Investigation of Different Fins across Engine by using Experimental and CFD Simulation

Mr. Ravi D. Suryawanshi[1], Prof. D. S. Nakate[2] and Prof. Avinash Patil[3]

[1,2,3]Department of Mechanical Engineering (ME-Heat Power)
Professor at Dr. D. Y. Patil School of Engineering, Department of Mechanical Engineering
Dr. D. Y. Patil School of Engineering, Ambi, Talegaon Dhabade Tq:Maval, Di:-Pune. Maharashtra.

---***---

Abstract- *Nowadays, it is observed in automobiles sector is that to improve the efficiency of vehicle its heat dissipation across engine must be instantly cooled naturally. To provide this fin of different shape is to be studied with its heat transfer parameters with the help of CFD simulations and manufacturing of optimum design to validate the results. In present research different fins of shape circular, triangular and parabolic shape are investigated to obtain maximum heat dissipation shape for 2-wheeler vehicle. The main use of fins is to increase the rate of heat transfer to the environment by increasing the convection. The amount of heat transfer is determined by the amount of conduction, convection, radiation. If the temperature gradient between the object increases then the heat transfer rate also increases. The main aim of our project is to increase the heat transfer rate at varying types of fin (circular, triangular and parabolic) arrangement geometry. The 3D model is fabricated in CATIA software. The experimental testing is to be carried out and then the result & conclusion is drawn.*

Keywords—ANSYS CFD, CATIA, Heater, Fin arrangement

1. INTRODUCTION

In Engine When fuel is singed heat is created. Extra heat is additionally produced by contact between the moving parts. Only approximately 30% of the vitality discharged is changed over into valuable work. The staying (70%) must be expelled from the engine to keep the parts from softening. For this reason, Engine have cooling instrument in engine to expel this heat from the engine some substantial vehicles utilize water-cooling framework and practically every one of the bikes utilizes Air cooled engine, since Air-cooled engine are just alternative because of certain favorable circumstances like lighter weight and lesser space prerequisite. The heat created during burning in IC engine ought to be kept up at more significant level to build warm productivity, yet to forestall the warm harm some heat should expel from the engine. In air cooled engine, expanded surfaces called balances are given at the outskirts of engine chamber to build heat move rate. That

is the reason the examination of balance is critical to expand the heat move rate. Computational Fluid Dynamic (CFD) investigation has demonstrated upgrades in balance productivity by changing fin geometry, S fin pitch, number of balances and balance material. The issue of this part is it commitment of heat move coefficient (h), Temperature (T), Velocity, Fin Geometry (thickness, hole), material and surface harshness. Heat move rate that contribute while running the engine relies on the speed of vehicle, fin geometry and the encompassing temperature. On the off chance that the cooling rate diminishes, it brings about overheating prompting seizure of the engine. Simultaneously, an expansion in cooling rate influences the turning over of the engine. In investigation of heat move, a fin is a surface that it is reaches out from an article to build the pace of heat move to or from the earth by expanding convection. The measure of conduction, convection or radiation of an article decides the measure of heat move. Expanding the temperature contrast between the item and the earth, expanding the convection heat move coefficient, or expanding the surface territory of the article builds the heat move. Some of the time it isn't efficient or it isn't practical to change the initial two choices. Adding a fin to an item, in any case, builds the surface region and can here and there be a practical answer for heat move issues. Bike fin is structured with a few kinds of it. It is steady territory straight fin, variable zone straight balance, pin balance and annular balance. The explanation of structuring is normally to investigation the impact of the heat moves wherein the kind of configuration is one of the variables that gave impact. The motivation behind planning fin is to get the adequacy of the balance. In spite of the fact that the balances altogether increment heat move from the chamber, impressive improvement could in any case be gotten by expanding the quantity of fins.

2. LITERATURE REVIEW

K. Sathishkumar et.al [1], In this paper the information on efficiency and viability of the fin are essential for legitimate structuring of balances. The fundamental target of paper is to perform investigation to decide the progression of heat at different notches accessible and the examination is finished by utilizing ANSYS – CFD Fluent software. The fin with different designs were demonstrated utilizing CREO 2.0 and investigations are finished by utilizing CFD – Fluent so as to discover the heat move rate. Unmistakably the outcomes from software and hypothetically says that the fin with rectangular score have more noteworthy heat move rate contrasted with that of the balances without openings, balances with gaps and V molded balances. Since the heat distribution rate is more in rectangular notch so we reason that the rectangular score balances are most effectiveness and best heat move notch among a wide range of notch.

