1st Automotive CFD Prediction Workshop

Case 2a – DrivAer Fastback Variant
Case 2b - DrivAer Estate Variant

V1.7 - 18/10/2019

Neil Ashton – neil.ashton@eng.ox.ac.uk – University of Oxford
Vangelis Skaperdas - vangelis@beta-cae.com – BETA-CAE Systems
Summary

- The DrivAer automotive model represents a modern automotive vehicle that is available in several rear-end variants. For this workshop the **Fastback (F)** and **Estate (E)** variants are chosen, as shown in *Figure 1*.

- According to the definitions set by TUM ([http://www.aer.mw.tum.de/en/research-groups/automotive/drivaer/geometry/](http://www.aer.mw.tum.de/en/research-groups/automotive/drivaer/geometry/)) the following configurations have been chosen:
  - F_S_wM_wW
  - E_S_wM_wW
  - i.e the fastback (F)/estate (E) with a smooth underbody (S) together with mirrors (wM) and with wheels (wW). In addition these are moving ground simulations so both the ground boundary will move and the wheels will rotate.

*Figure 1 – DrivAer variants*
Geometry

- The geometry is available on the website in .step format (additional formats can be provided upon request) - *DrivAer_Estate.step.gz* and *DrivAer_Fastback.step.gz*. Please use this geometry rather than from any other source. It is provided in 1:1 scale, in metres, and is a complete vehicle (not half-car).
- The provided CAD includes the required domain for the far-field boundaries, which are shown in the Figures below:
  - Domain inlet 56.7m upstream of car
  - Front wheel axis at x=z=0m (wheel axes are located at a distance of 0.3165m from the ground level)
  - Road located at z=-0.3165m
Surface Mesh

The surface mesh is available as separate .stl files for those wishing or needing a surface mesh to generate their own meshes (which should be provided to the organizers for reference). These are available on the website as DrivAer_Estate.stl.gz and DrivAer_Fastback.stl.gz (additional formats are available upon request). These are in 1:1 scale and in metres (m)
Properties of STL surface meshes

- The table to the right shows all the boundaries that are contained within the provided surface meshes. These have been split to provide the opportunity to provide more detailed splits of the forces as well as allowing greater flexibility for other mesh generators.

<table>
<thead>
<tr>
<th>ID</th>
<th>FastBack</th>
<th>Estate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A-pillar_common</td>
<td>A-pillar_common</td>
</tr>
<tr>
<td>2</td>
<td>B-pillar_fastback</td>
<td>B-pillar_estate</td>
</tr>
<tr>
<td>3</td>
<td>body_common</td>
<td>body_common</td>
</tr>
<tr>
<td>4</td>
<td>body_rear_fastback</td>
<td>body_rear_estate</td>
</tr>
<tr>
<td>5</td>
<td>bumper_fr</td>
<td>bumper_fr</td>
</tr>
<tr>
<td>6</td>
<td>bumper_rr</td>
<td>bumper_rr</td>
</tr>
<tr>
<td>7</td>
<td>C-pillar_fastback</td>
<td>C-pillar_estate</td>
</tr>
<tr>
<td>8</td>
<td>D-pillar_fastback</td>
<td>D-pillar_estate</td>
</tr>
<tr>
<td>9</td>
<td>domain_inlet</td>
<td>domain_inlet</td>
</tr>
<tr>
<td>10</td>
<td>domain_outlet</td>
<td>domain_outlet</td>
</tr>
<tr>
<td>11</td>
<td>domain_sides</td>
<td>domain_sides</td>
</tr>
<tr>
<td>12</td>
<td>domain_top</td>
<td>domain_top</td>
</tr>
<tr>
<td>13</td>
<td>front_intakes</td>
<td>front_intakes</td>
</tr>
<tr>
<td>14</td>
<td>lights_front</td>
<td>lights_front</td>
</tr>
<tr>
<td>15</td>
<td>road_far</td>
<td>road_far</td>
</tr>
<tr>
<td>16</td>
<td>road_near</td>
<td>road_near</td>
</tr>
<tr>
<td>17</td>
<td>side_mirrors</td>
<td>side_mirrors</td>
</tr>
<tr>
<td>18</td>
<td>spoiler_fastback</td>
<td>spoiler_estate</td>
</tr>
<tr>
<td>19</td>
<td>tyres_fr</td>
<td>tyres_fr</td>
</tr>
<tr>
<td>20</td>
<td>tyres_rr</td>
<td>tyres_rr</td>
</tr>
<tr>
<td>21</td>
<td>underbody_smooth</td>
<td>underbody_smooth</td>
</tr>
<tr>
<td>22</td>
<td>wheels_fr</td>
<td>wheels_fr</td>
</tr>
<tr>
<td>23</td>
<td>wheels_rr</td>
<td>wheels_rr</td>
</tr>
<tr>
<td>24</td>
<td>windows_common</td>
<td>windows_common</td>
</tr>
<tr>
<td>25</td>
<td>windows_fastback</td>
<td>windows_estate</td>
</tr>
</tbody>
</table>
Volume Mesh

