**Innovation in teaching learning process**

**[1] Name of the Innovation activity: CFD Analysis of Pump as a Turbine**

**[2] Course code and course name: 1MEPC301, Turbo Machinery**

**[3] Program and Class: Mechanical Engineering, T.Y. B.Tech (A and B)**

**[4] Name of Faculty: Mr. G. B. Pawar**

**[5] Introduction:**

In the ever-evolving landscape of engineering education, integrating innovative teaching strategies is essential to equip students with the skills and competencies needed to thrive in a rapidly advancing field. One such strategy that has gained prominence is the utilization of Computational Fluid Dynamics (CFD) software to solve case studies, particularly those focusing on the innovative application of pumps as turbines (PAT). This approach not only enhances students' understanding of fluid mechanics and turbo machinery principles but also empowers them to tackle real-world engineering challenges with precision and efficacy.

The conversion of pumps into turbines, known as the PAT concept, represents a paradigm shift in the utilization of hydraulic machinery. Traditionally, pumps are designed to impart energy to fluids, while turbines extract energy from flowing fluids. However, the PAT concept harnesses this reversible process, allowing pumps to efficiently generate electricity from natural water flows. By employing CFD software, students can delve into the intricate fluid dynamics involved in PAT systems, gaining insight into flow behavior, pressure distribution, and efficiency optimization.

The integration of CFD software into the study of PAT case studies offers a multifaceted learning experience for engineering students. Through hands-on simulation exercises, students can explore various design configurations, assess performance parameters, and troubleshoot operational challenges in a virtual environment. This interactive approach not only enhances their theoretical understanding but also cultivates practical skills in numerical modeling, data analysis, and problem-solving – all of which are vital for engineering practice in the digital age.

Furthermore, leveraging CFD software in the study of PAT case studies aligns with broader educational objectives, including the promotion of sustainability and renewable energy solutions. By simulating the performance of PAT systems under different operating conditions, students gain insight into the potential applications of this technology in harnessing renewable energy sources and mitigating environmental impacts. This interdisciplinary approach fosters a holistic understanding of engineering principles while instilling a sense of responsibility towards addressing global challenges such as climate change and energy sustainability.

**[6] Motivation/Purpose of innovative technique**

* Enhanced Understanding of Fluid Dynamics and Turbo Machineries
* Real-World Relevance
* Empowerment Through Simulation
* Interdisciplinary Learning Opportunities
* Promotion of Innovation and Creativity

**[7] Suitability of technique over course content**

Utilizing Computational Fluid Dynamics (CFD) software to solve case studies focusing on pumps as turbines (PAT) represents a cutting-edge and innovative approach to engineering education. By integrating CFD simulations into the curriculum, students are provided with a powerful tool to explore the intricate dynamics of fluid flow within PAT systems. This hands-on learning strategy not only enhances students' understanding of theoretical concepts but also fosters critical thinking, problem-solving skills, and interdisciplinary integration. Moreover, it aligns with industry practices and trends, preparing students for the challenges of modern engineering practice. In this paper, we delve into the suitability and effectiveness of employing CFD software in solving PAT case studies, highlighting its transformative potential in shaping the next generation of engineers.

**[8] Procedure Followed**

In the CFD laboratory, the simulation is performed on pump as turbine to analyze its performance under different loading conditions. Step by step process of simulation is discussed with the students. Same problem by different boundary conditions are given to the students for the analysis.

**[9] Evaluation process followed**

Case studies are evaluated on the basis of student’s results and flow pattern obtained.

**[10] Outcome**

1. PAT performance analysis under different boundary conditions.

2. Flow pattern interpretation

C:\Users\dell\Desktop\6.tiff

Fig. 1 Velocity vector at different impeller diameter at part load (Q=8.35 lps)