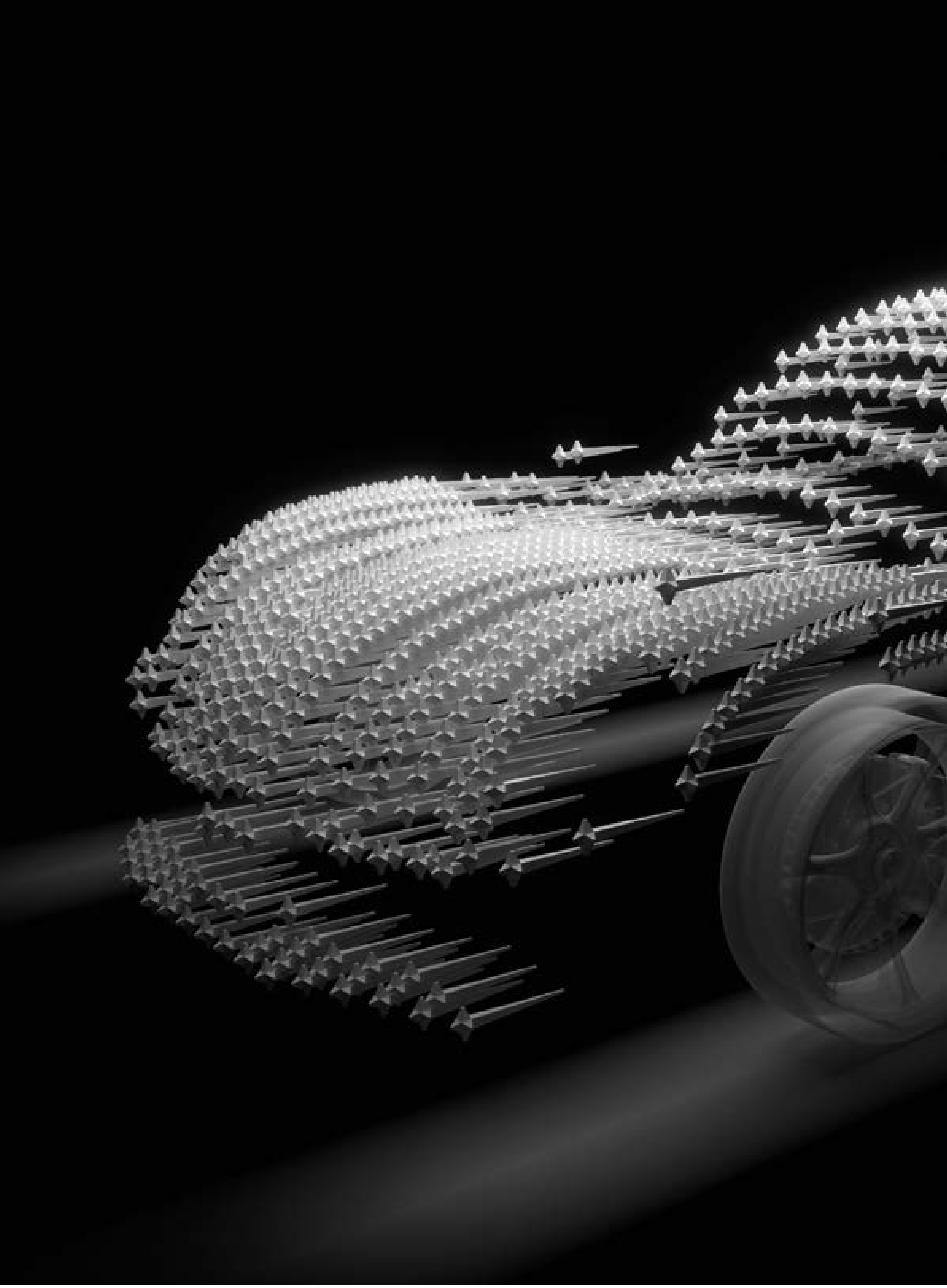

JSAE Benchmark of Automotive Aerodynamic Test Measurements

Ahmed-Type Car Body Versus CFD Software Predictions

By Boris Marovic, Automotive Industry Manager, Mentor Graphics

The Society of Automotive Engineers of Japan (JSAE) recently conducted a blind benchmark for commercial Computational Fluid Dynamics (CFD) software to demonstrate their accuracy against test validation data on a new car shape [1]. Participants simulated a ¼-scale wind tunnel car model. The aerodynamic test model consisted of the "Ahmed" vehicle body (see Figure 1) with a full vehicle length of 1,100mm without the "additional part" at the end of the vehicle (Case 1) and at a length of 1,250mm with the additional part attached (Case 2). The height of the vehicle was 355 mm, the width 320mm, and the underfloor vertical gap was 15mm.

Each CFD software package had to analyze the airflow around the model and compare their prediction accuracies to experimental data without knowing the data beforehand. In particular, JSAE wanted to verify how accurate each CFD tool was at predicting boundary-layer separation, pressure distribution, and body forces on the model. It was up to the participants in the benchmark to choose the best meshes and turbulence models in their CFD codes to offer their best prediction.

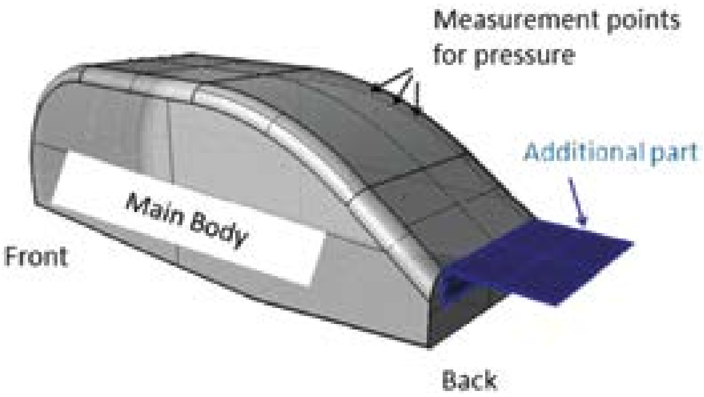

Figure 1. The JSAE Aerodynamic Test Ahmed Vehicle Body with and without the additional part at the end of the vehicle

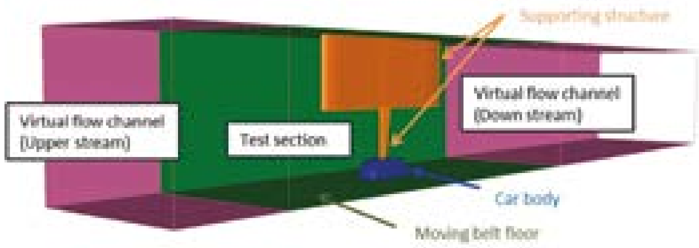

