Proceedings of the National Conference on Computational Modeling of Fluid Dynamics Problems (CMFDP-2019)
NIT Warangal, India – Jan 18-20, 2019
Paper No. XXX (The number assigned when the papaer is accepted)

CFD Analysis of PAT: A Case Study

Akash B G[a], Shivukumara B[a,], Hariprasad T S[a,], Naresh R[a,], Priyabrata Adhikary[a,]
[a]*Mechanical Engineering Department, New Horizon College of Engineering, Bangalore-560103, Karnataka, India*

*Corresponding author Email: **akashbggowda@gmail.com**

Abstract:

PAT are emerging as solution for sustainable, eco-friendly, long term and cost-effective water or renewable energy resource for future. According to the International Energy Agency (IEA), approx.22% (2008) of the world's populations, living without access to electricity, 85% of whom live in rural areas. Of the 1.5 billion people in the world who have no access to electricity, India accounts for over 300 million. Such an energy situation for the poor villagers is unacceptable. It is possible to achieve universal energy access in the foreseeable future, and modern renewable energy technologies can play a crucial role in achieving this goal. This paper describes design and development of low cost micro hydro turbine (converted from commercially available water flow meter) effective for hilly and/or rural area as basic electricity home systems for rural and/or hilly area electrification. Water flow rotates the turbine rotor inside stator whose speed of rotation changes with the different rate of flow of water. To the best of the author's knowledge these novel approach for CFD analysis of PAT are absent in renewable energy or fluid mechanics literature due to its assessment complexity.

1. Introduction

PAT systems are most suited for developing countries where conventional turbines are not as available, and may or may not be the best option for your village. This study is intended as a step-by-step guide for site assessment, system layout, sizing and purchasing system components, and installation. Some parts of the study are technical and may be best understood by someone with technical expertise. Typical hydropower systems convert the energy of falling water to mechanical energy with a turbine. In some cases, it may be more appropriate to replace the turbine with a centrifugal water pump, and run it in reverse. The words 'pump' and 'turbine' are used interchangeably in this study, as are the words 'motor' and 'generator'. Using a pump as a turbine has numerous benefits for rural pico-hydro projects in the developing world. Since centrifugal water pumps can usually be found locally, one avoids paying expensive import taxes, and since the pump is a familiar technology to local pump and motor technicians, it can be serviced if problems arise. This is far easier than finding renewable energy technicians specializing in pico-PAT. Furthermore, pumps are manufactured to operate under a wide range of conditions, are easy to install, and spare parts for these pumps are easy to find.[1-10]

2. Methodology Adopted

The governing equations of viscous flow are based on conservation of mass, momentum and energy which are Langrangian in nature. The governing equations are expressed using equations (notations used have their usual standard meanings) shown below:

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho\, \mathbf{u}) \tag{1}$$

$$\rho\left(\frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla\right)\mathbf{u} = -\nabla P + \rho \mathbf{g} + \frac{1}{c}\mathbf{J} \times \mathbf{B} \tag{2}$$

$$\rho\left(\frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla\right)e = -P\nabla \cdot \mathbf{u} + \rho \mathbf{u} \cdot \mathbf{g} + \frac{1}{\sigma}\mathbf{J}^2 \tag{3}$$

The numerical analysis of CFD in micro hydro turbine consists of incompressible fluid flow that reduces the conservation of mass and momentum to equations shown below respectively. In addition, the temperature effect is negligible during the analysis. Therefore, conservation of energy is ignored during analysis.

There are many commercial general-purpose CFD programs (Interdisciplinary field of study based on Physics, Engineering, Mechanics, Biology, Material Science supported by both Mathematics and Computer Science) available, e.g. Ansys-Fluent, Ansys-CFX, Star-CD, FLOW 3D, SoldWorks Flow Simulation. A very useful open-source program that can handle CFD problems is OpenFoam. However, the documentation and the user interface are not well developed as those for the commercial codes. Commercial CFD packages contain modules for CAD drawing, meshing, flow simulations, solver and post-processing. The CFD analysis for the same was carried out using the FLOEFD, for analysing the performance. For very complex systems the results are not very accurate, but CFD can still be very useful saving design engineer’s time-cost-effort. Experimental validation verifies the codes to make sure that the numerical solutions are correct and compare the results (making a provision for measurement errors).

