

Assignment 1

by Haiwei Hu

FILE	11978_HAIWEI_HU_ASSIGNMENT_1_176684_504934711.DOCX (1.31M)		
TIME SUBMITTED	29-APR-2019 07:48AM (UTC+0100)	WORD COUNT	2145
SUBMISSION ID	105621481	CHARACTER COUNT	10365

Simulation and Modelling Assignment 1: A CFD Analysis About Ahmed Body

Haiwei Hu

Background knowledge

CFD

CFD, or computational fluid dynamics, is a branch of fluid mechanics, referred to as CFD. CFD is a combination of modern fluid mechanics, numerical mathematics and computer science (Harlow, 2004). It is a cross-science with strong vitality. It uses electronic computers as a tool and applies various discrete mathematical methods to conduct numerical experiments, computer simulations and analytical studies on various problems of fluid mechanics to solve various practical problems.

FVM

Most of CFD software use the finite volume method which is called FVM, The Finite Volume Method which is like finite difference method or finite element method. The Finite Volume Method is also known as the Control Volume Method (Eymard, et al., 2003). The progress of FVM is: 1. The calculation area is divided into a series of non-repetitive control volumes, each of which has a node as a representative, and the conservative differential equation to be sought is integrated into space and time in any control volume and time interval; 2. Make assumptions about the time- and space-variant line or interpolation method of the function and its derivatives; 3. Make the integrals of the selected lines in step 1 and organize them into a set of discrete equations about the unknowns on the nodes.

Turbulence models

It is well known that the flow state of a fluid has a difference between laminar flow and turbulent flow. Most of the flow in the project is turbulent, so when using CFD software to simulate the flow in the project (Andersson, 2012), it is often necessary to use the "turbulence model", such as the k- ϵ model, the SST model, and so on.

I will discuss with two most used models, they are both 2 equation models:

1. Realizable K-epsilon(k- ϵ) model

The k- ϵ model is a two-equation model that requires two variables, velocity and length scales. In FLUENT, the standard k- ϵ model has been the main tool in engineering flow field calculation since it was proposed by Launder and Spalding. It has wide range of application, economical and reasonable accuracy. It is a semi-empirical formula that is summed up from experimental phenomena (Hanjalic & Launder, 1972).

The realizable k- ϵ model adds a formula to the turbulent viscosity. a new transfer equation is added for the dissipation rate. This equation is derived from an exact equation for laminar velocity fluctuations. The immediate benefit of the achievable k- ϵ model is a more accurate prediction of the divergence ratio of the plate and cylindrical jets. Moreover,

it has a good performance for the boundary layer flow, flow separation and secondary flow of the rotating flow, strong back pressure gradient.

2.SST k-omega (k- ω) model

The k- ω model solves the hydrodynamic problem by solving the kinetic energy k and the turbulent energy dissipation rate ω . The k- ω model predicts free shear flow propagation rates, such as wake, mixed flow, plate flow, cylindrical flow, and radial injection, and can be applied to wall bound flow and free shear flow (Wilcox, 2008).

The SST k- ω model is like the standard k- ω model with the following improvements: 1. The deformation of the SST k- ω model and the k-e model is increased by the blending function and the dual model. The mixing function is designed for the near wall region, which is valid for the standard k- ω model, as well as the free surface, which is effective for the deformation of the k-e model. The SST k- ω model incorporates cross-diffusion from the ω equation. 2. The turbulent viscosity considers the transmission of turbulent shear stress. 3. Different model constants These improvements make the SST k- ω model more accurate and reliable than the standard k- ω model in a wide range of flow fields.

Ahmed body

Ahmed body will help us simulate the normal vehicle, its aerodynamics are similar to the car, but it has a simple geometry. The aerodynamic properties of the rear of the Ahmed body are very close to that of a regular car.

Methodology

Pre-Processing

1.Geometry and domain

In the CFD process, Geometry function is always used to import the original model file (like .x_t file) and decompose the part of the aerodynamic model.

With Geometry software in Ansys, we can also create and size the domain, and if we use the Fluent mesh function, we will need this software to help us create density box. But in the assignment, we use ICEM CFD to mesh, the ICEM CFD contain the density box function.

The domain must be big enough for air to flow, but it shouldn't be too big to extend the calculation time.

2.Mesh

To analyze fluid flow, the watershed is divided into smaller subdomains. The governing

equations are then discretized and solved in each subfield.

With ICEM CFD, we can restrict the maximum size of the elements in any curves/edges, surfaces and spaces inside the domain we set.