K. Chinnarasu et.al [2], In this paper it presents concentrate on higher cooling limit of radiator, expansion of fins is one of the ways to deal with increment the cooling pace of the radiator. This technique follows the rule of expanding contact surface. The shifting balance structures are Box type, Sharp type, Round type. Sharp type radiator balances are created to assess the adequacy of the radiator. Additionally, correlation of regular coolant with SiC Nano liquid has been done by utilizing Solid works and Ansys software. The examination of the car radiator with Nano liquid and water as a coolant in disparate geometrical structure of balance is adequately conveyed. From reproduction of the radiator balances with ordinary coolant (water) as coolant it is discovered that heat drop from 375K to 362.1K for example 3.47% for round kind balance followed by box type fin from 375K to 364.01K for example 2.94%, sharp type balance from 375K to 365.51K for example 2.54% and typical type balance from 375K to 368.07K for example 1.85%. Round type balance show high temperature drops in the reproduction. From recreation of the radiator fins with SiC Nanofluid as coolant it is discovered that heat drop from 375K to 349.44K for example 6.82% for round type fin followed by box type balance from 375K to 35364K for example 5.7%, sharp type fin from 375K to 354.59K 5.44% and typical type balance from 375K to 360.67K for example 3.83%. Round type balance show high temperature drops in the simulation.

Mohsin A. Ali et.al [3] In this article it speaks to an air-cooled engine cycle engine discharges heat to the environment through the method of constrained convection to encourage this, fins are given on the external surface of the chamber. The heat move rate relies on the speed of the vehicle, fin geometry and the encompassing temperature. The cooling balances permit the breeze to move the heat away from the engine. Low pace of heat move through balances is the principle issue of air-cooling framework. An endeavour is made to reproduce the heat move utilizing CFD for various shape and geometry of Fins to break down consequences for pace of heat scattering from fins surfaces. The heat move surfaces are demonstrated in CATIA and mimicked in FLUENT programming. Models for three distinct states of Fins were created and impacts of wind speed and heat move coefficient esteems were examined. Heat move rate increments in the wake of changing fin geometry and it is seen that HTC and choppiness are more in the event of Step shape Fin model as contrast with 'S' shape Fin model. In view of non-consistency in the geometry of Fins choppiness of streaming air builds which brings about more heat move rate.

Rashin Nath KK et.al [4], in this paper it presents the heat move of various balance geometry under various constrained convection conditions. The effectiveness of heat move can increment by expanding the heat move coefficient. Motor cycle motor discharges heat to the climate through the method of power convection. The heat move rate is characterized relying upon the speed of vehicle, balance geometry and the encompassing temperature. In any case, unique balance geometries are demonstrated in CATIA V5 software and CFD investigation will be utilized to reproduce the heat move of the motor square. The distinction of heat move rate for ordinary fin are more prominent for low speed as for that of recently created wavy balance and crisscross balance. The heat moves rate increments for crisscross and wavy fin contrasted with that of customary level balance as we continue speeding up. Because of the improvement of bended and crisscross shape it can produce whirl between two balances which incites turbulences and thus higher heat move. Crisscross and wavy balance along these lines can be favoured over ordinary balances for higher speed vehicles as it prompts more prominent choppiness and, in this manner, more noteworthy heat move rate. More disturbance and vorticity bring about additional improved heat move rate. This assists with improving the motor productivity and capability.

Akash M Vyas et. al [5] In this paper it presents the fuel is burned in a motor, heat is delivered. Extra heat is likewise produced by contact between the moving parts. In an air-cooled I.C motor, broadened surfaces called balances are given at the fringe of motor chamber to expand heat move rate. The fundamental of point of this examination various sorts of fins to improve heat move pace of cooling balances by changing chamber balance geometry. To expand the chamber cooling rate the chamber ought to have a more prominent number of balances. In view of audit study chamber heat move rate additionally increment by changing the different kinds of geometry of fins mounted on it. From the current examination we can say that the serrated strip balance geometry gives successful arrangement. In this simulation we found that, base temperature conveyance of serrated strip balance is 29 % not exactly the rectangular fin and 3.7 % not exactly the round balance.

