Simplicity without compromise

KARALIT CFD VALIDATION: ASMO

ASMO
A validation study of KARALIT CFD has been carried out on the ASMO car body shape. It is a well known and publicly available test case for CFD codes validation in the automotive sector. Available data for comparison [1] are wind tunnel pressure coefficient distributions alongside the car's body at its mid-span section. A comparison is also made with numerical data obtained by Xflow [2]. The model's geometry is shown in Figure 1.

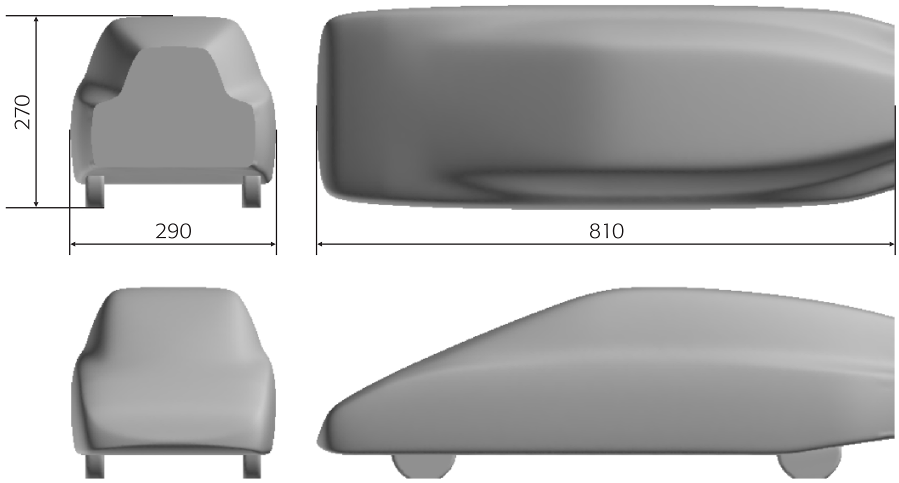

Figure 1: ASMO model geometry (dimensions are in mm)

SIMULATION PARAMETERS:

- Steady state 3D simulation
- Viscous turbulent flow
- Wind Tunnel App
- Number of cells in the computational domain : approximately 3.5 millions
- Spalart-Allmaras turbulence model, wall function
- Velocity inlet: 50 m/s
- Re = 2.75 x 10^6 based on the model length (0.81 m)
- Grid resolution: y+ = 96
- Numerics: implicit scheme, 2nd order symmetric TVD discretization scheme, CFL = 5
- Boundary conditions:
  - Slip boundary conditions on domain's side and top walls
  - Symmetry boundary conditions on symmetry plane
  - No slip conditions on wind tunnel ground

Figures 2 and 3 show the computational mesh at the symmetry plane and a magnified detail of the grid at the front of the car.

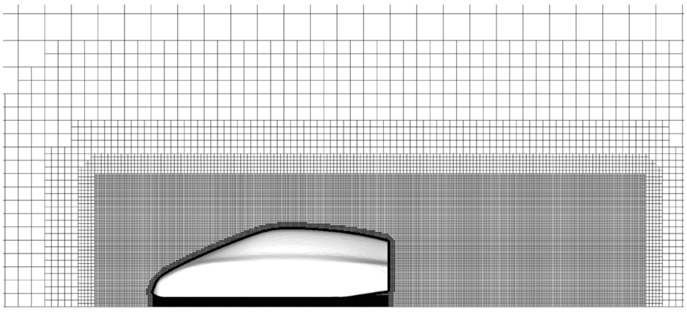

Figure 2: the ASMO computational grid at the symmetry plane

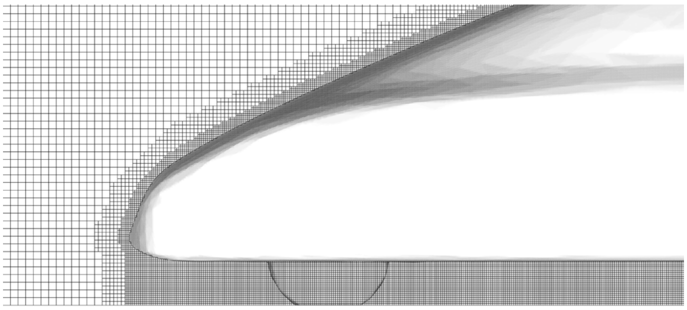

Figure 3: grid magnification at the symmetry plane close to the car's front

Figures 4, 5, 6 and 7 show the computed pressure coefficient profile around the car body at the symmetry plane. It is evident the generally good agreement both with experimental data and Xflow numerical prediction.

