

CERN openlab for CFD Applications



www.cern.ch/cfd

The term **Computational Fluid Dynamics (CFD)** covers all aspects of computational techniques that can be applied to the solution of problems involving fluids and related thermal phenomena. CFD studies are not limited to engineering. They cover fields such as weather forecasting, medical, chemical and environmental applications. In engineering, CFD is primarily used as a design support for predicting the performances of equipment involving fluid flow and heat transfer. The possibility to simulate heat transfer and fluid flow problems numerically before or instead building a prototype, cuts drastically the development cost and time of a project.

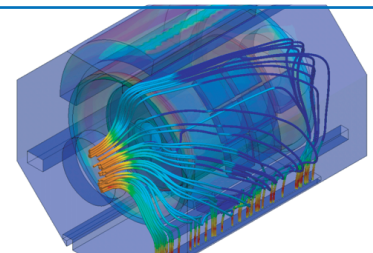
At CERN, a parallel commercial CFD code is used to provide assistance to the LHC machine and detectors during the prototype, design and installation phases of their components. Most investigations involve prediction of velocity and temperature fields in natural convection environments. Since May 2005, the CFD Team at CERN can rely on the high performances of the openlab cluster to perform compute-intensive computational fluid dynamics simulations.

Availability of the openlab cluster increased up to eight times the performance of the CFD simulations allowing to reduce the delivery time and the accuracy of the studies.



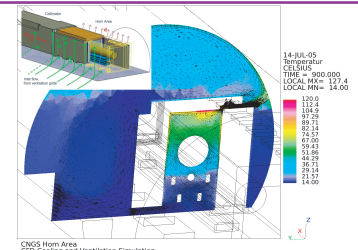
ALICE L3 MAGNET

Transient 3D
~500k cells
3 days



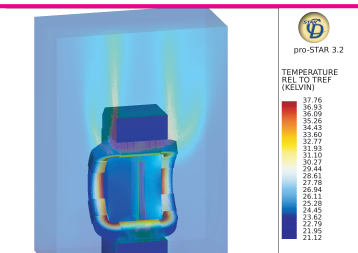
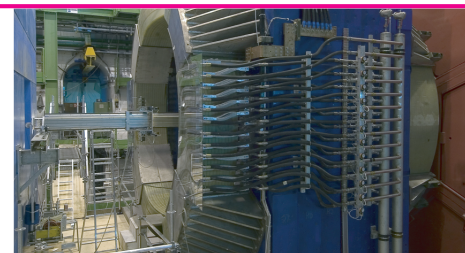
CNGS HORN

Transient 3D
~1000k cells
1 month



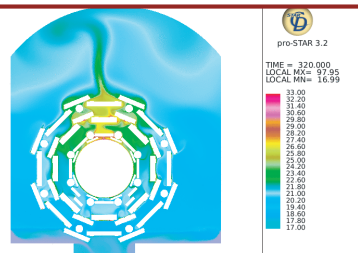
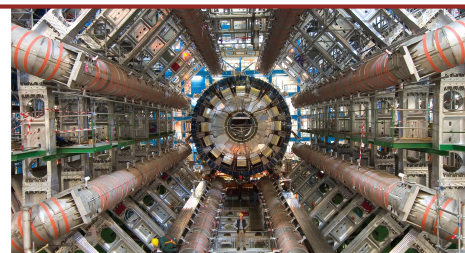
ALICE MUON

Transient 3D
Radiation
~400k cells
2 months



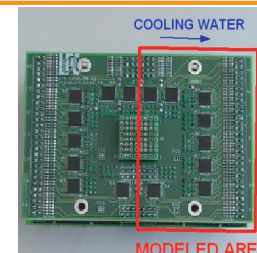
ATLAS CAVERN

Transient 2D
~250k cells
1 week



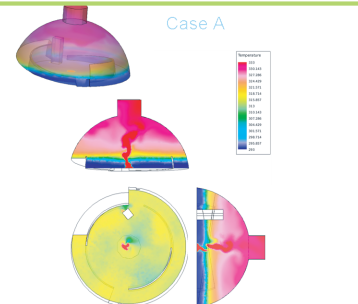
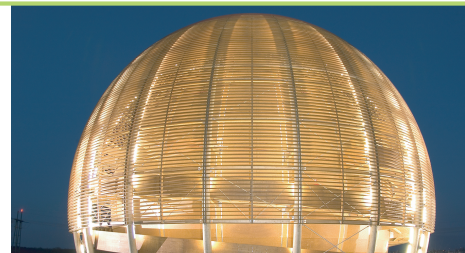
LHCb PS VFE Board

Transient 3D
~500k cells
1 week



GLOBE

Transient 3D
~500k cells
2 weeks



The CFD Team is part of the Detector Cooling (DC) Section of the Cooling and Ventilation (CV) Group in the Technical Support (TS) Department at CERN. Our mission is to provide engineers, physicists and scientists working on all CERN units, with flow and

thermal analysis using CFD tools and methods to solve fluid flow and heat transfer related problems.
www.cern.ch/cfd
cfd-team@cern.ch

European Organization for Nuclear Research

