

## Validation of open source CFD methods used for automotive defrost simulations

<b>Dr. Ries Bouwman</b> , r.bouwman@iconcf.com <b>DI (FH) Tobias Völker</b> , t.voelker@iconcf.com	Icon Technology & Process Consulting Ltd. SL4 1QN Berkshire, UK
---	--

### Abstract

The main goal of the research was the setup and validation of a coupled simulation method to predict the defrost process inside a car cabin.

Based on a real car geometry, the cleanup CAD data is automatically meshed using the meshing tools developed at the VW Group (VWG, [1]), which are based on snappyHexMesh. The final mesh was used to simulate the acclimatization inside a car using a coupled process based on the open source CFD toolbox OpenFOAM® [2] and the thermal simulation code AURA, used for heat convection, conduction and radiation [3].

In a first step, the flow field inside the car cabin is simulated using the steady state RANS solver, simpleFoam. The second step in the process is to freeze this steady state flow solution and to only simulate the temperature field using AURA. The influence of the temperature changes on the flow field are neglected. The main advantage of the method is the enormous speedup compared to a simultaneous simulation of the flow and temperature field.

In order to validate the RANS simulations, two methods of verification have been applied. First of all, measurements on the real vehicle have been conducted. Second, the code MGLET has been used to simulate a transient flow of the car cabin using a LES solver.

The work clearly shows the applicability of open source CFD tools in an industrial environment. The meshing is robust and provides very high quality meshes. The solvers are fast and accurate. The coupling of different tools makes it possible to obtain results in a timeframe workable for the industry.

OPENFOAM® is a registered trade mark of OpenCFD Limited, the producer of the OpenFOAM software

**Key words:** Climatisation, industrial application, open source CFD, OPENFOAM®, AURA, MGLET, convection, radiation, RANS, LES

### References

- [1] ICON vwgFOAM-1.5-1 Setup and Simulation Guide, London, 2009
- [2] OpenCFD Limited OpenFOAM User Guide, Boston, 2009
- [3] Thümmeler R. AURA Manual Version 1.6.0, CFD Consultants, 2008