CFD modelling of flow around
Ahmed body

K.V.S.Pavan

Indian Institute of technology Hyderabad

During internship (May 2012- July 2012) at CD-adapco Bangalore
Abstract

Validation study has been done for basic ground vehicle type bluff body called "Ahmed body" Fig-1 of back slant angle of 12.5 degrees. The models (Fig-1) length based Reynolds number is 4.25 million. The model has been tested using segregated and coupled solvers. Drag and pressure coefficients have been monitored and have been compared with the experimental results.

Introduction

Ground vehicles can be termed as bluff bodies moving close to the vicinity of the road. The air flow around the ground vehicles can be classified into two categories internal and external flows. The external flow includes the underbody flows, flow over body surface and wake behind body. The external flow is responsible for over 85% of the drag force on the bluff body. Most of the time results are obtained using experimental measurements in wind tunnels or numerical simulation. In the current trend numerical simulation is used most in order to shorten time and cost. However, the current state of the art in the computational fluid dynamics shows that over the last few year’s accurate results were obtained for the automotive aerodynamics. Aerodynamics of bluff bodies has been studied years ago. Ahmed body was one of the first published parametric studies, which have been with a generic car-like body. Slant angles and yaw angles are the major parameters which have been tested. The first experimental work has been done by S.R.Ahmed.
Model Description

The body geometry is shown in (Fig-1). The dimensions of the flow domain are 10L x 2L x 1.5L (L = 1.044 m) normalized with length along length, width and height of the domain respectively. Front end of the Ahmed body is at 2.4L from the inlet and back end is 6.6L from outlet of the domain. The body is suspended 50mm from the ground. Frontal curvature of 100mm is provided. The back slant angle used for this model is 12.5degree. In the experimental work conducted by S.R.Ahmed stilts were used to suspend the body form the ground and their effect on the drag is subtracted. Therefore in the present model stilts have not been modelled.

Meshing

Trimmer mesh has been used since the flow is directional. Ten Prism layers mesh was used with first layer thickness of 0.01mm. The first layer thickness was decided based on wall y+ by trial and error procedure. Y plus (Fig-2) was maintained around 5-10 near streamlined flow. Volumetric controls have been used near the front, under-hood and wake because of stagnation, boundary layer formation and flow separation respectively. The cell count for this mesh is 4.8 million.

Physics setup and boundary conditions

The simulation has been done for steady flow. K-ω SST turbulence model was used. The simulation was done using both segregated and coupled solver. Segregated solver was tested using Delta v-dissipation. Coupled solver was used along with Grid Sequential Initialization, Expert driver and Coupled sub solver.
GSI is used to provide better initialization for coupled solvers which are very sensitive to initialization. It solves the In-viscid irrotational Navier-stroke equation for a series of coarser meshes before solving for the final mesh. Expert driver is used to ramp the courant number and to set the maximum and minimum relaxation factors. CFL number used was 200 and it was ramped to that value in first 50 iterations. V-cycle pre-sweeps and post-sweeps were changed from 0 and 2 to 1 and 3 respectively to stabilize the CFL number at 200(Fig-3).

Based on the Reynolds number of 4.25 million and inlet velocity of 61.1m/s the density and viscosity were calculated. The bottom wall (ground) was given a tangential velocity of 61.1m/s. The realizibility coefficient was set to 0.6.

The drag coefficient and pressure coefficient were monitored to validate the results with the experimental results. The formulation of drag coefficient and pressure coefficient is given below.

**Drag and Pressure Coefficient Equation**

\[ F_d = \frac{1}{2} \rho v^2 C_d A \]

\[ C_p = \frac{(P - P\infty)}{1/2 \rho v^2} \]

\( F_d \) = Force of drag, \( \rho \) = density of fluid, \( v \) = velocity of object relative to the fluid

\( C_d \) = Drag coefficient, \( A \) = reference area, \( P \) = pressure at point pressure coefficient is calculated, \( P\infty \) = Pressure in the free stream
Reference area is the orthographic projection of the object in the direction of motion. The force of drag computed is the combination of force due to foam drag and skin friction drag.

The pressure coefficient at the top half of the Ahmed body along the symmetry plane was compared with the experimental results for both segregated (Fig-5) and coupled solver (Fig-4).

**Conclusion**

1. Coupled solver saves nearly half the time when compared to segregated solver.

2. Segregated Delta-Vdissipation saves time when compared to case without Delta-Vdissipation.

3. Results from STAR-CCM+ simulation match well with the experiments (Table-1) with deviation of (0.2-1)%.

**References**


3. Numerical simulation of the flow around the Ahmed body Gerardo Frank, Norberto Nigro, Mario A.Storti and Joerge D’Elia.
Model

FIG-1
Wall Yplus

CFL monitor
Pressure coefficient for coupled solver

![Graph showing pressure coefficient for coupled solver](image)

Pressure coefficient for segregated solver

![Graph showing pressure coefficient for segregated solver](image)
Comparison of solvers using 8 and 16 cores.

<table>
<thead>
<tr>
<th>No cores</th>
<th>Solver</th>
<th>Convergence time</th>
<th>( C_d ) (expt-0.23)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 cores</td>
<td>Segregated</td>
<td>10000 sec</td>
<td>0.2272</td>
</tr>
<tr>
<td>8 cores</td>
<td>Coupled</td>
<td>6200 sec</td>
<td>0.2295</td>
</tr>
<tr>
<td>16 cores</td>
<td>Segregated</td>
<td>4990 sec</td>
<td>0.2272</td>
</tr>
<tr>
<td>16 cores</td>
<td>Coupled</td>
<td>2970 sec</td>
<td>0.2295</td>
</tr>
</tbody>
</table>

Table-1