- The Domain contains 21 volumes, 1 main and 20 MRF volumes in the front and rear wheel pockets

- Axes of rotation are aligned to global y-axis. Origin points are shown here below in \textit{mm}:
Summary

- All of the provided meshes (surface and volume) were generated in ANSA v19.1.2 by BETA-CAE Systems. Three meshes are provided: **Coarse**, **Medium** & **Fine** which will now be described.

- The provided unstructured volume meshes contain prismatic near-wall layers to achieve $y^+<1$ layers and are combined with a hex-dominant mesh with poly transition zone and tetras zone to connect with layers. This is referred to as **HexaPoly** (according to ANSA terminology).
Summary

- Two further mesh types are provided for those codes that cannot use the hexpoly format or for those wishing to investigate the effect of different unstructured mesh types.
- These are shown below and are available for all mesh types.

HexaPoly
HexaInt
HexaIntConv
Setup of refinement zones based on superimposed initial simulation results

These size boxes are available in the same folder as the meshes as .stl’s for those wishing to produce their own meshes with similar refinements.
### Layer Growth rate

<table>
<thead>
<tr>
<th>Refinement level</th>
<th>Specs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>17 layers, 1&lt;sup&gt;st&lt;/sup&gt; height =0.018mm Variable growth rate from 1.3 to 1.5</td>
</tr>
<tr>
<td>Medium</td>
<td>22 layers, 1&lt;sup&gt;st&lt;/sup&gt; height =0.018mm Variable growth rate from 1.05 to 1.4</td>
</tr>
<tr>
<td>Fine</td>
<td>26 layers, 1&lt;sup&gt;st&lt;/sup&gt; height =0.018mm Variable growth rate from 1.05 to 1.3</td>
</tr>
</tbody>
</table>
Near wall imposed orthogonality

First five layers are purely orthogonal
## Volume mesh sizes (in millions)

<table>
<thead>
<tr>
<th></th>
<th>FastBack HexaPoly (million cells)</th>
<th>FastBack HexaIntConv (million cells)</th>
<th>Estate HexaPoly (million cells)</th>
<th>Estate HexaIntConv (million cells)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Coarse</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 mil trias on surface</td>
<td>93</td>
<td>76</td>
<td>92</td>
<td>75</td>
</tr>
<tr>
<td>17 layers</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Medium</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.2 mil trias on surface</td>
<td>165</td>
<td>140</td>
<td>162</td>
<td>138</td>
</tr>
<tr>
<td>22 layers</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Fine</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.5 mil trias on surface</td>
<td>258</td>
<td>224</td>
<td>250</td>
<td>217</td>
</tr>
<tr>
<td>26 layers</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Boundaries

- The table to the right shows all the boundaries that are contained within the provided meshes. These have been split to provide the opportunity to provide more detailed splits of the forces as well as allowing greater flexibility for other mesh generators.
Files available for download

The meshes are available in four mesh formats:
- Fluent (*.msh)
- StarCCM+ (*.ccm)
- OpenFOAM (binary double precision)
- CGNS ADF mixed (*.cgns)

Units are **metres** and additional file formats are available upon request.
Mesh refinement study – HexaPoly meshes

Coarse 93 million
Mesh refinement study – HexaPoly meshes

Medium 165 million
Mesh refinement study – HexaPoly meshes

Fine 258 million
Mesh quality metrics

Statistics for Fine HexaPoly mesh
253 million cells (168 million prisms, 90 million core)
Mesh quality metrics

