The book of abstracts

The Second International Conference on Heating, Ventilating and Air Conditioning

1-3 June 2010, Olympic Hotel
Tehran-Iran

(400 word abstracts)

www.hvac-conference.ir

Organized by:

Building and Housing Research Center

The Iranian Construction Engineering Organization

Sharif University of Technology

The Iranian e-Community of Mechanical Engineers

In Cooperation with:

Institute of Standard and industrial Research of Iran

Iranian Society of Consulting Engineers

Iranian Syndicate of M&E and Industrial Contractors

Iranian Scientific and Engineering Society of Heating and Refrigeration

Iranian Syndicate of Industrial Manufacturers

Iranian Combustion Institute

Sponsored by:

Iran Radiator Industrial Group

Chauffagekar Co.

Tahvieh Co.

Samsung Co.

Iran Industrial Vibrations Co.

Event Organizer:

Aria Group

Scientific Committee:

Prof. Abbas Abbasi	Amirkabir University of Technology – Iran
Mr. Ayoub Adeli	Iranian Combustion Institute
Dr. Abolfazl Ahmadi	(Iranian University of Science and Technology – Iran)
Prof. Mohammad Ali Akhavan	(University of Tehran – Iran)
Prof. Mahmoud Arab Yaghoubi	(University of Shiraz – Iran)
Prof. Mehdi Ashjai	(University of Tehran – Iran)
Prof. Mehdi Bahadori Nejad	(Sharif University of Technology – Iran)
Dr. Shahram Delfani	(Building and Housing research Center – Iran)
Prof. Fariborz Haghighat	(University of Concordia – Canada)
Mr. Asghar Haj Saghati	(Iranian Solar Energy Society)
Prof. Ghasem Heydarinejad	(Tarbiat Modares University – Iran)
Dr. Mostafa Hossein Ali Pour	(Iranian University of Science and Technology – Iran)
Prof. Mohammad Hosni	(Kansas State University – USA)
Dr. Mansour Jadidi	(Shahid Rajai Univeristy – Iran)
Dr. Farzad Jafar Kazemi	(Islamic Azad University – Iran)
Dr. Abdol Razagh Kabinejadian	(Iranian Scientific and Engineering Society of Heating and Refrigeration -Iran)
Dr. Behrouz Kari	(Building and Housing research Center – Iran)
Mr. Mohammad H. Khaksari	(Packman Co. – Iran)
Dr. Mehdi Marefat	(Tarbiat Modares University – Iran)
Dr. Kiumars Mazaheri	(Tarbiat Modares University – Iran)
Prof. Mozafar Ali Mehrabian	(Bahonar University of Kerman – Iran)
Prof. Mohamamd Moghiman	(Ferdowsi University of Mashad – Iran)
Mr. Heshamat olah Monsef	(Piraz Consultants Co. – Iran)
Mr. Manuchehr Motamedi	(Jarvand Co. - Iran)
Prof. Mojtaba Mousavi Nainian	(K N Toosi University – Iran)
Prof. Michel Ohadi	(University of Maryland – USA)
Mr. Mahmoud Rezai	(Tavan tech Consultants Co. – Iran)
Prof. Majid Saffar Aval	(Amirkabir University of Technology – Iran)
Prof. M. H. Saidi	(Sharif University of Technology – Iran)
Dr. Sepehr Sanaye	(Iranian University of Science and Technology – Iran)
Dr. Sorena Sattari	(Sharif University of Technology – Iran)
Dr. Asghar Shirazpour	(Iranian Construction Engineering Organization – Iran)
Prof. Hossein Shokouhmand	(University of Tehran – Iran)
Dr. Mohammad Taghi Zadeh	(Sharif University of Technology – Iran)

Dear Guests, Researchers and Students

On behalf of the organizing Committee, We would like to welcome you and express our deepest appreciation for your participation in The Second International Conference on Heating, Ventilating and Air Conditioning (ICHVAC-2). The organizing committee built on the success of the first conference held in the last year. We are looking forward to a successful second conference that will focus on the state of art Heating, Ventilating and Air Conditioning issues and facilitate a forum for exchange of ideas.
Over 80 research papers will be presented, 60 papers for oral presentation and 20 papers for poster session. All Contributions that will be presented in this conference have been reviewed by a team of local and International experts. The conference features three keynote presentations, which will focus on the latest developments on topics like:

1- Envelope of Building as a Mimic of Human Skin Behavior- The Main Factor of Energy Needs
2- Natural Ventilation and Cooling in Buildings
3- Renewable Energy Sources and Energy Efficiency for Building's Greening to a Approach Zero Energy Settlements and Sustainability

In addition to the keynote presentations, we have 18 workshops that will be conducted by the senior international and national research and industry professionals in different aspects of heating, ventilating and air conditioning.
The conference will host one technical panel on the critical issue of academic education and training in HVAC design and construction.
We would like to extend our appreciation to the Entities that gave generous support to this conference starting with Building and Housing Research Center, Iranian Construction Engineering Organization, Sharif University of Technology and The Iranian e-Community of Mechanical Engineers along

with international and national research and development centers and universities.

Finally, we would like to thank the scientific and international advisory committees for their continuous guidance and support throughout the various stages of the conference and the organizing committee for their persistent effort to formulate a successful conference.

I wish you an enjoyable, stimulating and productive conference.

Thank you

M.H. Saidi (Professor)

Chairman of the Organizing Committee (ICHVAC-2)

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1006

Simulation and Comparison of Air Blowing Angles in an Amphitheater

D. Rajaee[1], A. Abbassi[2]

[1]M.Sc Student, Amirkabir University of Technology, Mech. Eng. Dept;
dariush.rajaee@gmail.com
[2]Professor, Amirkabir University of Technology, Mech. Eng. Dept.; abbassi@aut.ac.ir

ABSTRACT

Flow pattern and heat transfer of Amirkabir University of Technology' Amphitheater cross section has been studied. The effect of different air entrance angles on temperature and velocity are compared using isotherm contours and CFD streams. The continuity, momentum and energy equations solved by using CFD method. To discretize the conserved equations, the projection method was applied based on the second order central discretization. The flow was assumed, laminar, steady state and two dimensional. Also, the air enters from two diffusers and exits from one diffuser. The people modeled as a thermal source. The numerical results and comparison of flow patterns of temperatures and velocities show that the entrance air angles close and higher than 45 degrees are suitable. In this angle, forced flow is able to take out the heat from occupied regions.

Keywords: Amphitheater, Air Entrance Angle, CFD, Projection Method

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1007

Investigation of Pollutant Source Motion Effect on the Particle Removal Efficiency of a Cleanroom

G.R. Molaeimanesh [1], B. Sajadi [2] , M.H. Saeidi [3]

[1] Ph.D. Candidate, School of Mechanical Eng., Shiraz University
[2] Ph.D. Candidate, School of Mechanical Eng., Sharif University of Technology
[3] Professor, School of Mechanical Eng., Sharif University of Technology

ABSTRACT

Nowadays cleanrooms have a growing application in a broad range of industries such as pharmacy, aerospace, nuclear energy, electronics and etc. So, it seems that numerical modeling of airflow in such spaces becomes more essential.
In this research, the role of two essentially effective parameters on the clean room efficiency which are inlet-outlet position and pollutant source motion has been investigated.For this investigation, particle flow in 10 cases of 3-D full scale cleanroom has been numerically modeled by Fluent 6.3, using Eulerian approach and standard k-ε turbulence model and by the aim of Final Efficiency (FE) all 10 cases where compared by each other. The results show that the more pollutant source motion intersect the streamlines which reach the outlet, the more efficient cleanroom we'll have.

Keywords: Clean room, Pollutant source motion, Inlet-outlet position, turbulence model, Eulerian approach

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1009

Analysis of Airflow inside the Windcatchers in Different Ambient Conditions and the Effect of Water Spray on it

M. Hojjati[1], H. Ahmadikia[2]

[1]Bachelor of Science, Mechanical Engineering, Bu-Ali Sina University in Hamedan;
Mohammadhojjati@yahoo.com
[2] Assistant Professor, Mechanical Engineering, Bu-Ali Sina University in Hamedan;
Ahmadikia@yahoo.com

ABSTRACT

The function of windcatchers consists of catching the required wind and directing it into the building's main rooms, ab-anbars (water reservoirs), or Sardabs. The historic city of Yazd is known as the city of windcatchers, and according to investigations, in comparison with other central cities in Iran, it has the most number of windcatchers. Located in this city, is the tallest windcatcher in the world, the Bagh-e-Dowlat-abad Windcatcher, which stands at about 34 meters high. The upper shaft of the windcatcher is 11 meters high. Wind flow analysis inside the Bagh-e-Dowlat-abad windcatcher was performed and the effect of water spray (water fountains) on temperature changes and relative humidity in different locations in the windcatcher were investigated. Wind flow analysis was performed with various wind speeds and with two different water spray methods, one against wind flow (under the windcatchers) and one perpendicular to wind flow (inside the octagon tower). The main aim of this experiment is to study the basis of the change in the average ambient temperature in the octagon tower and of the average humidity inside the windcatchers as a result of water spray.
To accurately measure the effect of the discharge volume of the water spray on the working of the windcatchers and the change in temperature and relative humidity water spray with discharge volumes of 0.9, 1.8, 2.7, 3.6, 4.5, 5.4 kg/s and input wind speeds of 3 and 6 meters per second, have been investigated.
Wind flow analysis inside the Bagh-e-Dowlat-abad Windcatcher was studied and the effect of water spray (water fountains) on temperature changes and humidity were investigated. By investigating different discharge volumes for two different sprays in two different locations and different input speed these results were achieved:
1) By using two different water spray methods it is observed that the first method almost results in the wasting of the spray water, and has no noticeable effect on the humidity and temperature. In contrast, the second method was effective and resulted in an increase in relative humidity and decrease in temperature on a larger scale.
2) By increasing the discharge of water with a constant speed, the averages of relative humidity and temperature were respectively increased and decreased linearly. For example, when the outdoor temperature is 45 °C and relative humidity is 20% and a wind with the speed of 3 meters per second is flowing, with a discharge volume of 5.4 kg/s (using 80 sprayers), the ambient temperature of the octagon room saw a 3.2 degrees Celsius decrease, and the amount of relative temperature saw an approximate 5.5% increase; which results in a more pleasant environment.

Keywords: Windcatchers, Water spray, Ambient condition

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1011

A Correlation to Predict Flow Boiling Heat Transfer Coefficient inside Inclined Microfin Tubes

S.M. Razavinasab, M.A. Akhavan-Behabadi*, S.G. Mohseni

School of Mechanical Engineering, College of Engineering, University of Tehan, Iran
* Corresponding Author, Email: akhavan@ut.ac.ir, Tel.: +98-21-88005677; Fax: +98-21-88013029

ABSTRACT

An investigation on heat transfer during flow boiling of R-134a inside a microfin tube has been carried out experimentally. The microfin tube has been provided with different tube inclination angles of the direction of fluid flow from horizontal, α. The experimental set-up was a well instrumented vapor compression refrigeration system. The test-evaporator was a copper microfin tube of 9.52 mm O.D and 1100 mm length. The averages outside wall temperatures of the microfin tube were measured at six axial locations. At each location four thermocouples were fixed at top, two sides and bottom positions.
A total of 224 test runs with four different refrigerant mass velocities of 53, 80, 107 and 136 kg/m^2-s were performed for seven different tube inclinations from α = -90˚ to α = +90˚ (with intervals of 30˚). The evaporation temperatures were between -5°C and -24°C. The calculated experimental heat transfer coefficients were compared with three existing correlations for horizontal and vertical up flow. It was concluded that these correlations predict the experimental data in a proper error band.
The experimental results indicate that the tube inclination affects the flow boiling heat transfer coefficient in a significant manner. The heat transfer coefficient increases with the increase of vapor quality until a vapor quality near 75–85% when it begins to decrease because of dryout. At low vapor qualities, the highest heat transfer coefficient is attained at the inclination angle of +90° and at high vapor qualities the highest heat transfer coefficient occurs when the microfin tube is horizontal or is inclined at -30˚. The vertical tube with inclination angle of -90° has the lowest heat transfer coefficient for the entire range of vapor quality.