The FLOEFD solves the Navier Stokes and conservation equations. The equations that we used are not closed, so we need to use Turbulence Modelling to close the equation set and then iterate towards a solution. We used what is called a Reynolds Averaged Navier Stokes (RANS) approach, (or we can use an Eddy Simulation technique which resolves the larger eddies in the flow and is only really required when you have separation or large re-circulating regions). The most commonly used models are the RANS models due to their low cost in terms of compute power and run times. The Eddy Simulation methods can be quite mesh sensitive but will yield much better results for separated and recirculating flow, but takes much longer run times. There are different turbulence models available in FLOEFD - Fluent as mentioned below: Spalart-Allmaras Model; k-Ɛ (k-Epsilon) Model-widely used; k-ω (k-Omega) Model; v2-f Model; Reynold’s Stress Model (RSM); Detached Eddy Simulation Model (DES); Large Eddy Simulation Model (LES) etc.[1-10]

3. Theory and Calculation

The modified k-ε turbulence model with damping functions proposed by Lam and Bremhorst (1981) describes laminar, turbulent, and transitional flows of homogeneous fluids consisting of the following turbulence conservation laws.

$$\frac{\partial \rho k}{\partial t}+\frac{\partial \rho k u_i}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+\tau_{ij}^R\frac{\partial u_i}{\partial x_j}-\rho\varepsilon+\mu_t P_B\ ,$$

$$\frac{\partial \rho \varepsilon}{\partial t}+\frac{\partial \rho \varepsilon u_i}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}\left[f_i\tau_{ij}^R\frac{\partial u_i}{\partial x_j}+C_B\mu_t P_B\right]-f_2 C_{\varepsilon 2}\frac{\rho\varepsilon^2}{k},$$

$$\tau_{ij}=\mu S_{ij}, \tau_{ij}^R=\mu_t S_{ij}-\frac{2}{3}\rho k\delta_{ij}\ , S_{ij}=\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k},$$

$$P_B=-\frac{g_i}{\sigma_B}\frac{1}{\rho}\frac{\partial \rho}{\partial x_i},$$

Where $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $\sigma_B = 0.9$, $C_B = 1$

If $P_B > 0$, $C_B = 0$ If $P_B < 0$, *The turbulent viscosity is determined from*

$$\mu_t = f_\mu . \frac{C_\mu \rho k^2}{\varepsilon},$$

Lam and Bremhorst’s damping function f_μ *is determined from* :

$$f_u = (1-e^{-0.025R_y})^2\ \left[1+\frac{20.5}{R_i}\right],$$

Where $$R_y = \frac{\rho\sqrt{ky}}{\mu},$$

$$R_t = \frac{\rho k^2}{\mu\varepsilon},$$

y is the distance from point to the wall and Lam and Bremhorst’s damping functions $f1$ and $f2$ are determined from:

$$f_1 = 1+\left[\frac{0.05}{f_\mu}\right]^3,\ \ f_2 = 1-e^{R_t^2}$$

Lam and Bremhorst’s damping functions $f\mu$, $f1$, $f2$ decrease turbulent viscosity and turbulence energy and increase the turbulence dissipation rate when the Reynolds number Ry based on the average velocity of fluctuations and distance from the wall becomes too small. When $f\mu =1$, $f1 =1$, $f2 =1$ the approach obtains the original k-ε model.[11-20]

3. Result and Discussion :

Comparison between numerical results and experimental data reveals a good agreement. PAT are not much accurate at low flow rates due to rotor/bearing drag that decelerates the rotor. Almost 5% of minimum rated flow capacity is required. It should not be run at high velocity because premature bearing wear and/or damage can occur. We need to be careful when measuring fluids that are non-lubricating because bearing wear can cause error. With the help of CFD the rotor driving torque (and power) is calculated on blades by using boundary conditions.

Case-1: Impeller with 5 number of blades

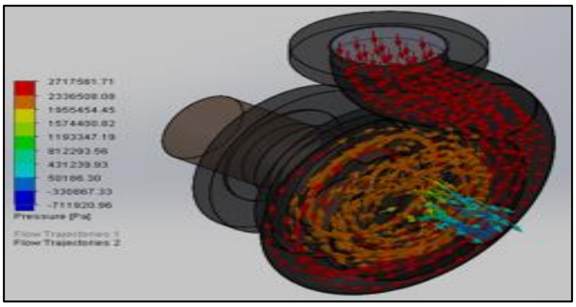

Fig. 1. Pressure – CFD Analysis (5 Blades)

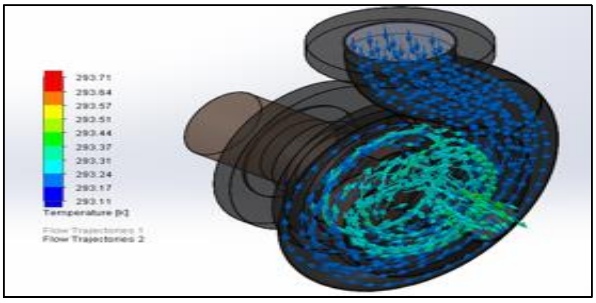

Fig. 2. Temperature – CFD Analysis (5 Blades)

