RENUDA

2013 *Code_Saturne* User Group Meeting

EDF – R&D
Chatou, France

9th April 2013

Thermal Comfort in Train Passenger Cars

For further information please contact:

Brian ANGEL
Director
RENUDA France

brian.angel@renuda.com

T: +33 6 14 72 31 70

www.renuda.com

Outline

1. Introduction

Introduction

- Over the last decade, ALSTOM Transport has used CFD to verify and optimise numerous designs of trams, metros, TERs and TGVS / AGVS for
  - thermal comfort of passenger cars and driver's cabins
  - internal and external aerodynamic performance

- Current practice for thermal comfort analysis models the entire ventilation system resulting in:
  - parallel calculations to simulate ever larger problems
  - a continuously increasing requirement for computer power

Introduction (2)

- The commercial CFD solver STAR-CCM+ has been used for several years by ALSTOM Transport for their thermal comfort and ventilation ducting simulations

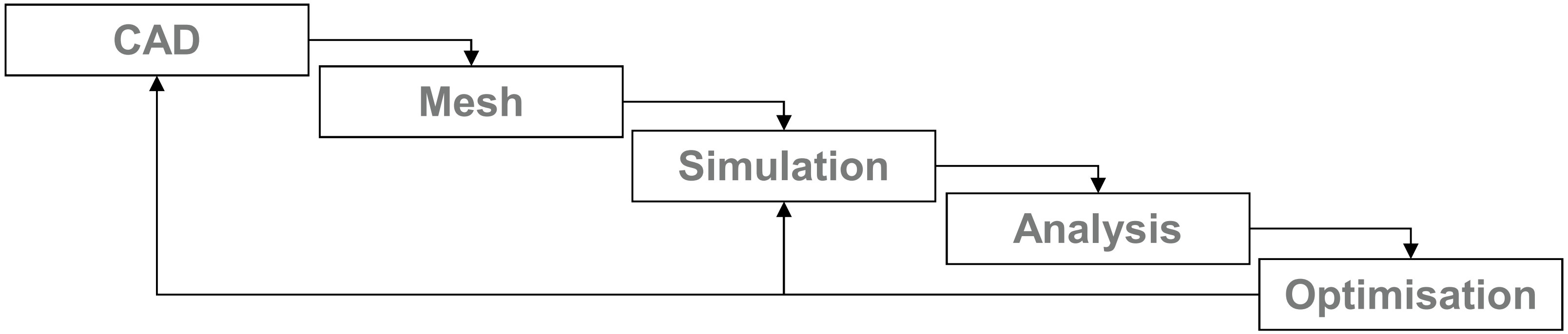
- Increases in licensing costs and the rise in the use of open source CFD in industry has lead ALSTOM Transport to investigate the use of open source CFD calculations chains for thermal comfort applications

- Renuda has worked closely with ALSTOM Transport since 2005 and, in close collaboration with ALSTOM, completed the testing and validation of open source CFD calculation chains for thermal comfort modelling

2. Context

CFD Cycle

- Typical CFD cycle for thermal comfort modelling at ALSTOM Transport:



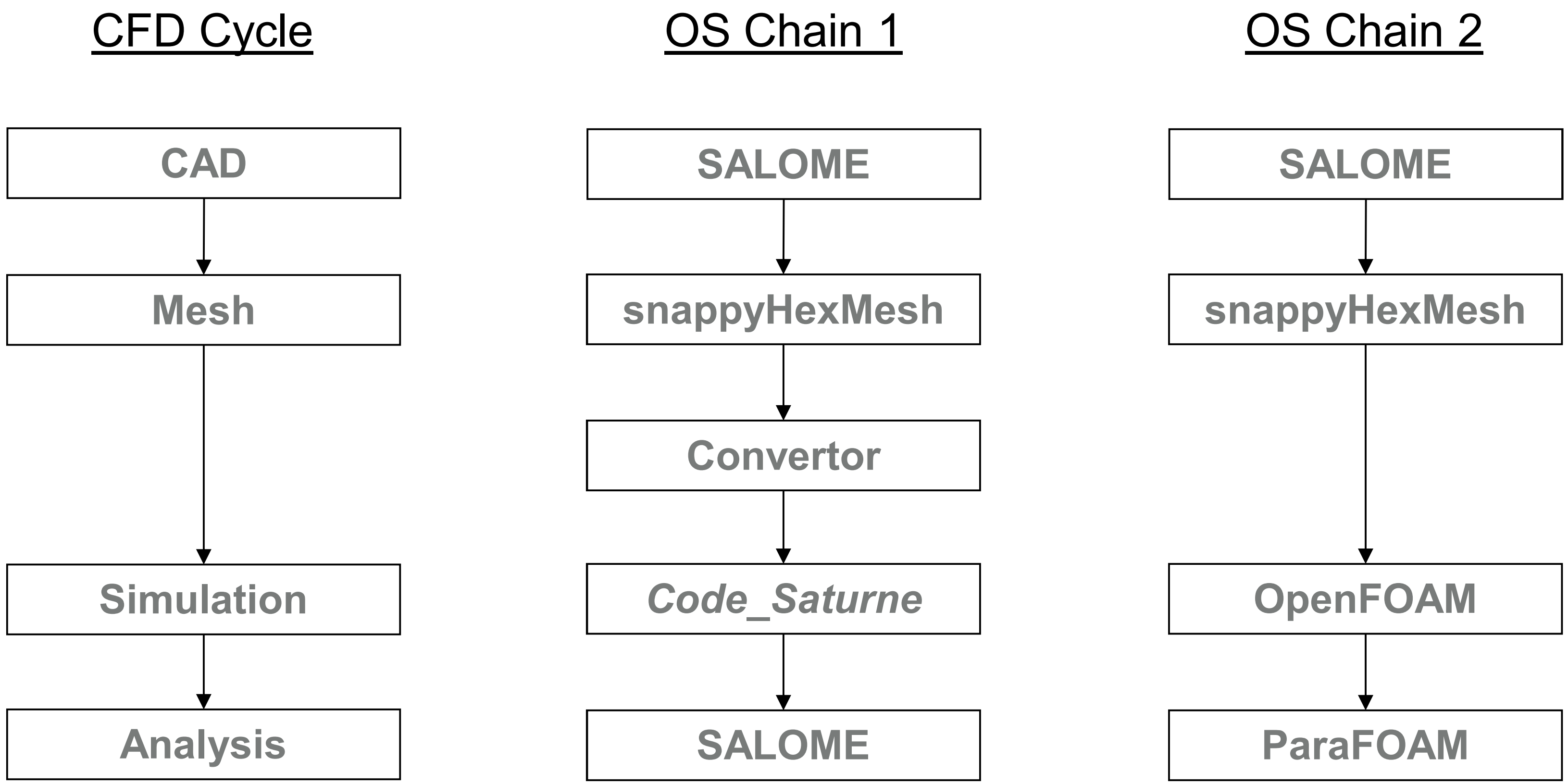
- The open source calculation chains need to replicate this cycle in its entirety

- Parameters which must be verified:

