Using a spectral/hp element method for high-order implicit-LES of bluff automotive geometries

Using a Spectral/hp Element Method for High-order Implicit-LES of Bluff Automotive Geometries

Filipe Fabian Buscariolo[1,2], Walid Hambli[1], James Slaughter[1*], Spencer Sherwin[1]

[1] Department of Aeronautics, Imperial College London, London, UK.
f.fabian-buscariolo16@imperial.ac.uk, w.hambli17@imperial.ac.uk, j.slaughter19@imperial.ac.uk, s.sherwin@imperial.ac.uk

[2] Department of Mechanical Engineering, University of São Paulo, São Paulo, Brazil.

Abstract

The combination of High-order methods and Large-Eddy Simulation (LES) is an ongoing research focus in turbulence due to the attractive dissipation characteristics of high-order methods. Whilst numerically speaking these methodologies are advantageous, their application is inhibited on industrial cases due to the inherent geometric complexities of such problems. Spectral/hp Element (SEM) solvers such as Nektar++, [1] have potential to be bridge the gap between high-order methods and industrial geometric complexity as shown by [2]. This study focuses on the intersection of the application of the SEM solver Nektar++ to an automotive geometry as well as the presentation of high-order mean flow characteristics for the SAE Notchback body. Using a 5^{th} order polynomial expansion at $Re_L = 2.3 \times 10^6$ on a curvilinear grid, results are compared with those empirically achieved by [3]. Implicit Sub-Grid scale modelling along with a novel Spectral-Vanishing Viscosity (SVV) approach is employed acting as an artificial diffusion operator preventing high-frequency instabilities and spurious oscillations. Suitable qualitative agreement between PIV and CFD methods is obtained, and quantitative agreement is demonstrated on C_D with 9% difference. More extensive backlight separation and subsequent bootlid impingement is observed in CFD than presented by [3]. This might be caused due to differing inflow characteristics, resulting in C_M and C_L variance to experimental values. Along with the mean flow field characteristics, the methodology and the pipeline used to achieve such results and agreement is presented. The use of a wall-conforming unstructured curvilinear grid allows for significantly greater geometric flexibility whilst retaining the advantages of the high-order polynomial expansion.

Key words: *Spectral/hp Element Methods; Implict LES; Vehicle Aerodynamics; Automotive Aerodynamics; High-order Methods*

1 Introduction

Understanding the aerodynamic behaviour of an automotive body is of significant interest to researchers and engineers due to the possibility of reducing fuel consumption and to have a positive contribution on reducing overall CO_2 emissions.

The use of reference or generic bluff bodies for automotive aerodynamic studies started in 1978 with [4], by proposing a bluff body with automotive characteristics, assembled close to ground. These bluff bodies have been utilised as a means to test novel numerical methods relevant to automotive flow fields due to their relative geometric simplicity whilst retain relevance of salient geometric and flow features.

The bluff body proposed by Morel was further improved and complemented by different authors and creating new geometries, including the Ahmed body which investigate in particular the flow topology of the backlight and vehicle wakes. Car-shaped bluff bodies, such as the SAE Notchback body, are gaining popularity in academic research due to the complex interaction between A-Pillar, backlight, bootlid and wake flows.

This particular case is based on the SAE 20° backlight angle notchback model, proposed by [5], however it incorporates a small diffuser to the geometry. The main dimensions of the 20% scale model are: length L of 840mm, width of 320mm and height H of 240mm with reference frontal area estimated in 0.076 m^2. The model is mounted in the wind tunnel with four pins at a ground clearance of 40mm and zero pitch.

Experiments for the SAE Notchback body were performed by [3], in the Loughborough University wind tunnel, considering working section of 1940mm wide and 1320mm high, with static floor condition and Reynolds number, based on the length of the body of $Re = 2.3 \times 10^6$. The static floor leads to the development of a boundary layer, where measurements indicated its thickness to be 40-60mm, which is considered very thick and might influence measurements. We present next the numerical method and simulation methodology employed in this study.

2 Spectral/hp Element Method iLES Simulations

The aerodynamic behaviour of automotive geometries is characterized by highly separated and recirculated flows. Numerical methodologies such as the Reynolds-Averaged Navier Stokes (RANS) methods, where turbulence is modelled are widely used in industry cases, have proven to be insufficient in spatio-temporal resolution to fully predict such complex flow. High-resolution methodologies that resolve turbulence rather than modelled, such as in DDES and uDNS/iLES, have become popular choices in for automotive applications to resolve wake interactions.

In combination with high-resolution uDNS/iLES simulation technique, we employed a high-fidelity spectral/hp element discretisation, which is on the frontier between spectral-element (SEM) and finite-element methods (FEM), as outlined by [6].The domain is decomposed into sub-domains of characteristic size h and the solution is approximated in each sub-domain by a high-order polynomial expansion of order p. These are the main characteristics of the CFD code Nektar++ ([1]) selected for this study. Nektar++ combines both the flexibility of methods based on domain decomposition and exponential convergence properties of SEM.