Figure 2. The model of the test chamber for the wind-tunnel experiment

The commercial CFD participants were provided with data that included support pole shape, test section shape, and vehicle model shape plus the specification of the wind tunnel test section (Figure 2). The participants were provided with reference tunnel data as well as the specified conditions for the test. Simulations were all to be at a velocity of 25.0 m/s; the fluid properties were given as a density of 1.17 kg/m^3 and a kinematic viscosity of 1.56 x 10-5 m^2/s, which resulted in a Reynolds number for the test of 1.76 x 106.

All CFD simulation software had to provide results for drag, lift, and pitching-moment coefficients as well as pressure coefficient at various sections of the vehicle body. Sections vertically to the car were compared to measurements at the center plane (y/W = 0.0), 12.5% off center (y/W = 0.125), and 25% off center (y/W = 0.25), where W is the width of the body. The underfloor section was only analyzed at the vertical center plane as it was not disturbed by the wind-tunnel fixture as the top side was. The section horizontal to the car was compared at 25% (z/H = 0.25) of the car height as shown in Figure 3 where 'H' is the height of the body.

The airflow wake predictions behind the car model were analyzed at the vertical lines of x = 1,000mm (line 1), x = 1,050mm (line 2), x = 1,100mm (line 3) and x = 1,200mm (line 4) – see Figure 4 - and compared to experimental measurements in the Blind Benchmark.