CAD	Mesh	Simulation	Analysis	Optimisation
Ability to simplify, clean and close complex geometries	Ability to generate quality volume meshes with geometry sizes varying from 1mm to 13m in current time frames	Facility to set up and simulate buoyancy driven flows with acceptable accuracy and in an acceptable time frame	Use methods that facilitate the understanding of product performance	Feedback loop into the CFD cycle with the aim of improving product performance

Open Source Calculation Chains

- Two open source calculation chains were tested for the CFD cycle



3. Test Case

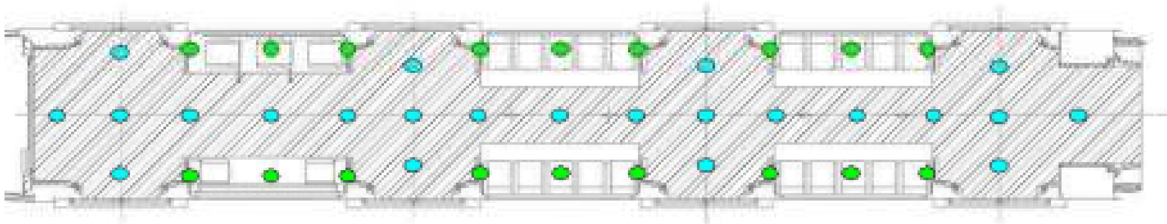
Description

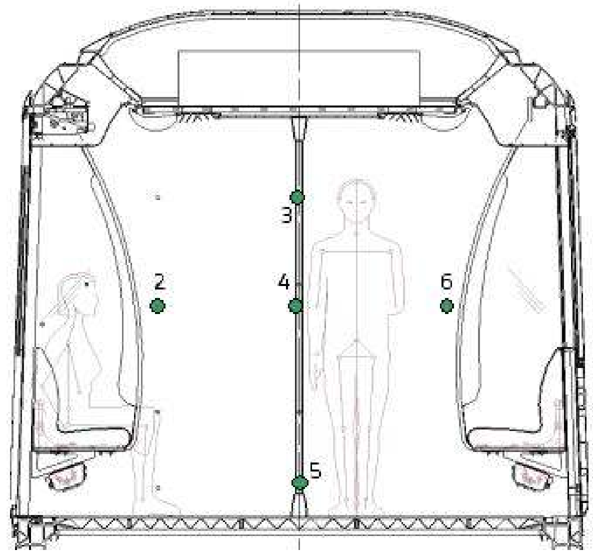
- **CAD**
  - Sophisticated ventilation duct design
  - Detailed passenger car zone to include seats, doors, windows and internal partitions
- **Mesh**
  - Large range in geometric size varying from 1mm thick panels to a 13m long passenger car
  - Highly detailed ceiling and floor level diffusion ducts
- **Analysis**
  - Experimental data for comparing with predicted data
  - Compare with a STAR-CCM+ simulation for the winter condition
- **Simulation**
  - 3D, turbulent buoyancy driven flow with heat transfer
  - Variety of outlet boundary conditions

- For winter mode (with fixed inlet air volume flow rate)

External air temperature (C)	-20.0
Inlet air temperature (C)	24.0
Fraction of return air	0.83
Fraction of extracted air	0.17

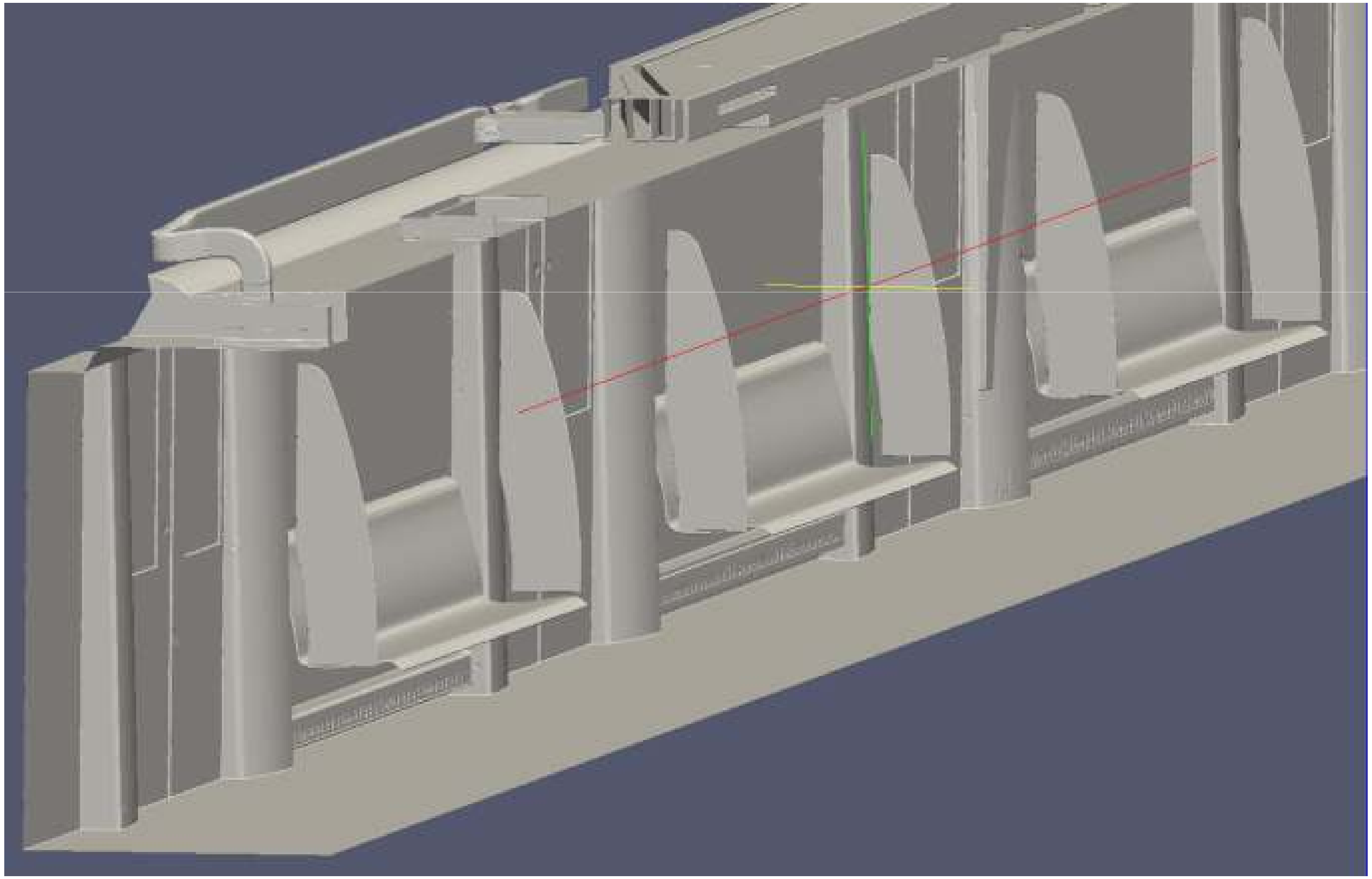
- Measurement locations
  - Conforming to EU Standards





