Industrial simulation of multiphase and moving body flow using OpenFOAM and Visual-CFD

Dr. Nima Abbaspour,
Wagner Group, Innovation and Development engineer, Nima.Abbaspour@wagner-group.com
Raj Kumar Barnawal
ESI-Group, Visual-CFD Product Manager, rbr@esi-group.com
Dr. Sebastien Vilfayeau
ESI-GmbH, CFD Team Leader, svu@esi-group.com

Wagner Holding Inc. is a market leader in advanced paint applicators. It develops and markets a wide array of painting and decorating products used in home improvement, commercial painting and industrial fine finishing. In the innovation and development center of Wagner, CFD is used to predict and improve performance of their paint spray guns, in particular of the pump inside the compact system, to minimize overspray, maximize efficiency, whilst producing a finer finish.

Wagner Group will present a vacuum fan simulation using OpenFOAM® and Visual-CFD for the W550 and W590 spray guns. From these spray guns, paint is dispensed at high volume and low pressure. OpenFOAM® was used to characterize the flow on the fan impeller and to compute the pressure drop for different geometrical designs and working conditions. The fan impeller momentum was modelled using MRF approach, assuming (liquid) incompressible flow, under steady state conditions. Comparison with experimental pressure drop results gave acceptable accuracy for decision-makers.

Later, pattern of the paint spray coming out from the device of nozzle will be investigated with LES and Volume of Fluid (VOF) simulations. Paint atomisation and droplet behaviour along jet will be captured and demonstrated. Various simulations will be done with static and adaptively refined mesh and results will be compared.

While dealing with such industrial applications user faces quite a few challenges, such as appropriate mesh sizing to resolve significant features/gaps and putting appropriate boundary layers as well as the numerical settings guaranteeing stability and robustness without compromising the accuracy and simulation speed. Visual-CFD helps out for applying best settings based on mesh quality and reduce their effort significantly to deliver accurate and time bound results with OpenFOAM®.