Azore® Software, LLC

Azore® CFD Software

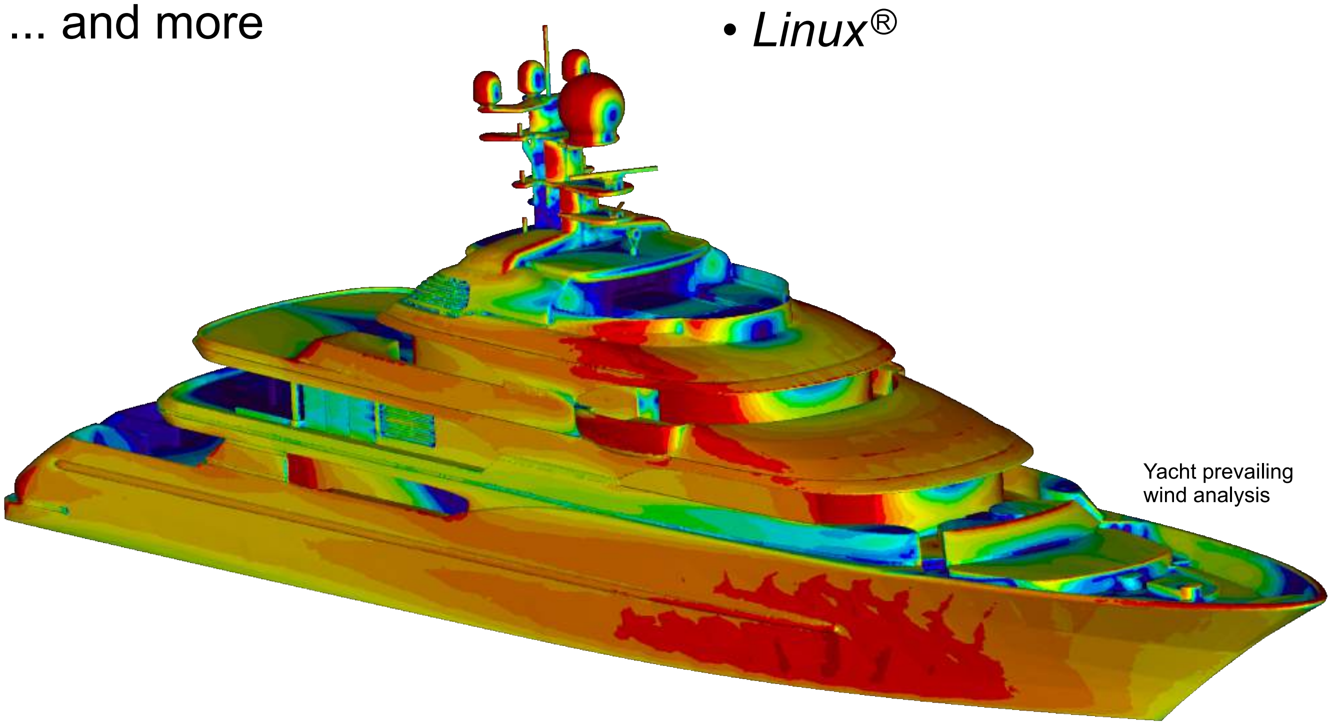
- Industrial flows
- Aerodynamics
- Heat transfer
- Conjugate heat transfer
- Two-phase flow
- Gas species mixing
- Particulate tracking

... and more

A cost-effective CFD tool ...
Just import your mesh
and run your analyses in parallel

Distributed for

- *Windows®*
- *MacOS®*
- *Linux®*



Yacht prevailing wind analysis

Used to design equipment and optimize flow systems worldwide since 2007:

Energy:
Boiler combustion optimization
Erosion/corrosion reduction
Heat transfer enhancement
Wind turbine design

Aerodynamics:
Automobiles and ships
Building and radar antennas
Rail car drag and tipover
Underwater research vehicles

Manufacturing:
Building materials and insulation
Food processing equipment
Mixing systems
Plant HVAC systems

Pollution Control:
Particulate capture (ESP, baghouse)
SCR, scrubber design
Sorbent / activated carbon injection
Stack flow optimization

HVAC:
Ductwork design
Exhaust systems/ventilation
Heating and cooling
Fan design and sizing

www.azorecfd.com

Azore® Software, LLC

FAQ

Azore® is a CFD software tool that utilizes the control volume method for discretization of the mesh topology. Traditional mesh topologies, as well as arbitrary polyhedral mesh topologies, are acceptable inputs to the Azore solver.