Seven organizations (Table 1) provided submissions to the JSAE blind benchmark (encompassing most of the main commercial CFD codes available in the market today) and they all had three months to submit their simulation results and technical information on their CFD computation approaches, physical models, and resolution scales. KKE Inc., a Mentor

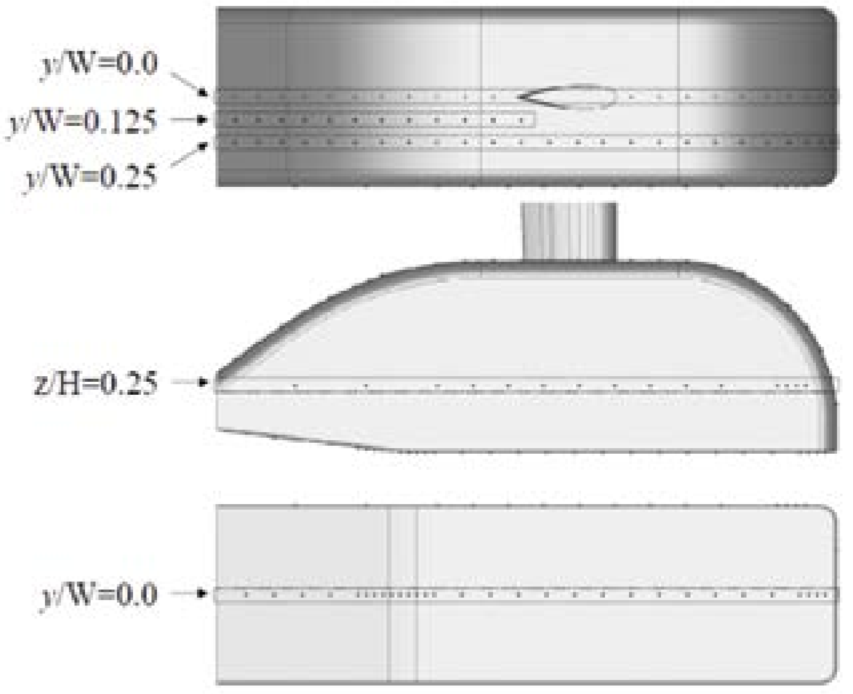

Figure 3. Pressure coefficient measurement point distribution on the JSAE car body

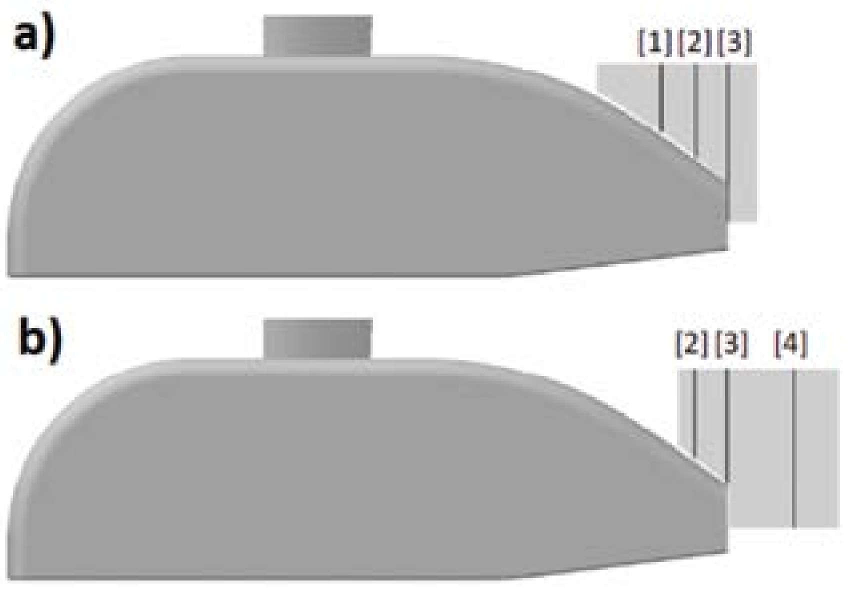

Figure 4. Wake measurements at y/W = 0.0 for the JSAE car body: a) the case without the additional part and b) the case with the additional part

Graphics reseller in Japan, submitted simulation results using the CAD-embedded CFD software, FloEFD™.