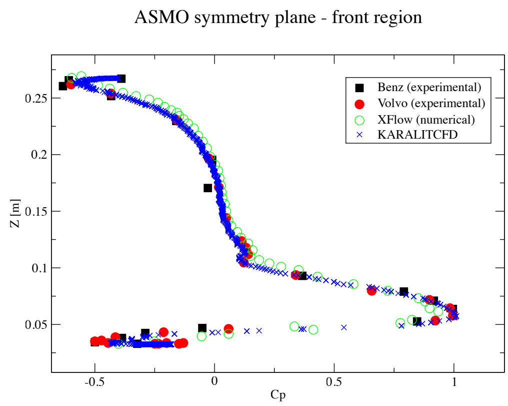

Figure 4: comparison of pressure coefficient distribution with experimental data [1] and Xflow numerical results [2] (car's front)

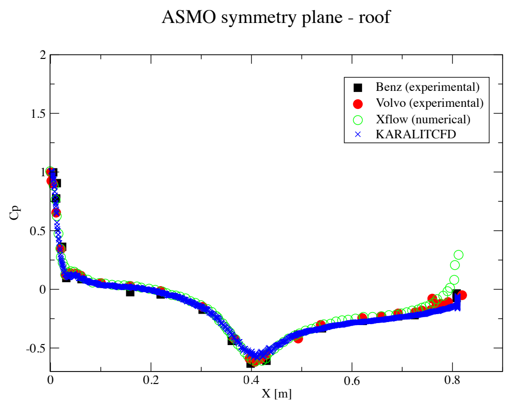

Figure 5: comparison of pressure coefficient distribution with experimental data [1] and Xflow numerical results [2] (car's roof)

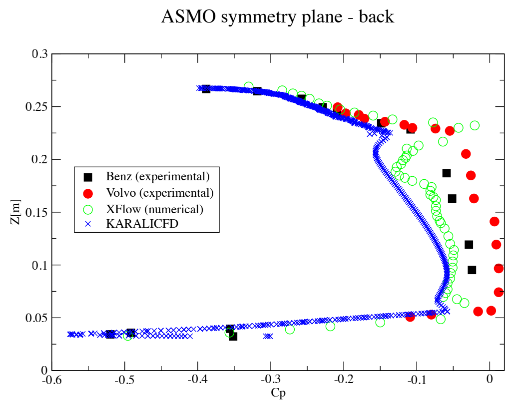

Figure 6: comparison of pressure coefficient distribution with experimental data [1] and Xflow numerical results [2] (car's back)

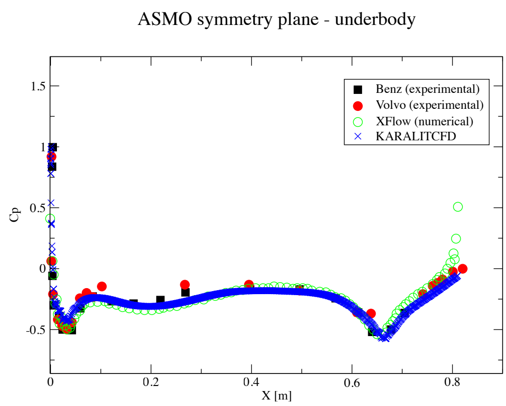

Figure 7: comparison of pressure coefficient distribution with experimental data [1] and Xflow numerical results [2] (car's under body)

Figure 8 shows the contour plot of the x-velocity component at the symmetry plane together with some streamlines.

It is evident the presence of two counter-rotating vortexes in the car's wake. This vortical structure has also a component aligned with the main flow direction. Figure 9 shows the velocity vectors and the contour of velocity magnitude on a plane at about 0.1m behind the car in the wake region. This complex vortical pattern is, in fact, unsteady in nature, as it has been shown by wind tunnel experiments [1].

Exam of the x-momentum convergence plot, that is shown in Figure 10, suggests that the phenomenon is in fact unsteady in nature. Although the simulation has been carried out using the steady option, a fairly good evaluation of the frequency associated with the unsteady behavior of the flow can be made through an FFT of the residuals data. Figure 11 shows the extracted peak frequency associated with the computation. The peak frequency is of about 500Hz.

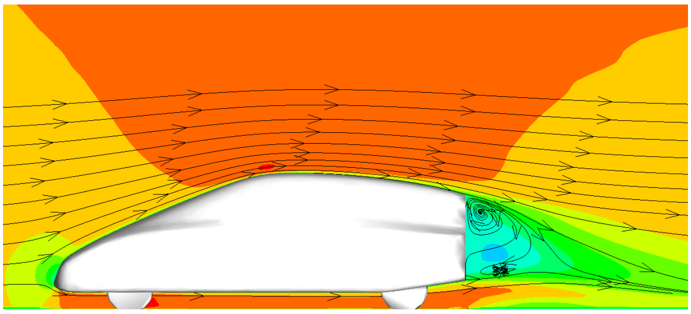

Figure 8: x-velocity component contour plot and streamlines at the symmetry plane; the presence of two counter-rotating vortexes can be observed in the wake region