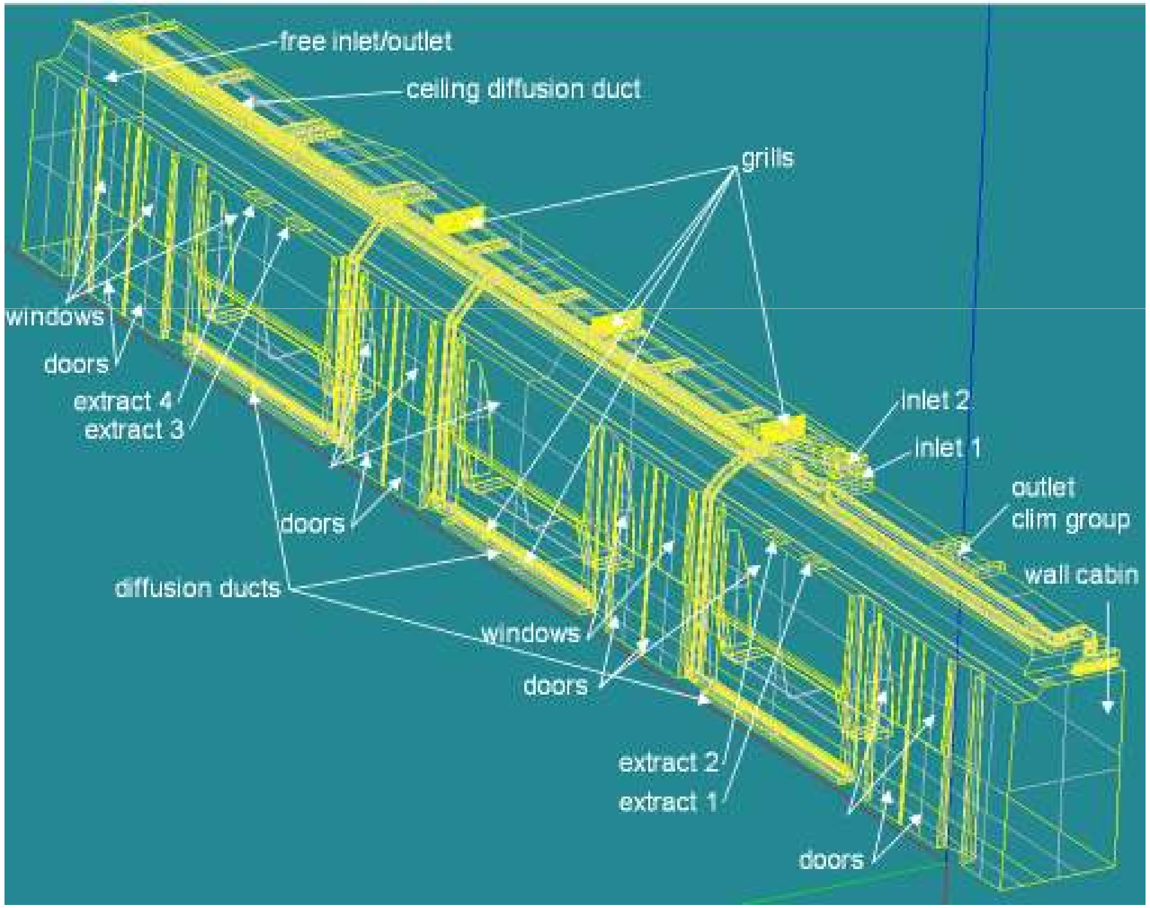
CAD : Geometry

- SALOME
  - IGES file imported with some surface errors
  - Geometry closed using the SEWING option – virtually instantaneous



CAD : Boundary Conditions

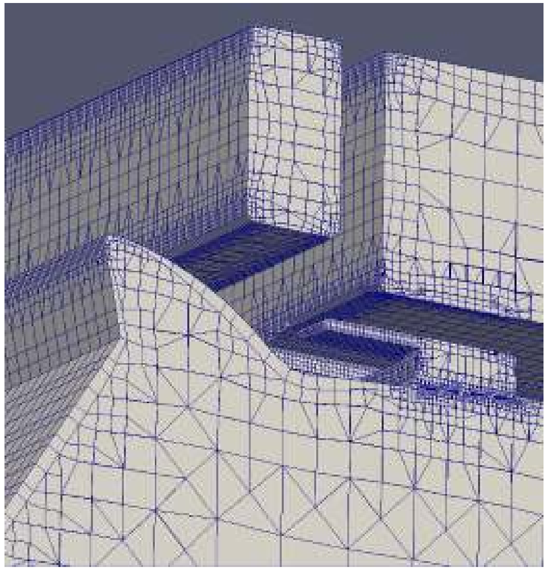
- SALOME
  - Creation of groups for meshing and boundary conditions
  - For complex geometries such as Amsterdam this was a relatively long process

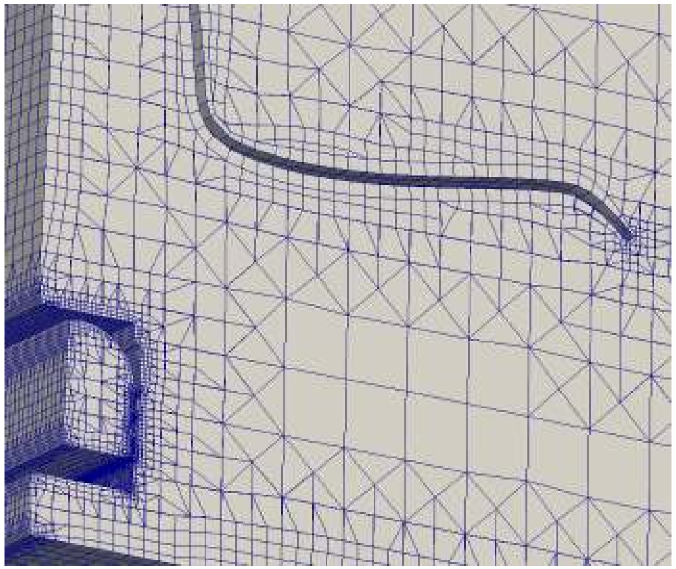


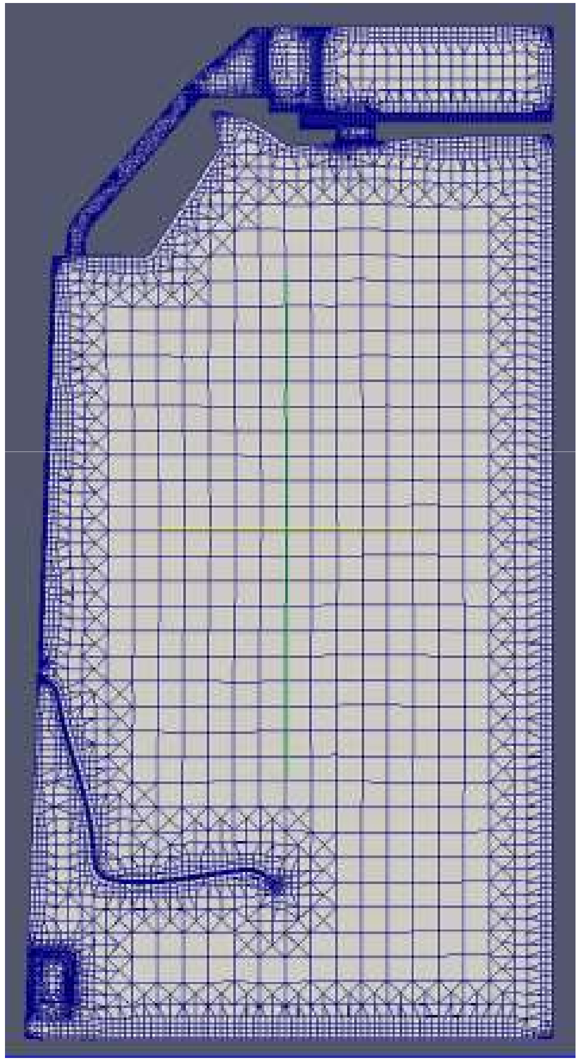
Volume Mesh

- snappyHexMesh
  - Surface mesh imported from SALOME
  - For the volume mesh

Mesh	Cell size (mm)	Extrusion layer		Total Cells
		Size (mm)	Layers	
M1	64	25 -> 0.5	1	10321082
M2	32	25 -> 0.5	1	11328880







