

Five-Day Skill/Faculty Development Workshop titled “**Applied CFD and Multiphysics Modelling – Hands-on with ANSYS Fluent**”, to be conducted in **online mode** during **27–31 January 2026**.

For registration, use fill the Google form using the link given below:

<https://forms.gle/G7UqFV2GG19HwCJA8>