Keywords: Boiling heat transfer, inclined microfin tube, experimental study

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1029

Influence of Radiative Source to Heat-Mass Transfer Intensity of Sprayed Water Droplets in Air Conditioning Processes

G. Miliauskas[1], S. Sinkunas[2], K. Norvaisiene[3]

[1] Kaunas university of technology, K.Donelaičio St. 20, LT-44239 Kaunas, Lithuania; gintautas.miliauskas@ktu.lt
[2] Kaunas university of technology, K.Donelaičio St. 20, LT-44239 Kaunas, Lithuania; stasys.sinkunas@ktu.lt
[3] Kaunas university of technology, K.Donelaicio St. 20, LT-44239 Kaunas, Lithuania; kristina.tiuksaite@stud.ktu.lt

ABSTRACT

Two phase water droplets-gas flows are common not only in natural phenomena, but are also met in important thermal processes such as waste heat utilization in ventilation; heat exchangers of building HVAC systems; air humidifying and drying in evaporation chambers of air conditioning systems. The rate of a droplet heating and evaporation is the essential factor of the state change of a two phase system droplets-gas.
In the evaporation chamber there is a small difference of energetic potential between air and water injected into air, therefore the processes of phase transformations can be inefficient. One searches for the methods to intensify heat and mass processes. For that it would be possible to use additional thermal energy, introduced into heating of water droplets. Radiation sources independent from air state - heated surfaces or even laser could supply additional energy. For development of such air-conditioning technologies, it is necessary to be familiar with the radiant influence on the heat and mass transfer processes of sprayed water droplets. Also, it is necessary to take into account the intensity of transfer processes in the air flow and inside the evaporating droplets. Additional radiant energy allow forming unsteady temperature field with negative gradient in sprayed water droplets and essentially changes the rate of phase transformations on the droplet surface. That allows the possibility to control more efficiently the air-conditioning process in the evaporation chamber by combining temperature of the radiant source and dispersity of sprayed water. Properly selected boundary conditions of heat and mass transfer allow humidifying and right air warming in the evaporation chamber.
Thermal state of water droplets and intensity of phase transformations on their surface is modelled in the evaporation chamber of air conditioning system. The modelling is performed using the method of combined analytic – numeric research of heat and mass transfer in a two-phase flow. System of parabolic type second power integral-differential equations describing complex heat transfer in the droplet by conduction and radiation is transformed into infinite series of integral equations. Calculating the rate of droplet evaporation and the intensity of convective heating, the influence of the Stefan's hydrodynamic flow is taken into account. Securing balance of energy fluxes in the droplet with confidence of one hundredth of percent and using the fastest sinking the droplet surface temperature is estimated.
The results are presented in the real time scale and the Fourier number – based time scale, described by the characteristic curves, which reflect dynamics of the phase transformation and thermal state parameters of water droplets, independently of the sprayed water dispersity.

Keywords: water droplets, air conditioning process

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1030

Design and Development of a Prototype of Air Cooling System for Truck-Cabin Using Heat from Engine Exhaust

H. Tiwari[1], G. V. Parishwad[2]

[1]Assistant Professor, Mechanical Department, Pimpri Chinchwad College of Engg. Pune 4110 44 India; sharvay12@rediffmail.com
[2]Professor, Mechanical Department, Government College of Engg. Pune 411005 India; parishwad@rediffmail.com

ABSTRACT

The present work is focused towards the development of a prototype of air cooling system for the cabin of truck using waste heat from exhaust. The available options in heat generated cooling have been reviewed. The adsorption refrigeration cycle using exhaust heat is found to be suitable for transport trucks cabin cooling. A scale of 5:1 is decided and a prototype of 0.2 TR has been designed, the development of the prototype is in progress. The prototype will be tested with various adsorber configurations.

Keywords: Prototype, Truck Cabin cooling, Engine exhaust, Absorption refrigeration

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1036

Thermodynamic and emission of pollution analysis of Hybrid SOFC-MGT cycle to Provide the Energy In Residential Building

A. Abdollahi[1], A. Alemrajabi[1] , M. Ghasemi[2], J. Pirkandi[2]

[1] Department of Mechanical Engineering, Isfahan University of Technology
[2] Mechanical Engineering Dept., Khaje Nasir. T. University of Technology, Tehran, IRAN

ABSTRACT

In this paper, the hybrid solid oxide fuel cell with gas micro turbine system has investigated in aspect of thermodynamic and proportion emission of pollution. Then the overall system performance was analyzed by analyzing the different components separately; and further applying thermodynamic laws for the entire cycle. Also, a parametric study was performed to illustrate the effects of various parameters such as compressor Pressure ratio, air mass flow rate on heat power, and electrical power and CO_2 emission.

Keywords: Thermodynamic analysis, emission of pollution, Hybrid cycle, Residential Building

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1069

Designing a Thermal Energy Storage (TES) System for Space Cooling an Office Building in Ahwaz and Comparison of them with Cooling Systems without TES

S. Mansoori[1],V. Mokarizadeh[2], M. Jabbar[3] , M. Noori[4]

[1]MS in Mechanical Engineering, Niroo Research Institute of Iran; shmansoori@nri.ac.ir
[21]MS in Mechanical Engineering, Niroo Research Institute of Iran; vmokarizadeh@nri.ac.ir
[3]MS in Energy Engineering, Tavanir ; jabbar@tavanir.org.ir
[4] BS in Mechanical Engineering, Tavanir ; mnoori@tavanir.org.ir

ABSTRACT

Thermal Energy Storage (TES) for space cooling is a technology that reduces electricity cost by shifting chilling activities to off-peak times. Generally, this system contains a compression chiller to ice making, thermal energy storage tank, brine, heat exchanger, pumps and valves. Basic of TES system is based on the heat storage properties of a material (water, ice or eutectic salts), which stores heat or alternatively cool for use later. TES systems are operated in two modes: full storage and partial storage. Full storage systems, also known as load shifting systems are designed to meet all on-peak cooling loads from storage. Partial storage systems meet part of the cooling load from storage and part directly from the chiller during the on-peak period. In this paper, an office building in Ahwaz has been selected to study the effect of using TES systems with different strategies such as partial and full storage operations on chiller and storage sizes and reduction of peak electric demand, energy consumption of the chiller for selected charging and discharging hours. and comparison of them with cooling systems without TES. Ice cool thermal storage systems have different types that a system with an internal melt ice-on-coil tank has been selected to use in this investigation. An important step to select the strategy is to size the chiller and storage capacities, a detailed analysis of the combined performance of the chiller and ice storage for each hour of the design day cooling cycle must be performed. However, an initial estimate can be performed by applying simple available formulas and by making some assumptions of the chiller capacity during the day and night time. The basic steps in sizing a cool storage system are as: 1. Determine accurate building load profile, 2. Select the design day system operating strategy, 3. Calculate the initial size and initial storage capacity, 4. Select the appropriate storage technology, and 5. Refine and finalize the chiller and storage equipment selection.
It has been found that the full storage operation has large chiller and storage capacity, energy consumption and peak electrical reduction. However, partial storage (load leveling) has the smallest chiller and storage capacities, and peak electrical reduction. Although, increasing price difference of electricity between on-peak and off-peak hours will economically improve using TES systems, the increasing rate of electricity demand during summer days in the southern of Iran, could be moderate using TES systems.

Keywords: Thermal Energy Storage Systems (TES), Partial Storage, Full Storage, Peak Load

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1077

Numerical Simulation of Indoor Air Flow and Particle Deposition in the Model Room

M. Yousefi[1], N. Abbaspoor[2], M. S. Javadein[3], S. Lafmejani[4]

[1]Department of Mechanical Engineering, Isfahan University of Technology ,
yousefi@me.iut.ac.ir
[2]Department of Mechanical Engineering, Babol University of Technology,
nima1386@gmail.com
[3]Department of Mechanical Engineering, Babol University of Technology,
m_javadein@yahoo.com
[4]Department of Mechanical Engineering, Isfahan University of Technology
,eng_s_sadeghi@yahoo.com

ABSTRACT

In modern buildings, both indoor air quality and airflow pattern of room are important. In this study, tow major parameters – flow pattern and particle deposition – in the chamber analyzed. Airflow in room is non-isothermal, 3-D and transient, there are some limitations on available tools to determine and predict building's indoor airflow.
8 particles size groups ranging from 0.1μm to 10μm are investigated. In this paper, a comparison between one numerical model and a semi-empirical expression considered, to obtain deposition fraction results. Density of particles inside a room has influence on the air quality. Continual breathe of these particles can cause illness. Indoor airflow pattern characteristics and demand for better measurement accuracy led to appear new methods to amend flow pattern, air quality and thermal comfort. Simulation has been done aided by fluent Engineering package and room geometry and grids has been crated aided by Gambit codes.
Room's space is length(x) ×width(y) ×height (z) =0.8m ×0.4m × 0.4m. Size of inlet and outlet's opening are the same (0.04m × 0.04m). Inlet velocity is 0.225m/s. The room air temperature is set as 27ºc and the room temperature condition is isothermal. In the chamber, low volume fraction and discrete are used Lagrangian approach. It's described particles state in 3 zones. In first zone Reason of particles behaviors were explained to this form that random movement of these particles increased with the decrease particle diameter and this phenomenon lead to exit fewer particles from room and more of them remain in room space. In particles with a high diameter, effect of Brownian force on this reduced and due to gravity forces show less inclination to stay in exit flow line of room and more particles remain in room space (third zone) with the kind of particles with have medial size and neither Brawn nor gravity force doesn't overcome the theme, there is no detectable change in concentration of them in this range with increase and decrease in particle diameter.

Key word: Particle deposition, indoor air flow, Lagrangian model, Brownian force, Fluent

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1088

CFD Analysis and Optimization of Swirl Air Diffuser

B. Sajadi[1], A. Mohebbian[2], M.H. Saidi[3]

[1] Ph.D. Candidate, Sharif University of Technology; sajadi@mech.sharif.edu
[2] B.Sc. Student, Sharif University of Technology; mohebbian@mech.sharif.edu
[3] Professor, Sharif University of Technology; saman@sharif.edu

ABSTRACT

The geometry of air diffusers has a key role in effective induction of airflow distribution through the indoor spaces. In this research, the airflow of swirl diffuser has been studied using computational fluid dynamics (CFD) and geometrical optimization has been done based on the results. The results show that both turning blade and slot angles have a vital role in making the ventilation airflow to swirl which leads to improve the diffuser performance. Based on the results, the optimum angle for the turning blade and slot are 32° and 55°, respectively. This study and its results are useful in gaining a better understanding about swirl diffuser airflow, the physical effects of its main geometric features, i.e. blade and slot angles, and also in design of more efficient diffusers.

Keywords: Swirl diffuser, Optimization, Computational fluid dynamics (CFD)

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1090

Ventilation System Design for Automobile Assembly Shops

B. Sajadi[1], P. Hanafizadeh[2], A. Irani R.[3], M.H. Saidi[4]

[1] Ph.D. Candidate, Sharif University of Technology; sajadi@mech.sharif.edu
[2] Ph.D. Candidate, Sharif University of Technology; hanafizadeh@mech.sharif.edu
[3] M.Sc. Student, Sharif University of Technology; abirani@mech.sharif.edu
[4] Professor, Sharif University of Technology; saman@sharif.edu

ABSTRACT

Industrial ventilation is a method for controlling worker exposure to toxic airborne chemicals or flammable vapors by exhausting contaminated air away from the work areas and replacing it by the fresh air. In this research, new design criteria have been developed for assembly shops ventilation system. By statistical study of automobile productions, pollutant emission in the roll test area of assembly shops has been estimated which leads to the required ventilation air flow rate. To design the best duct network, computational fluid dynamics (CFD)-aimed analysis has been used. The presented statistical-computational approach can be very useful in ventilation system design for applications which are not simple to use the common engineering procedures.

Keywords: Industrial Ventilation, Assembly Shop, Pollutant Level, CFD Analysis, Automobile

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1091

The Effect of Occupant Movement on the Cleanrooms Flow Pattern

J. Eslami[1], B. Sajadi[2], M.H. Saidi[3]

[1] M.Sc. Student, Sharif University of Technology; jabereslami@mech.sharif.edu
[2] Ph.D. Candidate, Sharif University of Technology; sajadi@mech.sharif.edu
[3] Professor, Sharif University of Technology; saman@sharif.edu

ABSTRACT

These days, according to increasing requirement of the clean rooms applications in industries particularly in the operating rooms, and within the developing industry of computer, positive indicators that show a real need to a compare comprehensive study and research about the operating rooms, in order to predict the distribution and dispersal of the particles. Correct determination of indoor airflow and particle concentration distribution results in new standard development for operating rooms. In this research we are going to consider the effects of a moving nurse on the operating room are studied numerically.

Keywords: Cleanroom, Occupant movement, Computational fluid dynamics (CFD)

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1104

Numerical simulations of fire-induced smoke control in tunnels with mechanical ventilation systems

M. Hosseinalipoor[1], F. Mohammadi[2], M. Khayat

[1]Associated Professor, Iran University of Science & Technology, alipor@iust.ac.ir
[2] Shahid Rajaee teacher training university, feraydoon.mohammadi@gmail.com
[3]Shahid Rajaee teacher training university, Zhian62@yahoo.com

ABSTRACT

In case of a tunnel fire, toxic gas and smoke particles released are the most fatal contaminations. When a fire occurs in a long tunnel, smoke control is crucial for obvious reasons of safety. Ventilation and extraction systems have to be designed with accuracy in order to control the longitudinal motion of the fire-induced smoke and to extract it efficiently in a zone close to the fire source. Longitudinal ventilation systems are commonly provided in long tunnels. A strong air flow driven by jet fans is used to push smoke to one side. This might be useful when a small fire occurs in a tunnel. But if the air flow due to operating the longitudinal ventilation system is too low, a smoke flow directed against the air flow (upstream direction) called as ‘‘back-layering” is induced. Smoke would then flow in both the upstream and downstream directions. This occurs easily for sloped tunnels because the air movement is naturally induced by buoyancy due to the temperature differences between the two ends. The main objective of this model is to represent, as well as make possible, the duality between inertial forces (due to ventilation) and buoyant forces. In recent years Computational Fluid Dynamics has been used as a tool to evaluate the performance of emergency ventilation systems. In this paper, Computational Fluid Dynamics technique is used to simulate a small scale tunnel model (scale reduction is 1:20) to study the fire-induced smoke control by longitudinal ventilation systems. The Numerical model is non-thermal and a buoyant release is used to represent the fire smoke plume. Radiation and heat losses at the walls are not taken into account in this model. For numerical simulations, eight different buoyant conditions are chosen in a wide range of heat release rate (from 0.1 to 10 kW). For each one, the nozzle diameter, the buoyant release density and flow rate are adjusted according to the semi-empirical model proposed by Me´gret et al for tunnel fire applications. Practically, the critical condition is reached by decreasing the longitudinal air flow until the back flow vanishes (visual control). Critical velocity (minimum longitudinal velocity needed to prevent smoke back flow when a fire occurs in a tunnel) is then measured. Numerical simulations were carried out on the reduced scale model to study the evolution of the critical velocity as a function of several basic parameters: fire HHR and nozzle diameter. Finally, Numerical simulation results are compared with former experimental results.