Physical and Numerical Modelling

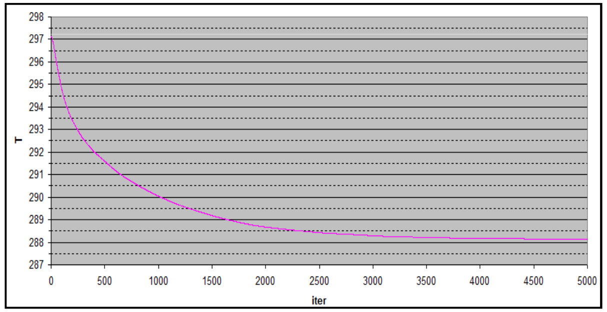
- Steady state, compressible, turbulent flow
- Air properties using standard atmospheric pressure and temperature
- *Code_Saturne*
  - Incompressible solver and $\rho = f(T)$
  - 1st order discretisation
  - k-ε LP turbulence model with the “two scale model” wall function
  - SIMPLE algorithm
- OpenFOAM
  - bouyantSimpleFoam solver
  - 1st order discretisation
  - k-ω SST turbulence model with the kqr wall function
  - SIMPLE algorithm
- Boundary conditions
  - Inlet, outlet and wall

4. Analysis

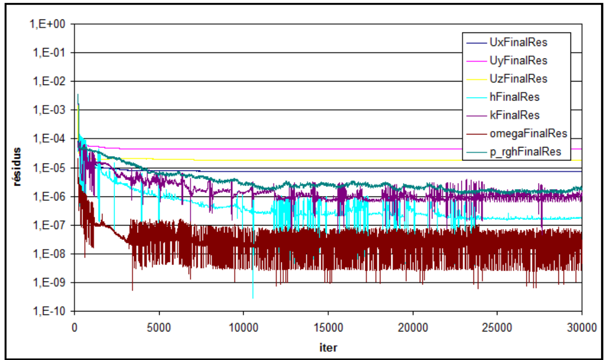
Simulations

Simulations		
Code	**Mesh**	**Simulation**
Code_Saturne	M1	S1
OpenFOAM	M1	S2
Code_Saturne	M2	S3
OpenFOAM	M2	S4

- *Code_Saturne*
  - Parallel calculation on 10 processors
  - Ran for 5000 iterations
  - Volume averages are stable after some 4500 iterations



- OpenFOAM
  - Parallel calculation on 10 processors
  - Ran for 30000 iterations
  - Volume averages are stable after some 15000 iterations

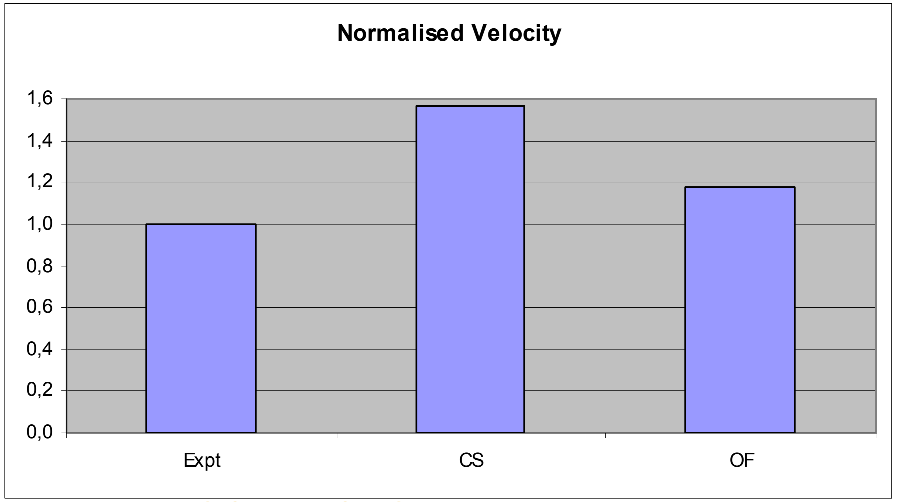


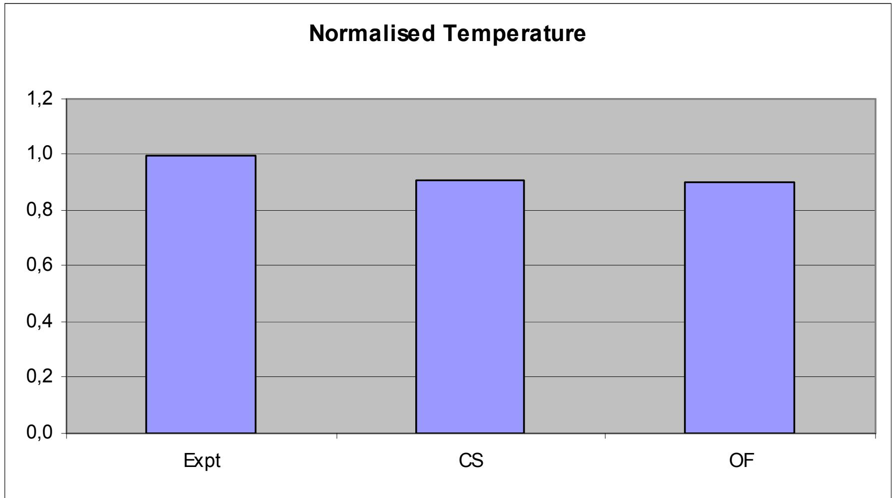
Global Performance

- Comparison of global performance (normalised to experiment)