Statistics for Fine HexaPoly mesh
253 million cells (168 million prisms, 90 million core)

Quality violations limited in areas with sharp corners and very thin layers for $y^+$ target value.
Boundary Conditions and setup

The required conditions are:

1. \( \text{Re} = 4.89 \times 10^6 \) (based upon the freestream velocity \( U = 40 \text{ms}^{-1} \), model scale vehicle length \( L = 1.84 \text{m} - 40\% \text{ scale} \), density \( \rho = 1.204 \) and viscosity \( \mu = 1.813 \times 10^{-5} \) all assuming atmospheric pressure i.e 101325pa at 20c) – this is to match the large body of work that has been done to date at the model scale Reynolds number (e.g original Heft et al. work).

2. Given that the supplied geometry and meshes are full-scale the velocity/viscosity must be scaled to match this Reynolds number. i.e reference full-size vehicle length is 4.6m, so freestream velocity should be: 16.0m/s rather than the 40ms\(^{-1} \) that would have been required using the model-scale (40%) vehicle. Best practices should be used for the particular turbulence model you are using i.e for the k-Omega SST variants a turbulent viscosity ratio of 10 and freestream intensity of 0.1%. Please supply the values which were used in your submitted results & presentation.

3. The values to non-dimensionalize the forces e.g \( C_L, C_D \) assuming full-size geometry are Velocity=\( 16 \text{ms}^{-1} \), Area=\( 2.16 \text{m}^2 \), density=\( 1.204 \)
Boundary Conditions and setup

- The side walls (domain_side) should be set as symmetry/far-field condition to replicate a free-air condition (i.e. we are not simulating the WT environment).

- To ensure a suitable floor boundary layer, the boundary underneath the car (road_near) should be set to match the freestream velocity with the farfield floor boundary (road_far) to a slip condition. Any changes to these recommendations should be justified.

- There are Moving Reference Frame (MRF) regions within all four wheels so these should be set appropriate to the chosen freestream velocity. Finally the rotating components of the tyres should be set to a rotational speed according to your freestream velocity and the tyre radius i.e. 0.3165m at full-scale.

- The time-step will be different depending on whether the temporal scheme is implicit or explicit but for an implicit scheme we recommend a time-step of $dt = 1.0 \times 10^{-5}$ as a starting point however we encourage participates to show the influence of the time-step in a similar way to the mesh grid sensitivity study.
Experimental data

- There have been numerous experimental campaigns for the DrivAers models. These have been in different tunnels at various Reynolds numbers/scales and the influence of the wind-tunnel itself is likely to have a noticeable influence on the force coefficients and flow-field.
- We therefore are going to be only using these datasets as a guide and to avoid the temptation to match the experimental data we will only present this data at the workshop itself.
- We will look for trends among turbulence models, mesh types, solver settings by looking at the submitted flow-fields and forces.
- Please submit your data as early as possible and we will contact you if your data is significantly out of the range of expected values.
- **It should be noted that Case 1a has been chosen to act as a stronger validation case against experimental data due to the modelling of the wind-tunnel itself.**
Postprocessing (1)

We require from all participants the following data for each case/mesh/model they run i.e separate files for each mesh level/time-step and turbulence model used. These should be compressed in a single .tar.gz (per mesh level/time-step/turbulence model) as described on the submission slide.

The first three files are text files but for the surface data, iso-surfaces and slices in X,Y,Z please could each participate contact neil.ashton@eng.ox.ac.uk to send a test-file to ensure compatibility well before the 25th November deadline.

We are only requesting the mean values (unless otherwise stated) but we encourage you to present unsteady data in your own presentation slot. For all the data shown in the next slides we are assuming the full-size vehicle.

