BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20-24 2008

VALIDATION OF CFD FOR THE FLOW UNDER A TRAIN WITH 1:7 SCALE WIND TUNNEL MEASUREMENTS

Mikael Sima[*], Andreas Gurr[†], and Alexander Orellano[†]

[*]Bombardier
Mainline and Metros, Dept. Specialist Eng., Aero- and Thermodynamics, SE – 721 73 Västerås, Sweden
e-mail: mikael.sima@se.transport.bombardier.com,

[†] Bombardier
Mainline and Metros, Dept. Specialist Eng., Aero- and Thermodynamics, Am Rathenaupark, 16761 Hennigsdorf, Germany
e-mails: andreas.gurr@de.transport.bombardier.com, alexander.orellano@de.transport.bombardier.com

Keywords: Underfloor aerodynamics, high-speed train, CFD validation, RANS, DES

Contact person:

Name: Mikael Sima

Affiliation: Bombardier

Postal address: Mainline and Metros, Dept. Specialist Eng., Aero- and Thermodynamics,
SE – 721 73 Västerås, Sweden

Email: mikael.sima@se.transport.bombardier.com

Phone: +46-21-317230

Fax: +46-21-317111

1 INTRODUCTION

The investigation of the underfloor flow field of high-speed trains is motivated by several aspects which affect system security and performance: cruising speeds are approaching values where ballast projection becomes a serious issue, requiring the optimization of the underbelly design in order to reduce the aerodynamic load on the track bed. Since relative movement between train and ground is required the alternatives for measurements are full-scale tests and wind-tunnel tests over moving belts or with moving model rigs. These all pose significant challenges to get even limited data about the flow and influence of train geometry, as well as being costly. It is therefore of great interest to use CFD as a tool for flow assessment and investigation of the effects of design changes, complementary to full-scale measurements.

The flow region in the gap between the underbelly of the vehicle and the trackbed poses a challenge to turbulence modeling since the highly-turbulent non-equilibrium flow is characterized by features which have shown to be difficult to capture within the classical steady RANS approach: the Couette type flow is regularly interrupted by the passing bogies and the cross-sectional changes at the inter-car connections. It is characterized by flow separation and formation of vortices at sharp edges and behind bluff bodied geometry details, by large-scale unsteadiness, and by intense secondary flow. With the complex geometry of the bogies, the large scale and high Reynolds number of the flow under the train, even when focusing on the short section from the middle of one car to the middle of the other, there will be a restriction on mesh size by available computational resources.

Thus, it is necessary to validate the numerical approach; by finding a sufficient and feasible mesh size, determining the influence of convective scheme and turbulence modeling. A specific generic underfloor geometry in scale 1:7 was designed for this purpose within the research cooperation “Aerodynamics in Open Air”. Four partners DB AG, Bombardier, Siemens AG and CAF dedicated resources to a laboratory investigation carried out in 2006 at the ISTA at TU Berlin, which is reported in an accompanying publication [1]. With a numerical model of the wind tunnel set-up, simulated and measured data is compared. The simulations cover the influence of mesh, convective discretisation and turbulence modeling including RANS, URANS and DES. The study was restricted to models and mesh available in commercial software.

2 GEOMETRY AND MESH

The wind tunnel test was on a generic underfloor geometry in scale 1:7 representing motor bogie, inter car gap and trailer bogie. Since the main flow features are expected – and was also seen in the measurements – to occur before the trailer bogie, the computational effort was reduce by leaving out the trailer bogie and cavity, Fig. (1). The domain extent was:

- *x*: -1085 to 833 *mm; y*: - 250 to 250 *mm; z*: -109.5 to 68 *mm*

The simulations were carried out with the code Star CD V4.0, mainly on a mesh with polyhedral cells, Fig (2), but to not have restrictions to one code also a tetrahedral mesh was used. The mesh was designed to cater for DES, and no coarsening for RANS was investigated. The inlet and outlet sections were extruded from the volume mesh. The general mesh size was 4.3 mm. Some refinements were used on the bogie surface mesh to resolve small cylindrical parts, Fig. (2). This was a restriction due to overall mesh size and gives only a coarse resolution of the complex bogie geometry. In this sense the study has an industrial engineering focus rather than scientific. Since the interest of the underfloor flow simulation is mostly on the track side rather than the train side, it can be acceptable to have local areas of lower accuracy if the effect on the overall flow field is small. The second aim with the study is to show the accuracy and deficiencies of various approaches with given restrictions. Mesh refinement was done in two steps, first the volume mesh shear layers where refined, Fig. (3), secondly the prism layers was refined normal to the walls such that the no.cells doubled within the same total prism layer height.

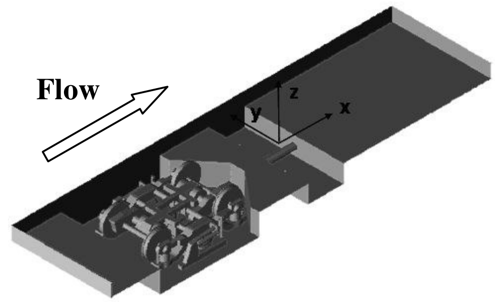

Figure 1: Computational domain.