Table 1 shows that the cell count was 4 to 11 times lower for FloEFD than the other CFD tools for the cases considered, it uses partial cells that can contain several sub-control volumes and a special boundary layer treatment that does not need a fine cell resolution of the boundary layer as other tools do. Also, it is worth noting that a large number of CFD calculations shown in Table 1 were conducted in transient mode, which usually results in very high CPU time for the calculation even with a high number of cores due to the large number of cells employed. More information on each code's CFD simulation set-up can be found in the original JSAE benchmark paper [1].

Results

For the FloEFD simulations, KKE Inc. used the Cartesian mesh approach with solution adaptive refinement (Figure 5) on an octree basis and local meshes around the body [2-3]. Each cell level refinement was easily set-up in FloEFD and the rest of the mesh generation adaptation process was automated. The adaptive refinement can also be limited to a certain region of the domain by a maximum level being applied to the cell count so that the code does not explode the mesh size and thus helping to prevent very high CPU times. Table 2 shows the computational effort used by the seven software tools used in the blind benchmark.

Compared to the other tools, FloEFD required less resources and less calculation time to come up with good overall results, and it shows quite good agreement with the wind-tunnel experimental measurements too (Figures 6-8).

Figures 6 - 8 show the simulation prediction results for all of the CFD codes employed in the benchmark plus the error margin of the test experiment data for drag, lift, and pitching moments. The blue dashed lines show the upper and lower error margin for Case 1, without the additional part, and the red dashed lines show the margin for Case 2, with the additional part. In Figure 6, the drag coefficient (CD) of AcuSolve (Inflow 2), FloEFD, and STAR-CCM+ (IDDES) were all within the margin of error for Case 1 followed by SCRYU/Tetra (DES) with a lower value and then iconCFD again with a little lower CD. For Case 2, none of

Participants	Software	Compressible/ Incompressible	Steady State/ Transient	Turbulence Model
JSOL Corporation	AcuSolve Incompressible	Incompressible	Steady state	Spalart Allmaras
ANSYS Japan K.K.	ANSYSFluent R14.5	Incompressible	Transient	Scale Adaptive Simulation (SAS)
Kozo Keikaku Engineering Inc.	FloEFD	Compressible	Steady state	Modified k-Σ
Icon Technology & Process Consulting Ltd.	iconCFD	Incompressible	Transient	Spalart Allmaras
ESI Group	PAM-FLOW	Incompressible	Transient	SGS
Software Cradle	SCRYU/Tetra	Incompressible	Transient	SST-DES, SST-SAS
CD-adapco	STAR-CCM+ v7.06	Compressible, Incompressible	Transient, Steady state	IDDES (SST), SST k-ω

Software	Mesh Type	Number of Cell Layers in the Boundary Layer	Number of Cells Without Rear Flat Panel (Case 1)	Number of Cells With Rear Flat Panel (Case 2)	Mesher Used
AcuSolve	Tetrahedral mesh	7	24,755,000	25,795,000	AcuConsole1.8b
ANSYSFluent R14.5	Unstructured grid	17	16,000,000	16,700,000	ANSYSMeshing R14, TGridR14
FloEFD	Cartesian mesh based on octree technology	-	3,520,000		FloEFD
iconCFD	Hexahedral dominant mesh	7	37,640,000	38,300,000	foamProMesh
PAM-FLOW	Tetrahedral mesh	6	38,260,000		PAMGEN3D
SCRYU/Tetra (DES, SAS)	Tetrahedral mesh with prisms	10	27,000,000		SCRYU/Tetra
STAR-CCM+ v7.06 (IDDES, SST k-ω)	Hexahedral dominant mesh	20	16,690,000	16,835,000	STAR-CCM+ v7.06

Table 1. Participant Companies and CFD Codes in the JSAE blind automotive aerodynamic benchmark