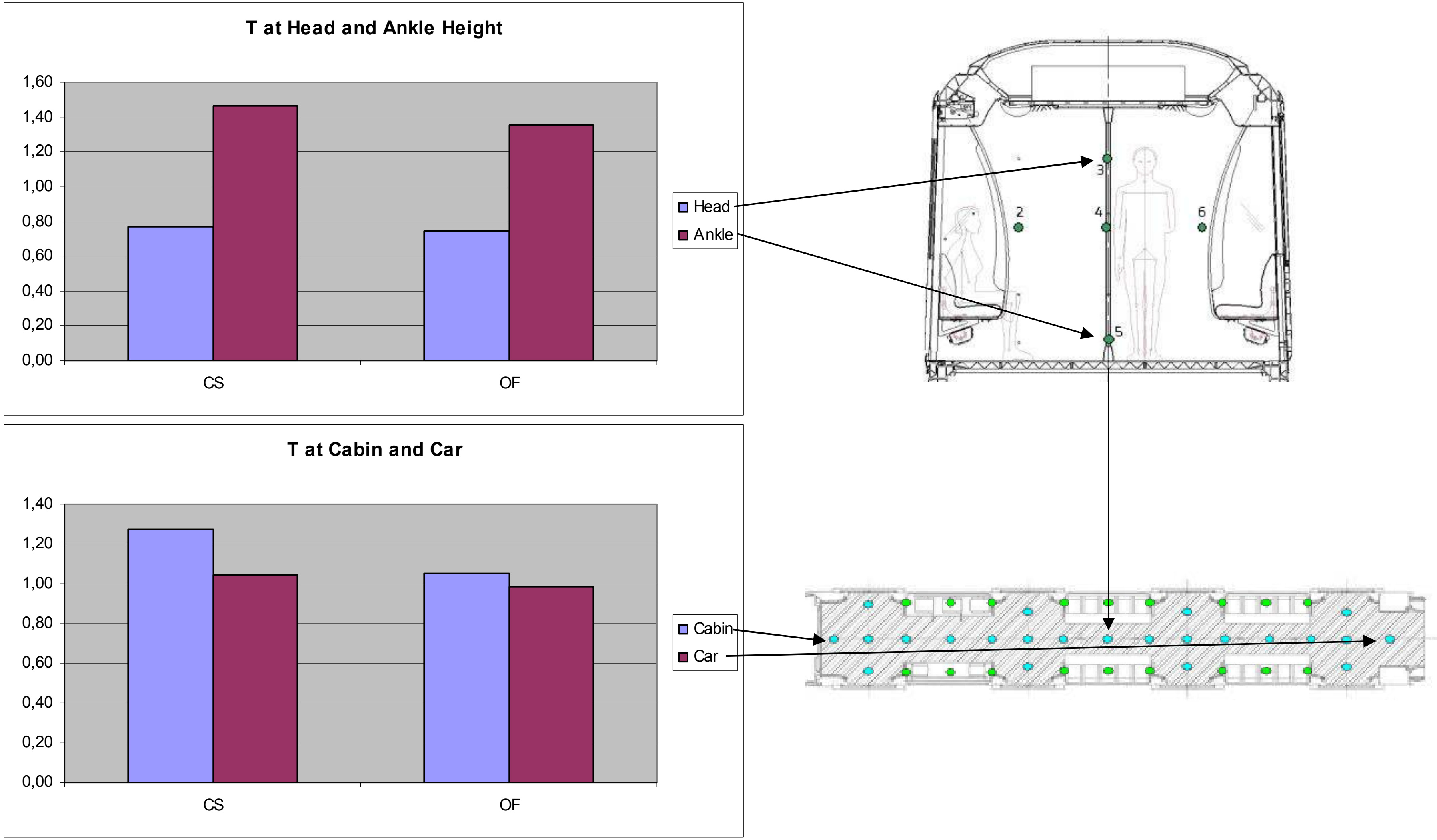
Source	T_{avg}	V_{avg}
Experiment	1.0	1.0
Code_Saturne (S1)	0.91	1.57
OpenFOAM (S2)	0.90	1.17

- The predicted values are within acceptable limits for the EU standards for thermal comfort
  - The velocities are difficult to accurately predict given the “unsteady” nature of the buoyant flow hence, the most important criteria is the predicted temperature which is in good agreement



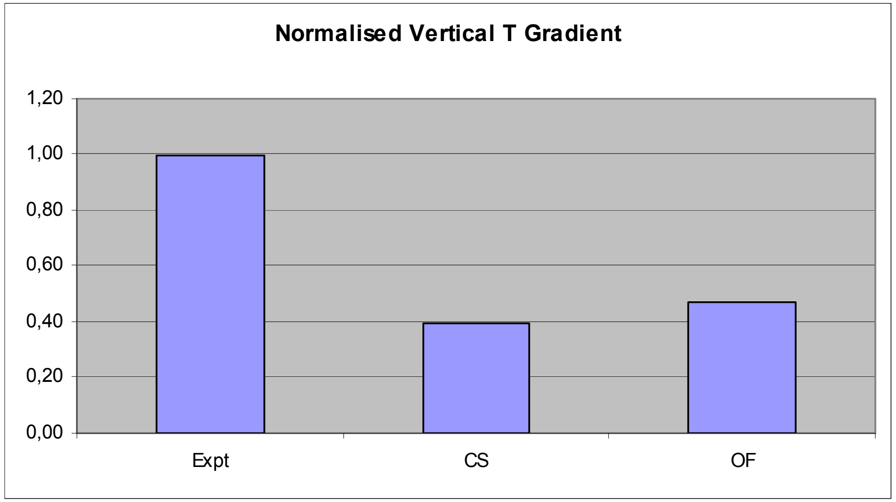


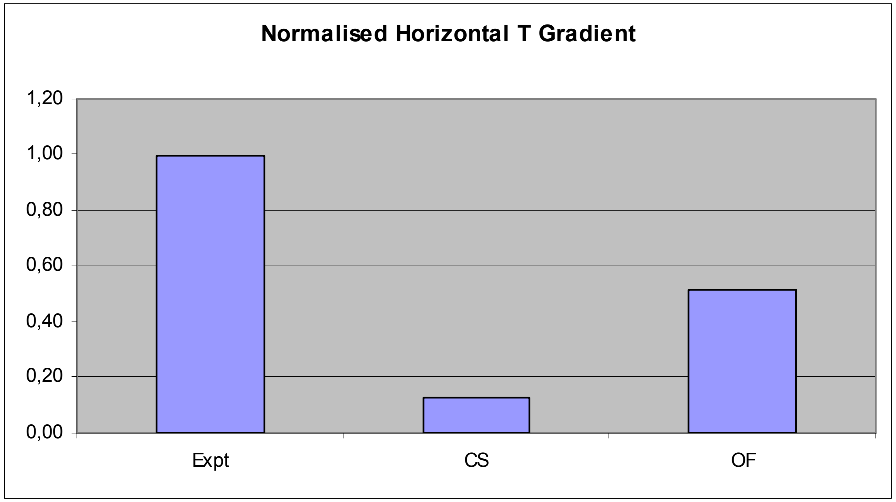
- Comparison of temperatures (normalised)



Temperature Gradients

- Temperature gradients (normalised to experiment)
  - The vertical gradients are less than measured experimentally for both solvers which indicates that these solvers predict more flow mixing than in reality
  - The horizontal gradients also indicate the same
    - For *Code_Saturne* it is thought that the boundary condition that is used to represent an adjacent passenger car is ill-defined in the model hence the lower horizontal gradient, this can be seen on slide 25

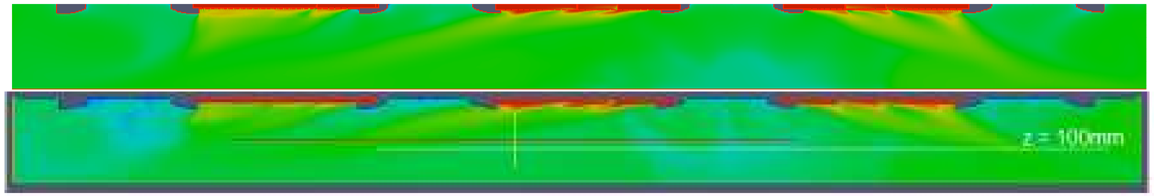




Temperature Fields

- Temperature contours on horizontal planes (identical scales)