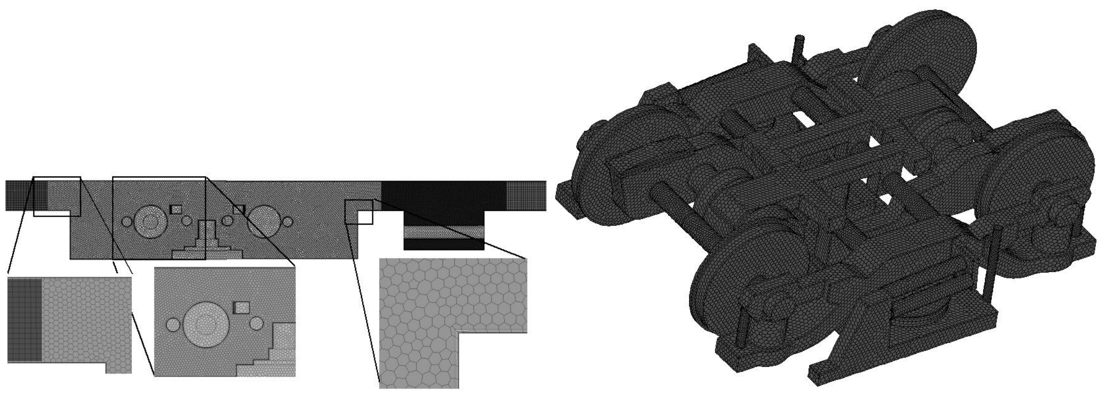

Figure 2: Computational polyhedral. Left: volume mesh. Right: bogie surface mesh.

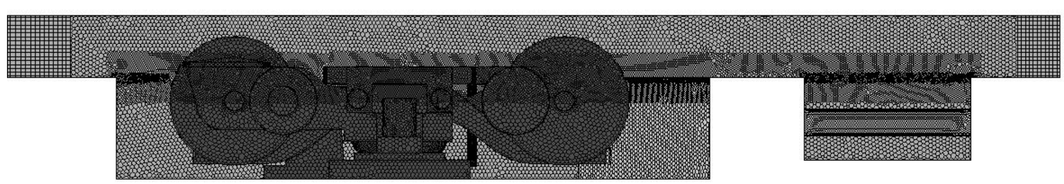

Figure 3: Surface mesh showing shear layer refinements.

3 SIMULATIONS AND RESULTS

Table 1 and 2 show the simulations performed. Within the AOA project Politecnico di Milano performed a RANS simulation using Fluent and realizable k-ε turbulence model and DB AG a LES simulation using CFX, both on the tetrahedral mesh. The reason for the LES on a non-purpose mesh was that the DES failed.

The simulations were mainly on a low-Re mesh with y^+ about 2. A high-Re mesh was also tested, however, the RANS solution was transient and with DES the transition to the LES region was already after the first cell and the results was not good. The conclusion was that the Reynolds number was to low to test a high-Re approach. The mesh refinements showed only small and local changes to the flow field in relation to the discrepancies with measurements.

In combination with the good correspondence to measurements achievable with DES, it was concluded that the reference mesh is sufficient to assess the influence of simulation approach.

No.	Name	Low / high-Re	Mesh no.	Turbulens model	Convective discretisation	Comment
1	**BT-Qke**	low-Re	1	Quadratic k-ε	Mars 0.5	
2	**BT-Qke tet**	low-Re	1,tet.	Quadratic k-ε	Mars 0.5	
3	**BT-Ske**	low-Re	1	Standard k-ε	Mars 0.5	
4	**BT-kOmegaSST**	low-Re	1	k-ω SST	Mars 0.5	URANS
5	**BT-Qke Vref**	low-Re	2	Quadratic k-ε	Mars 0.5	
6	**BT-Qke Vref Pref**	low-Re	3	Quadratic k-ε	Mars 0.5	
7	**BT-Qke M09**	low-Re	1	Quadratic k-ε	Mars 0.9	
8	**BT-Qke Lsmall**	low-Re	1	Quadratic k-ε	Mars 0.5	1/10th turb. length scale
-	**-**	low-Re	1	k-ω SST	Mars 0.5	Unsteady, no solution
-	**-**	high-Re	4	Quadratic k-ε	Mars 0.5	Unsteady, no solution

Table 1: RANS and URANS simulations.

No.	Name	Low / high-Re	Mesh no.	Turbulens model	Convective discretisation	Courant no.	Comment
9	**BT-DES**	low-Re	1	k-ε	Blend Mars 1.0 &CD	Max 57 Mean 0.11	Time step 0.00567 h/U_{ref}
10	**BT-DES dt5**	low-Re	1	k-ε	Blend Mars 1.0 &CD	Max 199 Mean 0.36	Time step 0.0189 h/U_{ref}
11	**BT-DES HiRe**	high-Re	1	k-ε	Blend Mars 1.0 &CD	Max 117 Mean 0.09	Time step 0.00567 h/U_{ref}
12	**BT-DES M09**	low-Re	1	k-ε	Mars 0.9	Max 177 Mean 0.36	Time step 0.0189 h/U_{ref}
13	**BT-DES SA M05**	low-Re	1	S-A	Mars 0.5	Max 7.4 Mean 0.27	Time step 0.0189 h/U_{ref}

Table 2: DES simulations.