3. PROBLEM STATEMENT

The majority of the heat dissemination forms incorporates constrained convection which is required to control the heat dispersal. In this way, disregarding the regular technique for expanding heat move, there is requirement for take a shot at the course of action of fins just as the material of balances utilized expectedly in the parts. The current investigation incorporates parametric and CFD examination of different geometric parameters of pin fin and furthermore the investigation of parameters, for example, heat move rate and heat move coefficient over the motor area.

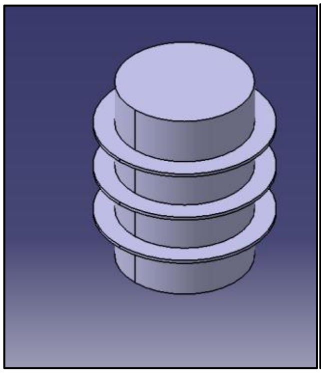
4. OBJECTIVES

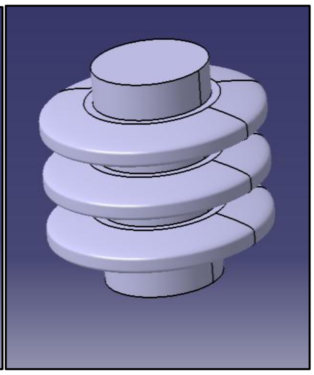
1. To investigate and evaluate different geometries (circular, triangular and parabolic) and experimental analysis on 2-wheeler engine changing the geometries of fin arrangement.
2. To find the effect of fins with different shape arrangement on forced convection heat transfer coefficient.
3. To obtain the optimized model for existing engine with different fins arrangement.
4. To obtain model with maximum heat transfer rate and heat transfer coefficient and manufacturing the best model for experimental results.
5. Experimental results comparison with numerical analysis results using temperature sensors.

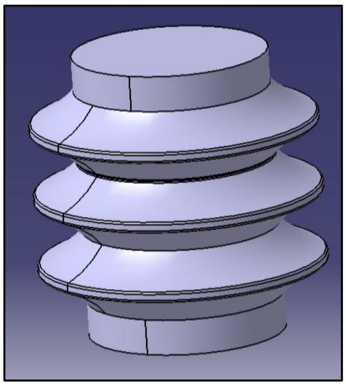
5. METHODOLOGY

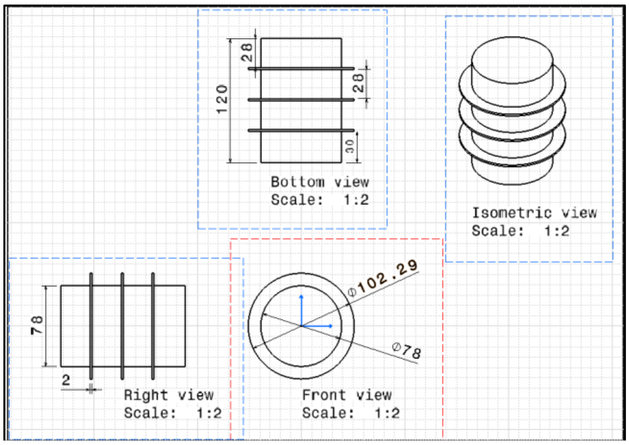
Step 1:- Initially research paper are studied to find out research gap for project then necessary parameters are studied in detail. After going through these papers, we learnt about different types of fin around engine.
Step2:- Research gap is studied to understand new objectives for project.
Step 3:- After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.
Step 4:-The Experimental Testing will be carried out on optimized design from CFD simulation.
Step 5:- Comparative analysis between the experimental & CFD simulation.