Software	Computer Characteristics	Cores	Calculation Time (h) Steady State	Transient	Time Step (s)
AcuSolve	HP ProliantDL360p Gen8, Xeon E5-2660 (2.2 GHz)	16	No Flat Panel: 4.2 With Flat Panel: 5.9	-	-
ANSYSFluent R14.5	Dell PowerEdge R720 (2.9 GHz)	32	4	60	2.0 x 10-4
FloEFD	HP Z600, Intel Xeon X5670 (2.93 GHz)	6	17	-	-
iconCFD	Intel® Xeon® Processor E5645 (2.4 GHz)	72	-	No Flat Panel: 254 With Flat Panel: 267	5.0 x 10-5
PAM-FLOW	HP BL460c, Intel Xeon E5-2680 (2.7 GHz)	16	40	155	5.352 x 10-5
SCRYU/Tetra (DES)	Intel Xeon E5-2690 (2.9 GHz)	48	-	33	1.0 x 10-4
SCRYU/Tetra (SAS)			-	34	1.0 x 10-4
STAR-CCM+ v7.06 (IDDES)	Dell Power Edge, Intel® Xeon® CPU X5675 (3.07 GHz)	120	-	~200	1.0 x 10-4
STAR-CCM+ v7.06v (SST k-ω)		12	17.5	-	-

Table 2. For the participating CFD Codes in the JSAE blind automotive aerodynamic benchmark, computational resource and time required for the calculations

the codes were exactly within the error margin but iconCFD made it the closest, followed by STAR-CCM+ (IDDES) and then AcuSolve (Inflow 2).

In Figure 7, the same dashed lines show the error margins also for Case 1 and 2 but here the graph shows the lift coefficient (CL). For Case 1 only, FloEFD was exactly within the margin. Slightly out of margin were STAR-CCM+ (SST k-ω), SCRYU/Tetra (SAS), and AcuSolve (Inflow 1) all at the same level, followed by AcuSolve (Inflow 2) with a larger CL. For Case 2, the test margin was very narrow and none of the codes were exactly within it. STAR-CCM+ (SST k-ω) was the closest, followed by ANSYS Fluent and then AcuSolve (Inflow 1), but with larger discrepancies.

The pitching moment coefficient predictions are shown in Figure 8 where they have the same margin color notation as before. In this graph, the error margin for Case 1 is very narrow and none of the CFD codes made it exactly within the margin. The closest were STAR-CCM+ (IDDES), followed by PAM-FLOW, and then SCRYU/Tetra (SAS) and STAR-CCM+ (SST k-ω) who were equally distant but on the lower margin compared to PAM-FLOW. Case 2 has a larger margin, and only FloEFD made it inside the margin, followed by ANSYS Fluent slightly outside the lower margin line and then with a slightly lower value SCRYU/Tetra (SAS) and AcuSolve (Inflow 2), with the same distance but on opposite sides of the margin.

It can therefore be concluded from the results of the JSAE benchmark that FloEFD was very accurate for both drag and lift in Case 1 and had the best pitching moment prediction in Case 2. STAR-CCM+ also comes out very well from the exercise if an expert user chooses the right turbulence model for the particular case

