NIET Journal of Engineering & Technology

Volume 10, Issue 2, 2019 ISSN: 2229-5828 (Print)

CFD Analysis of Fluid Flow in a Heat Exchanger Using ANSYS Software

Himanshu Dwivedi [1], Himanshu Goswami [1], Donald Singh, Somit Pratap Singh [1]

[1]Department of Mechanical Engineering, Noida Institute of Engineering and Technology, Greater, Noida 201306, India.

Abstract : ***The heat exchanger is necessary for heat transport and energy conservation. Because of their extensive design possibilities, ease of building, and cheap manufacturing maintenance costs, crossflow and counter-flow heat exchangers are widely used in the petroleum, petrochemical, air conditioning, food storage, and other industries. Shell and tube heat exchangers are frequently used in businesses as chiller plants to transfer waste heat from injection moulding machines to cooling water, hence increasing injection moulding machine efficiency. The heat exchanger capacity determines how waste heat from an injection moulding process is transported to cooling water. Heat exchanger optimization discovers the best heat exchanger parameter combination to maximise heat exchanger capacity. The prefix parameter (tube diameter) is employed as an input variable, while the most significant temperature difference between the shell and tube heat exchanger is used as an output parameter.***

Keywords: CFD; Heat Exchanger, Fluid Flow, ANSYS, Heat Transfer

1. Introduction

A heat exchanger is a device that transfers heat from one source to another in an effective manner. Media may either be separated by a solid wall and never interact, or they can directly interact. Space heating, refrigeration, air conditioning, power plants, petrochemical complexes, and petroleum refineries are just a few of the applications.

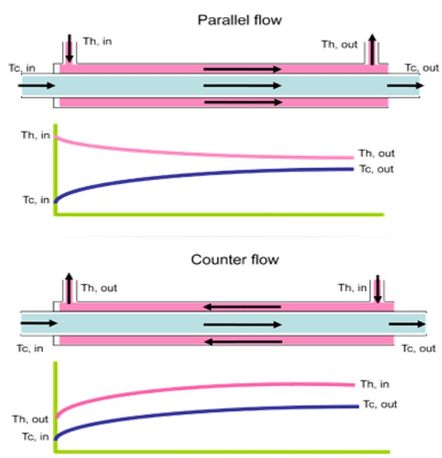


Fig1: Parallel flow & counter flow

Heat exchangers are classified according to their flow arrangement. The fluids flow perpendicularly across the exchanger in a crossflow heat exchanger. The fluids enter the heat exchanger from opposite ends in counter-current heat exchangers.

The heat exchanger is designed to increase the surface area of the wall between the two fluids while lowering fluid resistance across the exchanger for maximum efficiency. Fins or corrugations in one or both directions,

**Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.**

which increase surface area and may channel fluid or produce turbulence, can also reduce the effectiveness of the heat exchange

The effect of varied baffle inclination angles on fluid flow and heat transfer characteristics in a shell-and-tube heat exchanger was investigated in this study. Even if the driving temperature fluctuates throughout the heat transfer surface, a suitable mean temperature can be established. This is the "log mean temperature difference" in the most basic system (LMTD). Heat exchanger performance is measured using changes in mass flow rate and baffle inclination angle for a 36 percent baffle reduction. ANSYS CFX 12.1, a non-commercial CFD simulation programme, is used to examine the flow and temperature fields inside the shell [1].

A computational model of a simple fin plate heat exchanger was built. Plain fin performance was evaluated using CFD by changing different fin features and finding Colburn 'j' and fanning friction 'f' variables. Re, fin height h, fin thickness t, fin spacing s, and friction factor f have all been calculated [2].

In basic geometries, nanofluids have attracted interest as viable heat transfer fluids. This research looked at a three-channel corrugated PHE with a width of 127 mm, a length of 56 mm, and a channel thickness of 2 mm. The heated fluid circulates in the main channel, while the cold fluid circulates in the side channels. The simulations were carried out using commercially available CFD software (Ansys fluent). Higher fluid viscosity and convoluted flow regimes of PHEs cause this decrease in heat transmission [3].

The purpose of this paper is to provide a comprehensive overview of the literature on the use of CFD in the design of solar air heaters. CFD is a simulation approach that employs contemporary computers and applied mathematics to model fluid flow conditions in order to predict heat, mass, and momentum transfer while also optimising design. To model fluid flow through a common solar air heater, the current CFD code ANSYS FLUENT v12.1 is utilised. Five different turbulence models are compared to see how they affect the quality of the results. The Renormalization-group k– model appears to produce the best results for two-dimensional flow using conventional solar air heaters [4].