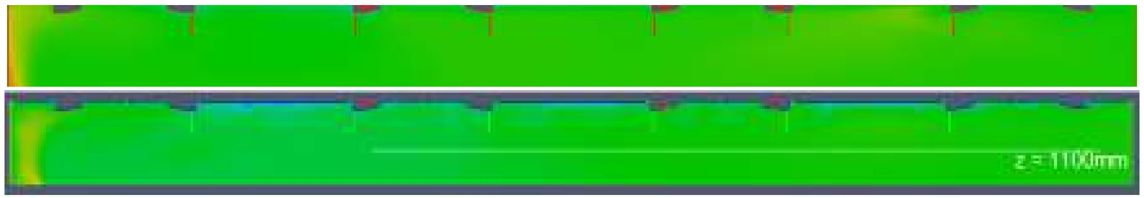
  - Ankle height (100mm)



Code_Saturne

OpenFOAM

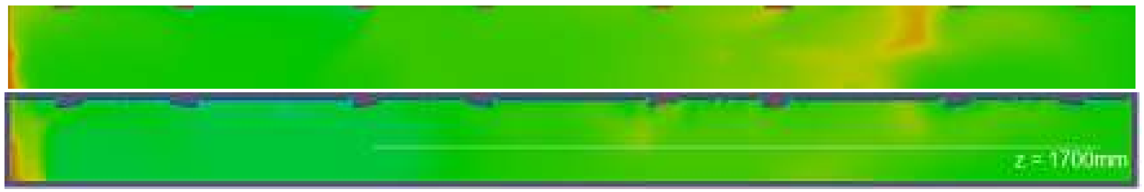
  - Waist height (1100mm)



Code_Saturne

OpenFOAM

  - Shoulder height (1700mm)



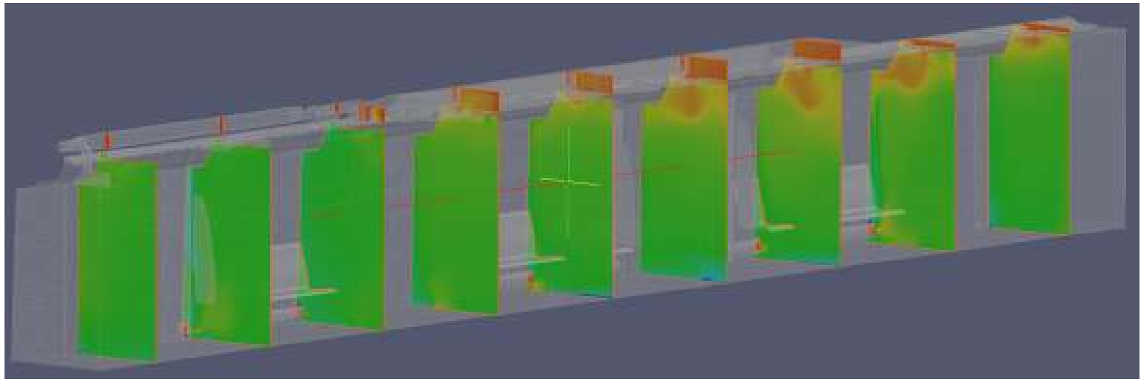
Code_Saturne

OpenFOAM

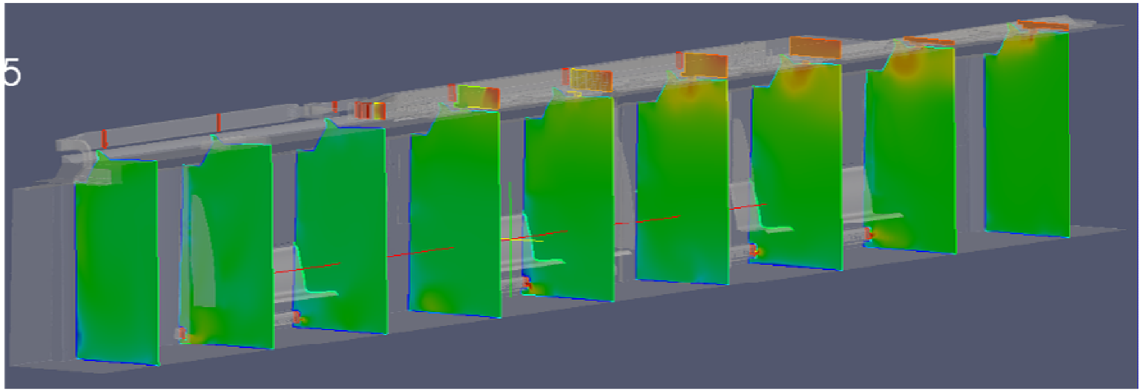
Temperature Fields

- Temperature contours in vertical planes (identical scales)

Code_Saturne



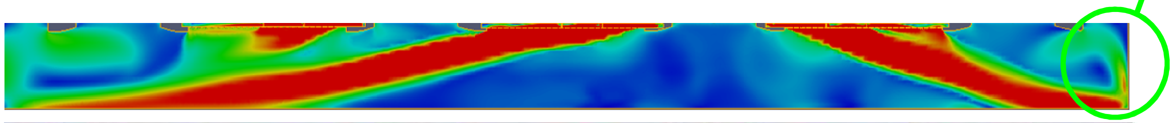
OpenFOAM



- Velocity contours in horizontal planes (identical scales)

Problem with boundary condition that needs to be rectified and which explains the low horizontal gradient in T

- Ankle height (100mm)

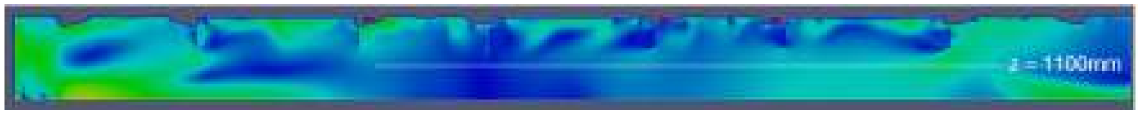


Code_Saturne

OpenFOAM

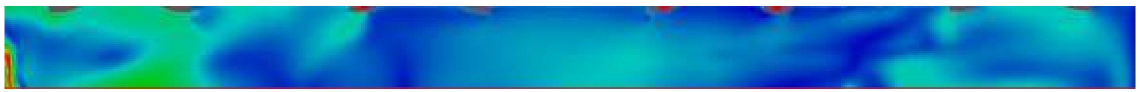
- Waist height (1100mm)

Code_Saturne

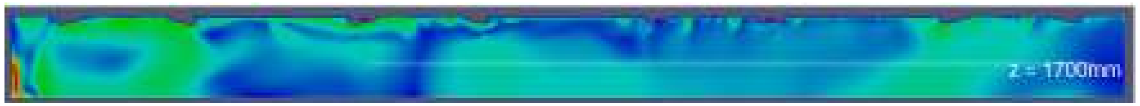


OpenFOAM

- Shoulder height (1700mm)