DESIGN OF DIFFERENT SHAPE FINS AROOUND ENGINE









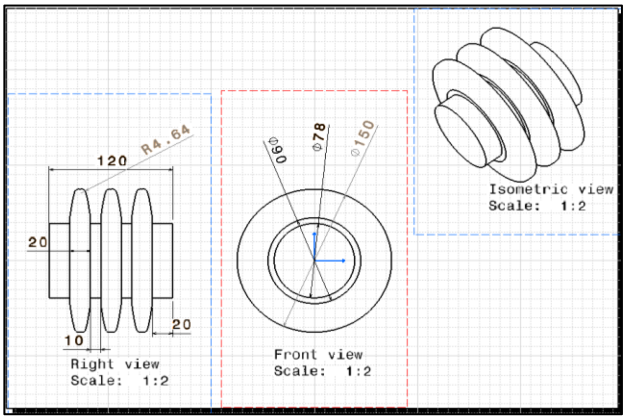


Fig. Geometry with enclosure for CFD simulation

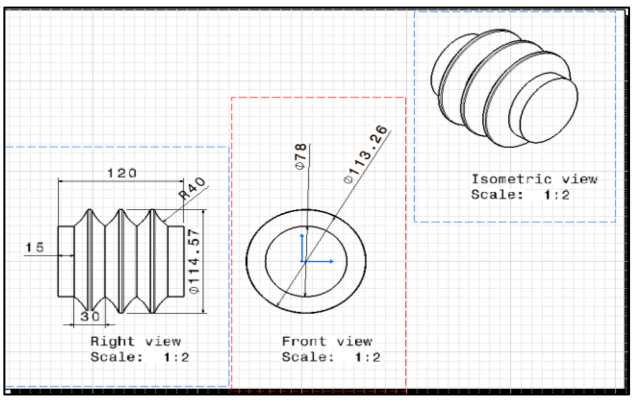
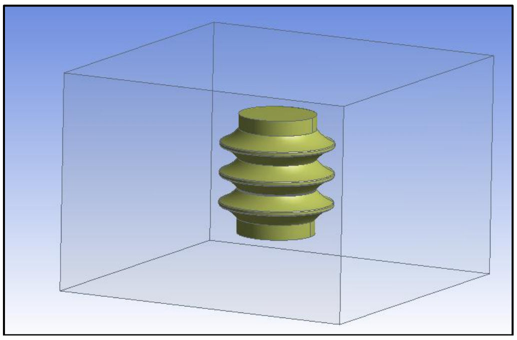


Fig. CATIA and drafting of different fins around engine

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modelling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

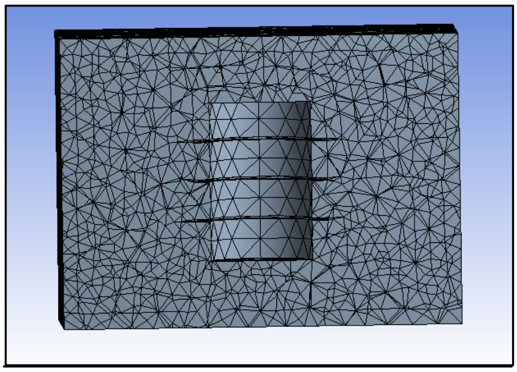
CFD PROCEDURE

- In CFD simulation bounding box is created across fin profile for simulation of velocity and pressure distribution across surface of fin.
- Fine meshing is performed for CFD simulation.
- Named selection is performed in CFD to define air inlet, outlet and fin surface.
- In general box model gravity is defined in perpendicular direction and energy is kept on to perform conservation of mass, momentum and energy equation to solve.
- In viscous model k epsilon, realizable and standard wall function is selected to maintain turbulence flow.
- Inlet velocity is defined as 12 m/s.
- Hybrid initialization is performed.
- 200 number of iterations is considered.



Mesh

In ANSYS meshing is performed as similar to discretization process in FEA procedure in which it breaks whole components in small elements and nodes. So, in analysis boundary condition equation are solved at this elements and nodes. ANSYS Meshing may be a all-purpose, intelligent, automated high-performance product. It produces the foremost acceptable mesh for correct, economical metaphysics solutions. A mesh well matched for a selected analysis may be generated with one click for all elements in a very model. Full controls over the options accustomed generate the mesh are accessible for the skilled user who needs to fine-tune it.



