Cavitation is a common problem in pumps, inducers, hydraulic turbines, propellers, fuel injectors and other fluid devices subjected to low pressures. Cavitation often causes performance breakdown and damage, which can be costly.

Because ANSYS CFX software has long been a technology leader in computational fluid dynamics (CFD) for rotating machinery, engineers and designers now look to ANSYS CFX to understand and reduce cavitation. The ANSYS CFX product delivers a robust, well-validated cavitation model to further solidify technical leadership in CFD analysis of rotating machinery.
The Cavitation Model
Cavitation is a phase change process. A usual starting point for analyzing cavitation is the Rayleigh-Plesset equation, which describes the growth and collapse of a vapor bubble subjected to a far-field pressure disturbance. Using this equation, it is possible to develop a relationship for the cavitation rate in terms of the fluid properties and the difference between the local pressure and vapor pressure. The particular model implemented in ANSYS CFX software has been tuned to model both the vaporization and condensation stages of cavitation, and it has been validated for a number of working fluids including water and diesel fuel. Using the powerful CFX Expression Language™ feature, users also may implement custom cavitation models.

Numerical Implementation
Cavitation can pose significant numerical challenges to a CFD code because of the large density ratio between the liquid and vapor. ANSYS CFX software has long been a leader in the simulation of multiphase flows so it is uniquely suited to address these challenges. The cavitation model is implemented as a homogeneous multiphase model, in which a zero slip velocity between liquid and vapor is assumed. The global continuity equation is expressed in volumetric form to minimize robustness problems at the liquid-vapor interface. With this formulation, the cavitation source appears as a volume source term that can be linearized in terms of pressure; the resulting equation set can be solved efficiently using ANSYS CFX software’s unique implicit coupled multigrid solver.

Comparison of experimental and computed surface pressure coefficients for midchord cavitation on hydrofoil for three cavitation numbers.