CASE STUDY /

Ansys + IIT Bombay

"Ansys Fluent enabled students of the Computational Fluid Dynamics & Heat Transfer course to quickly map the fundamentals of computational fluid dynamics (CFD) to a project analyzing ventilation in various labs and classrooms on campus. The Ansys Student free download provided students with easy access and the ability to download Ansys Fluent on their laptops, enabling them to ideate solutions on the go and successfully meet the learning goals of the course in a completely online mode. This hands-on experience not only helped students better understand foundational CFD concepts, but it also equipped them with the software skills highly sought after in industry."

Prof. Janani Srree Murallidharan
Assistant Professor / Mechanical Engineering Dept., IIT Bombay

CASE STUDY

Fourth-year Engineering Students Use Ansys Fluent to Analyze Ventilation in Shared Learning Spaces on Campus

Fluid and thermal engineers capable of tackling a wide range of complex, real-life problems are in high demand by industry. At the Indian Institute of Technology (IIT) Bombay, we created a course that provides students with Ansys Fluent experience through a semester-long project. Given the COVID-19 pandemic, we asked students to analyze ventilation in shared, on-campus learning spaces and suggest changes to improve safety. This project enabled students to synthesize abstract mathematical concepts of CFD and learn how advanced versions of these concepts apply to practical problems.

/ Challenges

- Enable students to solve complex problems that build on the foundational CFD theory they've learned in the course.
- Expose students to the robust numerical capabilities present in the background of a sophisticated, industry standard CFD tool.
- Provide students with a platform that will help them quickly ideate solutions for flow problems and assess their impact in a relatively short turn around time.
- Provide students with an easy-to-use CFD tool that can be easily accessed while taking a fully remote, online course.

/ Ansys Products Used:

- Ansys Fluent

/ Engineering Solution

The course required students to work on a socially relevant project and suggest solutions to a problem. With a view of bringing students back on campus after the COVID-19 lockdown, students evaluated the ventilation in various labs and classrooms on campus. They were required to model and simulate the flow field within a room and estimate how frequently the air inside the room needs to be circulated to minimize infection spread. Fluent's robust meshing and simulation capabilities enabled students to mesh large rooms efficiently and easily incorporate various flow effects such as ceiling fans, exhaust fans, air turbulence parameters, etc. The user-defined scalar (UDS) capability of Fluent was crucial for predicting the residence time of air in various parts of the room.

/ Benefits

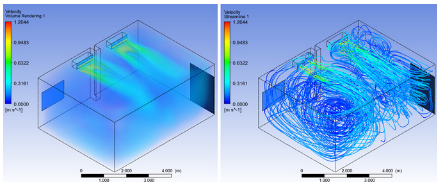
By trying out different simulation options available in Fluent and working with add-on numerical capabilities such as UDS, students applied the theory they learned to a socially relevant problem. Students also gained a better understanding of the importance of ventilating dead zones to limit infection spread by seeing flow visualized in the rooms using Ansys Fluent. Providing students with this hands-on experience not only helped them understand foundational CFD theory, but also equipped them with skills highly sought after in industry.

/ Company Description

Prof. Janani Srree Murallidharan is an assistant professor in the Department of Mechanical Engineering and associate faculty in the Interdisciplinary Programme in Educational Technology at IIT Bombay. Prof. Murallidharan covers fluid dynamics topics including the essentials of fluid dynamics and heat transfer in CFD, essentials of numerical methods in CFD, computational heat conduction, heat convection, physical law-based finite volume method, and computational fluid dynamics on a staggered grid.

Prof. Janani Srree Murallidharan, assistant professor, Department of Mechanical Engineering, IIT Bombay

Indian Institute of Technology Bombay



CFD simulation of flow field inside the Thermal Fluids Engineering Lab at IIT Bombay
(a) Flow circulation patterns within the room
(b) Velocity distribution within the room

ANSYS, Inc.
www.ansys.com
ansysinfo@ansys.com
866.267.9724