

# A Three-day Workshop on

**Harnessing Computational Fluid Dynamics in the Energy Sector : Techniques and Applications**

**(H\_CFD-2025)**

**(Hands on training in Hybrid mode)**

**23<sup>rd</sup> to 25<sup>th</sup> January, 2025**



VISAKHAPATNAM  
विद्या प्रशस्यते लोकेः

**ANSYS**

**CADFEM**

**Organized by**

**Department of Mechanical Engineering**

Indian Institute of Petroleum & Energy

Visakhapatnam - 530 003

Andhra Pradesh, India.

## About the Institute

Indian Institute of Petroleum and Energy (IIPE) is an autonomous institute under the Ministry of Petroleum & Natural Gas, Govt. of India, and is backed by public sector Oil and Gas giants viz. HPCL, IOCL, ONGC, GAIL, and OIL. The objective of establishing the Indian Institute of Petroleum and Energy (IIPE) is to meet the quantitative and qualitative gaps and to supply skilled manpower for Petroleum Sector. The vision of IIPE is to be an Energy University that is ranked among top 200 Universities of the world within 20 years of setup. The mission is to create and disseminate knowledge that facilitates progress and excellence in the field of Energy sector.

## About the Department

The key focus of the Mechanical Engineering department at IIPE is to impart theoretical and practical knowledge to students in the area of oil, gas, and energy sector. The department offers B. Tech and Ph. D programs (Full time, Part time & QIP) with focus on Renewable energy, Energy Storage, Desalination, Heat Transfer, Multiphase flow, Energy Conversion Devices, Turbomachines, Grid Integrated Systems, Data Analytics and AI for Energy Sector. The vision of the department is to be a globally recognized center of excellence in generating well-qualified mechanical engineers with expertise in oil, gas and energy sectors. The mission is to provide basic and advanced mechanical engineering education to the graduates through upgraded teaching-learning process using advanced tools.

## About the Workshop

Computational Fluid Dynamics (CFD) plays a crucial role in the energy and petroleum sectors by simulating fluid flow, heat transfer, and chemical reactions in various processes. The primary intend of this workshop is to provide a hands-on training on various aspects of Computational Fluid Dynamics (CFD) using ANSYS Simulation tool. The workshop can be an excellent opportunity for the students, researchers and professionals to deepen their understanding of Thermo-fluid behavior in complex systems.

## Workshop Themes

- ❖ **Geometry preparation using ANSYS Design modeler & space claim.**
- ❖ **Introduction to Fluent meshing & Fluent solver.**
- ❖ **Battery Thermal Management system.**
- ❖ **Boiling, Condensation, Melting & solidification**
- ❖ **Fuel Cell Modeling**
- ❖ **ANSYS Mechanical (Fluid-structure interaction)**
- ❖ **Topology Optimization**
- ❖ **Film cooling in Gas-turbine.**
- ❖ **Multi-phase flow in power plants and Petroleum sectors.**

