LSM Seminar

Thursday 18 April 2024 15:00 OFLG/402

"Introducing Splash-FOAM: A Dynamic CFD GUI Tool"

By Dr. Mohamed Sayed, (Simultica CFD Solution London)

In the realm of computational fluid dynamics (CFD), the Paul Scherrer Institute is renowned as one of the leading research institutes. As a former employee of PSI, I am excited to introduce an exciting initiative within the division of Scientific Computing, Theory, and Data – SplashFOAM

SplashFOAM represents a new approach, offering a user-friendly Graphical User Interface (GUI) tailored specifically for the renowned open-source CFD code, OpenFOAM®. It provides enhanced accessibility, flexibility, and cost-effectiveness in CFD simulations.

In this seminar, we will explore how SplashFOAM complements commercial packages like ANSYS Fluent, rather than replacing them. The objective is to empower the next generation of PSI students by bridging the gap between academia and the ever-changing industry demands.

Key features of SplashFOAM include an intuitive interface, robust solver capabilities, efficient meshing tools, and seamless integration with OpenFOAM. These features streamline complex workflows, making CFD simulations more accessible and efficient for both novices and experts. Moreover, this seminar will discuss the vision of establishing PSI's in-house version of SplashFOAM. This initiative reduces operational costs and fosters a collaborative environment wherein academics can tailor CFD simulations to their specific research needs.

Join me in this seminar as we explore what SplashFOAM can offer, empowering researchers and students alike to push the boundaries of fluid dynamics research.