PERFORMANCE IMPROVEMENT IN INTERNALLY FINNED TUBE BY SHAPE OPTIMIZATION

B.Velmurugan[1], S.Vijayakumar[2]
[1]Department of Mechanical Engineering, SASTRA University, Kumbakonam
[2]Department of Mechanical Engineering, Maharishi University of Information Technology (U.P)

ABSTRACT
Generally, fin is a surface that extends from an object to increase the rate of heat transfer to or from the environment by increasing convection. In this research, the Contours of Total Temperature for the optimized internal fin arrangement in the pipe, Velocity Vectors for the pipe without fin arrangement, Velocity Vectors for the optimized internal fin arrangement characteristics are analysed by CFD software , The internal Fin design with various dimensions such as height(H),thickness(t) and number of fins is made with the help of software GAMBIT and performance analysis of the heat transfer rate is calculated in the pipe with internal fin arrangement with flow of hot water as the working fluid and the results from CFD tool FLUENT are compared with experimental results.
KEYWORDS: Heat transfer, CFD Software, Fin, GAMBIT software

1. INTRODUCTION

With the increasing competition, the design of products of the highest quality and at less time has become a major necessity. To respond to these requirements, CFD tools are used to improve the process design in different engineering areas. They model and simulate the processes of real world avoiding the construction of prototypes and the task of performing experiments. With these tools, engineers can explore different design alternatives in less time and cost. Frequently, the main goal when investigating fluid flow processes is to optimize influencing parameters such that important engineering quantities become external, e.g. the pressure loss in a pipe system, the shear stress in a bio fluid, the drag force acting on a car body the degree of efficiency of a pump, the heat transfer in a cooling system, or the mixing efficiency in a stirrer should be optimized. Shape variation of the corresponding flow geometries is one possibility to reach that objective. In this context, the suitability of experimental measurement techniques to investigate and verify corresponding improvements systematically, is only limited, since the application of shape variations to a real model usually is very costly and time consuming or even not possible at all. As an alternative, numerical simulation techniques provide a great flexibility concerning geometrical parameter variations. To employ such techniques for optimization purposes an integrated approach combining flow simulation mathematical optimization, and shape variation is desirable. While for structural mechanics applications such approaches have been investigated and applied quite frequently (usually within the framework of the finite-element method), their application for general fluid mechanics problems up to now remained comparatively limited. Of course, this is mainly due to the fact that already the reliable simulation of a single flow configuration often can be a rather difficult task requiring a significant amount of computational resources. However, due to the rapid developments in computer technology and numerical techniques such methods more and more become feasible also for fluid flow problems of higher complexity.In the fin problem, CFD is a practical tool which can be used to predict the performance of heat transfer during the design process.

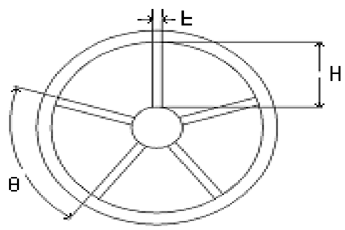

Figure 1 Cross-section of internal fins arrangement

Fig 1 shows the cross-section of internal fin arrangement. The condition in order to the design tool to be practically used concerns the computational costs. Since several expensive simulations have to be performed, the costs are usually prohibitive and the optimization strategy chosen should require as few evaluations as possible. Performance analyses of internally finned tube for a thermal performance improvement have been performed numerically and experimentally for many years . However, they have proposed the correlation equations for the design variables by considering only the flow and thermal characteristics of them.In recent years, the use of commercial CFD codes for analyzing the flow and thermal fields in industrial applications has been dramatically increased due to their advanced computational capacity and analytic algorithm. In addition, many optimization techniques have been developed to obtain the optimal solutions. Therefore, much attention has been paid to the optimization of fluid/thermal systems by combining the CFD and optimization algorithm. Finally the heat transfer rate without fin and with fin in the pipe is calculated and showing comparison results between CFD and experimental analysis.

