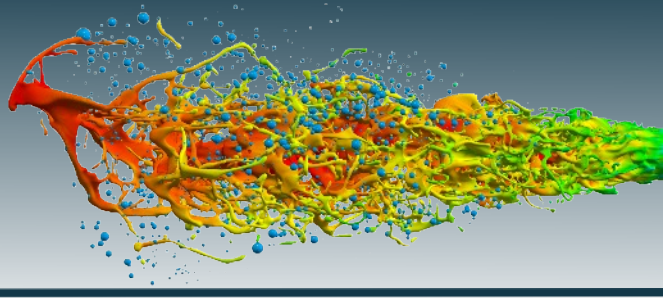
CFD -Tools for Disintegration and Spray Modelling



Solver for disintegration and spray propagation

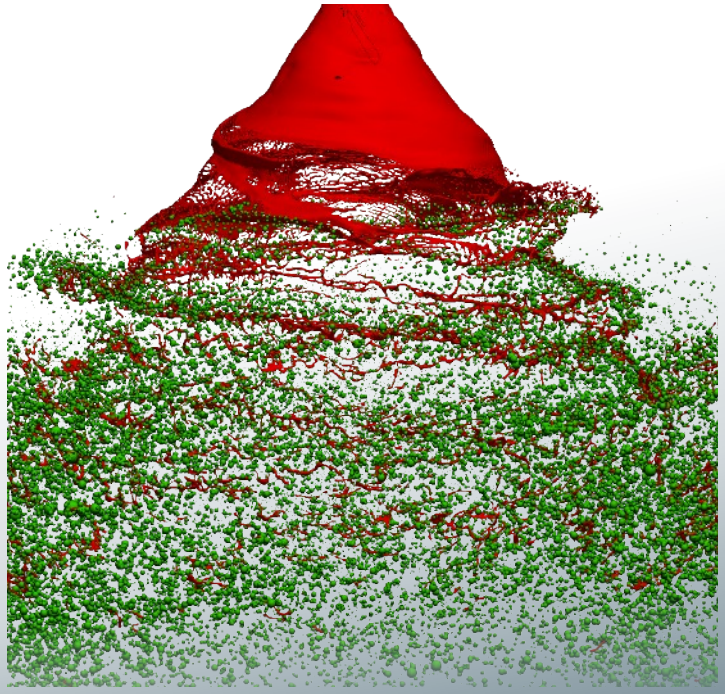
Transition modelling Volume of Fluid (VoF) to Lagrange

DHCAE has extensively extended the computational methods of OpenFOAM to realise a conversion of disintegrating continuous fluid regions (VoF method) into a discrete particle description. This enables a closed modelling of

- General disintegration processes of liquid jets into small droplets
- Atomisation processes
- Spraying processes
- Injection applications

The implementation is characterised by

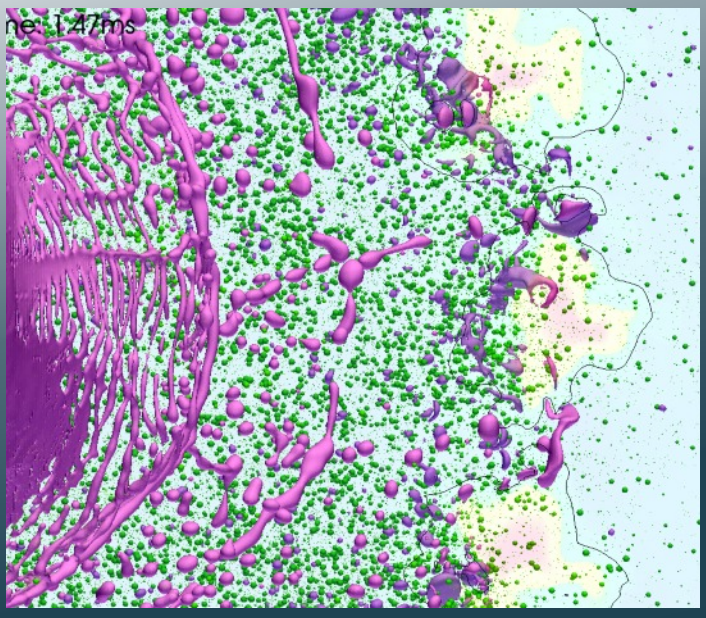
- a high degree of accuracy, e.g. with regard to the transmitted forces or conservation quantities,
- a particularly computationally efficient realisation for grid refinement and particle transport
- a high degree of parallelisation, as well as
- stable, reliable simulation runs with extensive monitoring (e.g. particle sizes, spray angles, etc.).



Transition from VoF areas (red) to Lagrangian particles (green) during atomisation from a swirl nozzle

Initial disintegration-VoF with adaptive meshes

In liquid decay processes, surface waves are first formed starting from an injector. From here larger, mainly separated liquid areas (ligaments) are formed, which then decay into smaller particles. During the initial wave break-up and the formation of ligaments in the liquid, this structure must be precisely resolved in the numerical grid. This is the only way to reproduce the decisive interaction of viscous forces, as well as surface and inertial forces. This is done particularly efficiently with the Volume of Fluid method (VoF) using an adaptive grid refinement around the fluid regions.



Interaction of ligaments and high velocity gas jets during atomisation

Spray Propagation - Lagrangian analysis

Once many small spherical droplets have formed, it is usually computationally impossible to resolve each individual droplet through several grid cells in order to model the spray dispersion. For this purpose, a transition model from VoF to Lagrangian particles was created in order to model the entire process from the disintegration of the liquid to the spray propagation.

Tool integration and validation

The disintegration tool is based on the renowned open-source CFD toolbox OpenFOAM. This guarantees the use of accurate and established simulation methods at low cost and fast simulation performance due to excellent parallelisation methods. The modelling tool was validated with different atomisation benchmarks and a very good agreement was achieved e.g. with measured droplet sizes in the spray.

DHCAE Tools GmbH - Friedrich-Ebert-Str. 368- 47800 Krefeld - Telefon: +49 2151 9490-200
www.dhcae-tools.com email: info@dhcae-tools.de