Statistics	
Nodes	9784
Elements	51124

Fig. Geometry and details of meshing of design

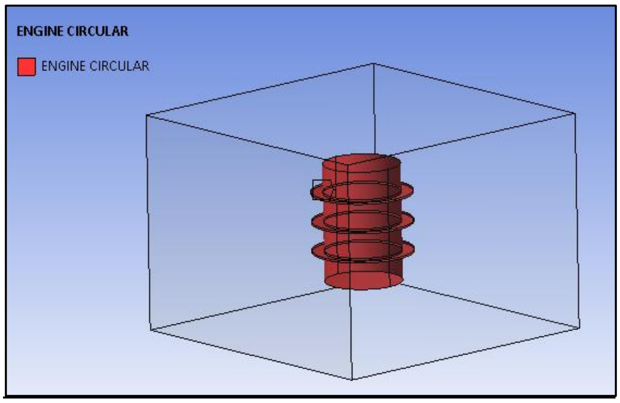


Fig. Named selection for CFD simulation

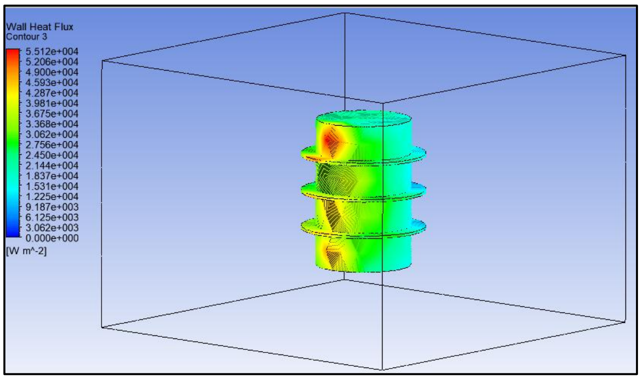


Fig. Wall heat flux contour for circular fin

In above plot it is observed that circular fin has maximum heat transfer rate as 55631 W/m^2 and heat transfer coefficient as 221 W/m^{2}k.

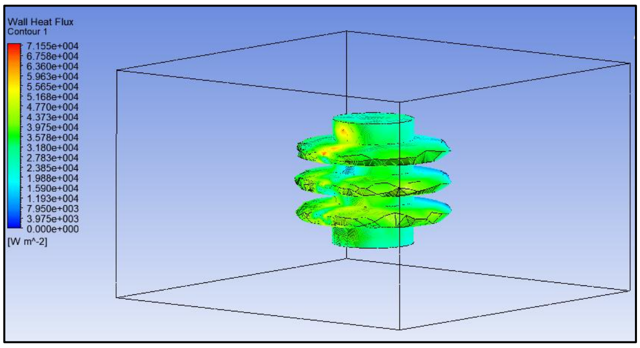


Fig. Wall heat flux contour for triangular fin

In above plot it is observed that triangular fin has maximum heat transfer rate as 72216 W/m^2 and heat transfer coefficient as 320 W/m^{2}k.

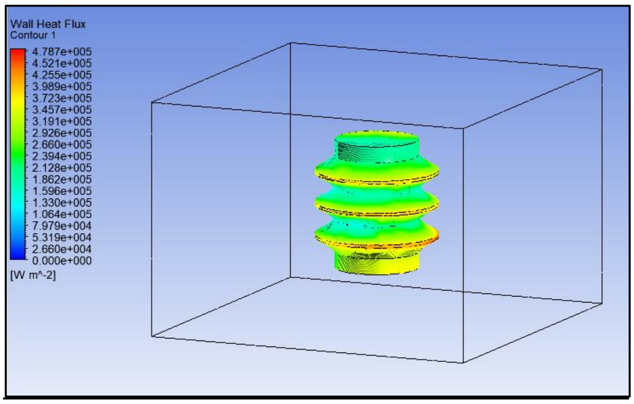
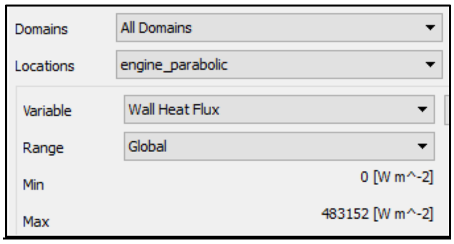