The influence of rib spacing on the average Nusselt number and friction factor is investigated in an intentionally roughened solar air heater (duct aspect ratio, AR = 5:1). The commercial product ANSYS FLUENT v12.1 is used to generate numerical solutions. With rib pitch-to-rib-height (P/e) ratios of 7.14, 10.71, 14.29, and 17.86, the hydraulic diameter to rib height ratio (e/D) is 0.042. Each rib spacing is evaluated at six Reynolds numbers ranging from 3800 to 18000 [5].

In modern businesses, shell and tube heat exchangers are frequently utilised as chiller plants. They collect waste heat from injection moulding machines and transfer it to water to cool them down. The heat exchange capacity of a heat exchanger determines how waste heat is transported to cool water. Companies nowadays are grappling with the issue of expanding heat exchange plant capacity. According to the issue statement, the goal of the work is to improve the efficiency of the shell and tubes. Geometric characteristics and operational considerations such as mass flow rate, cooling water entrance and exit temperatures, and so on influence the efficiency of heat exchangers.

2.Modeling of Shell and Tube Heat Exchanger

Modeling

After conducting the simple calculations, the modeling was completed using SOLIDWORKS 2016, and the analysis was completed using ANSYS 18.1.

Solid Works

SolidWorks is a Microsoft Windows-based computer programmer for solid modelling, computer-aided design (CAD), and computer-aided engineering (CAE). SolidWorks is compatible with Mac OS X, although Dassault Systems did not provide the programmer

Feature of solid works

When users tell the system to "mate," "insert," or "align" the components, the system will build them. The components "understand" how they're linked, and if one changes its shape or position, the others will follow suit. Solid Works is cross-platform compatible, having the same look and feel on all major UNIX and Windows NT systems. Parts may be built immediately in the assembly and defined by other components, so if those components move or change size, the part will automatically update.

Table. 1. Design and operating specification of exchanger

S. No.	Component	Value

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

NIET Journal of Engineering & Technology

Volume 10, Issue 2, 2019 ISSN: 2229-5828 (Print)

1	Shell inner diameter	250mm
2	Shell outer diameter	253mm
3	Length	1000mm
4	Tube inner diameter	22mm
5	Tube outer diameter	24mm
6	Tube length	1010mm
7	Pitch length	29mm
8	Total number of tubes	52
9	Inlet for shell side = water, mass flow rate and, temperature	0.065 kg/sec, 300K
10	Inlet for tube side = water, mass flow rate	0.320 kg/sec,
11	Inlet temperature and outlet temperature	320K, 305K
12	Design pressure	$19 kg/cm^2$
13	Operating pressure	$15.6 kg/cm^2$

Tube material: copper; Shell material: aluminum

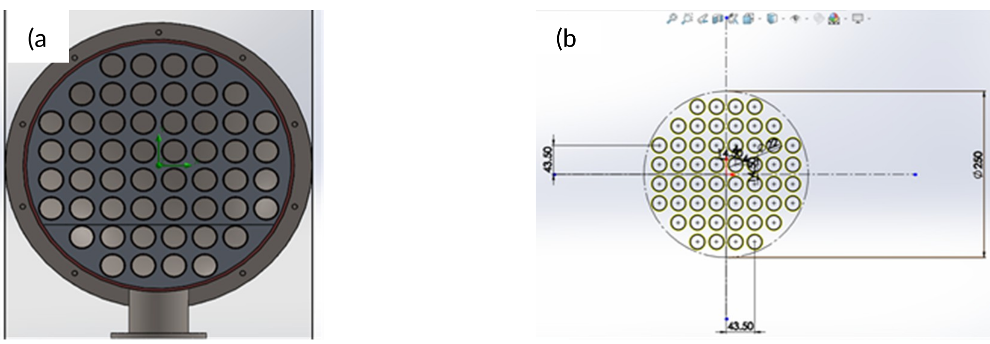


Fig. 2. (a): Cross-sectional view of the heat exchanger, (b) cavity model of the shell & tube (HE).