2. LITERATURE SURVEY

Kyoungwoo Park, (2007) has predicted of flow and heat transfer characteristics and shape optimization in internally finned circular tubes have been performed on three-dimensional periodically fully developed turbulent flow and thermal fields. For a trapezoidal fin profile, the effects of fin height h, upper fin widths $d1$, lower fin widths $d2$, and helix angle of fin γ on transport phenomena are investigated for the condition of fin number of $N = 30$. The CFD and mathematical optimization technique are coupled in order to optimize the shape of internally finned tube. The optimal solutions of the design variables (i.e., upper and lower fin widths, fin height and helix angle) are numerically obtained by minimizing the pressure loss and maximizing the heat transfer rate, simultaneously, for the limiting conditions of $d1$ = 0.5~1.5 mm, $d2$ = 0.5~1.5 mm, h= 0.5~1.5mm, γ = 10~30 degrees. The fully developed flow and thermal fields are predicted using the finite volume method and the optimization is carried out by means of the multi-objective genetic algorithm that is widely used in the constrained nonlinear optimization problem.

T. Lehnha¨user, M. Scha¨fer, (2005) has predicted a numerical method for the shape optimization of fluid flow domains. The procedure is based on a flow solver, a mathematical optimization tool, and a technique for shape variation, which are combined into an integrated procedure. The flow solver relies on the discretization of the incompressible Navier–Stokes equations by means of the finite-volume method for block-structured, boundary-fitted grids with multi-grid acceleration. The optimization tool is an implementation of a trust region based derivative-free method.

It is designed to minimize smooth functions whose evaluations are considered expensive and whose derivatives are not available or not desirable to approximate. The shape variation is obtained by deforming the computational grid employed by the flow solver. For this purpose, displacement fields scaled by the design variables are added to the initial grid. The displacement vectors are computed once before starting the optimization cycle by using a free-form deformation technique. Applications illustrating the functionality and the properties of the method are presented for some examples of engineering interest, such as the minimization of a pressure drop, the maximization of a lift force, and the optimization of a wall temperature.

M. El-Sayed, (2005) has predicted In many product design and development applications, computational fluid dynamics CFD has become a useful analytical simulation tool. CFD simulations are quite useful in predicting several response parameters for a given design condition. However, like any analysis tool CFD simulations provide limited insight into the design space and the changes needed to find the optimum design parameters. That paper deals with the shape optimization of fluid flows using

CFD and numerical optimization techniques. By integrating a commercial optimization code with a CFD code, a CFD shape optimization tool was developed. To study the effectiveness of the developed tool and its ability to produce results with reasonable CPU time, the shape optimization of an airfoil and S-shaped duct are studied with different numbers of design variables. The developed shape optimization tool along with the optimization and CPU time results are discussed.

Nathalie Marco-Blaszka, (1999) has predicted In this report, we present the numerical solution of four optimization problems by Genetic Algorithms (GAs). The test-cases involve two single-objective and two multi-objective optimization problems. In all four cases, the analytical functions to be optimized present a large number of local optima and the GA is demonstrated to be the most adequate optimizer. These four test-cases are part of the database developed within the INGENET European thematic network.

Nathalie Marco(1999) has predicted this report approaches the question of multi-objective optimization for optimum shape design in aerodynamics. The employed optimizer is a semi-stochastic method, more precisely a Genetic Algorithm (GA). GAs are very robust optimization algorithms particularly well suited for problems in which (1) the initialization is not intuitive, (2) the parameters to be optimized are not all of the same type (boolean, integer, real, functionnal), (3) the cost functional may present several local minima, (4) several criteria should be accounted for simultaneously (multiphysics, e_ciency, cost, quality, ...). In a multi-objective optimization problem, there is no unique optimal solution but a whole set of potential solutions since in general no solution is optimal w.r.t. all criteria simultaneously ; instead, one identi_es a set of non-dominated solutions, referred to as the Pareto optimal front. After making these concepts precise, genetic algorithms are implemented and _rst tested on academic examples ; then a numerical experimentation is conducted to solve a multi-objective shape optimization problem for the design of an airfoil in Eulerian flow.

N. Trigui(1999) has predicted the Intense competition and global regulations in the automotive industry has placed unprecedented demands on the performance, efficiency, and emissions of today’s IC engines. The success or failure of a new engine design to meet these often-conflicting requirements is primarily dictated by its capability to provide minimal restriction for the inducted and exhausted flow and by its capability to generate strong large-scale in-cylinder motion. The first criterion is directly linked to power performance of the engine, while the latter has been shown to control the burn rate in IC engines. Enhanced burn rates are favorable to engine efficiency and partial load performance. CFD based numerical simulations have recently made it possible to study the development of such engine flows in great details. However, they offer little guidance for modifying the ports and chamber geometry controlling the flow to meet the desired performance. This paper presents a methodology which combines 3D, steady state CFD techniques with robust numerical optimization tools to design, rather than just evaluate the performance, of IC engine ports and chambers.