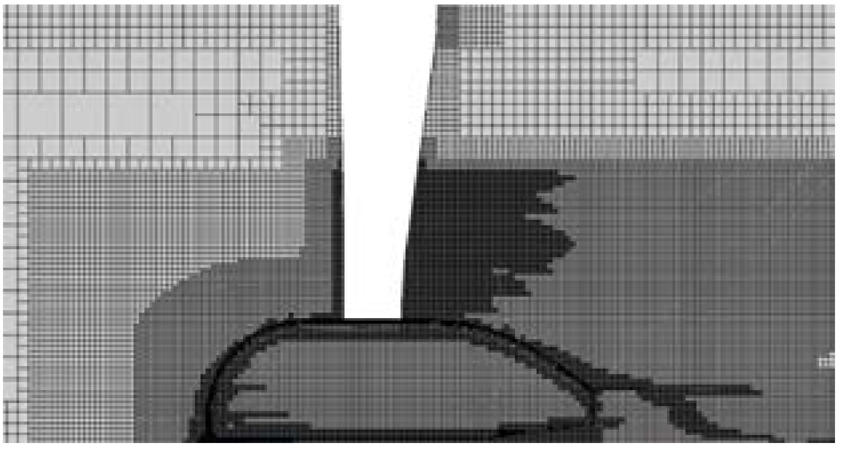

Figure 5. Computational mesh used by FloEFD for the JSAE benchmark model

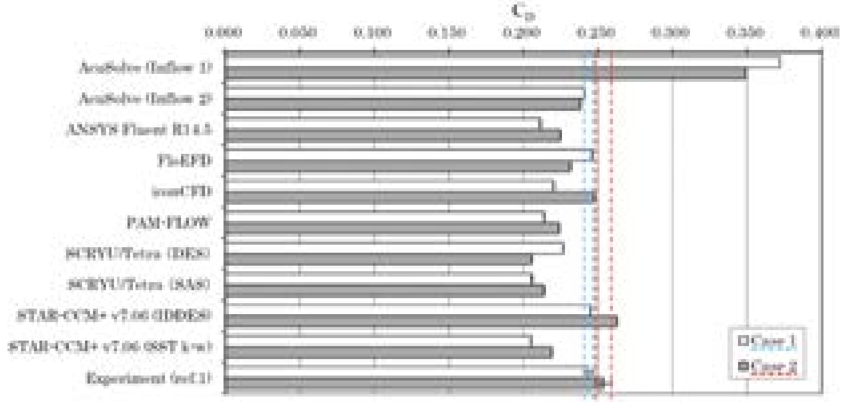

Figure 6. Drag coefficients for all CFD codes for Cases 1 and 2 with the experimental Test Data error margins in blue dashed lines (Case 1) and red dashed lines (Case 2)

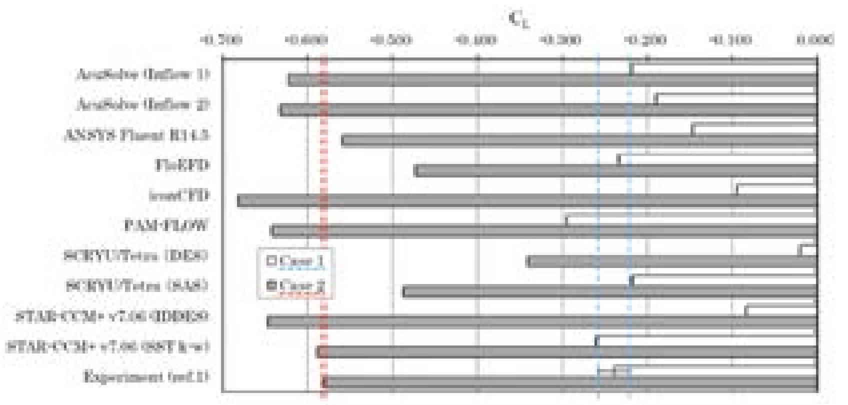

Figure 7. Lift coefficients for all CFD codes for Cases 1 and 2 with the experimental Test Data error margins.

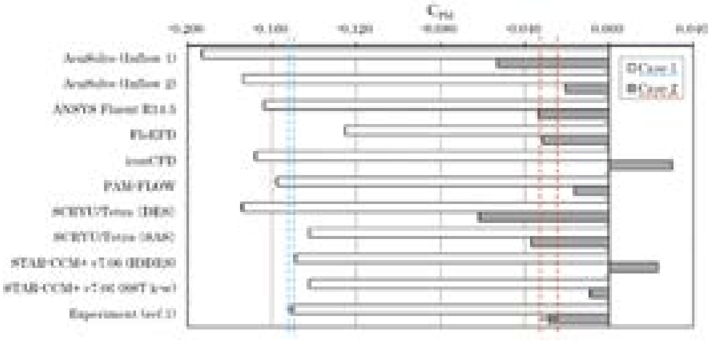

Figure 8. Pitching-moment coefficients for all CFD codes for Cases 1 and 2 with the experimental Test Data error margins.