Keywords: Tunnel; Smoke propagation; Numerical simulation; Heat release rate

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1097

Experimental Study of Indoor Airflows through Particle Image Velocimetry (PIV)

S.E. Kiaei[1], S.N. Saidi[2], B. Sajadi[3], M. Zabetian T.[4], M.H. Saidi[5]

[1] B.Sc. Student, Sharif University of Technology; e.kiaei@gmail.com
[2] B.Sc. Student, Sharif University of Technology; n_saeidi@sharif.edu
[3] Ph.D. Candidate, Sharif University of Technology; sajadi@mech.sharif.edu
[4] Ph.D. Candidate, Sharif University of Technology; zabetian@mech.sharif.edu
[5] Professor, Sharif University of Technology; saman@sharif.edu

ABSTRACT

In this research, the indoor airflow has been investigated experimentally and numerically using a down-scaled model. Experimental study has been performed through PIV technique and the results have been used to study the accuracy of common turbulence models. The results show that both standard and RNG k-ε turbulence models have acceptable accuracy; although RNG model depicts some priority. This research and its results are useful in getting a better understanding about indoor airflows and in providing a bench-mark to choose a proper turbulence model.

Keywords: Indoor airflow, Particle Image Velocimetry (PIV), Experimental study

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1122

Numerical simulation of flow field and heat transfer in a room with wall or ceiling cooling panel

H. Khorasanizadeh[1], G.A. Sheikhzadeh [1], A.R. Sabonchi[2], H. Botshekan[2]

[1]Associate Professor, University of Kashan, Kashan, Iran
[2]Master of Eng. Sc. Student, University of Kashan, Kashan, Iran

ABSTRACT

In this study, the fluid flow and temperature distribution in a room with a radiant cooling panel has been simulated numerically using Fluent software. A room which has dimensions of (3 × 3 × 3 m) with two external walls (East and North) and a window (1 × 1 m) on the eastern wall has been considered. Other surfaces have been considered as interior walls. The location of cooling panel has been considered on the ceiling also on the western wall. To evaluate the effect of occupant a cube has been considered in the center of the room with its external walls having constant heat flux in some models. To evaluate the effect of outside summer design temperature, simulations have been performed for Tehran and Semnan, with two different outside design temperatures.
Governing equations for turbulent incompressible flow (continuity, momentum and energy equations) coupled with radiation, have been solved using Boussinesq approximation with control volume approach and standard K-ε model also considering DO radiation model. For this purpose SIMPLE algorithm has been used.
The results show that at least 58% of the heat transfer to the cooling panel is due to radiation. The ratio of radiation rate to the total heat transfer is higher for the case of the wall cooling panel. By using radiant cooling panel on the wall or the ceiling, the vertical and horizontal distribution of temperature in the room is almost uniform and the average room air speed is less than 0.2 m/sec. For the studied conditions in this research, a comparative study showed that wall panel cooling uses less energy than the ceiling cooling panel, although both systems provide thermal comfort conditions. Therefore, for providing comfort condition and lower energy consumption using wall cooling panel is suggested.

Keywords: Ceiling radiant cooling panel, Wall radiant cooling panel, Thermal comfort

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1126

Economic Analysis to Propose the Optimum Capacity and Working Schedule of a Gas Engine Based Cogeneration Plant for an Industrial Sample Factory

A. Jafarian[1], J. Pakdaman[2]

Tarbiat Modares University; A.jafarian@avinpalayesh.com
Avin Palayesh Niroo Co; J.pakdaman@Avinpalayesh.com

ABSTRACT

Distributed power generation systems are employed with the aim of energy consumption optimization, decrease of the energy transfer losses and decrease of the pollution generated from fossil fuel combustion processes at high capacity power plants. In fact, Power generation and transfer in a large scale result in a huge energy loss. Recently, a great attention has been given to the distributed power generation plants in Iran, meanwhile, CHP systems have been employed more, because of their higher efficiency. In this paper, a mixed integer nonlinear programming model has been developed. The objective is to maximize the net present value (NPV) of the system for an industrial sample factory. The model reports the optimal CHP and auxiliary boiler capacities and the working schedule to supply the customer electrical and thermal energy requirements annually. A case study has been performed based on Iran's existence energy laws as well. The problem has been solved by using GAMS software to optimize the net present value of the plant from the factory owner's point of view. Furthermore, a sensitivity analysis has been done to present the variation of optimal solutions due to the changes of some key parameters such as fuel and electricity prices.

Keywords: Cogeneration, Gas Engine, NPV, Optimization

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1129

Study The Effect of The Fabric Ducts Nozzle Angle in a Pool

A. Niazmand[1], B. Ghadiri Dehkordi[2]

[1] Tarbiat Modares University, Depth. Of Mechanical Eng., a.niazmand@modares.ac.ir
[2] Tarbiat Modares University, Depth. Of Mechanical Eng., ghadirib@modares.ac.ir

ABSTRACT

Usual materials for designing and fabricating in HVAC systems are iron alloys. A major problem for using these alloys is corrosion. Rate of corrosion is increased by humidity in a wetted area like a pool. For solving this issue, designers use fabric ducts instead of metal ducts. They are fabricated from polymer tissues, so they can tolerate humidity very well. Another feature of these ducts is they can distribute air very smoothly, and users can have a good control on the air distribution. Because of fabricating easily, they can determine where they want to put their nozzles, and they can use many types of nozzles.

For designing HVAC systems efficiently, it is necessary to determining the pattern of flow in designing spaces. One of the knowing this is to simulate it numerically and this work is done by commercial software like Fluent, Ansys-CFX, Star-Adapco and so forth. The first step in simulating a case is to generate grids in the case for solving equations that are related to its condition. When there is a simple geometry, designers can use structure grids. There have several features like more accuracy and solving faster than other methods but when there is a complex geometry, we should use unstructured grids. These types of mesh generation fill our computational space with triangular nodes. For remedying the slower answering, they have to use parallel processing. For doing this, they should use some libraries like Openmp and MPI to divide the case into smaller parts and solve it faster.

The equation which we should solve is dividing to two main parts. First, we should solve physical equations like continuity, Navier-Stocks equation and energy equation. Because we study a complex geometry, the flow behavior can be turbulent. To simulating this turbulence effect, we should use some modeling like LES or $k-\varepsilon$. it is necessary to use reliable model. $k-\varepsilon$ has more features like high stability and high accuracy. In these types of models, two equations are solved to determining kinetic energy and energy dissipation of flow. By adding this to viscosity term, the viscosity effects are modeled.

The results, based on contour pattern of velocities and stream lines, show that changing the angel of nozzle can affect the nozzles pattern widely. It satisfies to use computational method for designing such types of HVAC systems. By looking these patterns, designers can study the zones which are created by these effects better and design a reliable system.

Keywords: Numerical simulation, Fabric ducts, CFD, Unstructured mesh, Parallel processing

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1039

Numerical analysis of cooling a three floor building by using combination of Solar chimney and Trombe wall system

M. Rabani[1], V. Kalantar[2]

[1] M.S. Student, Yazd University, rabaniasd@stu.yazduni.ac.ir
[2] Assistant of professor, Yazd University, vkalantar@yazduni.ac.ir

ABSTRACT

Nowadays, the energy used for heating or cooling the buildings makes up some 40 percent of the total energy consumption. This fact not only squanders our fossil fuels but also leads to the pollution of environment. Therefore, in the recent years, making use of renewable energies in the heating and cooling the buildings has caught the attention of engineers and architectures.
This paper, study a combination of the solar chimney and trombe wall system to cool a three floor building, in order to economize on fuel and mitigate environmental problems. Also, in this building a water injection system has been foreseen at the entrances to produce more moisture and desirable cooling.
The fact that the building has three floors warrants more attention to the natural ventilation of the building. We can solve this problem with a combination of the solar chimney and trombe wall system. The trombe wall which is made of copper acts as absorbent in the solar chimney system to enhance its efficiency and better ventilate air. The inner side of the trombe wall has been insulated, and also the rooms ceiling and floor have been insulated to prevent heat from escaping.
The airflow in the controlled volume has two-dimensionally been formulated according to the turbulence model of k-ε [realizable] and SIMPLEC algorithm. Moreover, in this system, radiation has been modeled with the aid of the model of Discrete Ordinates (DO). The software applied is Fluent 6.3.26 and a rectangular mesh has been considered for this issue.
According to the results, in summer the temperature inside the building is 25 centigrade. Average speed of the airflow in the chimney is about 1 m/s which is suitable, given that the building has three floors. The amount of energy can be saved above 70 percent.

Keywords: Solar chimney, Trombe wall, Natural ventilation, Solar radiation

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1045

Numerical simulation of building solar heating using thrombi wall

V. Kalantar

Mechanical Engineering Department, Yazd University, vkalantar@yazduni.ac.ir

ABSTRACT

At present, in Iran about 40 percent of using the total energy is related to the constructional parts that the significant part of it used for heating or cooling it, so in addition to irregular use of fossil fuels, it caused to the pollution of environment. In this essay for saving the use of fuel and decrease the environmental problems with presenting the idea of using solar energy for the automatic move of hot air inside the building by the solar chimney (thrombi wall) without using motive power or fossil fuels it's possible to provide a suitable and warm environment with appropriate temperature in cold and partly sunny areas according to the air conditioning standards, for this purpose with introducing the computational areas in two dimensional forms and using the conservation equations on the turbulent flow and exerting the suitable numerical methods, desired results were attained.

Keywords: solar energy, radiation, airflow, circulation

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1048

Determination of the slope and surface azimuth angles of solar panels for receiving the maximum solar radiation intensity for areas of southeast Iran

P. Talebizadeh[1], M. Abdolzadeh[2], M. A. Mehrabian[3]

[1]Shahid Bahonar University of Kerman; talebizadeh.pouyan@yahoo.com
[2] Islamic Azad University of Kerman; mabdolzadh@yahoo.com
[3] Shahid Bahonar University of Kerman; ma_mehrabian@alum.mit.edu

ABSTRACT

In this paper, the slope and surface azimuth angles of solar panels are determined for receiving maximum solar radiation intensity in four Iranian cities: Kerman, Yazd, Zahedan and Birjand. The energy obtained for Yazd is more than the other cities and as a result of that, this city is considered as the basis in this study. Optimum angels are calculated in different days, months, seasons and the whole year, employing different models. The results showed that the optimum azimuth angle is zero for receiving maximum solar energy. In addition, receiving maximum solar energy in different days of a year is accomplished in different slope angles, which are not economical to use a solar tracker in order to change the angle each day. However, the energy computed using the daily and monthly optimum angles is almost the same. On the other hand, comparison of the results in four cities showed that the optimum angles for these cities are close to each other due to the close latitude of the cities.

Keywords: Solar panel, Solar energy

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1060

Simulation and economic analysis of Solar Assisted air-conditioner

P. Ooshaksaraei[1] , K. Sopian[2] , R. Daghigh[3]

[1]Master Student, University Kebangsaan Malaysia, Solar Energy Research Institute,
p_ooshaksaraei@yahoo.com
[2]Prof., University Kebangsaan Malaysia, Solar Energy Research Institute,
ksopian@vlsi.eng.ukm.my
[3]PhD. Student, University Kebangsaan Malaysia, Solar Energy Research Institute,
rdodper@yahoo.com

ABSTRACT

Solar energy is one of the most well known green sources of energy in the world. It has many applications such as water heating, space cooling, solar drying and etc. This research presents a feasibility study of a small scale air conditioning unit consists of an aqua-ammonia ejector absorption system equipped by evacuated solar thermal collectors. Absorption system provides cooling during hot season and solar collector provides heating during the cold season directly. This refrigeration system consists of 2 pressure levels, a high pressure side in the condenser and the generator, and a low pressure side in the evaporator and the absorber. The idea of utilizing an ejector cycle is to reduce throttle losses. The high-pressure solution expands through the nozzle in the ejector and creates a vacuum at the other end of the ejector. As a vacuum occurs, the vapor from the evaporator is drawn into the ejector. The dynamic simulation has been performed using TRNSYS 16 environment. TRNSYS software is special software for energy system simulation. This software could be used in energy system simulation and specialized in the field of renewable energy. TRNSYS software is able to define a link to other software such as Fluent, MATLAB, Excel and etc for more complicated calculations and simulations. In this study, we modeled the absorption system in Excel, by implementing the derivative equation of binary working fluid (aqua and ammonia). Proper harvesting of solar energy will provide an invaluable thermal source of energy that could be investigated in different applications such as Air-conditioning (space cooling), space heating, drying and etc. Solar energy can run the system but not continuously. That why we need an auxiliary source to assure the system will functions properly. This study presents the solar fraction for a small scale aqua-ammonia ejector Air conditioner from 10% to 35% when the solar collector area is 5~50m^2. The economic analysis has also been conducted. Electrical cost less than 0.45 USD/kW.hr is not economical and leads to negative saving. In first glance, we may find solar energy as a non economic one; but the fact is; at this moment, the fuel cost is affordable, because of almost low cost non-renewable but invaluable Oil sources. We have to replace the conventional energy resources by renewable ones, for the time that there will be no oil any more. By economic analysis, we would be able to find the most economic system design and sizes.