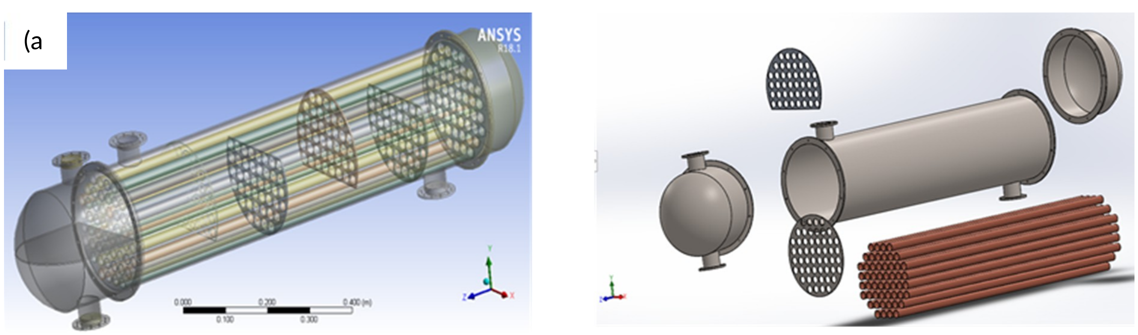


Fig. 3. (a) transparent view of (HE), (b) assembly model of the (HE)

3. CFD Analysis of shell and tube type heat exchanger

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

NIET Journal of Engineering & Technology

Volume 10, Issue 2, 2019 ISSN: 2229-5828 (Print)

Computer-aided design (CAD) can identify flaws in current components and lead to a more effective design. The initial stage is to locate the area of interest, after which the geometry of the region of interest is defined. After importing the mesh into the pre-processor, further elements such as the boundary condition and fluid characteristics are established.

***Geometry*:** The first stage in a numerical flow simulation is to specify the impeller's shape. The assumption that the flow is axis symmetric is the most important step in researching isolated impellers. Any modelling software may be used to create models, which can subsequently be transferred to another simulation software.

Meshing: The act of separating a modelled region into a number of smaller control volumes is known as mesh generation. The variables in each control volume are then calculated using numerical methods. These approaches approximate the conservation rules of mass, momentum, and energy. Meshing is a step in the engineering simulation process that breaks down complex shapes into fundamental components. Meshing has an impact on the simulation's accuracy, convergence, and speed. The meshing tools become better and more automated as the solution becomes faster and more precise.

The Structured grid technique gets its name from the fact that the grid is built out in a block pattern. Quadrilateral elements are utilised in 2D, whereas hexahedral elements are used in 3D. The mesh form of a decent, well-organized grid generator is automatically optimised for orthogonality and homogeneity.

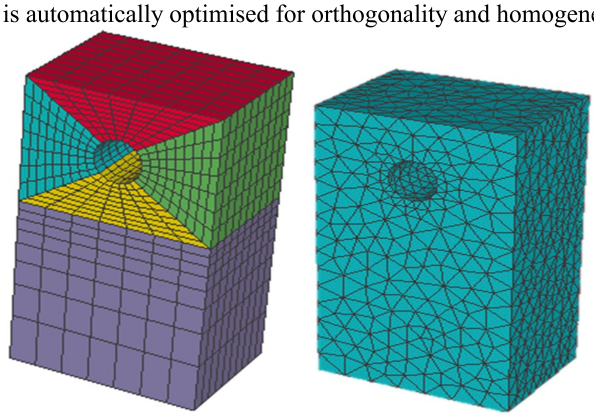


Fig. 4. Structured grid & unstructured grid\

Boundary condition: Boundary features (inlets, outlets, blockages, and so forth) can be connected to named 'object' descriptions during the grid building procedure. To create the grid and specify criteria, you no longer need to enter coordinates twice. The boundary condition is automatically adjusted if an item is later relocated or resized. The velocity and temperature inlet conditions may be specified using the grid profile. The intake turbulence intensity value may be used to calculate the turbulent kinetic energy (k) and its dissipation rate. Physically significant boundary limits, such as a specified pressure requirement, should be imposed to flow boundaries if at all practicable.

Solver: A differential equation can be invalidated in a variety of methods, but the finite volume method is the most popular. When dealing with coarse non-uniform grids and calculations where the mesh shifts to track an interface or a shock, finite volume approaches are advantageous. The values of the conserved variables are stored in the volume element rather than at nodes or surfaces.

Post-processing: After the solver has completed the simulation, the post-processing creates a graph of convergence criteria that has been checked. Different counter graphs, such as temperature, velocity, and pressure, may be constructed by loading a RES file into the CFX-post.

Turbulence Models: Turbulence is defined as a change in the flow of time and space. It's a tough technique since it's three-dimensional, unstable, and has several scales. Turbulence occurs when the inertia force in a flow surpasses the viscous forces. The length scales of turbulence flows are typically much smaller than the lowest finite volume mesh employed in numerical analysis. On the other hand, realistic Reynolds values encompass a

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

wide variety of turbulent length and time scales. These flows demand orders of magnitude more computing power than is currently available for Direct Numerical Simulation (DNS).

