Mechanical Engineering Department

TRAINING PROGRAM REPORT

- Title: Introduction to CFD
- Name of Resource Person: Prof. S. M. Gaikwad, Army Institute of Technology
- Name of Event Coordinator**:** Prof. M. N. Pohare
- Date and Time: 06/08/2021 10:30 AM To 12:30 PM
- Venue: Online Google Meet Platform and YouTube live
- Total Number of participants: 28

What is CFD?

Computational Fluid Dynamics (CFD) is the analysis of fluid flows using numerical solution methods. Using CFD, you are able to analyse complex problems involving fluid-fluid, fluid-solid or fluid-gas interaction. Engineering fields where CFD analyses are frequently used are for example aerodynamics and hydrodynamics, where quantities such as lift and drag or field properties as pressures and velocities are obtained. Fluid dynamics is involved with physical laws in the form of partial differential equations. Sophisticated CFD solvers transform these laws into algebraical equations and are able to efficiently solve these equations numerically.

How does CFD work?

A CFD analysis basically consists of the following three phases:

Pre-processing

In this phase the problem statement is transformed into an idealized and discretized computer model. Assumptions are made concerning the type of flow to be modelled (viscous/inviscid, compressible/incompressible, steady/non steady). Other

processes involved are mesh generation and application of initial- and boundary conditions.

Solving

The actual computations are performed by the solver, and in this solving phase computational power is required. There are multiple solvers available, varying in efficiency and capability of solving certain physical phenomena.

Post-processing

Finally, the obtained results are visualized and analyzed in the post processing phase. At this stage the analyst can verify the results and conclusions can be drawn based on the obtained results. Ways of presenting the obtained results are for example static or moving pictures, graphs or tables.

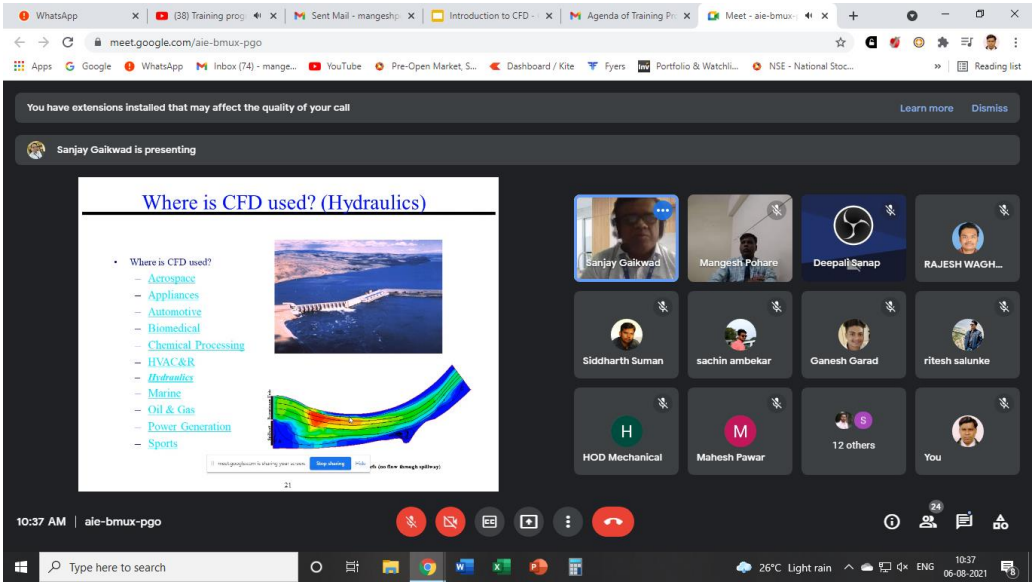
Ready to start your CFD analysis?

We hope you found this article to be both informative and useful. If you like to read more on the subject, we recommend to read our article: Bottlenecks in CFD analysis. Should you have any questions or do you want to know more about Femto Engineering's CFD projects, please contact us.

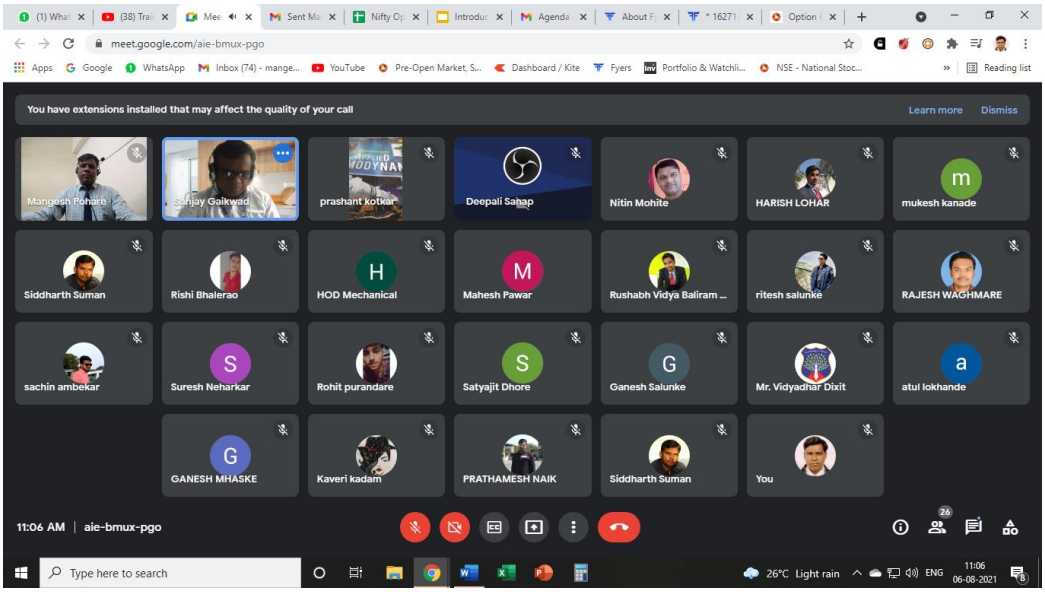
Outcome: After the end of this session students are_

- Able to create modal geometry and apply Ansys tools.
- Able to operate advanced futures of Ansys.
- Able to validate experiments results.

Photograph No: 01 Screenshot of slide presentation



Photograph No: 02 Screenshot of Google Meet attendance



Prof. M. N. Pohare	Prof. R. Y. Daspute	Dr. S. N. Mali
Event Coordinator	HOD Mechanical	Principal