A graphical user interface (GUI) is provided for preparation of simulation input and post processing of simulation results.

Solver key capabilities:

- Incompressible fluid treatment
- Steady state
- Continuity/Momentum coupling
- Support for the solution of the energy equation
- Support for species transport with an unlimited number of species
- Standard K-epsilon with standard wall function treatment
- Variable fluid properties (density, Cp, viscosity)
- Thin surface porous resistance
- Volumetric porous resistance
- Multiple reference frame support for rotating machinery
- Thin surface heat transfer

Post processor:

- Built-in post-processor
- Export capability to ANSYS® Fluent® or VTK formats

msh format import from:

- Trelis™ Meshing
- ANSYS® Fluent® (also case files)
- Gridgen®/Pointwise®
- TrueGrid®
- GridPro®
- Spider
- CUBIT
- CastNet
- ANSYS® Meshing™

Other mesh file formats upon request

Licensing options:

- Node-locked license
- Floating network license
- Full version trial license

(parallel licenses included)

Free Trial!

Correlation Study:
Coiled Tube and Shell Heat Exchanger

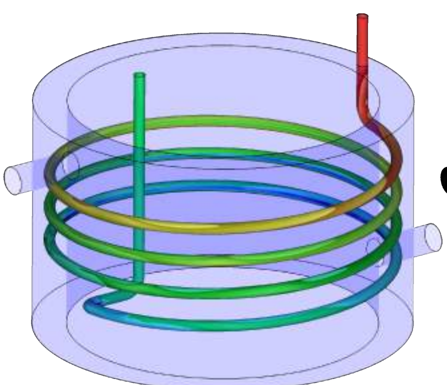


Figure 1

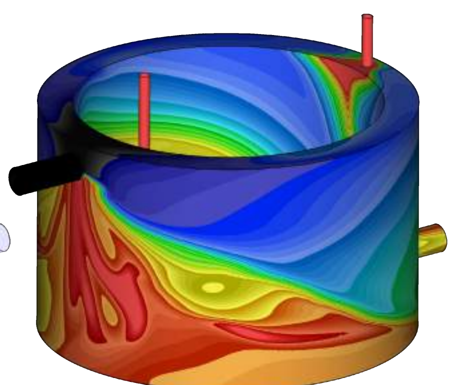


Figure 2

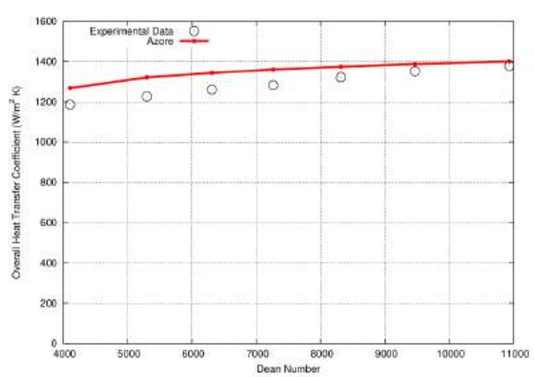


Figure 3

This CFD simulation is inspired by the experimental work of J.S. Jayakumar Et al.[1] This is a particularly complex simulation that includes predictions of heat transfer between two flow paths; one inside the coiled tube (Figure 1) and one inside the shell (Figure 2). Although the fluid flow paths are separate, energy passes from the hot coiled tube fluid to the cool shell fluid via convection and conduction through the tube wall.

The line graph (Figure 3) summaries simulation predictions for a series of flow rates (Dean Numbers) along with the experimental data collected.

[1]"Experimental and CFD estimations of heat transfer in helically coiled heat exchangers" Chemical Engineering Research and Design (2008) Volume 86 pages 221-232

Used to design equipment and optimize flow systems worldwide since 2007

www.azorecfd.com