Low numerical diffusion characteristics of high-order models allows for more accurate representation of vortices and vortex interaction over longer distances. However, spurious oscillations caused by the sub-grid scales (SGS) are a major source of instability in simulations due to its low numerical diffusion and must be damped after appropriate dealiasing techniques have been applied [7]. The simulations also consider a Spectral Vanishing Viscosity (SVV) operator, which can be defined as an artificial diffusion term, designed to cancel out the effect of such oscillations.

Moura et al. [8] proposed that the amount of SVV added should be proportional to the product of local velocity with the mesh spacing, turning the Péclet number to be constant and controlled all over the domain. Based on this base concept, [9] implemented a novel CG-SVV scheme with a DG mimicking Kernel. The basic concept of this SVV lies on matching the dissipation curve of CG of order p to the DG with order $p-2$, removing the instabilities on CG dissipation curve at very high Reynolds numbers. Simulations here presented consider the SAE Notchback body assembled in a virtual wind tunnel with similar test section as the Loughborough University wind tunnel from the experimental reference [3]. The length of the domain is defined as 7 L and the body is located at the centre, with a total distance of 3 L from both inlet and outlet.

The meshing generation process starts from a classical linear base mesh which is projects on the surface of the studied body resulting in a high-order mesh with curved elements. The curved mesh is generated by NekMesh, the meshing tool of Nektar++. Solution polynomial order in each element has been set to $p = 5$, resulting in approximately 56 million degrees of freedom in total, based on a relatively coarse linear base mesh. The base mesh and the high-order mesh can be seen on figure 1. The non dimensional time step, based on the free stream velocity U_∞ and the length L of the body has been set to $\Delta t = 2.5 \times 10^{-5}$, considering the Reynolds number of $Re = 2.3 \times 10^6$. Results for all quantities presented were averaged for 3.5 convective time units (CTU) based on the same length L. Sub-stepping approach with the condition $CFL = 1$ has been adopted and drag, lift and side force have been monitored to assess when numerical transience has been washed through the domain.

3 Results

Centreline Cp profile results in Figure 2 demonstrates reasonable agreement over the majority of the body, between simulation results and the experiment of [3]. However, due to the pressure tapping locations, the experiments fail to resolve the pressure peaks at the shield and backlight junctions, compromising the possibility of making proper comparisons at these locations.

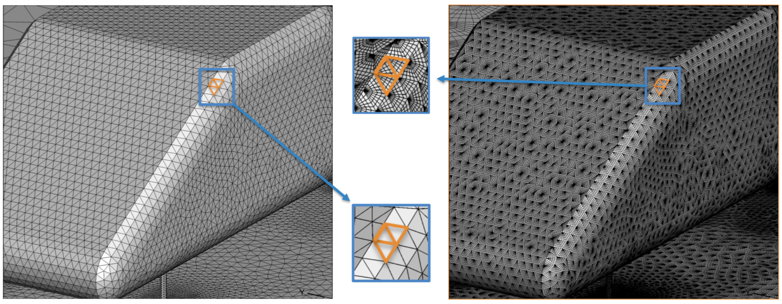


Figure 1: Mesh comparison analysis: linear mesh (left-hand side) and high order mesh, based on the linear mesh (right-hand side).

We observe result variance between $-0.45 < \frac{x}{L} < -0.2$ over the backlight and bootlid surfaces, where more extensive separation is seen, when compared to experiments. The higher Cp value observed from simulation results at $\frac{x}{L} \approx 0.45$ is caused by a slightly larger bootlid impingement from the wake.

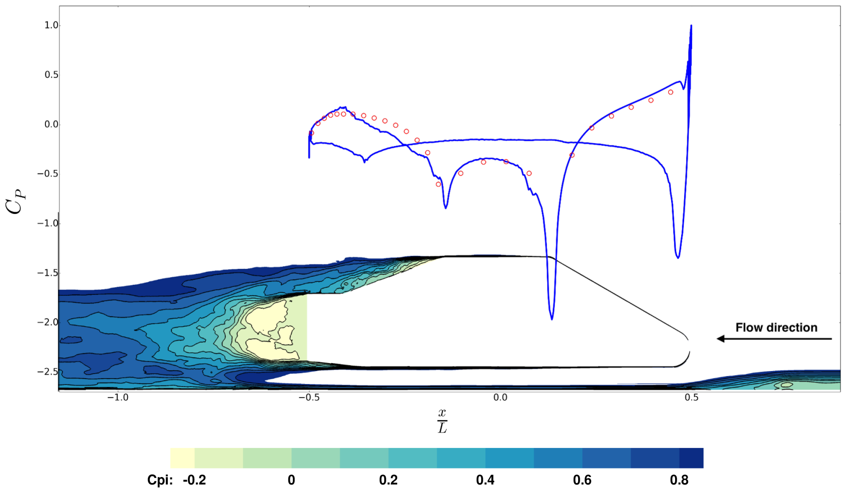
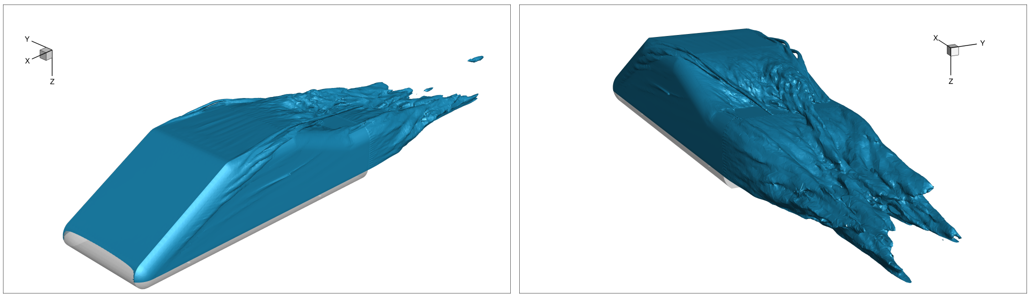