Waqar Ahmed Khan,(2004) has predicted In this study, an entropy generation minimization procedure is employed to optimize the overall performance (thermal and hydrodynamic) of isolated fin geometries and pinfin heat sinks. This allows the combined effects of thermal resistance and pressure drop to be assessed simultaneously as the heat sink interacts with the surrounding flow field. New general expressions for the entropy generation rate are developed using mass, energy, and entropy balances over an appropriate control volume. The formulation for the dimensionless entropy generation rate is obtained in terms of fin geometry, longitudinal and transverse pitches, pin-fin aspect ratio, thermal conductivity, arrangement of pin-fins, Reynolds and Prandtl numbers. It is shown that the entropy generation rate depends on two main performance parameters, i.e., thermal resistance and the pressure drop, which in turn depend on the average heat transfer and friction coefficients. These coefficients can be taken from fluid flow and heat transfer models. An extensive literature survey reveals that no comprehensive analytical model for any one of them exists that can be used for a wide range of Reynolds number, Prandtl number, longitudinal and transverse pitches, and thermal conductivity.

B. Daniel Marjavaara,(2006) has predicted the efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are, in next turn, two main factors that affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytic methods, experimental rules of thumb and model tests. In the last decade or two, the usage of computational fluid dynamics (CFD) has dramatically increased in the design process and will continue to grow due to is flexibility and cost-effectiveness. A CFD-based design search can further be aided with a robust and userfriendly optimization framework. Numerical prediction of the draft tube flow are, on the other hand, challenging and time consuming, caused by its complex flow features, e.g. unsteadiness, turbulence, separation, streamline curvature, secondary flow, swirl, and vortex breakdown. Hence, there is a great need of developing both accurate and reliable CFD models, together with efficient and effectiveoptimization frameworks.

3. DESIGN OF INTERNAL FIN BY GAMBIT SOFTWARE

Fig 2, 3 shows the modelling of the Pipe without fin arrangement, the internal fin arrangement is made by using the GAMBIT software. The model is meshed and specifying the boundary conditions. Also identify the solid and fluid path. Normally Tetrahedron mesh gives accurate results. The mesh file is exported from GAMBIT software.

Start up with 3Dimension solver, which is one of the type of solution approaches from FLUENT. The mesh file is imported to FLUENT software. Fig 4 shows the meshed fluid path. For heat transfer problem, select the energy equation. specify the values for each boundary condition. The values are initialized and it is iterated. Finally numerical and graphical results are taken.