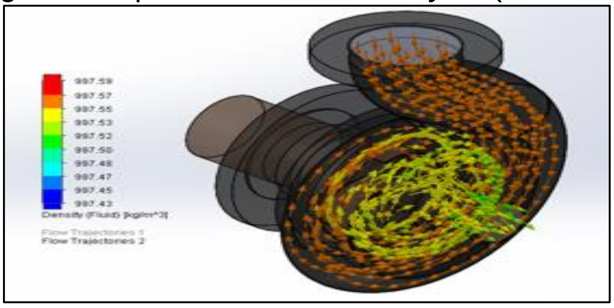

Fig. 3. Density – CFD Analysis (5 Blades)

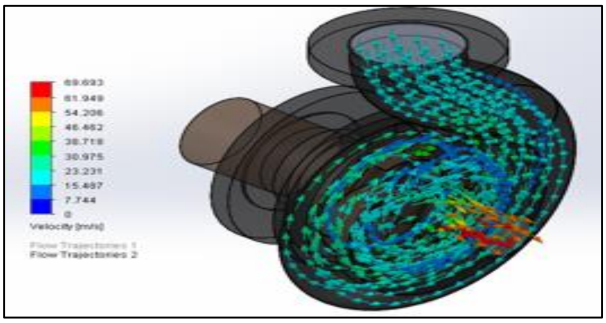

Fig. 4. Velocity – CFD Analysis (5 Blades)

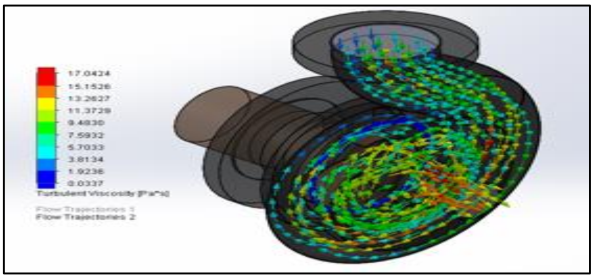

Fig. 5. Turbulent viscosity – CFD Analysis (5 Blades)

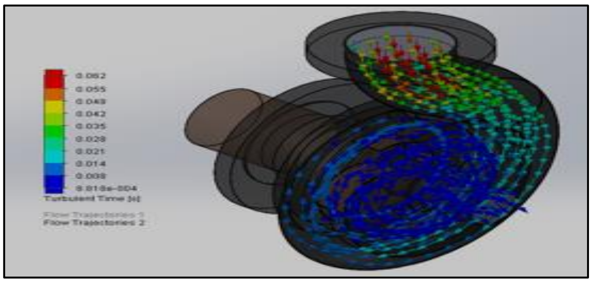

Fig. 6. Turbulent time – CFD Analysis (5 Blades)

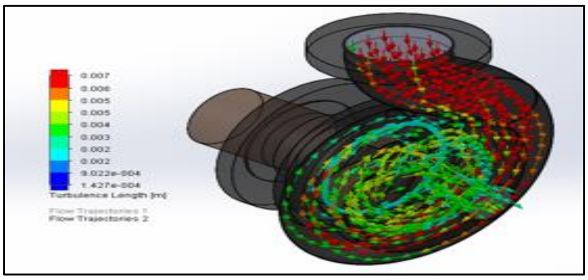

Fig. 7. Turbulent Length – CFD Analysis (5 Blades)

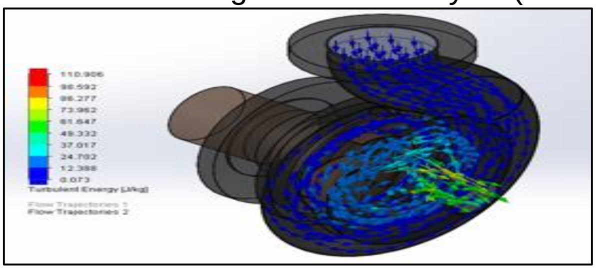

Fig. 8. Turbulent Energy – CFD Analysis (5 Blades)

Case-2: Impeller with 7 number of blades

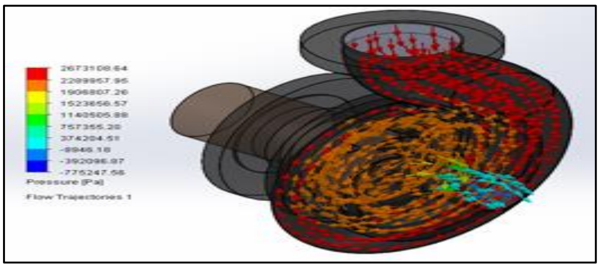

Fig. 9. Pressure – CFD Analysis (7 Blades)