Turbulence models are used to forecast the effects of turbulence on fluid flow. Some have highly narrow applications, while others may be safely used to a wider flow class. One of the most difficult aspects of turbulence modelling is predicting flow separation under negative pressure gradients. Equation-based turbulence models, on the whole, underestimate the amount of separation later and anticipate the commencement of separation too late. This is troublesome because it creates the impression of an airfoil with a performance characteristic that is unduly optimistic. The model created to solve this issue has shown impressive results.

k-ε Turbulence model: Most general-purpose CFD systems include the k-ε (k-epsilon) model, which is one of the most well-known turbulence models. It has a demonstrated predictive power and has shown to be stable and numerically robust. For general-purpose simulation, the model provides a good blend of accuracy and resilience. The scaled wall function approach is used in the ANSYS-k (turbulence CFX) to increase robustness and accuracy. Scalable wall functions, in comparison to standard wall functions, allow simulation on arbitrarily fine near-wall grids. Two extra variables are added to the set of equations in the k-model.

k-ω Turbulence model: Because it does not require the complicated nonlinear damping functions that the k-(model) requires, the k- (formula) model is more exact and resilient. Turbulence viscosity is assumed to be proportional to turbulence kinetic energy and turbulent frequency in the models. One of the advantages of this model is the near-wall approach for low Reynolds number computations.