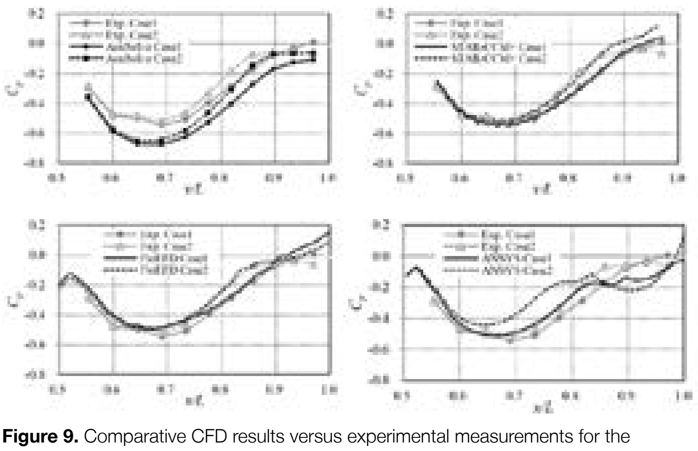

Figure 9. Comparative CFD results versus experimental measurements for the pressure coefficient at the top surface of the JSAE model at y/W= 0.125 for x/L > 0.5

being simulated. When looking at pressure coefficients measured on the car body surface (see Figure 3 for locations), Figures 9 through 12 show comparatively how well AcuSolve (Inflow 2), STAR-CCM+ v7.06 (SST k-ω), FloEFD, and ANSYS Fluent R14.5 matched the experimental data for the two cases being considered.

Finally, Figure 13 illustrates the experimental test results from particle imaging velocimetry (PIV) measurements as a contour plot post-processed to compare with most of the CFD simulation software predictions. FloEFD and STAR-CCM+ (SST k-ω) can be seen to most closely match the wind-tunnel experimental results the best.

Conclusions

Although the meshing and solver technology of FloEFD is a non-traditional CFD approach, this JSAE blind benchmark has proven that FloEFD is as accurate as, or better than, other traditional commercial CFD software in a difficult automotive external aerodynamic study. FloEFD ranks well alongside STAR-CCM+, whilst using fewer cells, a single sophisticated turbulence model, and lower CPU times to achieve the same level of results. In addition, it was worth pointing out that with FloEFD's out-of-the-box software, it also takes less time to set-up and optimally mesh the automotive body model, in addition to activating the single transitional k-Σ turbulence model compared to the many turbulence models some of the other codes employed, that required expert interventions.

References

[1] Nakashima, Takuji; Sasuga, Nobuhiro; Ito, Yuichi; Ikeda, Masami; Ueda, Iichiro; Kato, Yoshihiro; Kitayama, Masashi; Kito, Kozo; Koori, Itsuhei; Koyama, Ryutaro; Shimada, Yoshihiro; Hanaoka, Yuji; Higaki, Tatsuhiko; Fukuda, Kota; Yamamura, Jun; Li, Ye (2013). Benchmark of Aerodynamics CFD of Simplified Road Vehicle Model: JSAE. Paper Number: 20134343. Pages 8 – 28, http://bookpark.ne.jp/jsae/pdf_e.asp

[2] Advanced Immersed Boundary Cartesian Meshing Technology in FloEFD: Mentor Graphics, http://go.mentor.com/2gogl

[3] Enhanced Turbulence Modelling in FloEFD™: Mentor Graphics, http://go.mentor.com/2glzd

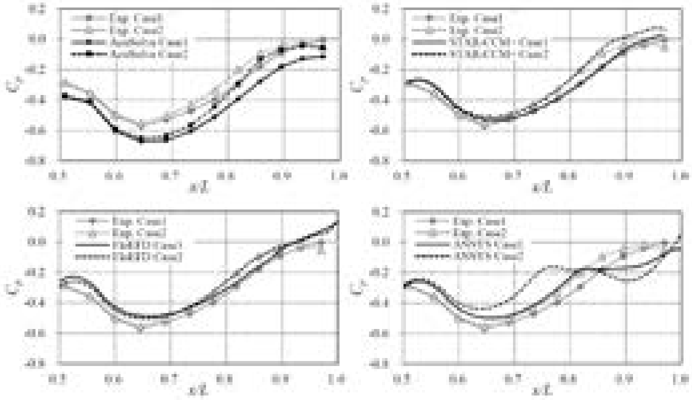

Figure 10. Comparative CFD results versus experimental measurements for the pressure coefficient at the top surface of the JSAE model at y/W = 0.125