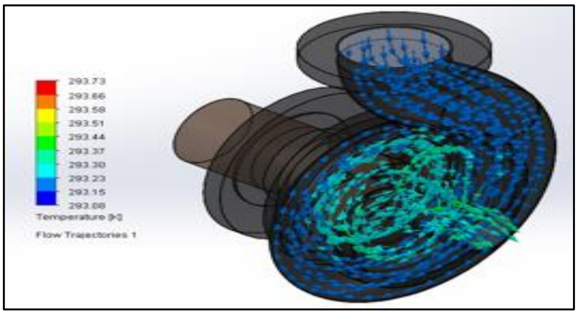

Fig. 10. Temperature – CFD Analysis (7 Blades)

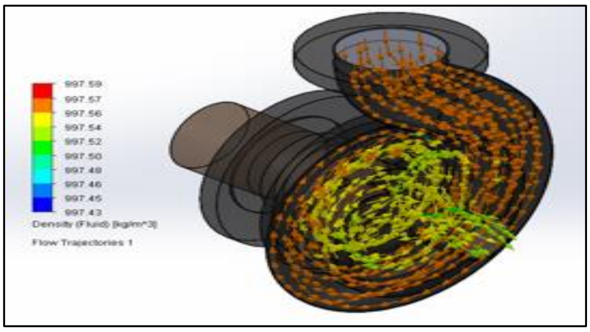

Fig. 11.Density – CFD Analysis (7 Blades)

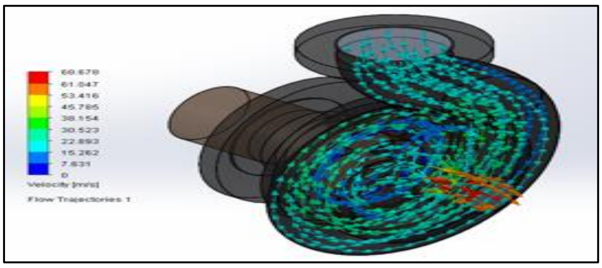

Fig. 12. Velocity – CFD Analysis (7 Blades)

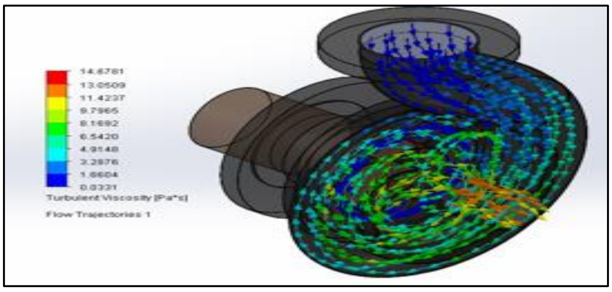

Fig. 13. Turbulent Viscosity – CFD Analysis (7 Blades)

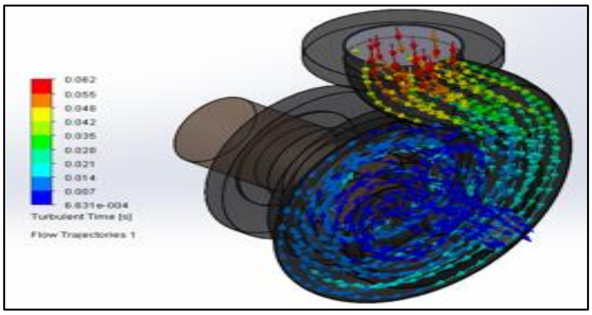

Fig. 14. Turbulent time – CFD Analysis (7 Blades)

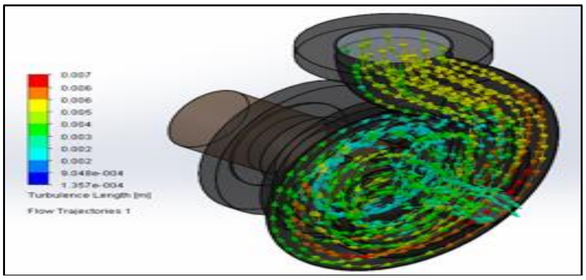

Fig. 15. Turbulent Length – CFD Analysis (7 Blades)

