
*Corresponding Author: *tondresomdatta@gmail.com, rajeshpokale79@gmail.com, yogibansod@gmail.com, shrikanty11@gmail.com, mahalleganesh56@gmail.com

ABSTRACT

Human Society is blessed with abundance of free solar energy, eco-friendly clean hence it is globally accepted as one of the most promising alternative energy sources. The effective use of solar energy is hindered by the intermittent nature of its availability and effectiveness in domestic and industrial applications especially in water heating. The requirement plenty of hot water for domestic, commercial and industrial purposes is seen recently. Various resources i.e. coal, diesel, gas, electricity etc, are used to heat water are costly due to their limitations. In traditional solar water, due to some limitations all the users was not getting uniform temperature of water for use. To overcome above limitations a modified solar water heating arrangement with extra features developed to meet the following requirements 1) To deliver constant temperature output as per requirement of user 2) To work effectively in winter season also. A prototype of a modified solar water heating system was constructed and tested. In modified solar water heater, the user will receive uniform hot water temperature as per set value of temperature. In this modified solar water heater the heating storage tank (primary storage tank) and hot water storage stank are separated due to this modified solar water heaters are comparatively more efficient than traditional solar water in winter season also. This modified solar water heater can fulfill requirement of users at no cost effectively and efficiently.

Keywords *: Solar water heating system, Solar Energy, eco-friendly*

1. Introduction

1.1 Overview

Mother earth blessed with Solar Energy in abundance without cost. The solar radiation falls on the surface of the earth can be effectively utilized for the benefit of human society. One of the popular devices that harness the solar energy is solar hot water system (SHWS). The use of hot water for domestic purposes is a simple and effective way of utilizing solar energy. The setup and installation cost of solar water heating system is very high without operating cost. It is a natural solar thermal technology. In this system, incident solar radiation is converted into heat and transmitted to a transfer medium such as water [1].

Solar energy is one of the most capable of the alternative energy sources with rising demand with cost of fossil fuel for energy. Solar energy is considered an alternate and efficient source of renewable energy for various applications in both homes and industry. It is reported that heating water consumes approximately 20% of total energy consumption for an average family [2–5]. Solar water heaters provides hot water at cheapest easily available green energy to households as per requirement. It can operate at any climatic seasonal conditions and their performance various upon availability of solar energy at a geographical location [6–9].

1.2 Solar Water Heating System

A solar water heater consists of a collector to collect solar energy and an insulated storage tank to store hot water. The solar energy incident on the absorber panel coated with selected coating transfers the hat to the riser pipes underneath the absorber panel. The water passing through the risers get heated up and is delivered the storage tank. The re-circulation of the same water through absorber panel in the collector raises the temperature to 80 C (Maximum) in a good sunny day. The total system with solar collector, storage tank and pipelines is called solar hot water system. Broadly, the solar water heating systems are of two categories. They are : closed loop system and open loop system. In the first one, heat exchangers are installed to protect the system from hard water obtained from bore wells or from freezing temperatures in the cold regions. In the other type, either thermosyphon or forced circulation system, the water in the system is open to the atmosphere at one point or other. The thermosyphon systems are simple and relatively inexpensive. They are suitable