3.Setting boundary condition and other conditions

We must set boundary conditions like velocity, pressure, and tell the software where those numbers applied. We also need to set density and viscosity of fluid, the temperature and so on.

When using Fluent, this step is taken together with the solver.

Solver

The most used CFD software: CFX, Fluent, Phoenics, Star-CD, comsol, star-ccm+, flow-3D, AUTODESK CFD. With the Ansys software, we can use both CFX and Fluent to simulate the air flow. CFX is more advanced than Fluent in Ansys, but Fluent offer a faster calculation. In Fluent, we can even use GPU to accelerate the calculation. Under the limitation of the hardware, in this assignment, I use the Fluent software in Ansys.

We need to choose the turbulence model in this step, it has huge influence on the result. And we will set the monitor of the value we need like force or speed in some area or point.

Post-Processing

After calculation, we should get some value and visualized and analysed fluid dynamics results, as a result or just as a comparison.

Compare and re-mesh

We need to compare the value we get with the experimental value and compare the contour with the mesh method to find out where can we improve in the next mesh. Keep that loop until get the closest data.

Practice and analysis

Import symmetry and create domain

At this step, I used the Geometry Function in Ansys Workbench.

First, I create the Ahmed body model in the by importing the given SOLIDWORKS file. Name the Ahmed body's surface for further use.

Then I create a domain box, which is big enough for the air flow and suitable to the boundary condition around the Ahmed body. The domain box is created from the symmetry plane which is depicted in Fig.01 (which is named 'symmetry'). The plane 'symmetry' is just a virtual plane for convince of further calculation (we just need to calculate the air in right half side) and display of the Ahmed body. The bottom surface of the domain, which is named as 'floor' (Fig.02) is coplanar with the bottom surface of the Ahmed body. After setting the size of the box as Fig.03, the box is created. Since the plane in front of Ahmed is where the air flows in, I call it 'inlet', and the plane behind the body is called 'outlet', I named them for setting the boundary conditions.

The final step is cut the Ahmed body out of the box like Fig.04. The final product we get in this step is a box filled with air.