We will cross-plot this data and present it during the workshop as well as making a summary presentation available on the website. We encourage you to show these quantities in your own presentations –the exact scale and image angle/zoom is your choice to best explain your results.
Postprocessing (2)

We require the following text files:

- **Submission_info.dat** (information on your code/mesh and setup: see example file)
- **HPC_info.dat** (information on the hardware and speed of your simulation: see example file)
- **Forces.dat** (iteration/time, $C_D$, $C_L$, $C_{LF}$, Mean $C_D$, Mean $C_L$, Mean $C_{LF}$, Mean $C_{LR}$: see example file)
- **CpWss_top.dat** ($x, y, z, Cp$, $WSS_i$, $WSS_j$, $WSS_k$) taken along symmetry plane and top surfaces: see example file
- **CpWss_bottom.dat** ($x, y, z, Cp$, $WSS_i$, $WSS_j$, $WSS_k$) taken along symmetry plane and bottom surfaces: see example file

- $Cp$=Pressure Coefficient, $WSS$=Wall-Shear Stress in $i,j,k$ components

- Example files available at: [http://autocfd-transfer.eng.ox.ac.uk/Case2a/Submission_Example/](http://autocfd-transfer.eng.ox.ac.uk/Case2a/Submission_Example/)
Postprocessing (3)

For the following files we request the vtk or Ensight format which should be available from the majority of CFD codes. Please include a folder with the following titles which the vtk/ensight should be within with the same naming convention.

- **Case2a_cp_wss**
  - *(Mean Pressure Coefficient and Mean Wall-Shear-Stress (i,j,k components))*

- **Case2a_iso_vel**
  - *(Isosurface of Mean Velocity (i) <0 i.e separated flow)*

- **Case2a_X_XXX**
  - *Mean Velocity (i,j,k and Magnitude), Mean Total Pressure Coefficient, Mean Pressure, Mean Turbulent Viscosity, Mean Spanwise Vorticity, Instantaneous Spanwise Vorticity*

- **Case2a_Y_XXX**
  - *Mean Velocity (i,j,k and Magnitude), Mean Total Pressure Coefficient, Mean Turbulent Viscosity, Mean Spanwise Vorticity, Instantaneous Spanwise Vorticity*

- **Case2a_Z_XXX**
  - *Mean Velocity (i,j,k and Magnitude), Mean Total Pressure Coefficient, Mean Turbulent Viscosity, Mean Spanwise Vorticity, Instantaneous Spanwise Vorticity*
Postprocessing (4)

• Where XXX should be changed to the following locations/numbers for X (based upon full-size car dimensions):
  • 001 – X=-1.25m
  • 002 – X=0.00m
  • 003 – X=1.25m
  • 004 – X=2.75m
  • 005 – X=3.00m
  • 006 – X=3.25m
  • 007 – X=3.50m
  • 008 – X=3.75m
  • 009 – X=4.00m
  • 010 – X=4.25m
  • 011 – X=4.50m
  • 012 – X=4.75m
  • 013 – X=5.00m
  • 014 – X=5.25m
Postprocessing (5)

• Where **XXX** should be changed to the the following locations/numbers for **Y**:
  • **001** – \( Y=-0.625\)m
  • **002** – \( Y=-0.3125\)m
  • **003** – \( Y=0.0\)m (symmetry plane)
  • **004** – \( Y=0.3125\)m
  • **005** – \( Y=0.625\)m

• Where **XXX** should be changed to the the following locations/numbers for **Z**:
  • **001** – \( Z=-0.1875\)m
  • **002** – \( Z=0.00\)m (Wheel axis)
  • **003** – \( Z=0.25\)m
  • **004** – \( Z=0.50\)m
  • **005** – \( Z=0.75\)m
  • **006** – \( Z=0.875\)m

• Velocity should be scaled by your Inlet Velocity e.g \( U=16\)m\(^{-1}\) and spanwise vorticity by \( L/U \) where \( L \) is the length of the vehicle and \( U \) is the inlet velocity. All are full-size dimensions.
Submission

All the requested data should be sent to neil.ashton@eng.ox.ac.uk by November 25th with the subject line being “AUTOCFD: XXX_Lastname” – where XXX is your submission ID (provided once you complete the intention to participate form on the website) and the Lastname of the lead investigator.

The data should be zipped up with the following convention:

XXX_Lastname_Code_Model__CaseXX_v1.tar.gz

Where XXX is the submission ID, Lastname is the lead investigator last name, Code is the CFD code used and Model is the turbulence model and CaseXX is the case number i.e 001_Ashton_Openfoam_SSTDDES_Case2a_v1.tar.gz

Please contact neil.ashton@eng.ox.ac.uk prior to submission if you believe the data will be too large and access to a FTP site can be arranged.