Fig. Wall heat flux contour for parabolic fin



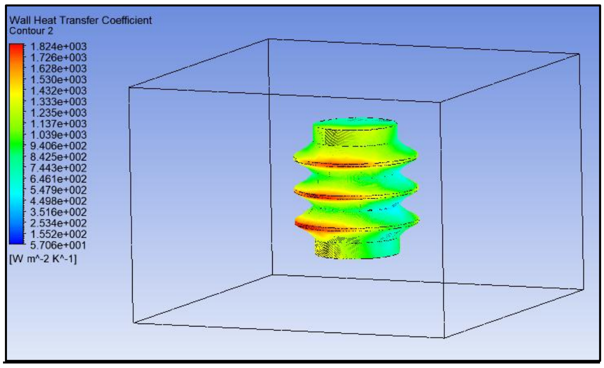


Fig. Wall heat transfer coefficient contour for parabolic fin

CONCLUSION

1. In present study CFD simulation of different fins are performed to obtain optimized model. It is observed from study that fins with parabolic shape have maximum heat flux and heat transfer coefficient compared to circular and triangular.
2. In parabolic shape due to its shape it has maximum heat transfer area across its surface.

REFERENCES

1) Computational Analysis of Heat Transfer through Fins with Different Types of Notches, K. Sathishkumar, K. Vignesh, N. Ugesh, P. B. Sanjeevaprasath, S. Balamurugan, International Journal of Advanced Engineering Research and Science (IJAERS) [Vol-4, Issue-2, Feb- 2017], ISSN: 2349-6495(P) | 2456-1908(O)
2) Analysis of Varying Geometri Structures of Fins using Radiators K. Chinnarasu1, M. Ranjithkumar , P. Lakshmanan, K. B. Hariharan, N. K. Vigneshwaran and S. Karan, Journal of Applied Fluid Mechanics, Vol. 11, Special Issue, pp. 115-119, 2018. Selected papers from International Conference on Newer Techniques and Innovations in Mechanical Engineering (ICONTIME 2K18), ISSN 1735-3572, EISSN 1735-3645.
3) Finite Element Thermo-Structural Methodology for Investigating Diesel Engine Pistons with Thermal Barrier Coating, Paolo Baldissera, Cristiana Delprete, Politecnico di Torino, Italy, SAE International by SAE MOBILUS Open Platform, Thursday, January 10, 2019
4) J.-J. Shu, I. Pop, "Thermal interaction between free convection and forced convection along a vertical conducting wall, "International Journal Of Heat and Mass Transfer
5) Wei Du, Lei Luo, Songtao Wang, Xinghong Zhang 'Effect of the dimple location and rotating number on the heat transfer and flow structure in a pin finned channel', "International Journal of Heat and Mass Transfer 127 (2018) 111–129"Published on 13 August 2018
6) Heat Transfer Analysis by CFD Simulation for Different shapes of Fins, Mohsin A. Ali and Prof. (Dr.) S.M Kherde, International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 3 Issue II, February 2015 ISSN: 2321-9653
7) Design Modification and Analysis of Two Wheeler Engine Cooling Fin, Rashin Nath.KK, Jayee K

Varghese, International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE) Vol 2, Issue 7, July 2017, ISSN 2456-1290

8) Analysis of I.C. Engine Fins for Effective Cooling Performance, Akash M Vyas Ruchir Parikh, IJSTE - International Journal of Science Technology & Engineering | Volume 5 | Issue 1 | July 2018 ISSN 2349-784
9) Heat transfer simulation of engine cycle fins under varying velocity using CFD method K.Shahril, Nurhayati Binti Mohd Kasim, and M.Sabri, 2nd International Conference on Mechanical Engineering Research (ICMER2013) IOP Publishing IOP Conf. Series: Materials Science and Engineering 50 (2013)
10) Thermal Analysis of Engine Fins with Different Geometries, L.Natrayan , G.Selvaraj , N.Alagirisamy , M.S.Santhosh, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 5, May 2016
11) Heat Transfer Simulation by CFD from Fins of an Air Cooled Enginecycle Engine under Varying Climatic Conditions, Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan, Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K.