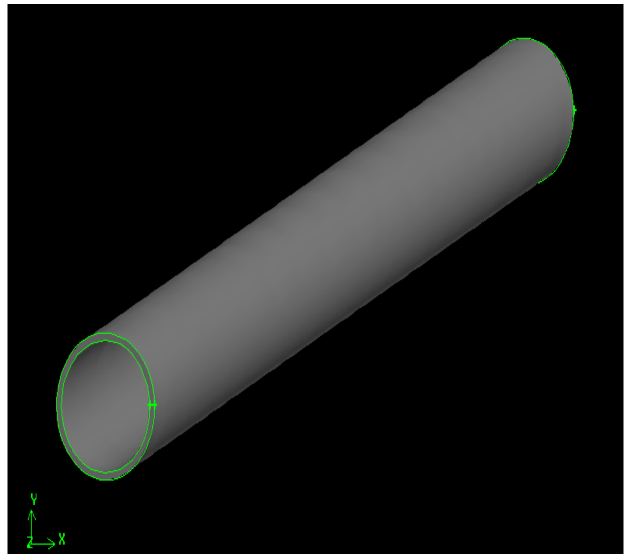

Fig 2 Pipe without fin arrangement

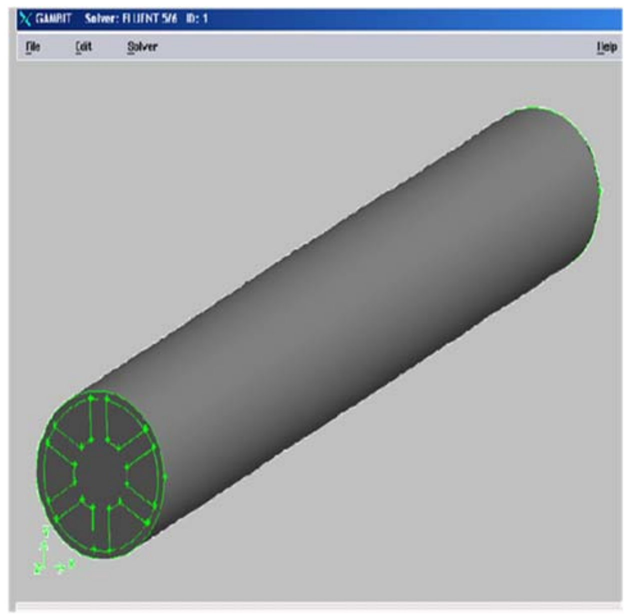

Fig 3 Pipe with internal fins arrangement

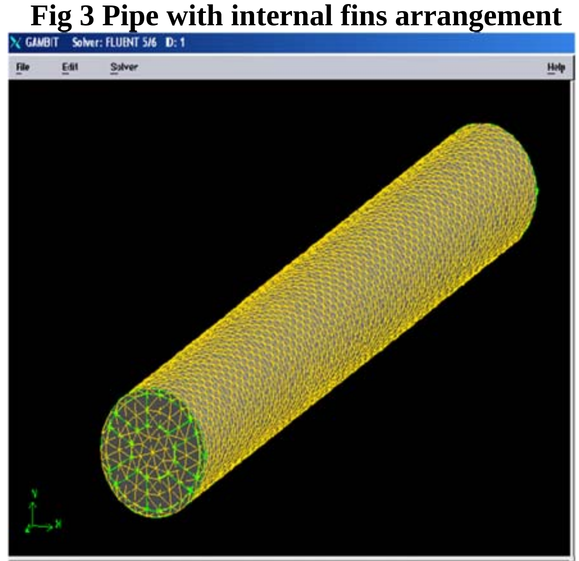

Fig 4 Meshed fluid path with internal fins arrangement

3.1 ASSIGNING PARAMETERS FOR CFD ANALYSIS

The internal flow and external flow are analyzed by using the commercial CFD tool FLUENT software. The hot water is flows inside the pipe(without fin arrangement).The outside of pipe is surrounded by atmospheric Air. The heat transfer coefficient, velocity of water is calculated analytically.

For heat transfer problem, select the energy equation. specify the values for each boundary condition. The values are initialized and it is iterated. Finally numerical and graphical results are taken. To analysis the Heat Transfer Coefficient for Internal flow and External flow

Water inlet temperature = 55 ˚c,
Ambient temperature = 36 ˚c
Mass flow Rate = 0.042 kg/s
Velocity of water = 0.03074 m/s
Length of pipe = 1500 mm
Characteristic diameter = 42 mm
Outlet diameter of pipe (d_c) = 47 mm
Surface temperature (Ts) = 52.5 ˚c

3.2 ASSIGNING PARAMETERS FOR EXPERIMENTAL ANALYSIS

For validation purpose, considered the Bench mark problem. The CFD results is compared with experimental results. The hot water is flows inside the pipe(without fin arrangement). The length of pipe, thickness and outer diameter of pipe are 1500mm, 2.5mm and 47mm respectively. The Fig 2 shows the Pipe without fin arrangement for experimental purpose. The experimental result is taken under the following condition,

Water inlet temperature = 55 ˚c,
Ambient temperature = 36 ˚c
Mass flow rate = 0.042 kg/s

4. RESULTS AND DISCUSSIONS

Table 1 Numerical Results of Heat Transfer Rate(Q) for various fin profiles

Sl.no	H (mm)	n (nos)	t (mm)	dc (mm)	h_{in} ($w/m^{2\circ}k$)	h_{out} ($w/m^{2\circ}k$)	T-out (˚c)	Q (w)
1	10	5	6	12.31	468.27	6.80	51.85	818.94
2	13	5	5	13.97	419.69	6.80	52.06	769.57
3	10	4	6	13.65	442.06	6.80	52.29	699.93
4	13	4	4	16.09	385.53	6.80	52.28	712.03
5	14	5	4	14.69	399.23	6.80	52.01	778.95
6	11	5	6	12.82	453.99	6.80	52.03	766.92
7	12	5	5	13.61	428.53	6.80	52.01	783.78
8	11	4	6	14.36	426.23	6.80	52.22	703.39
9	14	4	4	16.58	377.08	6.80	52.25	690.88
10	13	5	4	14.38	407.73	6.80	51.97	794.99

Table 1 shows the result taken from the Fluent software for the analyses of heat transfer rate, flow of water as the working fluid. The hot water is flows inside the pipe (including fin arrangement).The outside of pipe is surrounded by atmospheric Air. The fin profile such as Height of fin (H), Thickness of fin (t) and Number of fins (n) are varied. The velocity of water is depended on fin profile. Also the heat transfer coefficient is depending upon the fin profile.

