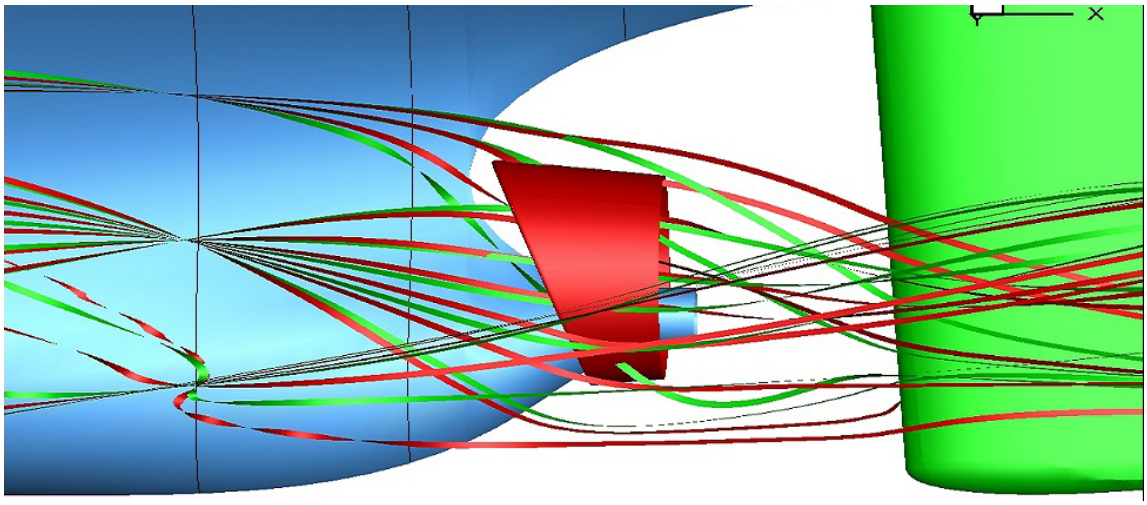

CFD - Computational Fluid Dynamics

Plan, make optimal basic form, create drawings.

The use of CFD in ship design has become standard procedure when developing and ranking new hull designs. Model tests are still superior when quantifying absolute values of resistance, propulsive power, and trim, but CFD can offer flow details not available in standard model tests. The flow around the hull (and many other other applications) can be visualised, used to identify possible problems and to find solutions.

SSPA mainly uses two CFD solvers. They are SHIPFLOW and FLUENT. SHIPFLOW is developed by FLOWTECH International AB [1], a company within the SSPA group. It is marketed worldwide, and is comprised of several different solvers (potential flow, thin boundary layer and RANS). SHIPFLOW is designed to be very efficient for simulating flows that can be considered standard flows around a hull shape. For simulations where this is not the case, SSPA often uses FLUENT. These types of flows could be non-stationary flows, multi-phase or species transport flows, Volume Of Fluid (VOF), internal flow, and many others.

Beyond the two main solvers, SSPA runs several support packages. They are ICEMhexa (grid generation), SPOST and TECPLOT (postprocessors) and mPuff (propeller wake). Together with SSPA staff CFD experience and powerfull calculation servers these tools offers solutions to almost any incompressible flow imaginable. Either as standalone simulations or as a suplement to model tests.

The lists below is an extract of recent projects at SSPA where CFD has been used.

CFD projects involving SHIPFLOW

- Hull design optimisation (parametric and manual)
- Multi-hull positioning
- Shallow water investigation
- Pressure signature
- Lifting device analysis
- Appendage positioning and design
- Wake calculations with and without appendages

CFD projects involving FLUENT

- Submarine flow
- Propeller flow (open water, inclined flow, cavitation inception, bottom erosion effects)
- Appendage flow
- Squat effect simulation
- Wind simulations around superstructures or external atmospheric boundary layers
- Release from multiple outlets of FPSO (multiphase or species transport)
- Exhaust gas simulations
- Oil flow simulation in gas turbine under motion
- Free surface simulation in harbours
- Oil scooping device
- Internal flow in general