Computational domain

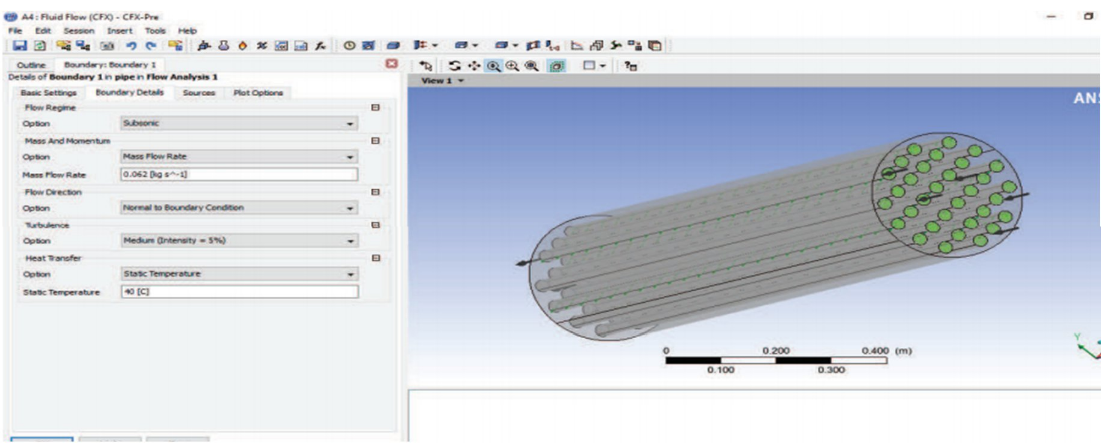


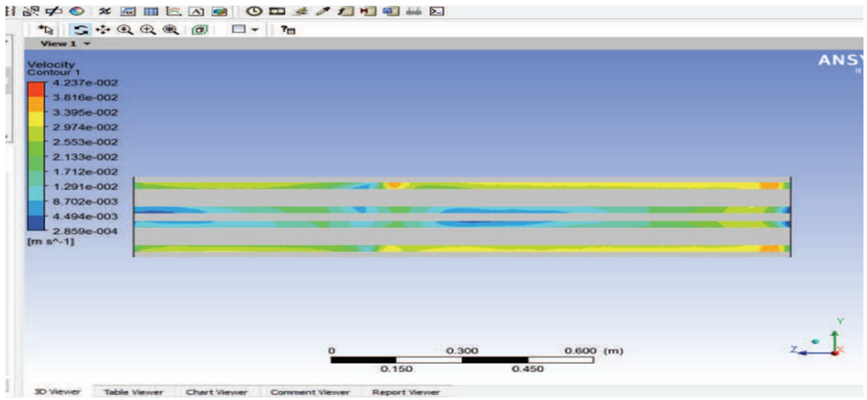
Fig. 5. CFD computational domain

4.Results and Discussion

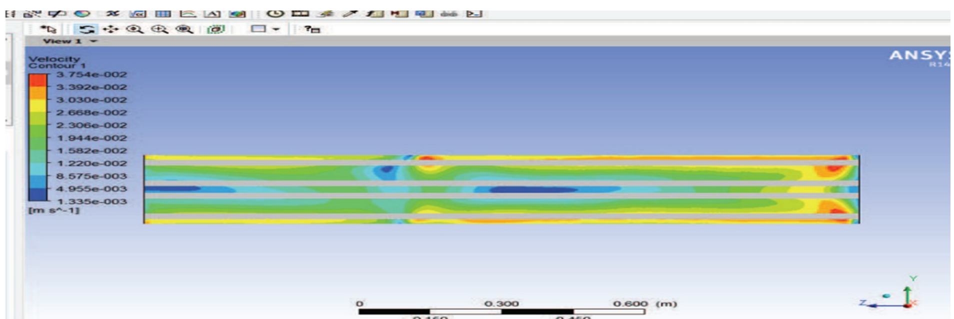
The most pressing issue affecting industry right now is heat exchanger efficiency. The Taguchi technique was used with the L9 array to improve the efficacy of heat exchanger properties. Performance requirements include diameter, mass flow rate, pitch length, longitudinal pitch, tube material, and other factors.
Taguchi's method has shown to be a valuable tool for determining which components and interactions are important in improving the efficiency of shell and tube heat exchangers. According to the practical and ANSYS findings, the mass flow rate and tube diameter are the most important parameters, while pitch length is the second factor that affects the heat exchanger's efficiency. According to the Taguchi analysis, the tube diameter should be 28 mm, the pitch length should be 36 mm, and the mass flow rate should be 0.34 kg/s to maximise heat exchanger efficiency. When the ANSYS and experimental data are compared and shown to be in good agreement, the model's strength is demonstrated. Following a review of the results of the CFD analysis, we can conclude that CFD Analysis