From the Table 1, the heat transfer rate is varied in various fin profile arrangement. So we need to identify the maximized heat transfer rate. Here, Genetic Algorithm is calculated the optimized Results.

4.1 COMPARISON OF EXPERIMENTAL AND CFD RESULTS

The results obtained from the CFD were compared with the experimental results in bench mark problem. Here calculated the heat transfer rate in the pipe without internal fin arrangement with flow of hot water as the working fluid. The outside of pipe is surrounded by atmospheric Air. The heat transfer coefficient, velocity of water is calculated analytically. The hot water flows inside the pipe with 55˚c temperature is come to outlet of pipe with 52.24˚c temperature of water by solving numerically.

Fig 6 shows the pipe without fin arrangement for calculate the heat transfer rate. The hot water flows inside pipe with 55˚c temperature is come to outlet of pipe with 52.24 ˚c temperature of water. Figure 8 shows the Velocity Vectors for the pipe without fin arrangement.

Table 2 Comparison of numerical and experimental results

Description	Numeric (CFD) Result	Experiment Result	% of Error
Water outlet Temperature	52.24˚c	50.00˚c	4.48%

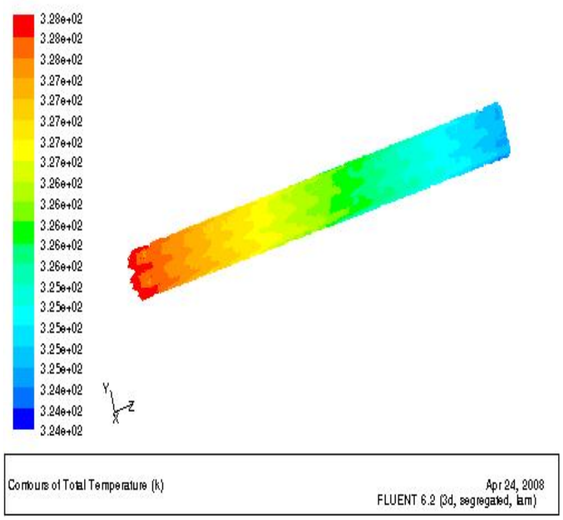

Fig 5 Contour of Total Temperature for the optimized internal fin arrangement

From the Table 2, the result of percentage of error is 4.48% .The numerically predicted results were in close conformance with that of the experimental values. The results shows that it can be easily predicted using the numerical analysis rather than going for experiments.