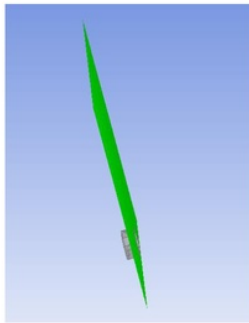


Fig.01

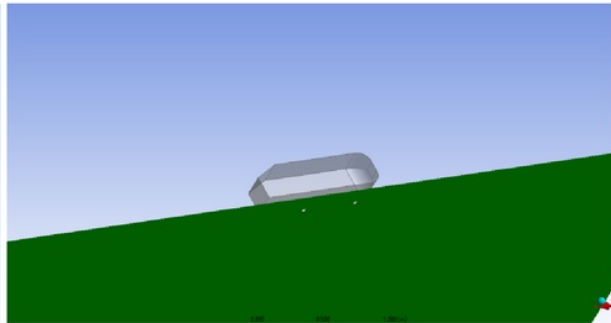


Fig.02

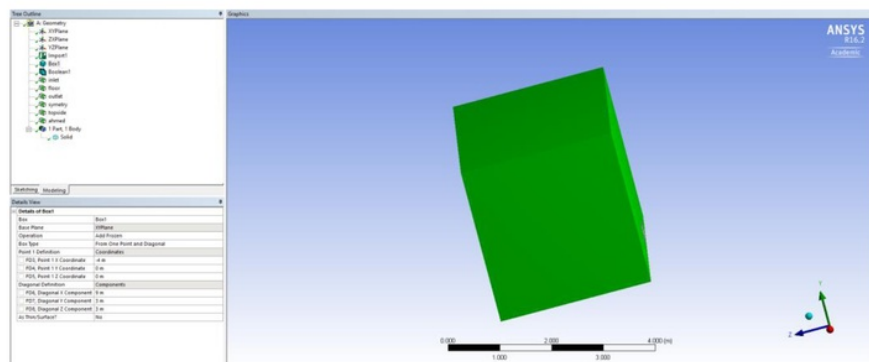


Fig.03

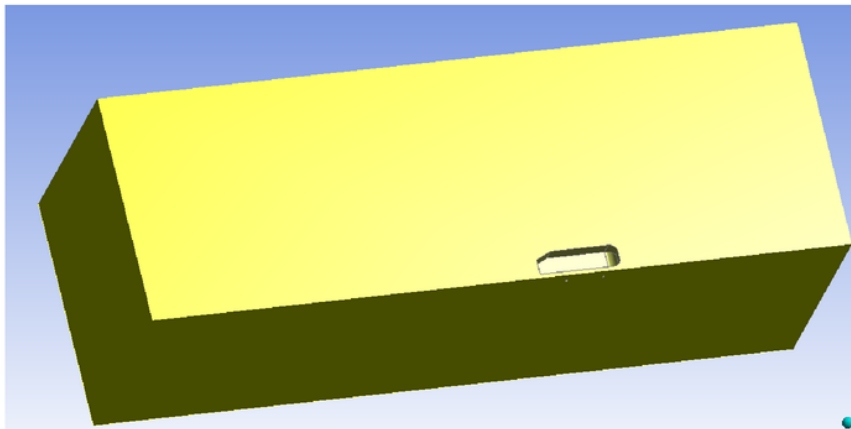


Fig.04

First Mesh

mesh without setting any mesh density

At this step, I used the ICEM CFD Function in Ansys Workbench.

The mesh method will be verified so the setting will be changed every time,

Since the whole car's length is just 1044mm, so I set the maximum element size as 0.1m like Fig.05. And then I run mesh, the result of mesh is showed in Fig.06



Fig.05

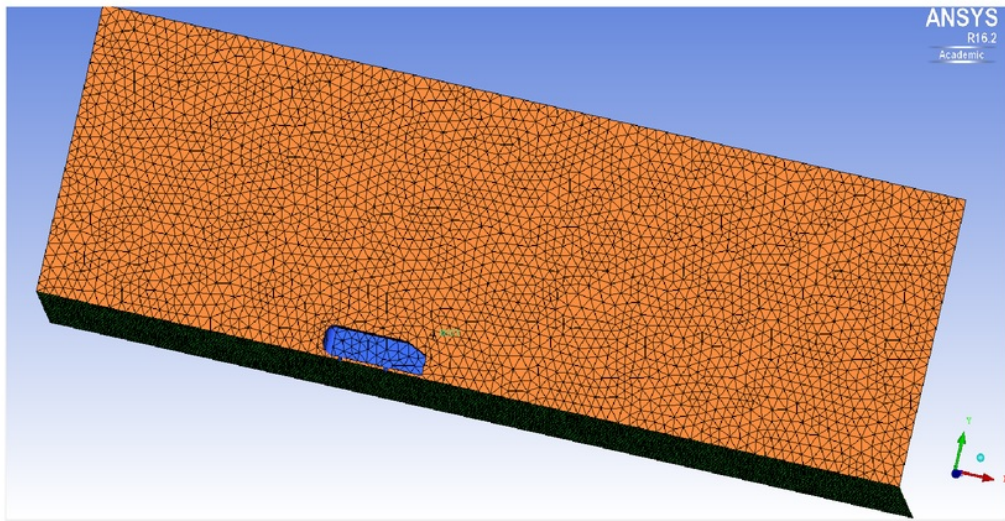


Fig.06

Leave most of the options as default. And since the CPU of the computer in the university is i7-7700k, a CPU with 4 cores and 8 threads, we set processing option to 'parallel' the solver processes to 7, leave 1 for the recording.

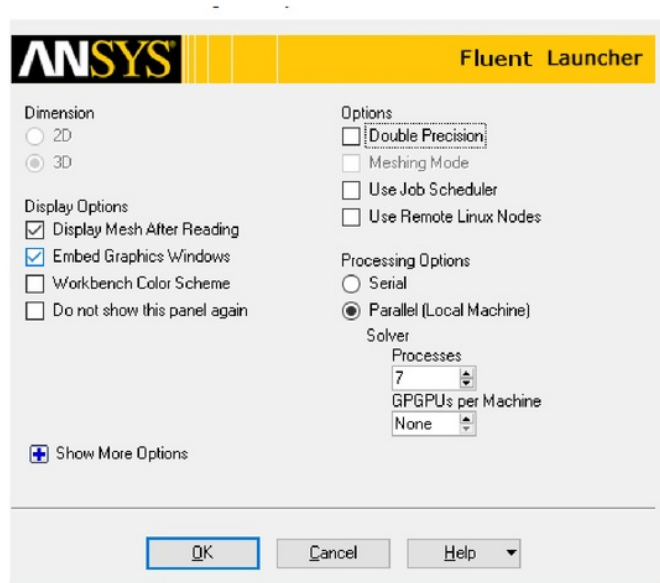


Fig.07

As I said in the method part, the SST k- ω is a mostly used and relatively precise method, so I set it and leave other settings as default.