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

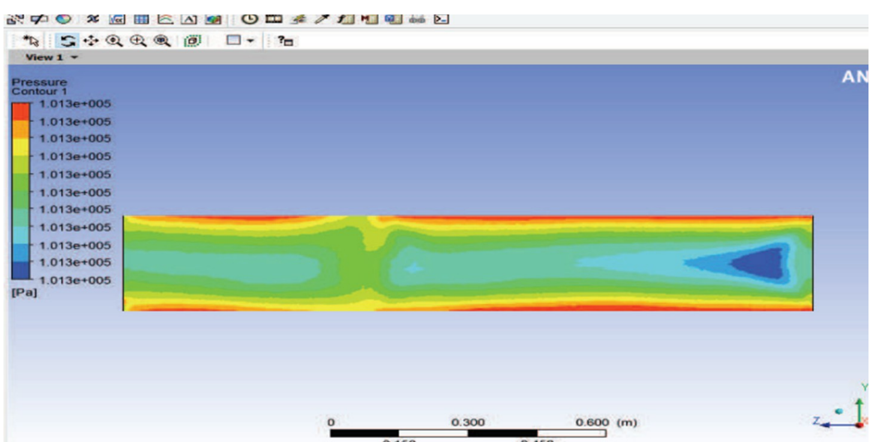
Outlet Fluid Velocity for Tube Side



Outlet Velocity for Shell Side

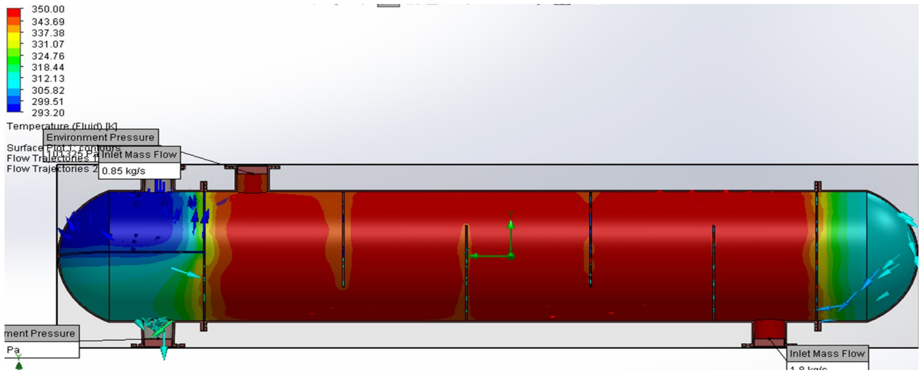


Pressure Contour

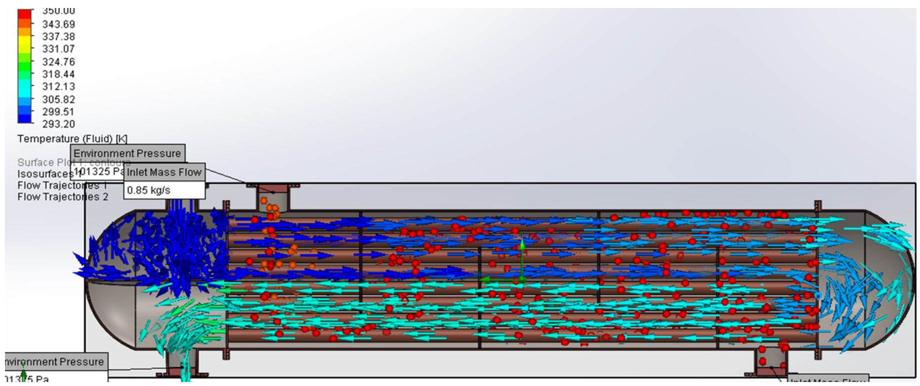


Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

Temperature



Flow path



5.Conclusion

The heat exchanger was subjected to a CFD study. For the research design, the analysis used Taguchi's method. Diameter, mass flow rate, pitch length, longitudinal pitch, and tube material were the factors used in the study design. The CFD study was utilised to investigate the flow behaviour of flow velocity, pressure, temperature, and flow route based on Taguchi's design.

References

[1] Raj Karuppa Thundil R., G. Srikanth, Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow using CFD, thermal Science, 16(4), 2012, 1165-1174.

[2] R. Bala Sundar Rao, G. Ranganath & C. Ranganayakulu, Development of colburn 'j' factor and fanning friction factor 'f' correlations for compact heat exchanger plain fins by using CFD, Heat and Mass Transfer, 49, 2013, 991–1000.

[3] Amir Jokar, Steven P. O'Halloran, Heat Transfer and Fluid Flow Analysis of Nanofluids in Corrugated Plate Heat Exchangers Using Computational Fluid Dynamics Simulation, J. Thermal Sci. Eng. Appl. 5(1), 2013, 011002.

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.

[4] Anil SinghYadavJ.L.Bhagoria, Heat transfer and fluid flow analysis of solar air heater: A review of CFD approach, Renewable and Sustainable Energy Reviews, 23, 2013, 60-79.
[5] Anil Singh Yadav & J. L. Bhagoria, Heat transfer and fluid flow analysis of an artificially roughened solar air heater: a CFD based investigation, Frontiers in Energy, 8, 2014, 201–211.
[6] Shashi Prakash Dwivedi, Bhavaya Gupta and Devesh Chaudhary; 'The Effect of Process Parameters on Mechanical Stir Casting Process',Volume No.1,Issue No.2,2013,PP.001-007,ISSN :2229-5828
[7] Shyam Lal, Sudhir Kumar,Z.A.Khan, A.N.Siddiquee; 'Research and Development in Wire Electrical Discharge Machining(WEDM) :A State of An Review',Volume No.1,Issue No.2,2013,PP.008-013,ISSN :2229-5828
[8] Bhavana; 'ERP- A Brief Structure and Discussion of its Success and Failure',Volume No.1,Issue No.2,2013,PP.014-017,ISSN :2229-5828
[9] Mohanty, Pramodini; 'An Efficient Baugh-Wooley Architecture for Signed &Unsigned Fast Multipication',Volume No.1,Issue No.2,2013,PP.018-021,ISSN :2229-5828
[10] Jitendra Kumar Saroj, Aruna Rana, Nitin Kathuria, Jitendra Kr.Mishra; 'High Gain Aperture Coupled Patch Antenna',Volume No.1,Issue No.2,2013,PP.022-024,ISSN :2229-5828
[11] Himanshu Tiwari, Monika Agarwal & Ankita Sagar; 'Supplying A Low-Voltage Continuous- Load From An Electrostatics Generator',Volume No.1,Issue No.2,2013,PP.025-028,ISSN :2229-5828
[12] Shefali Goyal, Prashant Singh; 'A Survey Based Comparative Study of DSR and AODV Routing Protocols In Adhoc Network Using NS-2',Volume No.1,Issue No.2,2013,PP.029-032,ISSN :2229-5828

Publisher: Noida Institute of Engineering & Technology,
19, Knowledge Park-II, Institutional Area, Greater Noida (UP), India.