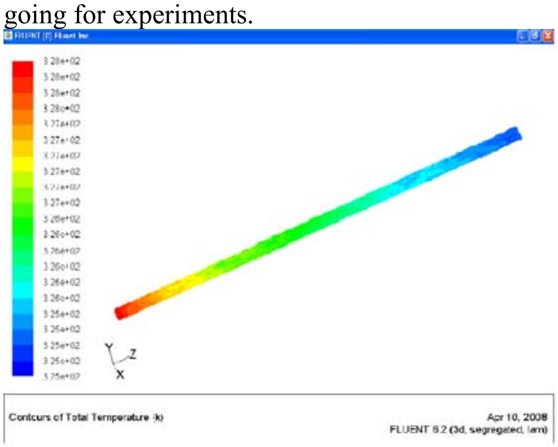

Fig 6 Contours of Total Temperature for the pipe without fin arrangement

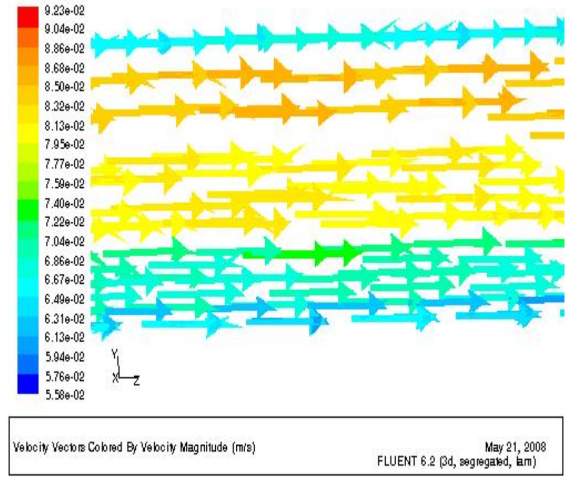

Fig 7 Velocity Vectors for the optimized internal fin arrangement

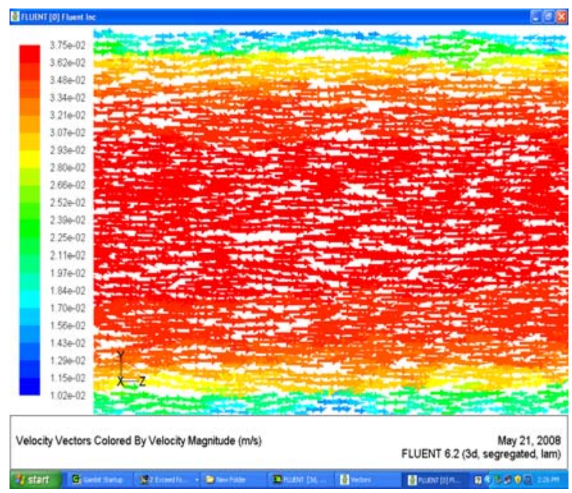

Fig 8 Velocity Vectors for the pipe without fin arrangement

5. CONCLUSIONS

The Internal fin arrangement was analyzed by using the CFD tool FLUENT software. In our project we have calculated the heat transfer rate in the pipe with internal fin arrangement with flow of hot water as the working fluid. The outside of pipe is surrounded by atmospheric Air. The combination (n, H, t) identified the optimization problem. G.A finds the optimal solution.The maximum heat transfer rate was calculated by using the optimized combination (n, H, t) of internal fin arrangement.

The results obtained from the CFD were compared with the experimental results in bench mark problem. Here calculated the heat transfer rate in the pipe without internal fin arrangement with flow of hot water as the working fluid. The numerically predicted results were in close conformance with that of the experimental values.

The efficiency of the Heat Transfer Rate is achieved by using fin arrangement with 500mm length of tube instead of using 1500mm length of tube without fin arrangement. The results shows that, it can be easily predicted using the numerical analysis rather than going for experiments.

REFERENCES

1. T.Lehnhauser,M. Schafer.(2005) ‘A numerical approach for shape optimization of fluid flow domains’ in the journal of Computer Methods in Applied Mechanics and Engineering vol. 194 pg 5221-5241.
2. Kyoungwoo park, byeong sam kim, hyo-jae lim, won han, park kyoun oh, juhee lee, and keun-yeol yu, (2007) ‘Performance Improvement In Internally Finned Tube By Shape Optimization’ in the journal of PWASET; Vol. 22; pg 225-230.
3. R.K. Jha and S.Chakraborty. ‘ Genetic algorithm based optimal design of fins’ in the journal of proc.IMechE vol.219 part C.J Mechanical Engineering Science.
4. M.EL-Sayed, T.Sun, J.Berry, (2005) ‘Shape optimization with computational fluid dynamics ‘ in the journal of ELSEVIER 36 pg 607-613.
5. Nathalie Macro-Blaszka And Jean Antoine Desideri (1999) ‘ Numerical solution of optimization test cases by Genetic Algorithm’ in the journal of Rapport de recherché n 3622-fevrier-23 pages.
6. Nathalie Macro-Stephane Lanteri And Jean Antoine Desideri,(1999) ‘Multi objective optimization in CFD by Genetic Algorithm ‘ in the journal of Rapport de recherché n 3686- Avril--40 pages.
7. N. Trigui, V. Griaznov, H. Affes and D. Smith,(1999) ‘CFD Based Shape Optimization of IC Engine’ in the journal of Oil & Gas Science and Technology – Rev. IFP, Vol. 54 , No. 2, pp. 297-307.
8. Dr. S. Pierret,(2002) ‘An Integrated Optimization System for Turbo machinery Blade Shape Design’ in the journal of RTO-MP-089 pg. 22-25.
9. Waqar Ahmed Khan, (2004) ‘Modeling of Fluid Flow and Heat Transfer for Optimization of Pin-Fin Heat Sinks’ presented to the University of Waterloo Ontario, Canada.