Multiphase Flow Module
General CFD Applications

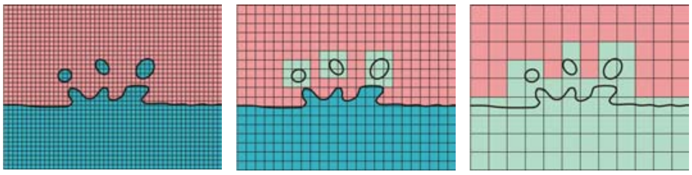

Figure 1.

SWIFT provides three basic models for the simulation of the multiphase flows. The models are classified according to the available grid resolution with respect to the interface length scales, as shown in **Figure 1.**

- Volume-of-Fluid (VOF) is used when there is enough resolution to resolve all interface details.
- Hybrid Model is used when the available resolution can resolve large-scale interface details, while the unresolved small-scale effects are modeled.
- The Multi-Fluid Model is used when the surface details cannot be resolved and the interface interaction terms are modeled.

Summary of the most important features of the SWIFT multi-phase flow module:

- Robust capability to simulate *N*-phase flows.
- Simulations can be performed on unstructured grids in-cluding arbitrary interfaces, sliding, moving boundaries, multiple frames of reference, and mixing planes.
- Arbitrary number of phases can be specified within the momentum equilibrium (homogeneous flow) inside the *N*-phase mixture.
- Capability of the coupling between the multi-fluid and VOF models in the same multiphase flow simulation, resulting in the great flexibility for model development and imple-mentation.
- Possibility to include multiple porosity regions with non-uniform volume fraction distribution.
- Compressibility effects available for multiphase isothermal flows.
- Available cavitation, evaporation/condensation (boiling) models.
- Cavitation, boiling, and VOF models can be used with more than two phases.
- User functions available for the implementation of the customer specific models.

Presented application samples were performed using the VOF and the multi-fluid model.

APPLICATION: Droplet Impact with Water Surface

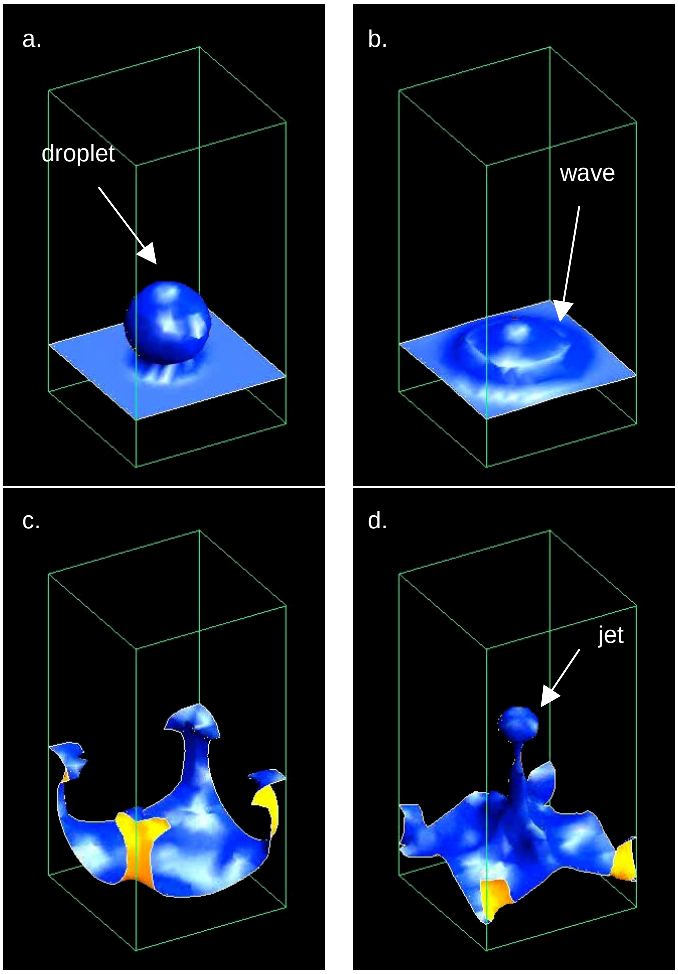

Figure 2.

Simulation results for a single droplet impact with the water surface in a rectangular vessel are shown in **Figure 2.** The calculation was performed using the Volume-of-Fluid (VOF) model. A droplet is placed close to the pool surface and pushed into the water (**a.**). The droplet gets immersed into the water and a circular wave is generated on the surface (**b.**). The wave travels toward the walls and reflects back (**c.**). The re-coiling wave forms a jet in the center (**d.**).

It can be seen that the results obtained with the Volume-of-Fluid model can be very detailed. The method is particularly suitable for the study of fundamental physics of multiphase flows.

Sales & Marketing, Tel.: +43 (316) 787-1851, Fax: +43 (316) 787-1462, Email: ast@avl.com

APPLICATION: Fuel Tank Filling

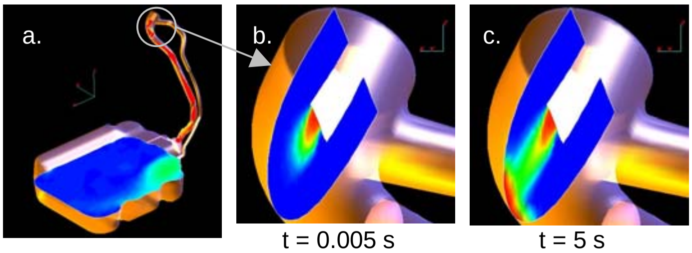

Figure 3.