With the k-ω *SST* RANS turbulence model it was not possible to have a steady solution. It was instead necessary to solve this with unsteady RANS, however, the solution was less accurate than both with the quadratic and standard k-ε models.

The detached eddy simulations solves in RANS mode near walls and switches to LES away from walls, employing the RANS model for the sub-grid scales. Hence the simulation is transient by nature and resolves the larger scales of the turbulence. When there is large scale separation the DES approach should improve on the accuracy of the solution. The flow in the BIAC at the forward edge of the bogie cavity separates much like a backward facing step. A shear layer forms and hits the trackside parts of the bogie. Around these there is separation about rounded shapes and sharp edges, reattachment and split of the flow where some goes into the cavity. At the funnel shaped downstream bogie cavity end the flow is collected and directed towards the track at the apex, causing massive separation and a recirculation zone downstream. Around the angled funnel edges two counter rotating longitudinal vortices form. The inter car gap is a cavity with recalculating flow with a lateral component.

With the flow features mentioned above it is expected that the RANS approach have difficulties that potentially DES can resolve. However, DES is still developing and as discussed in [2] there are known deficiencies. The simulations showed that for this flow the DES model and convective scheme had significant impact on the results. Examples of the results are shown in Fig. (4), comparing the “best” RANS and DES, simulations no. 1 and 9, with measurements. Over the bogie the RANS is better and after the funnel, in the central part of the channel the RANS has difficulties whereas DES captures the flow well.

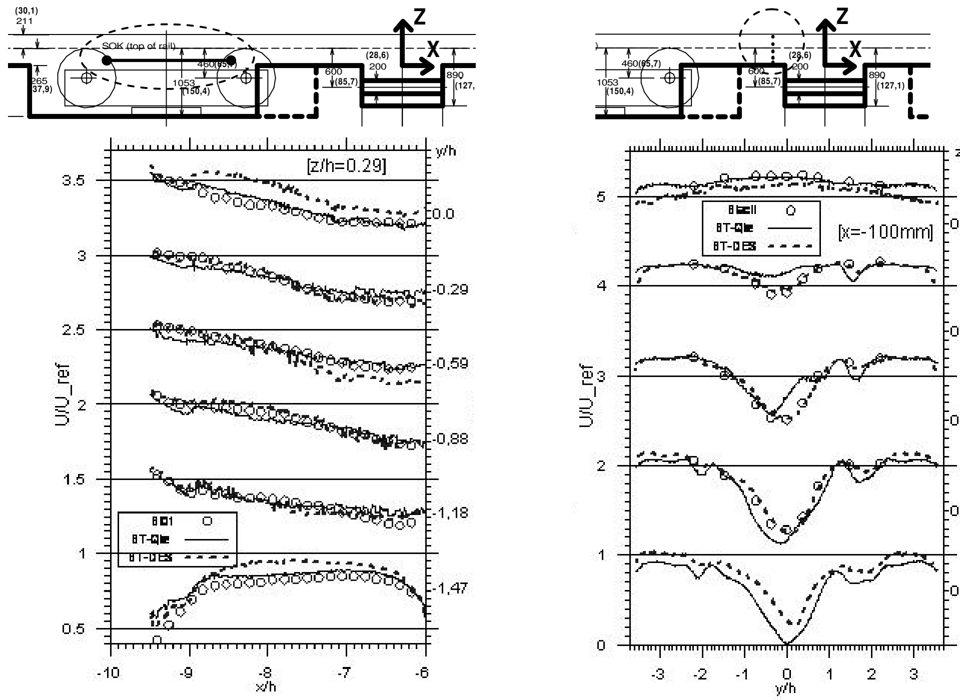

Figure 4: Non-dimensional axial velocity. Left: just above the bogie at various lateral positions offset by 0.5. Right: In the channel just upstream of the inter car gap at various heights offset by 1.0. Coordinates non-dimensional with the channel height, *h*.

4 SUMMARY

The study shows the drawbacks and benefits of numerical approaches available in commercial software for an engineering application by comparison with measurements on a generic train underfloor arrangement.

ACKNOWLEDGEMENTS

Partial funding under grants 19 S 5009 A and 19 S 5009 B from the German Ministry of Economics is acknowledged.

REFERENCES

[1] H.-J. Kaltenbach, I.A. Portillio and M. Schober. A generic train-underfloor experiment for CFD validation. *Abstract submitted to BBAA VI, Milano, 2008*

[2] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. Kh. and Travin, A.: A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theor. Comput. Fluid Dyn.,* **20**, 181-195, 2006