Keywords: Solar energy, Evacuated tube, Absorption, Air-conditioning, Economic analysis

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1065

Subsoil Heat Exchanger

F. Nasrollahi

Berlin University of Technology, nasrollahi@sustainbuilder.com

ABSTRACT

This paper deals with the potential of energy saving and economic viability of application of subsoil heat exchangers in buildings. The ground, during winter, has a higher temperature than the outdoor air, and during the summer a lower temperature. Therefore, it is possible to preheat fresh air in an earth buried duct during winter, or to cool it in summer. This can be done directly with air ducts in the ground, which called subsoil heat exchanger. Therefore, application of subsoil heat exchangers will lead to an energy saving in buildings.

In winter, the cold air flows through the underground pipes; it gets heated by the earth before it reaches the building. In summer, the outside warm air becomes cool by the earth to a pleasant temperature, thus decreasing the supply air temperature and room temperature. Therefore, the use of a subsoil heat exchanger for preheating the air in winter and pre-cooling it in summer will reduce the energy need for both heating and cooling.

In order to determine the amount of energy a subsoil heat exchanger can save in buildings, a sample subsoil heat exchanger is simulated in the city of Tabriz. The heat exchanger is 52.20m long, and can be located in the courtyard of a small residential building. Based on these simulations, the energy saving of such a heat exchanger is 2409.3kWh in winter and 450.4kWh in summer. Dependent on the built area of the house, this exchanger can save 12kWh/m²a heating energy and 2.25kWh/m²a cooling energy for example for a 2-story residential building with 200m² built area.

The economic evaluation of use of subsoil heat exchangers also shows that with the energy price in Iran, the use of subsoil heat exchanger is not economically viable because the payback time for investment for a subsoil heat exchanger is more than its utilization time. However; with the average energy price in the other countries, the subsoil exchanger is economically viable there. With the average energy price in the other countries, the payback time is 18.2 years and the internal interest rate is 20.7%.

The subsidization of energy carriers, which are paid from national capital, is the reason of low energy prices in Iran. Therefore, investment for subsoil heat exchangers is to the benefit of the national capital in Iran, even if there is no high public demand for such investments.

Keywords: Subsoil heat exchanger, Energy saving

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1093

Evaluation of a combined heat and power generation system with PV solar panels for a building

A. Shahsavar[1], M. Salmanzadeh[2], M. Ameri[3], P. Talebizadeh[4]

[1] M.Sc. Bahonar University of Kerman; amin.shahsavar@yahoo.com
[2] Assistant Professor, Bahonar University of Kerman; msalmanz@clarkson.edu
[3] Associated Professor, Bahonar University of Kerman; ameri_mm@mail.uk.ac.ir
[4] M.Sc. Student, Bahonar University of Kerman; p.talebizadeh@gmail.com

ABSTRACT

Photovoltaic (PV) panels are able to convert the maximum value of 20 percent of the receiving solar energy to electricity and major part of the absorbed solar energy would be dissipated by heat. The dissipated energy by heat could be used for preheating of the outdoor air required for an air handling unit in a building. On the other side as the electrical efficiency of the PV panels increases by temperature decrease, then the pre heating of the outdoor air can cause cooling of the PV panels and increases their electrical efficiency. In the present work, effect of certain numbers of PV panels on the outdoor air passing under these panels have been modeled for sample days in January, February and March for Kerman city in Iran and the total heating load required for outdoor air heating and the contribution of the PV panels in supplying this load have been calculated. Furthermore, the cooling effect of the outdoor air on the electricity generation of the PV panels has been studied. Results showed that using 80, 60 and 40 PV panels can supply all the required energy for heating of the outdoor air for sample days in January, February and March for Kerman city. On the other side, this heating process can also cause cooling of the PV panels and would increase their electrical power output.

Keywords: Photovoltaic, Photovoltaic/thermal, Fresh air heat load, building, Solar energy

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1095

Investigation and Comparison of the Performance of two solar heating systems, collector-floor heating and collector-radiator system for heating of the poultry houses

M. Sabzpooshani, H. Khorasanizadeh

Faculty of Engineering, University of Kashan

ABSTRACT

Due to an increase in conventional energy prices and environmental effects, the use of solar energy has increased. On the other hand, solar energy is being seriously considered for satisfying a part of the energy demand in Iran, as in the world. Fuel use constitutes a large percentage of the energy costs in the poultry houses. The objective of this study is to determine the feasibility of using the two solar heating systems, collector-floor heating and collector-radiator, in order to space heating of a broiler houses to reduce fuel consumption. Results are shown that collector-floor heating system is better than a collector-radiator system. On the other hand, collector-floor heating system was suitable in the cold and milder climate of Iran.

Keywords: Solar Heating, Active Solar Systems, Floor Heating, Radiator, Collector, Poultry Houses

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1112

Investigation and economic feasibility study of the using of trombe wall system in the poultry houses

M. Sabzpooshani, M. Monemzadeh

Faculty of Engineering, University of Kashan; spooshan@kashanu.ac.ir

ABSTRACT

Due to an increase in conventional energy prices and environmental effects, the use of solar energy has increased. On the other hand, solar energy is being seriously considered for satisfying a part of the energy demand in Iran, as in the world. Fuel use constitutes a large percentage of the energy costs in the poultry houses. The present paper describes the development, modeling and simulation of a solar passive system, trombe wall system, which can provide heating of the air in the poultry houses throughout the year. Results are shown trombe wall system, can provide about 15% of total fuel consumption in Tehran. Besides of low cost of initial installation of tromb wall system, payback was estimated less than two years.

Keywords: Solar Heating, Passive Solar Systems, Trombe Wall, Poultry Houses

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1127

The Renewable Energies and Sustainable Development of Iran

M. Barimani Varandi [1] , A. Kaabinejadian [2]

[1] Mazandaran Regional Electric Company –Iran; m_barimani1@yahoo.com
[2] Iran Renewable Energy Organization; kaabi@iranenergy.org

ABSTRACT

Limited fossilized energy resources and the problems which are due to spreading of greenhouse gasses makes it necessary to pay a close attention to new energies. Since there is a suitable and reasonable potential renewable energy in Iran, it is justifiable to have a Logical development in these resources, and in this way we can move on towards sustainable development goals.
The most important factor which is very effective in sustainable development is the energy resources. Having suitable energy is the main economical factor in industrial societies and that is after the labor force. Because energy is the basic need for continuing the economical development, social welfare, improving the life quality and security. Increasing the price of fossilized fuels, the importance of ecology environment, the security for providing energy, Technology improvement, and economical justification in some cases are important for determining the future of renewable energy. Iran has also done some activities in these fields in recent years. This is in such a way that, the social cast of ecological destruction by fossilized fuel consumption according to performed studies by world bank and Iran ecology protection was measured 92195 billion Rials in 2006, which is equal to 19.3% of NGP in that year.
Also, the fossilized and renewable energies in Iran (solar and wind) and hydro electrical energy in electricity net production in this year is 90.4%, 0.075% and 9.5%. And these proportions are 66.8%, 15.3% and 17.8% in OECD countries of North American for fossilized fuels, the renewable and Nuclear energies. The fossilized fuels share in European OECCD countries is 53.4% and in Asian and oceanic OECD is 69.7%.
Reaching to these expectancy objectives (goals) and move more than this share needs the private investors (internal and external) for investing in this substructure developing of the country. And this needs some structural changes and economical motives for the cooperation of the private sector to develop, and the programmers and planers and policy makers should pay attention. Apparently, these energies need more basic investment than fossilized fuels, while they don't need fuel during the work, they actually do not need recent or current expenses. The main problem in developing the application of renewable energies is the high investment (primary investment) and in such situation the government should support the private and public sectors and develop the investment in these sections.

Keywords: Renewable energy, Sustainable energy

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1021

Determination of Thermal Comfort Regions of Amirkabir University's Amphitheatre

D. Rajaee[1], A. Abbassi[2]

[1]M.Sc Student, Amirkabir University of Technology, Mech. Eng. Dept;
dariush.rajaee@gmail.com
[2]Professor, Amirkabir University of Technology, Mech. Eng. Dept.; abbassi@aut.ac.ir

ABSTRACT

In present paper, thermal comfort condition of AUT's amphitheatre has been studied using CFD modeling. A CFD computer code was used to calculate 3D temperature and velocity fields using the solution of continuity, momentum and energy equations for amphitheatre ventilation. Comparisons of the results with those measured in different points were satisfactory. Determination of thermal comfort indices PMV and PPD by applying calculated temperature and velocity components show that thermal comfort condition inside the saloon is satisfactory in 18 C and only 13% of the people have dissatisfaction.

Keywords: Thermal comfort, Amphitheater, CFD, PMV, PPD

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1046

Air Condition and Natural Ventilation; Thermal Comfort at Interior Environment

M. Ranjbar Jourjadeh[1]

[1]Department of architecture, University of Tehran, mohammad.ranjbar.arch@gmail.com

ABSTRACT

Main sections of interior environment comfort are thermal comfort, acoustical comfort, visual comfort and interior air quality. The Air conditioner created for thermal comfort providing of interior residence, also for more efficiency, some supplied of interior air quality considering in it. Natural ventilation is a traditionally method which was used to building for reach to such aims, and it currently applying now. As for residents thermal comfort providing these two methods can apply synchronously and their comparison from interior thermal comfort providing point of view will be useful.

This paper has paid attention to comparison of these two methods of cooling for thermal comfort at interior environment by means of field study. The field study has done for 12 residential building which the share of natural ventilation and Air conditioner was equal to each other and was 6 buildings for each way. For this purpose, the inside temperature of building recorded and besides the building residents completed questionnaires in each 2-hour period of time.

The result has shown that the residents were more comfort with the natural ventilation, and they have felt comforts in the wider range of temperature. Also, as the energy consumption is higher in the air conditioned, the value of resident's satisfaction is not so higher, which is the remarkable point. The thermal comfort equation according to air temperature in Air conditioner and natural ventilation were $y= 0.32x–8.16$ and $y=0.21x–5.27$ respectively. According to adaptability equations, the neutral temperature of Air conditioner and natural ventilation are 25.5 and 25.1 respectively. Then, this has shown that the people feel comfort by natural ventilation at the lower temperatures.

Key words: natural ventilation, air condition, thermal comfort, field study.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1018

Evaluation and simulation of surgery room air-conditioning system using Fluent

I. Tasdighi[1], J. Ghazanfarian[2]

[1]Zanjan University; i.tasdighi@me.iut.ac.ir
[2] Mechanical engineering branch, Faculty of Engineering, Zanjan University,
j.ghazanfarian@aut.ac.ir

ABSTRACT

In this paper various methods used for ventilation of operating room and optimized air conditioning systems are investigated. 3-D viscous fluid flow is simulated numerically in order to model and compare different modern ventilation methods. Generally, two common methods exist for ventilation of operating room called laminar flow and turbulent flow systems. Several operating rooms with different physical conditions and various configurations of inlet and exit port positions but with the same geometry are considered. Four doctors and one patient with an operating lamp have been located in the room. Specific velocity is imposed at the inlet port. In addition to the comfort conditions, reduction of pollutants and suspended particles are considered as important engineering parameters. Different configurations such as a vertical air flow system in order to reduce the number of stagnation points (dead points), with FLUENT software were simulated. Good agreements achieved between obtained results and experimental data.

Keywords: Surgery room air-conditioning, Simulation, Fluent

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1002

Factors that Influence Comfort Conditions in Buildings in Rasht

A. Ebrahimpour[1] , M. Maerefat[2] , B. M. Kari[3]

[1] Islamic Azad University Branch of Tabriz, Department of Mechanical Engineering; Salam_ebr@yahoo.com
[2] Tarbiat Modares University, Department of Mechanical Engineering; maerefat@modares.ac.ir
[3] Building and Housing Research Center, Materials section; kari@bhrc.ac.ir

ABSTRACT

Considering the high amount of energy required for thermal comfort conditions in buildings, energy use reduction strategies are essential. One way to reduce it is the appropriative selection of the external component of the buildings. In this study, the hourly effect of using the thermal insulation in building external envelope and use of natural ventilation on the level of thermal comfort conditions for the city of Rasht in warm seasons, using Energyplus software has been investigated. The results show that by using of natural ventilation at night in upper or lower floors, the thermal comfort level increases.