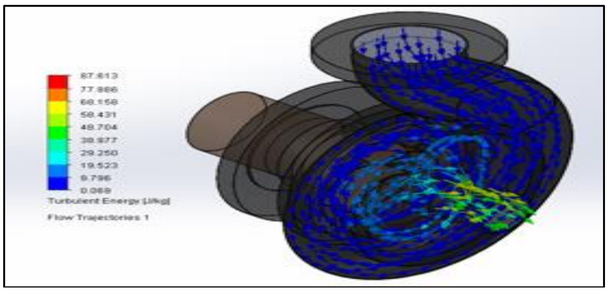

Fig. 16. Turbulent Energy – CFD Analysis (7 Blades)

5. Conclusion

The advanced CFD model used in this research solves the Navier-Stokes equations, which are formulations of mass, momentum and energy conservation laws for fluid flows. This CFD model is able of predicting both laminar and turbulent flows. Most of the fluid flows in engineering practice are turbulent, so this model uses the Reynold-Averaged-Navier-Stokes (RANS) equations, where time-averaged effects of the flow turbulence on the flow parameters are considered. Through this procedure, extra terms known as the Reynolds stresses appear in the equations for which additional information must be provided. To close this system of equations, it employs transport equations for the turbulent kinetic energy and its dissipation rate (k-ε model). This research shows the utility of the CFD numerical simulations as a tool for design and optimization of PAT performance and flow behaviour through hydro mechanical devices or hydraulic structures at minimum time-cost-effort.

Acknowledgements

The authors wish to thank N.H.C.E. Bangalore, M.G.P.L. Bangalore and S.W.R.E., Jadavpur University, Kolkata for the valuable technical support.

References

1. P.Adhikary;Design and CFD analysis for pump impeller;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P25

2. P.Adhikary;CFD analysis of concept car for improvement of aerodynamic design;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P20

3. P.Adhikary;Design and CFD analysis of centrifugal pump;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P11

4. P.Adhikary;12V DC pico hydro for hilly rural electrification: performance analysis by C.F.D. (Best Poster Award);2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P1

5. P.Adhikary;Micro Hydropower Generation from Rural Drinking W.T.P.: C.F.D. Analysis & It's Validation;FMFP 2016, International Conference, MNNIT Allahabad

6. P.Adhikary;C.F.D. ANALYSIS OF 12V 10W DC MICRO HYDRO TURBINE: A CASE STUDY;NCETPFS 2016, National Conference, Jadavpur University, Kolkata

7. P.Adhikary;Design and development of Arduino controlled self balancing duct cleaning robot: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:121-124

8. P.Adhikary;Conference room AC system performance analysis: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:117-120

9. P.Adhikary;Performance analysis of an office space HVAC system: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:110-113

10. P.Adhikary;Data Centre cooling system performance analysis: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:106-109

11. P.Adhikary;CFD analysis of a Call Centre HVAC system: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:102-105

12. P.Adhikary;CFD analysis of dairy cold room: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:98-101

13. P.Adhikary;CFD analysis of HVAC Scroll Compressor performance – case study using two refrigerants (R407C and R410A);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:73-76

14. P.Adhikary;Performance analysis by CFD of Scroll Compressor using two high pressure refrigerants (R134A and R410A);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:69-72

15. P.Adhikary;CFD analysis of high pressure Scroll Compressor - comparison with two refrigerants (R134A and R407C);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:65-68

16. P.Adhikary;High pressure Rotary Compressor CFD analysis - Performance comparison of two R gases (R134A and R407C);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:61-64

17. P.Adhikary;CFD analysis of Twin Screw Compressor using low pressure refrigerants (HFD1234yp and R407C);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:55-58

18. P.Adhikary;CFD analysis of low pressure Screw Compressor using two refrigerants (HFD1234yp and R123);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:51-54

19. P.Adhikary;Rotary Compressor performance analysis by CFD using two high pressure refrigerants (R134A and R410A);International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:43-46

20. P.Adhikary;12V DC pico hydro for hilly rural electrification: performance analysis by C.F.D. (Outstanding Paper Award);25th WBSSTC, Science Congress, 2018 (GoWB)

21. P.Adhikary;Numerical studies on indoor air flow in air-conditioned space: A case study of CFD application;ICMAAM 2018 Department of Mathematics, International Conference, Jadavpur University

22. P.Adhikary;Performance prediction method for pump as turbine (PAT) using CFD modelling: A case study;ICMAAM 2018 Department of Mathematics, International Conference, Jadavpur University

23. P.Adhikary;Thermal CFD study and improvement of domestic refrigerator evaporator: A case study;ICMAAM 2018 Department of Mathematics, International Conference, Jadavpur University

24. P.Adhikary;Design optimization of chilled water pump impeller using CFD: A case study;ICMAAM 2018 Department of Mathematics, International Conference, Jadavpur University