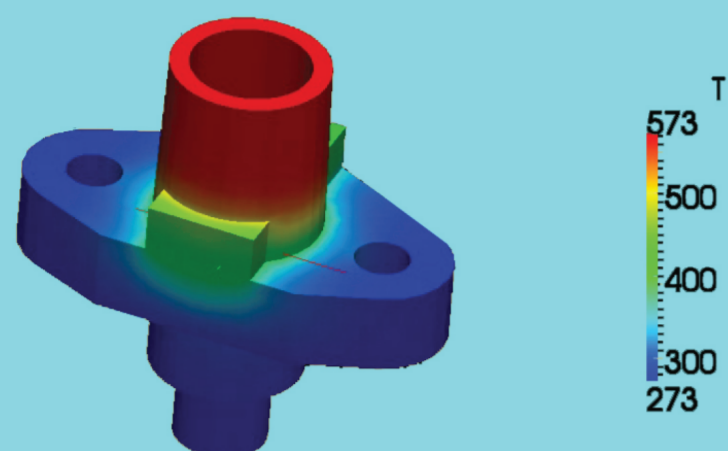


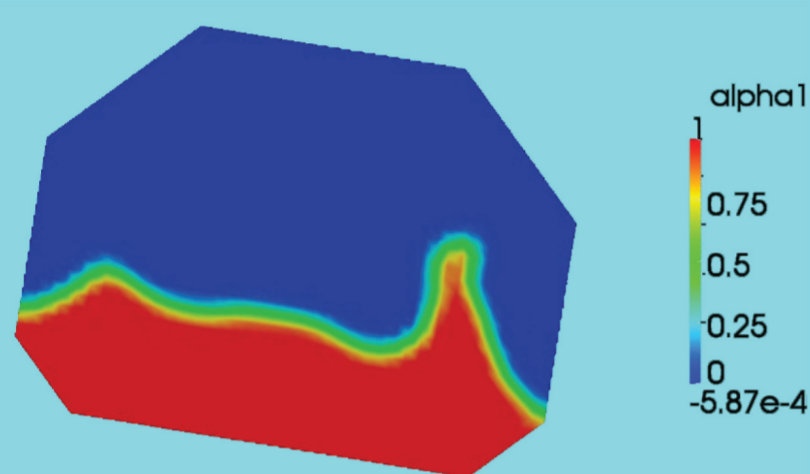


OpenFOAM: Free and Open Source alternative to Fluent

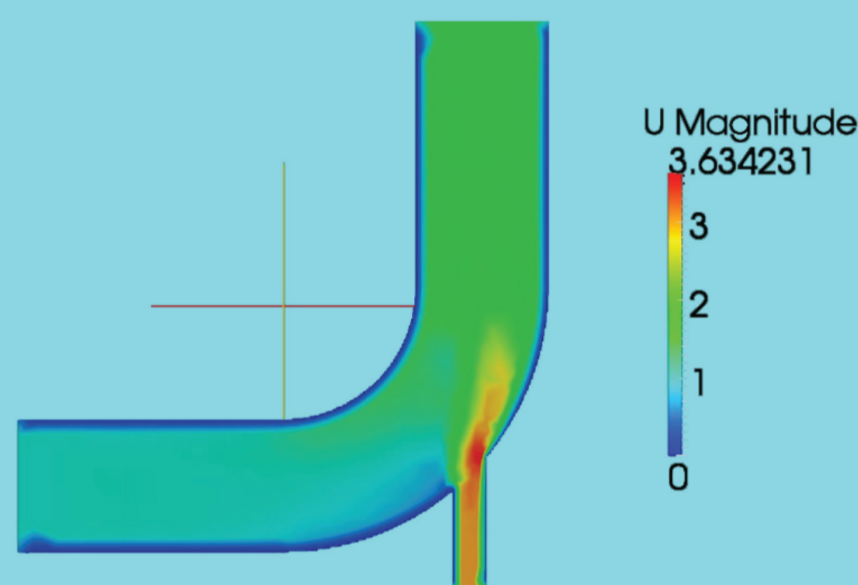
<http://cfd.fossee.in>



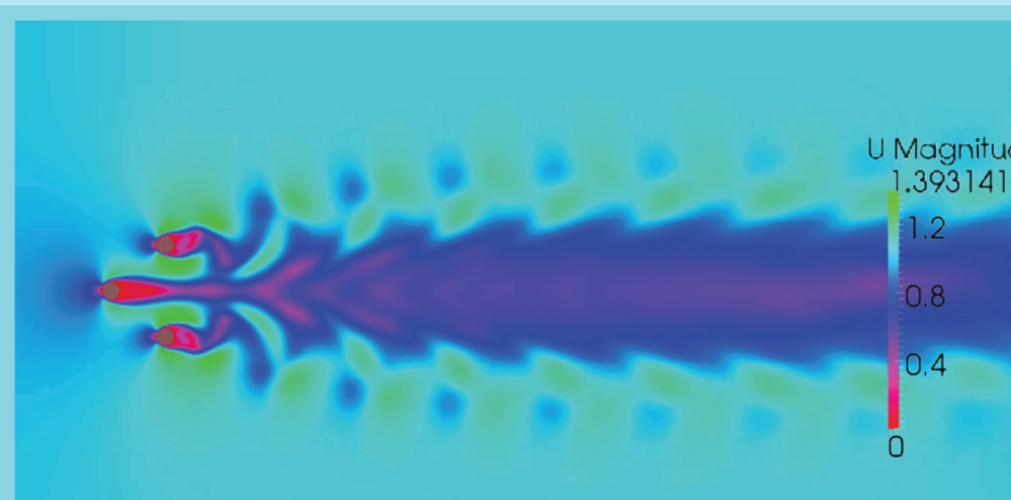
Temperature distribution over a heated flange



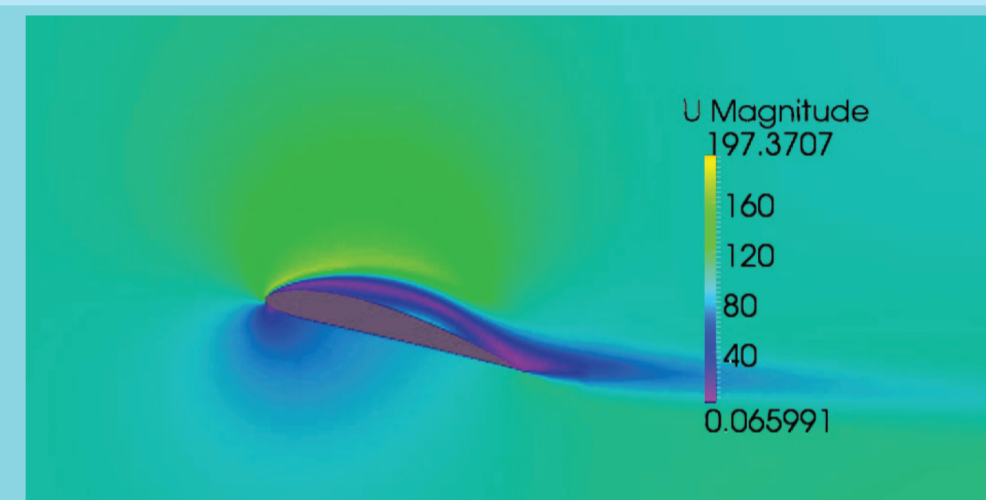
Water sloshing in a tank



Fluid mixing in a D-elbow



Flow over bluff bodies in a V configuration



Flow over an airfoil

Supports popular mesh file formats

Friendly syntax for PDEs

Wide range of solvers

Supports OpenMPI

Export results to different viewing formats

SELF Workshops: We conduct Spoken Tutorial based workshops on OpenFOAM free of cost

Applications

Incompressible Flows	MHD Flows	Financial Problems
Compressible Flows	DSMC	Particle Tracking
Multiphase Flows	Combustion	Heat Transfer

We help migrate to OpenFOAM free of cost through Spoken Tutorials, Lab Migration & Textbook Companion

Contact Us:

General Queries: contact_cfd@fossee.in

Text Book Companion: cfd_tbc@fossee.in



IIT Bombay

Funded by

National Mission on Education through ICT, MHRD
<http://sakshat.ac.in>



This effort is not approved or endorsed by ESI Group, the producer of OpenFOAM® software and owner of OpenFOAM® trademark.