Keywords: Comfort Conditions, Wall Construction, Natural Ventilation, Thermal insulation

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1050

Application of LBM in Prediction of Natural Convection Occurring in Building Components

M. Nour Mohammadzadeh[1], M. Rahnama[1,2] and S. Jafari[1]

[1]Shahid Bahonar University of Kerman, Kerman, Iran; Mohammad.nmz@gmail.com
[2]Kerman University of Technology for Graduate studies, Kerman, Iran

ABSTRACT

In this article, a double-population thermal lattice Boltzmann method is proposed to solve the problem of the heated cavity with imposed temperatures. This family of problems can be considered as a test model for building physics application. A double population based lattice Boltzmann method has been implemented in order to simulate natural convection in the laminar flow fields $(10^3 \le Ra \le 10^6)$. The numerical results, concerning the heat transfer in the cases tested, are in good agreement with those from the literature. In order to demonstrate the possibilities of the method described in the article, applications are described covering double-skin facades and solar collectors or local heaters.

Keywords: double-population thermal lattice Boltzmann, natural convection, long cavity, double-skin facades, solar collectors

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1054

Environment and Occupants Expectation Effects on Thermal Comfort in Interior Environment

M. Ranjbar Jourjadeh[1]

[1]Department of architecture, University of Tehran, mohammad.ranjbar.arch@gmail.com

ABSTRACT

'Thermal comfort' is defined as ''that condition of mind which expresses satisfaction with the thermal environment". For occupants, thermal comfort assessment in the indoor environment have been brought up several methods that heat balance and adaptive theory more used in late decades. Heat balance method search for effective factors among physical factors which affect on thermal comfort and by determining of physical factors value on thermal comfort and inserting them in physical equation, the thermal comfort with numerical quantity was shown. But adaptive method says that people in their thermal environment are not passive about thermal comfort, and when unsatisfied about their thermal comfort, they will show reaction and behaviors to reach thermal comfort and adaption with the environment. In fact, they attempt to get comfort by use of several methods which these methods are distinctive at different conditions.

Behavioral model which used by the people at different environment has shown their perception to environment and differences in occupants behavioral model at different environments could be a reason for their different expectations and needs in that condition. For reach to better knowledge about occupant behavior from an energy point of view, we try to investigate occupant's behavior and analysis their thermal comfort by use of adaptive method in the indoor environment and on three residential, official and educational domains in this paper. Then not only subjective and objective factors in thermal environment but people behavior in different thermal condition recorded. The analysis has shown not only there are different occupants behavioral model at different environments, but this has caused difference for heat balance to the environment with different activities. Neutral temperature is 26.6 °C, 24.4 °C and 22.8 °C for residential, official and educational respectively. Also they have y=0.29x-7.74, y=0.15x-3.5 and y=0.27x-6.24 comfort equation respectively, which show their different about an expectation from environments.

Keywords: behavior pattern, thermal comfort, various activities, expectation, thermal regression.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1087

Numerical Analysis of Operating Theatre Ventilation

B. Sajadi[1], M.H. Saidi[2], G. Ahmadi[3]

[1] Ph.D. Candidate, Sharif University of Technology; sajadi@mech.sharif.edu
[2] Professor, Sharif University of Technology; saman@sharif.edu
[3] Professor, Clarkson University, USA; gahmadi@clarkson.edu

ABSTRACT

Surgical site infection (SSI) is one of the most challenging problems in design of ventilation system for operating rooms. In this research the effect of fixed and removable partitions on the risk of wound infection has been studied using CFD approach. The results show that such partitions, especially removable ones, have a major influence in reduction of airborne particle deposition on the wound and are highly recommended to control the risk of SSI especially in critical surgeries such as joint replacement.

Keywords: Operating theatre, Ventilation, CFD

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1004

Effect of Ambient Air Temperature on the Performance of Split-Air-Conditioner with Evaporative Condenser

E. Hajidavaloo[1], H. Eghtedari[2]

[1]Mechanical Engineering Department, Shahid Chamran University;
hajidae_1999@yahoo.com
[2]Mechanical Engineering Department, Shahid Chamran University;
eghtedari_hashem@yahoo.com

ABSTRACT

Increasing the coefficient of performance of split type home-air-conditioners is a challenging problem, especially in hot weather conditions. Application of evaporative cooled air condenser instead of commonly used air condenser is a solution to this challenge.
In this paper, an evaporative cooler was designed and coupled to the existing air condenser of a split air conditioner to evaluate its effect on the cycle performance under various ambient air temperatures up to 49°C. Experimental results show that application of evaporative cooled air condenser has a significant effect on performance improvement of the cycle and the improvement increases as ambient air temperature increases. It is found that by using evaporative cooled air condenser in hot weather conditions, power consumption can be reduced up to 21% and performance of the cycle can be improved up to 52%.

Keywords: evaporative cooler, split air conditioner, energy saving, condenser

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1010

An investigation on performance and governing equations of two bed adsorption chillers

M. Mahdavikhah[1], M. Vahidi[2], S. Vahidi far[3]

[1]Ferdowsi University of Mashhad; mmahdavikhah@yahoo.com
[2] Islamic Azad University of Mashhad; masood_vahidi_ok@yahoo.com
[3]Ferdowsi University of Mashhad; S_vahidifar@yahoo.com

ABSTRACT

Heating and cooling systems are widely used in industrial and comfort applications. Nowadays, the share of the energy for heating and cooling purposes in total energy consumption increases. Although there are numbers of types of devices for transferring heat from a cool source to a warm sink, only three are currently proven capable of rejecting the heat load for a house:
(1) Mechanical vapor compression (reversed Rankine cycle) is the overwhelmingly prevalent heat pump technology for automobiles, residential/commercial space cooling, and refrigeration.
(2) Absorption (liquid-vapor) heat pumps are the one common alternative, and are widely used as industrial/ commercial ice makers and water chillers, applications in which they are more economical than mechanical heat pumps.
(3) Adsorption (solid-vapor) refrigeration is analogous to liquid–vapor absorption, except the refrigerant is adsorbed onto a solid rather than absorbed into a liquid (dissolved) as in liquid–vapor heat pumps.
Adsorption heating and cooling can be a good alternative to classical vapor-compression machines. Adsorption cooling units are attractive, since they can be operated at temperature levels where liquid absorption systems cannot work. A basic adsorption chiller cycle consists of four main parts: an adsorber, which is a container filled with an adsorbent (such as silica gel, active carbon, zeolite, etc.) a condenser; an evaporator; and an expansion valve. An adsorption cooling cycle consists of four steps, which are isosteric heating, isobaric desorption, isosteric cooling and isobaric adsorption. The present paper covers the working principle of adsorption chillers, recent studies on advanced cycles, developments in adsorbent–adsorbate pairs and design of adsorbent beds. To optimize system performance, many numerical models have been proposed. The performance of an adsorption heat pump is controlled by many parameters such as refrigerant, adsorbent properties, system design and operating conditions. But the efficiency of the adsorption systems is largely dependent on the heat transfer rate and the mass transfer limitations. The heat and mass transfer (HMT) model for transportation process in adsorbent bed is studied in relation to heat transfer (HT) model, which is dependent on assumptions such as the conduction dominance and negligible mass transfer resistance.

Keywords: two bed absorption chiller, Heat and mass transfer

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1016

Effects of Geometrical Parameters on Exergy of Cold Water Storage Tanks

M. Farmahini[1], G. Heidarinejad[2], M. Maerefat[3], S. Delfani[4]

[1] Department of Mechanical Engineering, Tarbiat Modares University,
moienfafar@gmail.com
[2]Department of Mechanical Engineering, Tarbiat Modares University,
gheidari@modares.ac.ir
[3] Department of Mechanical Engineering, Tarbiat Modares University,
maerefat@modares.ac.ir
[4]Building and Housing Research Center (BHRC), Building Installations Department,
delfani@bhrc.ac.ir

ABSTRACT

Thermal stratification is applied in the field of energy storage to augment efficiency of energy use. Research on energy storages has revealed that the thermal performance or energy saving of a water storage tank can be increased by maximizing the level of thermal stratification within the storage tank. In this paper, impacts of geometrical parameters of tanks on the thermal stratification within a cold storage tank are analyzed. Seven two-dimensional models have been numerically simulated by using the computational fluid dynamics program, Fluent, with realistic boundary and initial conditions applied. The level of thermal stratification in each model has been quantified using exergy analyses. The results show increasing the tanks aspect ratio of height to width, decreasing inlet/outlet diameter, and moving the inlet/outlet position to the outer extremities of the tank all result in increasing levels of thermal stratification.

Keywords: Cold storage tanks, thermal stratification, exergy analysis, CFD.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1033

Exergy and Energy Analysis of a Diffusion Absorption Refrigeration Heat Pipe Cycle for Cooling

B.M. Zia pour[1], M .Tavakoli

Department of Mechanical Engineering, University of Mohaghegh Ardabili, Ardabil, Iran

ABSTRACT

This work is an attempt to propose an energetic and exergetic thermodynamic analysis of a diffusion absorption refrigeration heat pipe cycle. A thermodynamic model was developed for an ammonia–water absorption diffusion refrigeration heat pipe (ADRHP) cycle with helium as the auxiliary inert gas. Mass and energy conservation equations were developed for each component of the cycle. The performances of the system and exergetic coefficient of performance were examined parametrically by computer simulation. The validity of the simulated model was confirmed by comparison with previously published experimental data for ADRHP systems. Investigation of cycle performance under different conditions indicated that the best performance was obtained for a concentration of the rich solution of 0.35 ammonia mass fraction and that the recommended concentration of the weak solution was 0.1. Maximum performance coefficient is obtained in high evaporator temperature, a low thermo-syphon temperature. There is an optimized temperature for thermo-syphon for known operative temperature in which performance coefficient and Exergy performance become maximum value. In this temperature, also Exergy losses become minimum value. The effects of heat pipe dimensions on heat absorption from evaporator are investigated. Results showed that by increasing heat pipe diameter to 25 mm, absorption heat value increased. Exergy losses in the evaporator, condenser and dephlegmator are small compared to thermo-syphon and absorber. The total Exergy destruction of ADRHP cycle increases with increasing evaporator temperature and increasing thermo-syphon temperature.

Key words: Diffusion, Exergy, inert gas, performance, absorption, heat pipe.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1044

The Effect of Lewis Factor on the Counter Flow Cooling Tower Performance

M. Karami[1], G. Heidarinejad[2], S. Delfani[3], H. Pasdarshahri[4]

[1] MSc., Department of Mechanical Engineering, Tarbiat Modares University, m.karami53@gmail.com
[2] Professor, Department of Mechanical Engineering, Tarbiat Modares University, gheidari@modares.as.ir
[3] Assistant Professor, Building and Housing Research Center (BHRC), delfani@bhrc.ac.ir
[4] Ph.D. Candidate, Department of Mechanical Engineering, Tarbiat Modares University, hadi.pasdar@gmail.com

ABSTRACT

One of the main purposes of a building is to provide a comfort condition for its occupants. In modern societies, man spends the greater part of his life time indoors. Cooling towers are one of the most important parts of air conditioning devices that are used in a wide range of domestic applications.
High energy cost and problems associated with environmental concerns, call for efficiency improvement of conventional air conditioners and all related systems. Cooling towers commonly used in a medium to large cooling systems to reject the waste heat to the atmosphere via a water loop between building and outdoor. One of the advantages of regular cooling tower over the air-cooled condenser is the lower temperature approach. In the former, water approaches to wet bulb temperature, is much lower than dry bulb temperature.
The Lewis factor relates the relative rates of heat and mass transfer in wet-cooling towers. In this paper, the Lewis factor effect on wet cooling tower performance is investigated. First, the Lewis factor is introduced; then, governing heat and mass transfer equations is presented. The history and development of the Lewis factor and its application in wet-cooling tower heat and mass transfer analyses are discussed. Also, the Lewis factor effect on different parameters such as water outlet temperature, water loss rate is considered.
Results show that the amount of water that evaporates, however, is a function of the actual value of the Lewis factor. If the inlet ambient air temperature is relatively high, the influence of the Lewis factor, on tower performance diminishes. It is very important, in the view of the Lewis factor that any cooling tower fill test be conducted under conditions that are as close as possible to the conditions specified for cooling tower operating conditions.

Key Words: Cooling tower, Counter flow, Numerical simulation, Lewis factor

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1055

Thermodynamic analysis of water/lithium bromide double effect absorption cooling system with a boiler and cooling tower

H. Talesh Bahrami[1], S. Eftekhari[2], K. Javaherdeh[3], H. Saffari[4]

[1]Iran University of Science and Technology
[2] Islamic Azad University of Science and Researches
[3]Iran University of Science and Technology
[4]University of Gilan, Rasht, Iran

ABSTRACT

Refrigeration and ventilation have an important role in the human comfort. If very expensive buildings are built but don't have suitable ventilation systems, they will be intolerable for living. Therefore, with development of construction technology, refrigeration and ventilation technology also have been a considerable growth. Different systems according to weather conditions and fuel prices and many other factors are used in different regions for providing cooling of buildings. Absorption chillers are one of the cooling systems that do not harm the ozone layer. Wide engineering research has been conducted in the field of absorption chillers. In this paper, a system consisting of a double effect lithium bromide absorption chiller, boiler and cooling tower is studied. First, a system includes a cooling tower, a double effect lithium bromide chiller and a boiler was considered. In the first step, the first law analysis for all components of this system was made. Then, with known properties in different parts of the system second law and exergy analysis for the components was conducted. It should be mentioned that for system analysis software MATLAB was used.

