Special Issue on Computational Fluid Dynamics

Call for Papers

Computational fluid dynamics (CFD) is the use of applied mathematics, physics and computational software to visualize how a gas or liquid flows -- as well as how the gas or liquid affects objects as it flows past. Computational fluid dynamics is based on the Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related.

In this special issue, we intend to invite front-line researchers and authors to submit original research and review articles on exploring **Computational Fluid Dynamics**. Potential topics include, but are not limited to:

- Turbulence
- Two-phase flows
- Heat transfer
- Chemical reactions and combustion
- Acoustics
- Unsteady flows
- Free-surfaces
- Fluid-solid interaction
- Navier-Stokes solution techniques
- Discretisation methods and schemes
- Convergence acceleration procedures
- Grid generation and adaptation techniques
- Mesh-free methods
- Distributed computing
- CFD simulation and analysis

Authors should read over the journal's For Authors carefully before submission. Prospective authors should submit an electronic copy of their complete manuscript through the journal's Paper Submission System.

Please kindly specify the "**Special Issue**" under your manuscript title. The research field "**Special Issue - *Computational Fluid Dynamics***" should be selected during your submission.

Special Issue Timetable:

Submission Deadline	December 30th, 2016
Publication Date	March 2017

Guest Editor:

For further questions or inquiries, please contact Editorial Assistant at
ojfd@scirp.org.