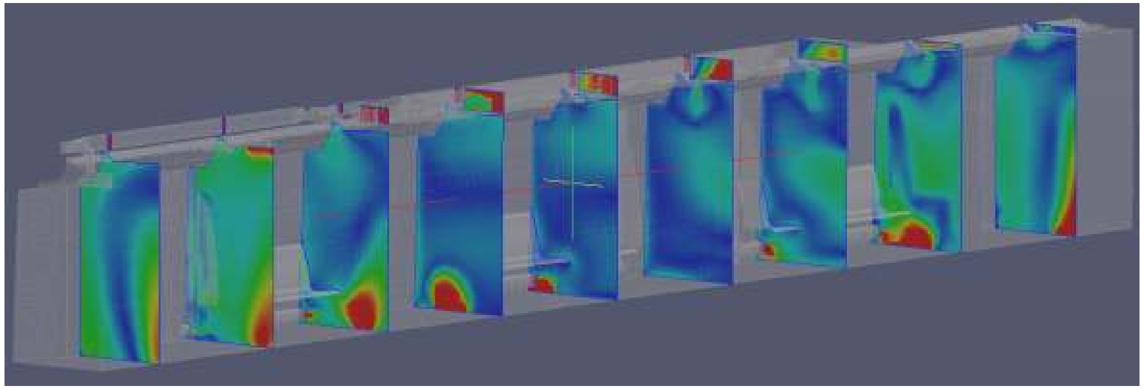
Code_Saturne



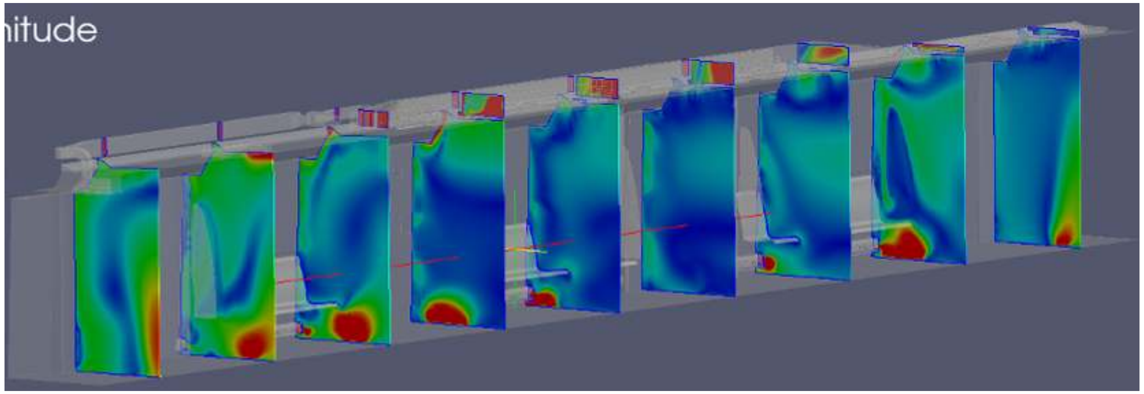
OpenFOAM

- Velocity contours in vertical planes (identical scales)

Code_Saturne



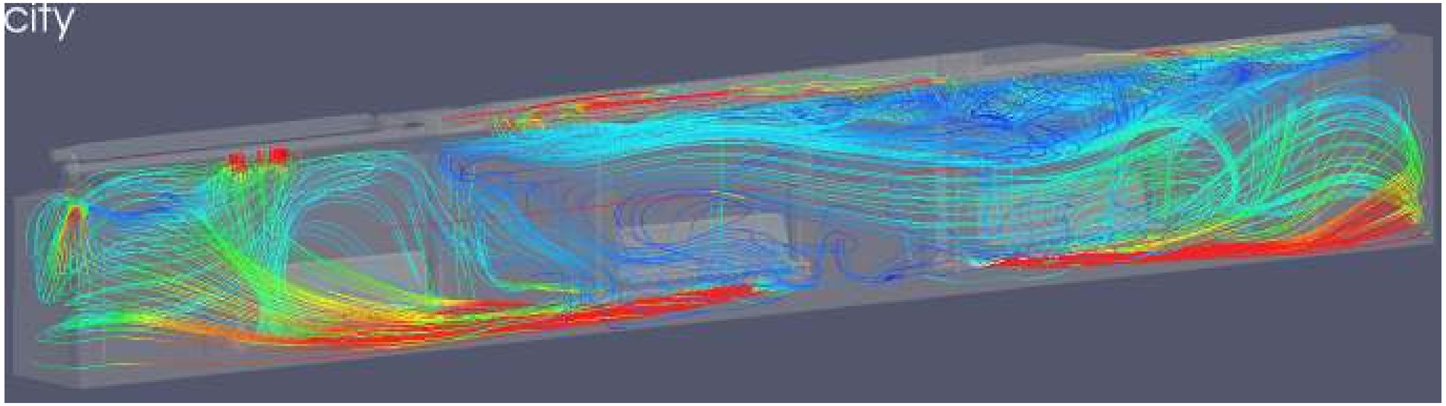
OpenFOAM



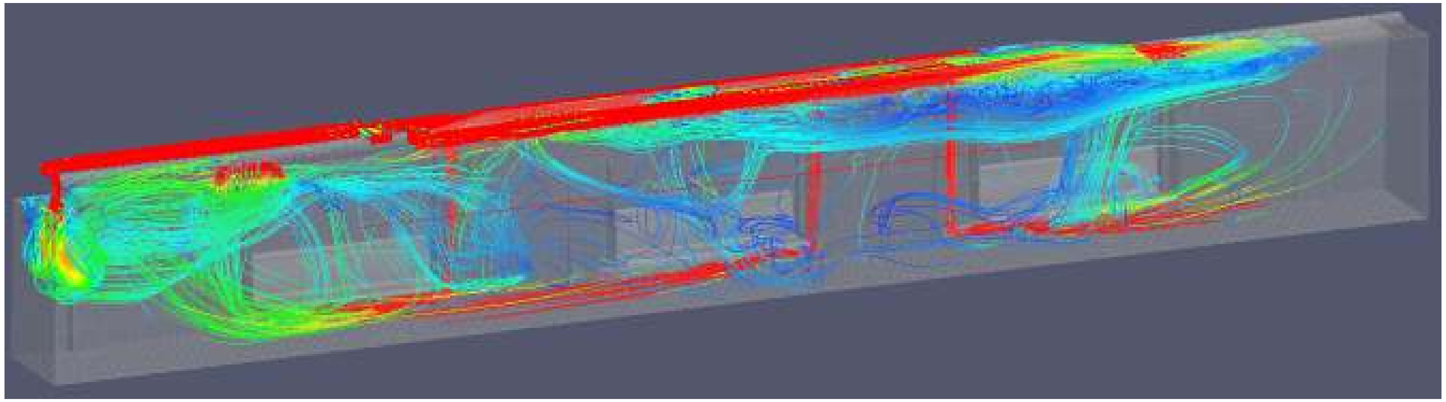
Flow Patterns

- Streamlines (identical scales)