Keywords: thermodynamic analysis, double effect absorption cycle, cooling tower

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1056

Thermodynamic Analysis of Series and Parallel Flow Water/Lithium Bromide Double Effect Absorption System with Two Condensers

S. Sedeigh[1], H. Talesh Bahrami[2], H. Saffari[3]

[1]Iran University of Science and Technology; saeed_sedigh@ yahoo.com
[2] Iran University of Science and Technology; h_tbahrami@yahoo.com
[3] Iran University of Science and Technology; saffari@iust.ac.ir

ABSTRACT

Many analyses have been conducted on single effect and double effect absorption systems that had just one condenser. In this paper, a series double effect and parallel double effect absorption systems are analyzed. First the systems were studied thermodynamically according to the first and second law of thermodynamics. Then a computer code was developed to simulate the thermodynamic analysis. Finally with this code, the performance of these cycles is simulated and compared. Results showed that the coefficient of performance of the parallel was higher than the series one. And also exergy loss of evaporator, absorber, condenser, low temperature generator and high temperature heat exchanger of parallel was lower than series one. But exergy loss of low temperature heat exchanger of parallel was higher than the series one.
The amount of heat that is exchanged in the high temperature generator and low temperature het exchanger of parallel was less than series one. As a result we can say that the first and second law performance of parallel is higher than series one.

Keywords: absorption chiller, Thermodynamic analysis, Exergy

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1086

Numerical Simulation of Indoor Air Flow and Particle Deposition in the Model Room

N. Abbaspoor[1], M. H. Jafari Marandi[2] , S. Rekab Talayee[3]

[1]Department of Mechanical Engineering, Babol University of Technology,
nima1386@gmail.com
[2]Department of Mechanical Engineering, Arak University of Science &Technology,
eng.jafari.m@gmail.com
[3]Department of Mechanical Engineering, Arak University of Science &Technology,
Sara_re_t@yahoo.com

ABSTRACT

Today, in industry nozzles, venture nozzles and orifices plates widely used to measure the fluid discharge. This means, it provides values to calculate discharge with measuring the pressure above and downstream in pipeline or channel. The fundamental condition of this term in standard to be that is pipeline or channel is fully developed. The principle of this study is surveying of non fully developed convergent flow such as ISA1932 nozzle with convergent ring that these devices what percent errors can create and is which conditions of non fully developed convergent flow can use the terms in standard with errors less that 10%.
Flow meter is an instrument for measuring the flow rate of a fluid flowing in special surface per time unit (s). Nozzles are one of them. In this search, it has been studied, the method for measurement flow rate with using the nozzles. Nozzle is an instrument that determines the flow rate of the fluid flowing in the conduit and causes pressure drop in pipeline or channel. This device consists of divergent section, upstream face that its diameter equals with a pipe's diameter, convergent section.
Set dimensions the situation of digital pressure gauge and requisite equation to determine mass flow rate have been stated on ISO5167, British standard1042.
In this part, it has been considered the sample of calculation on nozzles in different ratio in 4stages. Calculation has been used for $\mathrm{Re} = 5 \times 10^5$. It means the maximum allowable limit for air.Turbulent production in channel line causes upstream pressure drop and thus reduced the pressure and create errors in the calculated mass flow.

Keywords: fully development flow, ISA1932 nozzles, discharge coefficient, convergent turbulence

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1109

Design and optimization of cooling tower for Zarand electrical power plant

M. A. Mehrabian[1], G. A. Sheikhzadeh[2], F. Ahmadi[3]

[1] Professor, Department of Mechanical Engineering, Shahid Bahonar University of Kerman;
ma_mehrabian@yahoo.com
[2] Assistant Professor, Department of Mechanical Engineering, University of Kashan;
sheikhz@kashanu.ac.ir
[3] MSc Student, Department of Mechanical Engineering, University of Kashan;
farzaneh.ahmadi.z@gmail.com

ABSTRACT

The cooling tower is one of the important parts in industrial units, which is used to cool the recirculated water and remove heat during the processes. The wet cooling towers are usually used in hot and dry weather conditions. The Zarand electrical power plant like most domestic power plants takes advantage of wet cooling towers to cool the water during the power generating process. Because of inefficiencies in cooling system construction, the cooling process does not take place properly, especially in high heating loads. The company is thus planning to add a new cooling unit. In this paper, the Merkel method is used to design and optimize the new cooling tower. Taking into account the local weather conditions and variable parameters such as air and water flow rates, tower cross sectional area and height, and the packing model, the tower characteristic curves with respect to mass flow rate were drawn. The variables were then limited and the intersection of characteristic curves with the packing curves were found. The optimized power for the fan is then selected based on the intersection points. This design results in increasing the rate of latent heat.

Keywords: Tunnel Cooling tower; Optimized design; Heat transfer; Packing

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1124

Influences of using the excess air-high velocity burners to warm-up furnaces

A. Adeli, S. Valipour Givi

Sholesanat Co., info@sholehsanat.com

ABSTRACT

In the present research, excess air-high velocity burners are introduced and main advantages of these burners such as high velocity and excess air and the influences of these specifications in the warm-up furnaces are presented.
The studies indicate due to using the excess air-high velocity burners instead of medium velocity burners and displacement of burners in the furnace relative to the exhausting position of combustion products, in addition to fuel consumption, the temperature of furnace and amount of NO_x are reduced.

Keywords: high velocity burner, excess air, warm-up

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1008

Performance Evaluation of U-type Tubes in Boiling Heat Transfer and its Effect on Pressure Drop

M.A. Akhavan-Behabadi*, M. Jamali, R. Khosroshahi, M. Kazemianfar

School of Mechanical Engineering, College of Engineering, University of Tehan
* Corresponding Author, Email: akhavan@ut.ac.ir, Tel.: +98-21-88005677; Fax: +98-21-88013029

ABSTRACT

In the present work, an experimental investigation on flow boiling heat transfer and pressure drop characteristics of R-134a inside a straight tube and the U-type tubes with different bends radiuses is carried out. The experimental set-up which is used in this investigation is a well instrumented vapor compression refrigeration cycle. This set-up consists of a test evaporator which all the experiments are carried out on it. The test section has two straight tubes of 1100 mm length and a 180º return bend. The inner and outer diameters of the tube are 8.25 mm and 9.52 mm, respectively. Refrigerant R-134a flowing inside the test evaporator is heated by an electrical coil heater wrapped around it. The tests are conducted with three different mass velocities of 55, 77, and 99 kg/m^{2}s and also at five different bend ratios (2R/D) of 3.37, 5.16, 7.27, 9.71 and 12.62.

Analysis of the collected data indicates that, both the heat transfer coefficient and pressure drop are increased when U-type tubes are used instead of straight tubes. Similar to straight tubes, in U-type tubes also the heat transfer coefficient and pressure drop increase with the increase of vapor quality and mass velocity. The maximum pressure drop appears at the maximum mass velocity of 99 kg/m^{2}s, vapor quality of almost 0.70, and radius of curvature of 2R/D=3.37. The pressure drop in this situation is almost 11 times more than that of the straight tube. However, the enhancement of heat transfer coefficient reaches to an extreme of 30% compared to the similar straight tube.

Also, the performance evaluation of flow boiling in U-type tubes from the point view of heat transfer enhancement and pressure drop increasing is done. The best performance in all bend ratios happen at low vapor qualities and mass velocity of 99 kg/m^{2}s, while the worst situation happens at low vapor qualities and mass velocity of 55 kg/m^{2}s. Finally, it is concluded that, considering the heat transfer and pressure drop increasing, the straight tube has the best performance. In other word, the use of U-type tubes instead of straight tube is not for the purpose of heat transfer enhancement, but it is used in order to have a compact heat exchanger.

Keywords: U-tube, Boiling heat transfer, Pressure drop

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1025

Effect of Coiled Wire Insert on Heat Transfer and Pressure Drop in Horizontal Evaporator

H. Najafi[1], M.A. Akhavan-Behabadi[2]

[1]School of Mechanical Engineering, College of Engineering, University of Tehran, Iran; hassan_najafi@yahoo.com
[2]School of Mechanical Engineering, College of Engineering, University of Tehran, Iran; akhavan@ut.ac.ir

ABSTRACT

The present study investigated the effect of coiled wire insert on the heat transfer enhancement and pressure drop increasing in an evaporator. For this purpose, Wire coils having different coil pitches and wire diameters were inserted into a horizontal plain copper tube of 7.5 mm inside diameter, 9.54 mm outside diameter, and 1200 mm long. The coil pitches were 5, 8, 10 and 13 mm, and the wire diameters were 0.5, 0.7, 1.0 and 1.5 mm. Refrigerant HFC-134a flowing inside the tube was heated by a flexible electrically coiled heater wrapped around it. The ranges of refrigerant mass fluxes were between 54 and 136 kg/m^{2}s.

Analysis of the collected data indicated that, both of heat transfer coefficient and pressure drop increase by increasing the mass flux and vapor quality in coiled wire inserted tubes similar to the plain tube, and the insertion of coiled wires inside horizontal tubes increases the flow boiling heat transfer coefficient but with a higher penalty due to the increasing of pressure drop, in comparison to that for the plain flow. This enhancement depends upon the test conditions and the geometry of coiled wire. The maximum heat transfer enhancement of the wire coil inserted tubes over the plain tube in the best condition was 98% on a nominal area basis in the high vapor quality region at mass flux of 114 kg/m2.s with the coil pitch of 10 mm and wire diameter of 1.5 mm (the coil with thickest wire) , in the worst condition, it increases the pressure drop up to 1000% relative to a similar plain tube pressure drop rates , This is done in high vapor quality region at mass flux of 85 kg/m2.s with the coiled wire of 1 mm wire diameter and of 5 mm coil pitch (the lowest pitch).

By considering the heat transfer and pressure drop, it has been found that the plain tube has the best performance. Therefore, the coiled wire inserts are recommended only under special conditions and for certain applications by considering the appropriate consistence between heat transfer performance and the rate of pumping power increasing. Using the present collected data, a new correlation was developed for estimation of pressure drop during evaporation inside a horizontal coiled wire inserted tube. This correlation predicts the experimental data of this study within an error band of ± 20%.

Keywords: Coiled wire, Heat transfer, Pressure drop, Evaporator

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1031

Calculation of Mean Temperature in a Room with Floor Heating System Using Neural Network Method

E. Mehrabi[1]

[2] Islamic Azad University of Kerman; mehrabi.ehsan@gmail.com

ABSTRACT

In this study a 3 dimensional room with floor heating system stimulated using Fluent Software. Considering, the inner air of the room incompressibility, Boussinesq model, reformed model of $k-\varepsilon$, radioactive model DO and simple algorithm, the stimulation of fluid flow and temperature distribution performed. After change of effective various parameters on the floor heating system in the model (room) such as pipe depth, water temperature, internal emissivity, air absorption Coefficient, window sizes, object size(object was in room) , distance between object and floor, are calculated mean temperature for every case and they are used as educational data for neural network. The neural network produced by two layers: hidden layer, with 20 neural neurons and arbitrary function, outer layer, with 1 neural neuron and liner Function, in the Matlab software with 6 inputs and 1 output.

Comparing the achieved results with the corresponding results by numerical methods which done by the new data showed that the difference between the results was very insignificant so the neural network may estimate mean temperature for floor heating system of room carefully.

Keywords: Floor heating, Neural network

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1064

The Effect of Airchange Rate on Energy Consumption of Buildings

F. Nasrollahi

Berlin University of Technology, nasrollahi@sustainbuilder.com

ABSTRACT

This paper deals with the effect of airchange rate on energy consumption of office and residential buildings, for which dynamic energy simulation is used.
The results of the simulation and analysis show that, increasing the airchange rate will strongly increase the heating energy consumption of both office and residential buildings. The influence of airchange rate on cooling energy demand is very less than that of heating energy demand. The increasing of the air change rate of buildings up to a specific rate will decrease the cooling energy demand. Increasing the airchange rate over this rate will increase the cooling energy demand. The total energy consumption of office and residential buildings will strongly increase with increasing the airchange rate too.
Increasing of heating and also total energy consumption of buildings with increasing of airchange rate is approximately a linear effect, while the effect of airchange rate on cooling energy consumption of buildings is not a linear effect. The results show also that, office and residential buildings have very similar behavior regarding the amount of airchange rate. Therefore, for reducing the energy consumption of buildings, the airchange rate of buildings must be minimized. But in airtight buildings the required fresh air is not supplied through infiltration. Therefore, to ensure the indoor air quality in airtight buildings a mechanic ventilation system is required.
In buildings with a mechanical ventilation system, for example in winter, a specific amount of warm indoor air must be exchanged with an equal amount of cold outdoor air to supply the required fresh air. Therefore, a significant amount of energy must be consumed to increase the temperature of cold outdoor air to indoor air, during winter. It is, however, possible to use the heat from the exhaust air for heating the incoming air with application of an air-to-air heat exchanger. There are currently heat exchangers available that achieve up to 95% recovery rate for heat losses.
With the use of a ventilation system with a heat exchanger in airtight buildings the following objectives are achieved:
Heat recovery from exhaust air and thus energy saving
Supplying sufficient fresh air for occupants to living spaces
Removal of air pollution from spaces with more pollution
Controlling the incoming air and elimination of dust, pollution, pollen, odours, etc. from entering the building

Keywords: Airchange, Energy consumption

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1066

The effect of changing the time in the first six-month period of the year on the heat load of a building

D. Malek Mohammadi[1], D. Jalali Vahid[2]

[1]Islamic Azad University of Brojen; d.malek.m@gmail.com
[2] Industrial University of Sahand Tabriz; davoudjalali@sut.ac.ir

ABSTRACT

Daylight saving is customary in most countries of the world. Economizing of energy consumption is the purpose of this act. In this study, the effect of this action on the heat losses (cooling load) of an administrative building in three cities of Iran (Tehran, Tabriz and Abadan) has been investigated and the peak of cooling load has been calculated for a sample building in July (Mordad) for daylight saving mode and normal mode. Moreover, the effect of direction of building for 4 status on the cooling load has been investigated.

