

# SC/Tetra - CFD in Perfect Balance

## Ease of Model Set-up, Performance and Accuracy

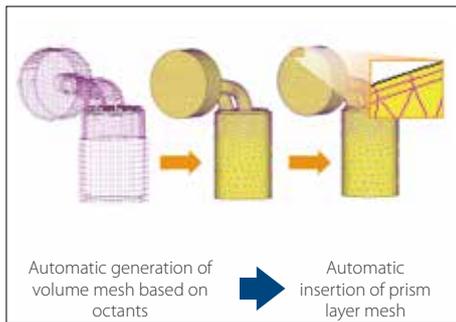
SC/Tetra efficiently solves a wide range of challenging CFD problems, leaving you more time for innovation

# SC/Tetra

SC/Tetra is general purpose thermo-fluid simulation software which uses a hybrid mesh to represent the surface shape with high accuracy. It features a sophisticated mesh generation system, low computational times, low memory consumption, and user-friendly operation. You can obtain highly accurate CFD results faster.

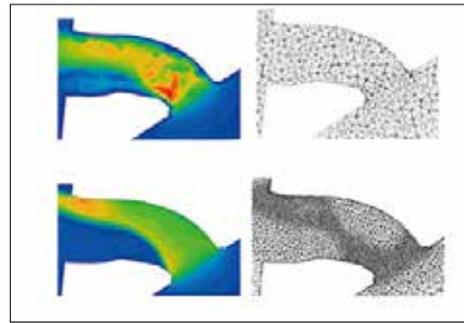
### High-speed Automated Meshing

SC/Tetra automatically generates a high quality mesh quickly, using an octree and Advancing Front method. Prism layer elements, providing high simulation accuracy, can be created automatically with the right layer thickness. The software leverages parallel computing for even faster mesh generation and supports large scale meshes.



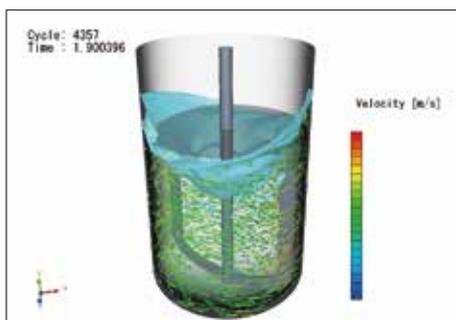
### Automatic Mesh Refinement

With this function in a steady-state analysis, a mesh is automatically refined in areas where the flow or pressure changes greatly. The function greatly simplifies flows in tubes with complex shapes.



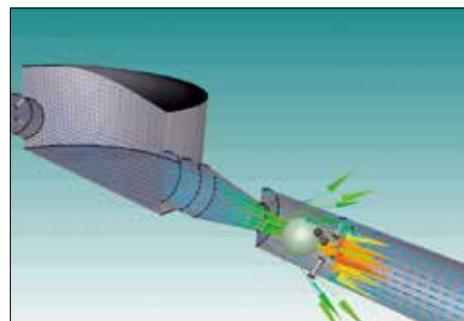
### Free Surface Flows

SC/Tetra accurately simulates free surface flows - the interface between a gas and a liquid. The unique Interface Capturing Method enables high-speed simulation and can be used with other functions such as moving boundary, overset mesh, and particle tracking. Applications include simulating the effect of waves on a ship, the effect of vibration on a gasoline tank, and similar analyses.



### Dynamics

Translation and rotation of a rigid object caused by fluid forces can easily be simulated with SC/Tetra's Dynamics capabilities. Applications include ball valves, wind power generators, and blades of wave power generators.

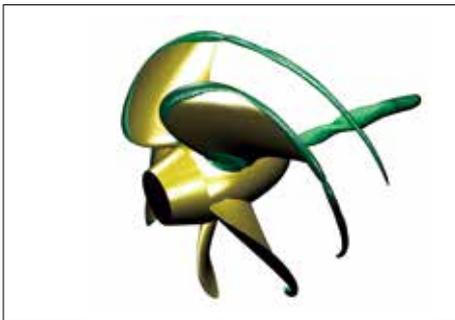


## Unstructured mesh (Tetrahedral, pentahedral, and hexahedral elements)

- Aerodynamic simulation for automobiles
- Evaluation of rotational devices such as fans and pumps
- Prediction of cavitation and erosion
- Design of household electric appliances such as refrigerators and wash machines
- Internal flow analysis of ducts and nozzle
- Analysis involving chemical reactions including reactor, catalyst, furnace, combustor, and CVD
- Multi-phase flow analysis such as mixing, spray, solidification, melting, boiling, and condensation

### Cavitation

This function enables simulation of a vaporization phenomenon called cavitation, which is caused at an area where pressure of a liquid becomes lower than in the surrounding area, such as with a propeller rotating at a high speed under water. The occurrence of cavitation can be predicted by applying the cavitation model based on the pressure values. The software also supports problems caused by cavitation such as erosion.



### Overset mesh

By overlapping mesh for a stationary region with mesh for a moving region, a motion which was not able to be simulated with standard CFD tools can be easily simulated. In addition, contact of objects and overlap of multiple moving regions are supported. Applications include opening and closing of a valve of an engine port, and a gear pumps.