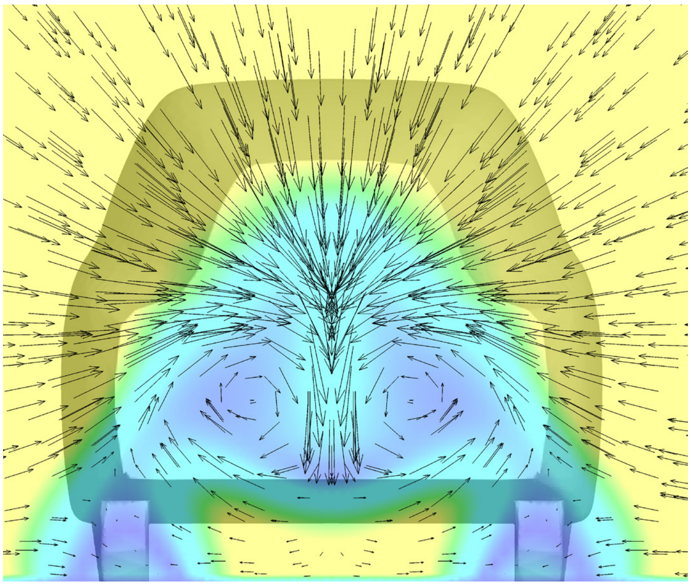

Figure 9: velocity vectors and velocity magnitude contour plot over a plane normal to the main flow direction in the wake region at 0.1m behind the car

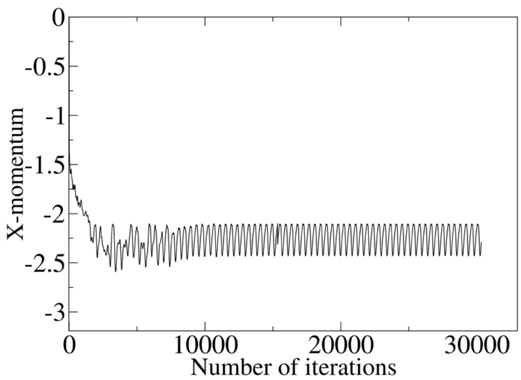

Figure 10: normalized x-momentum residual's convergence history

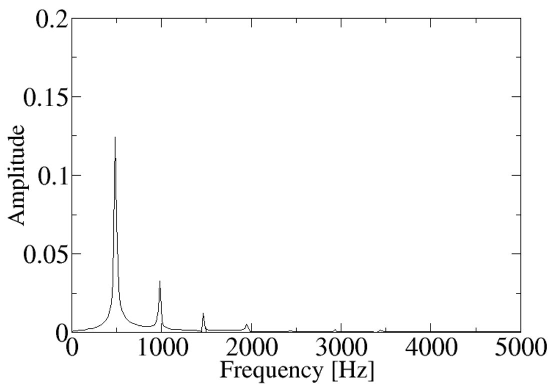

Figure 11: FFT of the residual's convergence history; peak frequency is of about 500Hz

CONCLUSIONS

The calculations undertaken with the KARALIT CFD code using the Immersed Mesh technique predict the correct trends of the flow- field streamlines and Cp distributions on the ASMO shape, in comparisons against available tests. Considering the high y+ used for the simulations and therefore the dissipation introduced by the coarseness of the grid, the following results can be considered in good agreement with the experimental results.

The predictions undertaken with the KARALIT CFD code demonstrate the degree of understanding that can be made available to the user with minimum setup for quite challenging flow-field scenarios.

The calculations can be extended invoking different turbulence models in future efforts toward this validation. Given that some of the important wake fields have been shown to be unsteady in nature, it would be important to provide for transient prediction in future validations.

REFERENCES

1. Perzon S. and Davidson L., On Transient Modeling of the Flow Around Vehicles Using the Reynolds Equations, In ACFD 2000 Beijing, Oct 17-20 2000, China, Eds. Wu J.-H., Zhu Z.-J., Jia F.-P., Wen X.-B. and Hu W., pp 720-727, 2000 - http://www.tfd.chalmers.se/~lada/postscript_files/sven_acfd_paper_2000.pdf

2. Aerodynamic analysis involving moving parts with XFlow - Project: Vehicle aerodynamics ©2010 Next Limit Technologies & qpunkt GmbH - http://web.mscsoftware.com/Submitted-Content/Pdf/XFlow-Application-Brief-Project01-ASMO.pdf

Contact your sales office or **sales@karalit.com**
Headquarter: Pula (CA), Italy - **Operating Office:** Via Mameli 191, Cagliari, Italy
Tel: +39 070 278202 - info@karalit.com - **www.karalit.com**

Italy Office: Milan
Tel: +39 02 87188454

UK Office: London
Tel: +44 (0) 8006126912

USA Office
Tel: +1 855 KARALIT (527 2548)