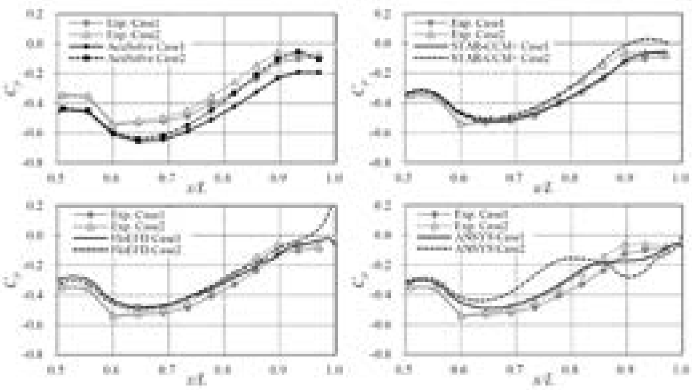

Figure 11. Comparative CFD results versus experimental measurements for the pressure coefficient at the top surface of the JSAE model at y/W = 0.25 for x/L > 0.5

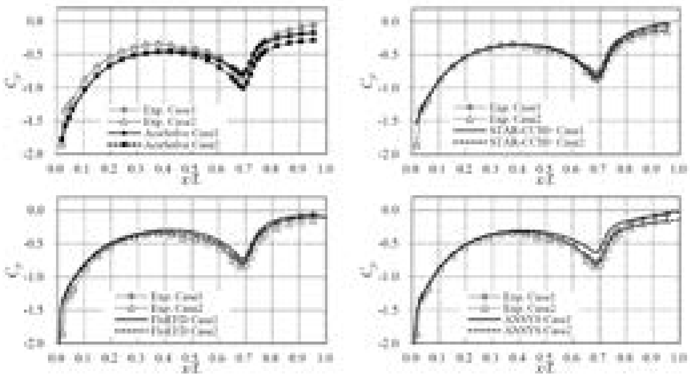

Figure 12. Comparative CFD results versus experimental measurements for the pressure coefficient at the top surface of the JSAE model at y/W = 0.0

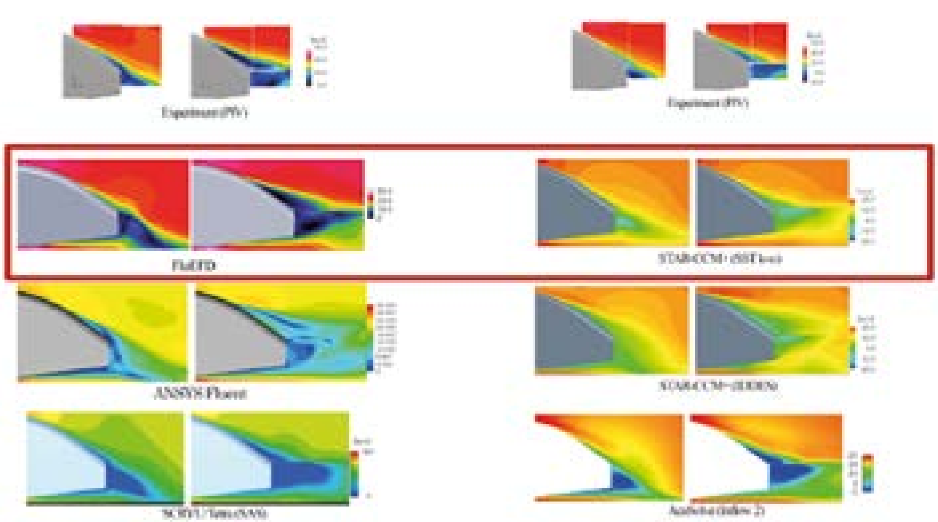

Figure 13. Centerline velocity distribution test results versus CFD code predictions for JSAE Cases 1 and 2