Fuel tank filling system is shown in **Figure 3.a**. Flow details at two different time instants of the filling process are shown in **Figure 3.b** and **Figure 3.c**. Fuel enters into the pipe through the pistol, hits the opposite wall and spreads around the pipe forming a typical annular-flow regime. Simulations allow the analysis of typical issues of the design interest like early shut-off and the back-splash during the filling process. It is also possible to analyze the filling of the entire tank system and to include the acceleration effects for the sloshing simulation [1].

APPLICATION: Channel Boiling (Water-cooling jacket)

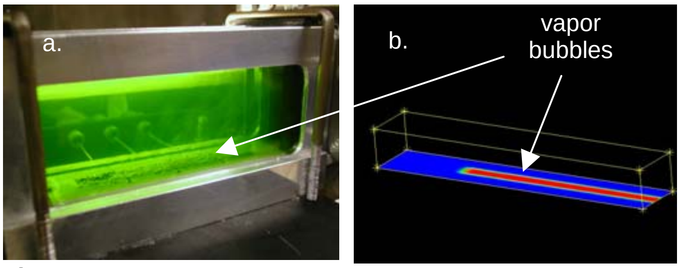

Figure 4.

Boiling is a very efficient heat transfer process. The formation of vapor bubbles off the heated surfaces and the related mixing in the near wall area help enhancing the heat transfer. At the same time, the boiling rate needs to be controlled in order to prevent too fast vapor generation or the formation of vapor pockets. Both can lead to local dry-out which can cause the rupture of the material. This can have catastrophic effects if occurred in the engine water-cooling jacket. **Figure 4.a** shows the measurement of the boiling in the water/glycol mixture in simplified geometry. Formation of the bubbles can be seen in both experiment and simulation, **Figure 4.b**.

APPLICATION: Cylinder Head Quenching

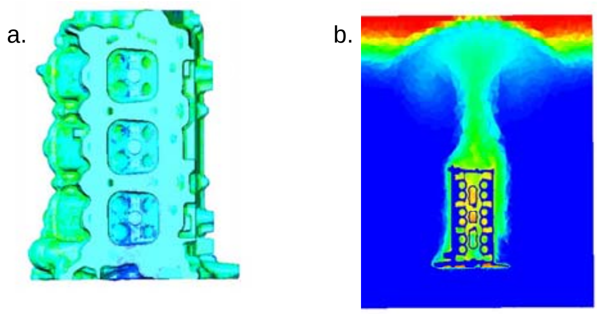

Figure 5.

The process of quenching over the cylinder head is extremely complex [2]. The cylinder head geometry consists of many small details. When the hot cylinder head mold gets immersed into the pool of water, abrupt boiling commences. Boiling produces a very good heat transfer rate between the cylinder head and water and therefore fast cooling of the mold. At the same time, heat transfer distribution on the surface needs to be uniform in order to prevent the formation of internal stresses within the material. These can cause surface cracks and further material failure.

Figure 5 shows the simulation of the quenching process [3]. Calculated surface temperature can be seen in **Figure 5.a**. Vapor generation is very rapid and vapor plum rapidly reaches the water surface, **Figure 5.b**. Simulation allows the analysis of the vapor generation, surface temperature, and heat transfer rate. Quenching simulations can help the optimization of the process.

APPLICATION: Flooding

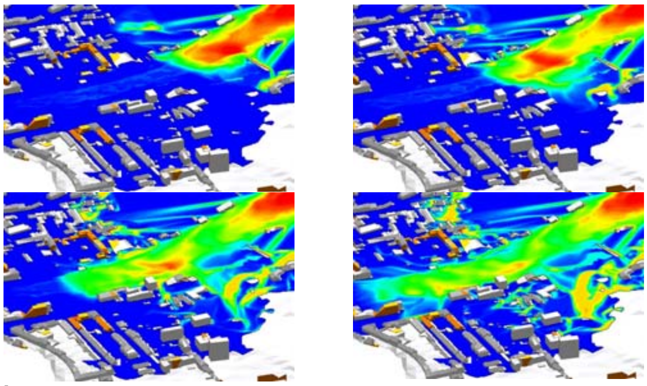

Figure 6.

Figure 6. shows the flooding simulation results obtained by VRVis and AVL [4] using SWIFT. The way in which flood is distributed in a city or the countryside, water height, velocity, and pressure are calculated at each location in 3D space and at each instant in time. The simulation helps to identify endangered areas. It also supports the planning of protective measures, as well as showing the effectiveness of these measures.

REFERENCES

1. Greif, D., Wiesler, B., Alajbegovic, A., "Two-Phase Tank Filling Simulations of an Automobile Tank-System," 2nd MIT Conference on Computational Fluid and Solid Mechanics, Boston, Massachusetts, June 2003.
2. Wang, D.M., Alajbegovic, A., Su, X., Jan, J., "Numerical Modeling of Quench Cooling Using Eulerian Two-Fluid Method," ASME IMECE 2002, New Orleans, Louisiana, November 2002.
3. Wang, D.M., Alajbegovic, A., Su, X., Jan, J., "Numerical Simulation of Water Quenching Process of an Engine Cylinder Head," FEDSM2005-45538, Honolulu, Hawaii, July 2003.
4. Gouda, M., Karner, K., Tatschl, R., "Dam Flooding Simulations Using Advanced CFD Methods," 5th World Congress on Computational Dynamics, Vienna, July 2002.

Sales & Marketing, Tel.: +43 (316) 787-1851, Fax: +43 (316) 787-1462, Email: ast@avl.com