Keywords: heat load, energy consumption

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1071

Surveying and Comparison of the Code No. 19 of the National Regulations on Buildings with the Laws, Regulations and Standards in the Leading Countries in the Field of Energy Conservation in Buildings and Representing Procedure

F. Rahimi Mougouei

Student in Master of Industrial Engineering, Amirkabir University of Technology;
Rahimi.Farzad@aut.ac.ir

ABSTRACT

About 3 decades ago, the countries that consumed energy in large amounts decided to plan and implement various programs aimed at decreasing energy losses and optimization policies in fuel consumption. In this case, not only did they economize energy costs, but also the environmental destruction was prevented effectively.
To improve the energy efficiency of buildings via compliance to regulation in Iran, Code No. 19 was devised in 1991. The code lacks high level aims and objectives, addressing the characteristics of Iranian buildings. As a consequence, the code has been revised and is not completely implemented in practice, and still remains inefficient. As with any energy coding system, this code has to identify the right balance between the different energy variables for the Iranian climate and way of life.
This paper focuses on surveying and comparing the regulated standards and codes in the field of economizing energy consumption in building and the code for energy conservation in buildings in Iran (Code No. 19).
In order to assist improvements to high level objectives of Code 19, this code is compared with a trail country building codes to understand how these have adapted international standards to national features. In this case, we begin with introducing Code No. 19 and then comparing it with various indexes and details about building energy code in Australia, USA, Britain, Canada, New Zealand and Hong Kong. The output of this paper can be used in recognizing regulated codes in these countries, and complementing the code framework of energy conservation in buildings in Iran.
In order to test the appropriateness of Code 19, eleven indexes introduced and the results are compared. The results demonstrate that Code 19 is efficient in some indexes, but it needs improvements in the areas of ventilation, gains from internal and solar sources. The paper concludes by offering suggestions for improving the code.

Keywords: Building Energy Code, Energy Efficiency, Iranian Building Code, Code 19, Energy Management

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1098

Energy Consumption of Floor Heating System in Buildings

V. Golkarfard [1], M. Salmanzadeh[2], P. Talebizadeh[3]

[1]M.Sc. Student , Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; vahid.golkarfard@gmail.com
[2] Assistant Professor, Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; msalmanz@clarkson.edu
[3]M. sc. Student , Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; talebizadeh.pouyan@gmail.com

ABSTRACT

High energy demand in recent years has caused the countries to change their energy consumption policies. Buildings are one of the important energy consumers in the world. High temperature gradients in heating systems of buildings can increase the heat loss of the envelopes during the cold season and consequently, increase the energy consumption. Floor heating systems generate lower temperature gradients in compare with other convective heating systems. In this work, the CFD simulation has been done to investigate the energy loss in rooms with floor and radiant heating systems. The required energy to achieve the thermal comfort in a room by floor and radiant heating systems has been calculated by modeling the velocity and temperature fields and compared with each other and with the data calculated by a conventional method of heating load calculation. The outdoor temperature is considered for the Kerman city climate in winter and the ASHRAE standard is used for the indoor air design temperature. The calculated results showed that the total heating load for radiant heating system is 58 percent higher than the floor heating system for the tested room.

Keywords: Floor heating, Radiant heating, Energy consumption

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1103

Evaluation and Optimization of Refrigeration Cycle With Economizer

F. Eskandari-Manjili[1], M. Noori[1]

Assistant Professor, Bo-Ali-Sina University; eskandari_m2@yahoo.com
B.Sc. Student, B0-Ali-Sina University; milad85me@gmail.com

ABSTRACT

Nowadays, the issue of energy supply and the destructive effects of high energy consumption gets more serious, improving the efficiency of industrial equipments, is one of the main goals of researchers and designers. The refrigeration industry is not excluded from this issue & efficiency improvement with reduced in power consumption is one of the main objectives of this industry. About 15 percent of the world electricity industry is consumed in a refrigeration system including cooling and air conditioning. This study has tried to study the issue of economizer usage in the compression refrigeration cycle to reduce irreversibility and improving the efficiency. Therefore, a thermodynamic model based on the principles of mass, energy and exergy balances is developed for R22 and R134a. Coefficient of performance changes and irreversible losses changes have been evaluated and optimized pressure of economizer has been calculated in two ways of maximizing the coefficient of performance and minimizing irreversibility. One of the interesting results of the modeling is that the optimum inter stage pressure is the same according to the first and second laws of thermodynamics and using the economizer will be more effective improving the efficiency of the refrigeration cycle with refrigerant R134a. The optimum reduced inter stage pressure of the cycle for refrigerant R22 is in the range of 0.32 to 0.35 and for the refrigerant R134a is in the range of 0.30 to 0.34. The effect of condenser temperature on the cycle performance & its optimum pressure has been studied. The results indicate that the more temperature difference between condenser and evaporator increases, the more effective is the usage of economizer on efficiency improvement of the simple refrigeration cycle, and it is observable that as the condenser temperature increases, optimum reduced inters stage pressure decreases for both R22 and R134a. The analysis is performed on each of the system components to determine their individual contribution to the overall system irreversible losses. It is found that the economizer plays the main role in the reduction of irreversible losses.

Keywords: Refrigeration Cycle, Economizer, Energy consumption

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1105

Optimum Design of Solar Buildings in Various Climate Conditions of Iran Using Genetic Algorithm

H. Tarmahi[1], S. Edalati[2], H. Mansouri[3]

[1]M.Sc., Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; tarmahi.h@gmail.com
[2] M. sc. Student, Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; adalati.s@gmail.com
[3] Professor, Mechanical Engineering Department, Shahid Bahonar university of Kerman, Iran; mansouri@alum.mit.edu

ABSTRACT

Direct gain of solar energy is the simplest way of building heating. In this method, building itself functions as a collector and storage unit and as the enclosure in which people live, work, and are protected from an often harsh exterior environment. South facing window is an ideal way to gain maximum of solar energy in winter and minimum in summer.
In this paper, at first, the incident solar radiation on a sample building in 3 different cities with different climates in winter is calculated. Then the heating load of this standard building which is insulated within national Iranian standard of building design (19-th section) is calculated. By using these results, the best building aspect ratio and best south- facing window area is calculated. The auxiliary energy for one to four stories building in this climate and the saved energy by adding these stories is calculated and also this is calculated for a single building two to four side by side building. Next the best patio angle by considering the maximum solar energy gain in winter and minimum in summer is calculated and the best patio area for this building is designed. At the end, the best window and patio area for common building dimensions is designed and the results are optimized by genetic algorithm.

Keywords: passive solar heating, direct gain systems, optimum window area

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1125

Simulation of Energy Storage System with Phase Change Material (PCM)

M. Rostamizadeh, M. Khanlarkhani, M. Sadreameli

M.Sc. Student, Chemical Engineering Department, Tarbiat Modares University;
m_Rostamizadeh@yahoo.com
M.Sc. Student, Chemical Engineering Department, Tarbiat Modares University;
mkhanlarkhani@yahoo.com
Professor, Chemical Engineering Department, Tarbiat Modares University;
sadramel@modares.ac.ir

ABSTRACT

Thermal energy storage plays an important role in a wide variety of industrial, commercial and residential applications. Phase change material (PCM) is used in these systems in order to store heat. Latent heat storage in a PCM is very attractive, because of its high energy storage density and isothermal behavior during the phase change process. Increasing of building energy storage capacity can be achieved with decreasing the indoor air temperature variations so that the indoor temperature goes near comforting temperature for a long time which results in more comfort for human. In this paper, a mathematical model was developed based on an enthalpy formulation and the effect of PCM thickness on temperature distribution in the PCM and melting fraction was investigated. Results show that melting time change with the amount of PCM linearly and lower thickness is suitable for a better PCM performance.

Keywords: Energy storage, PCM, Simulation

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1005

Operation analysis of an energy storage system in order to supply the heating load for residential buildings

J. Pirkandi[1], A. Keshavarz [2], M. Ghasemi [3]

[1,2,3] Mechanical Engineering Dept., Khaje Nasir. T. University of Technology, Tehran, IRAN

ABSTRACT

Daily life of the world people depends on the energy generation and use and hence, its supply and demand is continually increasing in the human societies. Regarding to the high capacity and efficiency of phase change materials in energy storage systems, use of them in these systems is interesting for many advance countries in the world. The aim of this article is operation analysis of an energy storage system in order to supply the heating load for residential buildings.
The system is consisted of two concentric cylinders which its working fluid flows through the outer cylinder and the inner pipe has been filled with phase change material. The system works periodically and the governing equations of heat transfer for the working fluid and the phase change material are solved numerically. The differential equations of flow and heat transfer, with initial and boundary conditions have been discreted by control volume approach and then, solved by using an iterative procedure. Finally, temperature distribution in the energy storage system is obtained for charge and discharge period of time and the results of two cases are compared together.

Keywords: Thermal storage system, phase change material, energy optimization , solar energy

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1038

The Experimental Study of Outdoor Temperature Effect on Performance of gas engine heat pump

S. Sanaye[1], M. Chahartaghi[2]

[1] Associate professor, Department of Mechanical Engineering, Iran University of Science and Technology
[2] Assistant professor, Department of Mechanical Engineering, Shahrood University of Technology; chahartaghi@iust.ac.ir

ABSTRACT

In order to supply energy needs, the use of various cooling and heating systems are common in different regions. Gas engine heat pump is one of the most significant equipment in heating, cooling and air conditioning section in recent years Due to. To reliable resources of natural gas in Iran, Using gas engine heat pumps (GEHP) can be considered as one of the useful devices and therefore study of their performance in different conditions is important.
In this paper, the effect of outdoor temperature on fuel consumption and coefficient of performance (COP) of a GEHP were studied experimentally in both cooling and heating modes of operation.
Results showed that the fuel consumption of GEHP increased with increasing the outdoor temperature and the COP of GEHP decreased with increasing the outdoor temperature in cooling mode. In heating mode the fuel consumption of GEHP decreased and the COP of GEHP increased with increasing the outdoor temperature.

Keywords: Gas engine heat pump, outdoor temperature, fuel consumption, coefficient of performance

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1040

Numerical simulation of heat transfer from a double window and investigation of optimum dimensions and utilized gas type

G. Sheikhzadeh[1], A. Fatahi[2], A. Khoram[3]

[1] Faculty of Engineering, University of Kashan; sheikhz@kashanu.ac.ir
[2] Faculty of Engineering, University of Kashan; abolfazlfa2008@yahoo.com
[3] Faculty of Engineering, University of Kashan; khoram.ali@gmail.com

ABSTRACT

In this paper, we simulate a double window using FLUENT and Gambit software. Effects of some parameters such as the air space gap, type of utilized gas in the air space, the temperature difference between two sides of window and height of the window, are numerically investigated to predict the efficiency of a double window in winter. It is seen that Krypton gas causes less heat loss than Air and Argon. However, the amount of the decrease of heat loss for Air is higher than that of Krypton and Argon. When gas space width is equal to 10mm, increasing the temperature difference from 10°C to 40°C leads to increasing heat loss to 4 times for Air and 5 times for Krypton. When gas space width is small, the conduction heat transfer is predominant and the isotherms are similar to the parallel lines. But, with increasing gas space width, dominant heat transfer mechanism is the convection and flow pattern tends to a circulating cell. In the top and the bottom zone of the gas space, the heat transfer is more than that for the other zones. Also, increasing gas space width up to 20mm has a negligible effect on the heat loss. Increasing height of the window is also increased heat loss. When height of the window is equal to 500mm, heat loss for Air is increased up to 1.5 times of that for Krypton. The increase is also raised for higher height. The obtained results show that energy loss due to increasing the temperature difference is more intense for the Air, but, amount of the increase is more for Krypton.

Keywords: Numerical simulation, Heat transfer, Fluent, Double window

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1052

Two Stage Refrigeration Systems Revisited

F. Jafarkazemi[1], M. Golkhani[2]

[1]Islamic Azad University, South Tehran Branch; fj_kazemi@azad.ac.ir
[2] Islamic Azad University, South Tehran Branch; mahdi.golkhani@gmail.com

ABSTRACT

It is the aim of this paper to have a review on possible two stage refrigeration systems and investigate the correctness of some traditional rule of thumb calculations in these systems. Further to a review of the basics of two stage refrigeration methods, different alternatives are compared. Effect of inter-stage pressure and volume ratio are calculated and their effects on system performance are evaluated. The calculation results of the paper shows that the optimum displacement ratio increases with increasing the temperature lift.