Code_Saturne

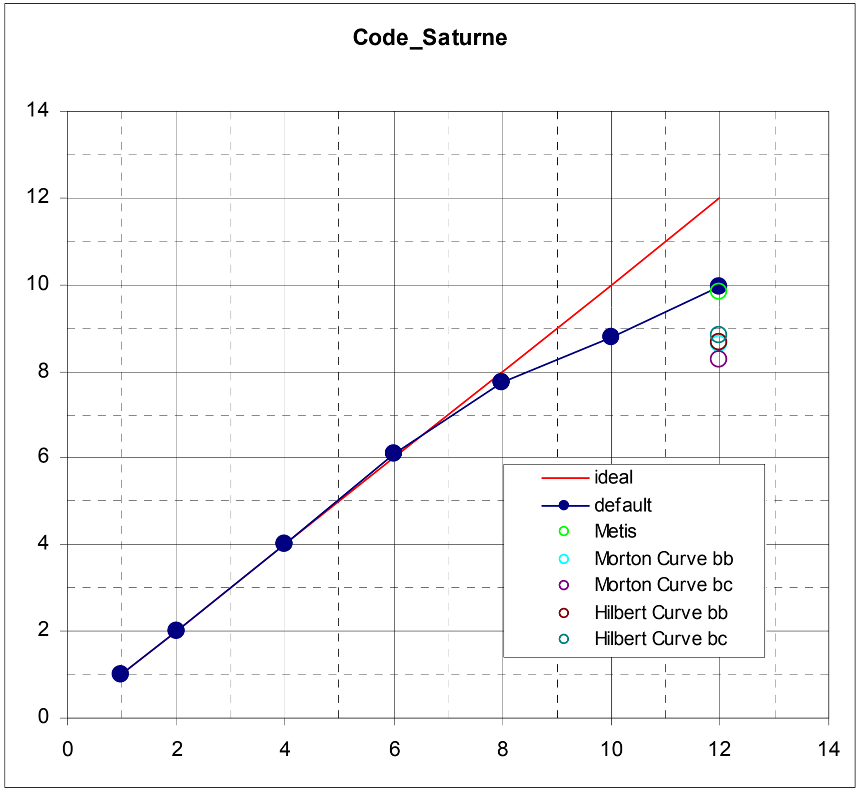


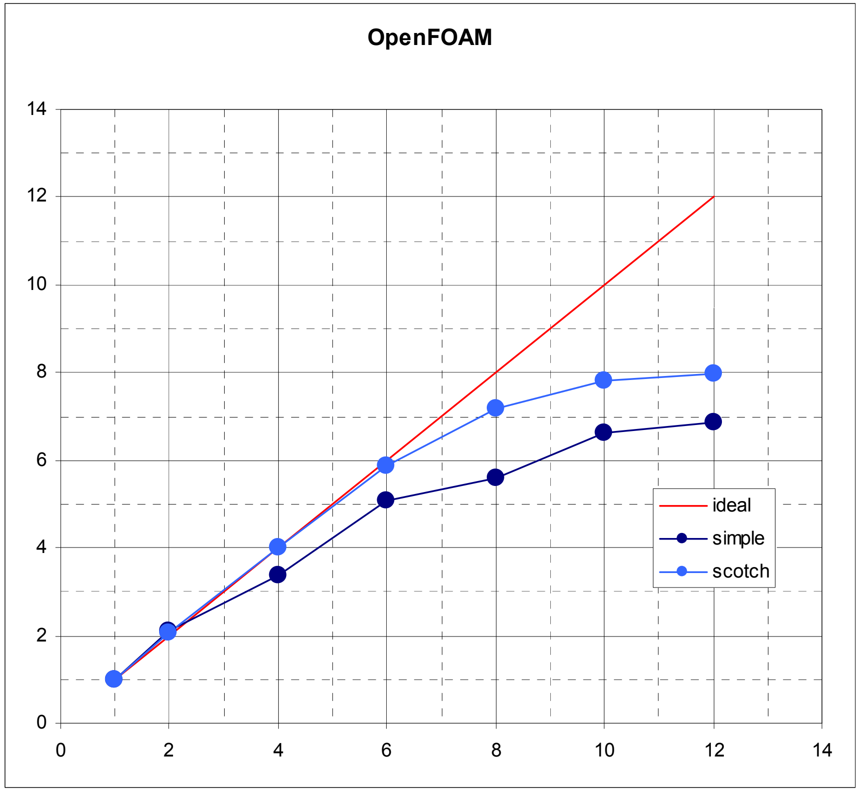
OpenFOAM



Parallel Calculations

- Speed Up for Mesh M1 – 10.3 million cells





5. Summary and Perspectives

Summary

- A combination of open source software including SALOME, snappyHexmesh, *Code_Saturne,* OpenFOAM and Paraview was used to reproduce the industrial CFD simulation process traditionally carried out at ALSTOM Transport using commercial CFD codes:
  - Simple CAD cleaning and closing
  - Volume meshing
  - Problem set-up and simulation
  - Solution analysis
- The full simulation suite was tested and validated for thermal comfort applications in winter conditions using the Amsterdam metro passenger car
- Results have been compared against experimental data for the winter operating condition
  - There is acceptable agreement between the predicted results and experimental data
  - *Code_Saturne* and OpenFOAM have been shown to give comparable results to STAR-CCM+

Remarks

- SALOME and *Code_Saturne*
  - \+ SALOME
    - Generated simple geometry of a driver's cabin
      - Appears to have the capability to generate complex geometries
    - Simple cleaning and closing of a complex geometry
    - ParaView integration (ParaVis) for post processing
  - \+ *Code_Saturne*
    - Integrated into the SALOME platform (CFD study module)
    - GUI significantly facilitates problem set up, running and monitoring
    - Achieves a stable solution in the same time frame as Star-CCM+
  - \- SALOME
    - The generic Linux version, V6.5.0, was found to be relatively unstable and could crash or "freeze" with Ubuntu
  - \- *Code_Saturne*
    - Requires its own open source automatic unstructured meshing tool
    - Needs a surface-to-surface (s2) radiation model and a solar load model for simulating summer conditions

Remarks (2)

- OpenFOAM
  - +
    - Has its own automatic meshing capability
    - Has a perfect gas flow solver
    - Large choice of solvers, discretisation schemes, etc.
    - ParaView is integrated into the OpenFOAM suite
  - -
    - Requires a GUI to facilitate problem set up
      - All set up is done manually
      - Some 10 files to set up correctly
    - Large choice of solvers, discretisation schemes, etc.
    - Lacks a solar load and s2s radiation models for simulating summer conditions
    - Needs significantly more iterations to achieve a “stable” solution for this case
      - Other solvers / AMG pre-conditioners can reduce this time?
    - Does not have as good a speed up capability for parallel calculations as *Code_Saturne*

Perspectives

- Open source calculation chains can be used in an industrial environment

- However, additional functionalities are required before *Code_Saturne* or OpenFOAM can be used to replace commercial CFD solvers for thermal comfort modelling

- Further validation for thermal comfort is also required on other configurations
  - TGVs, Tram, TER, …

- Test for other types of flow in the rail industry
  - External aerodynamics
  - Cross winds
  - Under body flows
  - Interaction between two trains and trains and tunnels
  - …..