Figure 2: Cp Distribution on centreline of the SAE body overlayed with contour of Cp_{tot} on the centre plane.

The backlight separation in Figure 2 is limited to a narrow region around the centre line as shown by the Cp_{tot} isosurface presented in Figure 3a. Away from the centre line this separation does not seem to occur. A-pillar vortices roll up over the top of shield and roof before seemingly separating from the body before the C-pillar and backlight at the roof junction. The filaments after rolling up on the A-pillar separate over the backlight and then converge towards the centre line as shown in Figure 3b.

4 Conclusions

This study presented numerical simulations using the spectral/hp element method applied to the SAE Notchback automotive bluff body. Latest developments on SVV stabilization and meshing generation techniques are employed to deliver high-fidelity results and numerical results are compared with experimental reference. Comparative results over the body centreline indicate similar Cp distribution trend on the top surface. Larger separation on the backlight is observed on the simulation, when comparing flow structures defined by isosurfaces of Cp_{tot}.
Drag and pitching moment coefficients obtained are, respectively 0.190 and -0.1233. Comparing to experimental results, the drag coefficient has a 9% difference, whereas pitching moment magnitude



(a) A-Pillar detail. (b) C-Pillar and Bootlid details.

Figure 3: Isosurface plots of Cp_{tot} over the SAE Notchback body.

is higher but in the same direction. The lift coefficient obtained is -0.078, similar value as the experiment, however with opposite sign. A similar tendency for the last two coefficients was presented by most participants of the 1^{st} Automotive CFD Prediction Workshop. Due to the body characteristics negative lift is expected and an additional experimental campaign would enhance reliability of currently available results. A longer averaging period here could yield results closer to experimental data.

References

[1] C.D. Cantwell, D. Moxey, A. Comerford, A. Bolis, G. Rocco, G. Mengaldo, D. De Grazia, S. Yakovlev, J.-E. Lombard, D. Ekelschot, B. Jordi, H. Xu, Y. Mohamied, C. Eskilsson, B. Nelson, P. Vos, C. Biotto, R.M. Kirby, and S.J. Sherwin. Nektar++: An open-source spectral/hp element framework. *Computer Physics Communications*, 192:205–219, jul 2015.

[2] F. F. Buscariolo, S. J. Sherwin, G. R. S. Assi, and J. R. Meneghini. Spectral/hp iles-svv simulation methodology study on an ahmed body squared back. Technical report, SAE Technical Paper, 2018.

[3] D. Wood, M.A. Passmore, and A.K. Perry. Experimental data for the validation of numerical methods-sae reference notchback model. *SAE International Journal of Passenger Cars-Mechanical Systems*, 7(2014-01-0590):145–154, 2014.

[4] T. Morel. Aerodynamic drag of bluff body shapes characteristic of hatch-back cars. Technical report, SAE Technical Paper, 1978.

[5] A. Cogotti. A parametric study on the ground effect of a simplified car model. *SAE transactions*, pages 180–204, 1998.

[6] G. Karniadakis and S. Sherwin. *Spectral/hp Element Methods for Computational Fluid Dynamics*. Oxford University Press, jun 2005.

[7] G. Mengaldo, D. De Grazia, D. Moxey, P.E. Vincent, and S.J. Sherwin. Dealiasing techniques for high-order spectral element methods on regular and irregular grids. *Journal of Computational Physics*, 299:56–81, oct 2015.

[8] R.C. Moura, S.J. Sherwin, and J. Peiró. Eigensolution analysis of spectral/hp continuous galerkin approximations to advection–diffusion problems: Insights into spectral vanishing viscosity. *Journal of Computational Physics*, 307:401–422, feb 2016.

[9] R. C. Moura, M. Aman, J. Peiró, and S. J. Sherwin. Spatial eigenanalysis of spectral/hp continuous galerkin schemes and their stabilisation via DG-mimicking spectral vanishing viscosity for high reynolds number flows. *Journal of Computational Physics*, 406:109112, apr 2020.