Keywords: Refrigeration, Two stage, Volume ratio

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1053

Thermal-Economical Multi-Objective Optimization of Desiccant Cooling System Using Genetic Algorithm

S. Sanaye[1], S. Sedghi Ghadikolaee[2]

[1]Associate Professor, Energy Systems Improvement Laboratory, Department of Mechanical Engineering, Iran University of Science and Technology; Sepehr@iust.ac.ir
[2]Msc Student, Energy Systems Improvement Laboratory, Department of Mechanical Engineering, Iran University of Science and Technology; Sh.sedghii@gmail.com

ABSTRACT

In order to remove the latent heat, the traditional refrigerant vapor compression system (VCS) or the not yet traditional vapor sorption system (VSS), cools the process air down below its dew point in order to condense out water vapor contained therein. The dehumidified air is then reheated to meet the required indoor temperature conditions. If the latent load is handled by another means than by this deep cooling, two components of the burden on the conditioner, brought about by the presence of latent load, will be avoided. Those are, namely, (1) the energy required to bring the air from the supply temperature down to the temperature of condensation of water vapor contained in the process air (below the dew point of the air), and (2) the energy needed to reheat the air from that temperature up to the supply air temperature. When the sensible heat ratio (SHR) of the conditioned space is low, the sum of these two components increases dramatically. Furthermore, the VCS is actuated by electricity, the generation of which involves most often the utilization of fossil fueled power plant with the consequent emissions of carbon dioxide (CO_2) into the atmosphere. The desiccant cooling can be either a perfective supplement to the traditional vapor compression air conditioning technology to attenuate the effects of its drawbacks, or an alternative to it for assuring more accessible, economical, and cleaner air conditioning. Still more importantly, when powered by free energy sources such as solar energy, and waste heat, it can significantly reduce the operating costs and increase considerably the accessibility to the air conditioning for the populations in remote areas, especially in developing countries. In this paper, thermal-economical multi-objective optimization of the hybrid cooling system is considered. Hybrid cooling system is a combination of the vapor compression cooling system and desiccant system in which, first, the humidity is absorbed by a desiccant and then the air temperature decreases up to the desired temperature by a vapor compression cooling system. Coefficient of performance and annual costs, are two objective functions that through optimized two variables have been examined. For better comparison of traditional cooling systems and hybrid systems, Ahvaz city has been studied. Calculations show that using this system, total costs will be reduced to the amount of 0.068 dollars and power consumption rate of 0.822 KW per ton of refrigeration in one year than vapor compression cooling.

Keywords: Hybrid desiccant-vapor compression system; Total annual cost; Multi-objective optimization;

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1121

Analysis of a Vapour Ejector Refrigeration System

E. Afshari[1], Y. Mollayi Barzi[2]

[1] Department of Mechanical Engineering, Faculty of Engineering, University of Isfahan, P.O.Box 81746-73441, Isfahan, Iran
[2] Islamic Azad University, Kashan Branch, Kashan, Iran

ABSTRACT

This paper describes the performance of vapour ejector refrigeration system driven by solar energy. A computer code based on the ejector theory has been developed to simulate the operation of the system and the related thermodynamics. Then by calculating the produced solar energy, the required energy in the refrigeration system of a 10kW and solar refrigeration plant using the system is determined.

Keywords: Ejector, Refrigeration, Solar energy, COP

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1130

An Investigation of Integration Strategies for Oxide Fuel Cells (SOFC) in HVAC Systems

F. Ommi [1], S. Haghshenas [2]

[1] Associate Professor of Aerospace Engineering, Tarbiat Modares university,
fommi@modares.ac.ir
[2] Graduate Student of Aerospace Engineering, Tarbiat Modares university,
s.haghshenas@modares.ac.ir

ABSTRACT

Since oil and gas demand is increasing and we have serious problems such as exhausting of fossil resources and global warming, we need efficient energy systems and new conversion processes. Fuel cells and hybrid systems are advanced thermodynamic systems which have excellent capabilities in achieving high energy and reduced environmental loads. Among several fuel cell types available currently, SOFC was chosen for this study because of its ability to be utilized as a cogeneration system. In addition, the operating temperature of SOFC fuel cells is high enough to provide sufficient heat for the fuel reforming process. This allows SOFCs to reform fuel internally which removes the need for expensive external reformers used to produce hydrogen in SOFCs. The high operating temperature also enables SOFCs to use the residual heat created as a byproduct of the power generation process for space and water heating, allowing SOFC to be used as a cogeneration system. This paper reviews different concepts/strategies for SOFC-based integration systems, which are opportune transformational energy-related technologies.

Keywords: Solid oxide fuel cell, Integration strategy, Hybrid system

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1017

Experimental Investigation on Heat Transfer and Pressure Drop of Nanodiamond-Engine oil Nanofluid Laminar Flow inside a Horizontal Tube under Constant Heat Flux

E. Rasuli, M.A. Akhavan-Behabadi*, M. Ghazvini, M. Raisee

School of Mechanical Engineering, College of Engineering, University of Tehan
* Corresponding Author, Email: akhavan@ut.ac.ir, Tel.: +98-21-88005677; Fax: +98-21-88013029

ABSTRACT

Very interesting results about enormous enhancement of heat transfer coefficients by dispersing nanoparticles, encouraged researches for doing more studies in this field. So far, the majority of researches have focused on thermal enhancing characteristics of nanoparticles dispersed in common heat transfer fluids like water, ethylene glycol and oil. Consequently, the researchers rarely have been considered the side effects of using nanoparticles like causing more pressure drop.
In the present work, the heat transfer enhancement and the pressure drop increasing of engine oil-nanodiamond nanofluid is experimentally investigated. A plain tube with the internal diameter of 6 mm was used as the test section. The nanofluid flowing inside the test section was heated by an electrical coil heater wrapped around it to produce a constant heat flux. The data are collected for laminar flow. 20W50 engine oil and four nanofluids with 0.2, 0.5, 1 and 2 percent weight fractions were utilized as the working fluids. The experimental data has been collected under 3 to 18.5 (kW/m2) heat fluxes and Reynolds number up to 100.
Results showed that thermal conductivity was significantly affected by adding nanodiamond and increased by about 35% at most. The highest heat transfer enhancement was about 64%. Moreover, investigations clarified that pressure drop increased while using nanofluids in comparison to base fluid (engine oil) and this increase; mount up in higher particle concentrations. Also the pressure drop decreased in constant Reynolds number with an increase in heat flux.

Keywords: Heat transfer, Pressure drop, Nanofluid, Horizontal tube, Experimental study

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1027

Experimental Study on Pressure Drop of CuO-Base Oil Nanofluid Laminar Flow in a Horizontal Wire Coils Inserts Tubes under Constant Heat Flux

M. Saeedinia[1], M.A. Akhavan-Behabadi[2]

University of Tehran, College of Engineering, School of Mechanical Engineering,
[1]mohamad.s@gmail.com, [2]akhavan@ut.ac.ir

ABSTRACT

In this work, an experimental study is carried out to investigate on pressure drop characteristics of CuO-Base oil nanofluid laminar flow in a plain tube with different wire coil inserts. Particle weight fractions of nanofluid range from 0 to 2%. The test section is a horizontal copper tube of 14 mm inner diameter and 1.2 m longitude in which wire coils inserts with different coil pitches and wire thicknesses are used. The test section is heated by an electrical heating coil wrapped around it to produce a constant heat flux conditions.

The effect of different parameters such as mass velocity, wire diameter, coil pitch, nanofluid particles concentration and heat flux on the pressure drop is studied. For the tube with a given wire coil insert and a specific fluid, the results show that by increasing of mass velocity, the pressure drop is increased as it was expected. Moreover, applying coil wires with lower pitches and higher thicknesses would make more increases in the pressure drop at a given mass velocity. Also, the effect of the increase in nanoparticles concentration on the pressure drop has been investigated. Observations show that by using nanofluid instead of base fluid, the pressure drop increases and this enhancement grows at higher nanoparticles concentration.

Keywords: Pressure drop, Nanofluid, Wire coils inserts, Cuo, Laminar flow

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1059

Procedure for Design and Implementation of TAB (Testing, Adjusting and Balancing) Manual, HVAC Ducting System in Sample Project

M. khorasani

Master of science in mechanical engineering, Iran atomic energy organization;
Mehrzad_khorasani@yahoo.com

ABSTRACT

TAB is defined as testing, adjusting and balancing of an air duct system, in a way that, the amount of inlet and outlet air in closure function in maximum with 10% error consistent with a design amount. TAB design and implementation are employed in order to be utilized in air and water HVAC system. Amount of used air current has some differences with existing amount in designed plans, which means that designing and balancing in an air rotary current is not implemented automatically by the system.

The only possible solution to reach design process in all system branches is to make balance in the assembly, measure and perform accurate adjustment based on designed condition. TAB guarantees accurate performance of the system (consistent system with design) and is an approach to facilitate the maintenance and continue correct performance of installed HVAC systems. To achieve this purpose make an internal desired condition, decrease maximum costs of energy consumption, prevent functional problems of the system, whole HVAC system should be controllable.

In order to make each HVAC system controllable, three fundamental terms should be considered:

1. Design current should be existed in all terminals and branches during maximum load.
2. Pressure differences in both sides of dampers and control equipment should not be more than design.
3. Current should be compatible in all system branches.

Balance of the HVAC ducting system is possible in tow way:

A: Trial and error method

B: using of Federspiel and Walton models and current equation of the low pressure system and their solution.

Considering the complication of air current equations this, method can be just implemented by software coding.

It should be mentioned that in, this method (performance curve)all parts of the system should be determined for coding. This method is applicable in projects with high ducting volume tunnels; Some of the world big companies which are active in this field provide software and use them.

In this project first method has been used. Work was started by studying and coding plans then flow and pressure of air in canals closures, and dampers (CFM) in designed TAB form are entered. If room has also a ventilation fan, is switched on and CFM amount of the main canal should be controlled and adjusted by lateral Pitot measurement, according to plan.

After studying plans all, parts are controlled in the project site, and it is made sure that manual and also automatic dampers are open. Inlet and outlet status of room is investigated, and it needs to be open then they will be totally open. It fresh air fan is out of the conditioning system, it should be switched on. The amount of fresh air on main branch of fresh air riser adjusted by lateral measurement of air conditioning if, the current was more than 10% of the design amount outlet, entry of air conditioning is adjusted.

At the end, all closures and dampers of the system should be investigated carefully one by one and if, if it is required, probable problems should be removed, also in order to remove the leak, all system inlets and outlets should be inspected.

Keywords: TAB system design implementation TAB, forms, correct measurement equipment data record, closure, and dampers adjustment.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1092

Assessment and investigation on existing potentials in boiler's blow down systems

M. R. Aligoodarz[1], S. Keshmiri[2], N. Nasiri[3]

[1]Associate Professor, Rajaei University , School of Mechanical Engineering; maligoodarz@srttu.edu
[2]Fluid Mechanical Expert, Khosh Souz Engineering Company; sydream74@gmail.com
[3] Mechanical Expert, Tamin Engineering Company; navid6850@gmail.com

ABSTRACT

Important role of industrial, commercial, domestic and power plant boilers in heat production is obvious. Blow down is one of the essential boiler maintenance activities, and it has a considerable potential of saving energy. Outlet water of boiler during blow down as well as carrying lots of energy outside the boiler, the cost of chemical additives, filtration and demineralization and boiler water will be wasted. There are two solutions for minimizing these costs. The first one is "blow down reduction" or optimization (in fact preventing the unnecessary water outlet), secondly re-using the water energy by using regeneration devises.
In this paper, the first solution will be discussed and a model is presented for blow down determination, blow down reduction method, specifying the potential for blow down, and a practical solution for using these potentials.

Keyword: blow down, boiler maintenance, water exiting.

Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning
June 1-3, 2010, Olympic Hotel, Tehran, Iran
ICHVAC2-1094

Assessment on Hydronic Systems with Variable Speed Distributed Pumping System

M. R. Aligoodarz[1], S. Keshmiri[2], N. Nasiri[3]

[1]Associate Professor, Rajaei University , School of Mechanical Engineering; maligoodarz@srttu.edu
[2]Fluid Mechanical Expert, Khosh Souz Engineering Company; sydream74@gmail.com
[3] Mechanical Expert, Tamin Engineering Company; navid6850@gmail.com

ABSTRACT

In a Hydronic system, any method that reduces the design temperature difference consequently increases the flow rate, will cause increasing energy consumption. Some of these methods are using of a by-pass tube, 3-way or 2-way control valve with by-pass tube. Although in most of the Hydronic systems at least one of the above-mentioned energy consumed method is used, but these systems have an advantage such as stabilizing the design temperature difference in part-load condition and consequently, flow rate reduction and pumping cost reduction compare with the constant flow system.
In a constant flow system, there is no consideration for consistency between consumed energy and instantaneous load. However, in Hydronic system with variable speed distributed pump, with separating, producing and distributing circuit and using variable speed drive pumps in a distribution circuit, it could be possible to change the flow rate with load changes and with a flow reduction and constant temperature difference for each circuit, energy consumption cost and pumping cost will be reduced.
Secondly, due to not having same time peak loads, it could be possible to select the pump for production and chiller circuit according to flow for total pick load and with reducing the head of the route with maximum pressure lost from producing pump, the total power consumption for pump will be reduced.
Thirdly, for parallel chiller arrangement, with turning off one chiller in part-load its pump in an initial loop will be turned off, and consequently, energy consumption will be reduced with instantaneous load.

Key-word: Hydronic system, pumping cost, variable speed distributed pumping system