Model

- ☐ Inviscid
- ☐ Laminar
- ☐ Spalart-Allmaras (1 eqn)
- ☐ k-epsilon (2 eqn)
- ☒ k-omega (2 eqn)
- ☐ Transition k-k-omega (3 eqn)
- ☐ Transition SST (4 eqn)
- ☐ Reynolds Stress (7 eqn)
- ☐ Scale-Adaptive Simulation (SAS)
- ☐ Detached Eddy Simulation (DES)
- ☐ Large Eddy Simulation (LES)

k-omega Model

- ☐ Standard
- ☐ BSL
- ☒ SST

Fig.08

Then I set the materials and zone, the 'body' means the space between the domain box and the Ahmed body. Fig.10 is the cell zone condition setting.

Materials

Fluid
air
Solid
aluminum

Fig.09

Zone

body

Phase	Type	ID
mixture	fluid	11

Fig.10

Then the boundary conditions, I set their type like Fig.11-Fig.15, and the set the 'inlet' face as '40m/s velocity-inlet at x direction', the 'outlet' face is set as '0 pascal pressure-outlet'. The 'symmetry' and 'topside' are set as symmetry because they are just virtual plane.

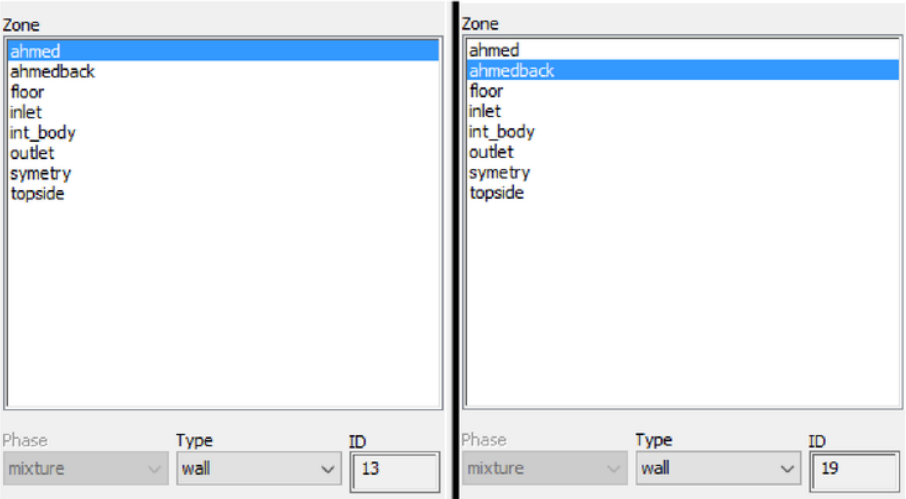


Fig.11

Fig.12

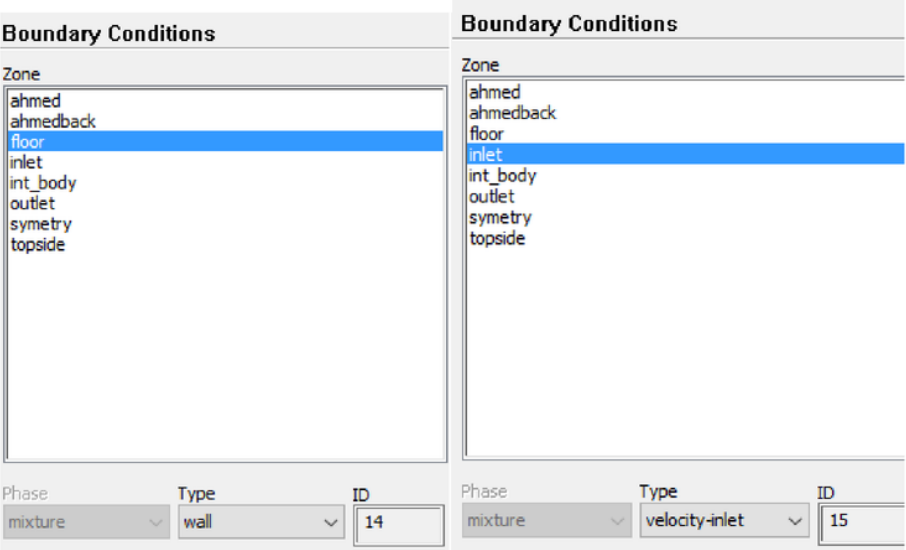


Fig.13

Fig.14

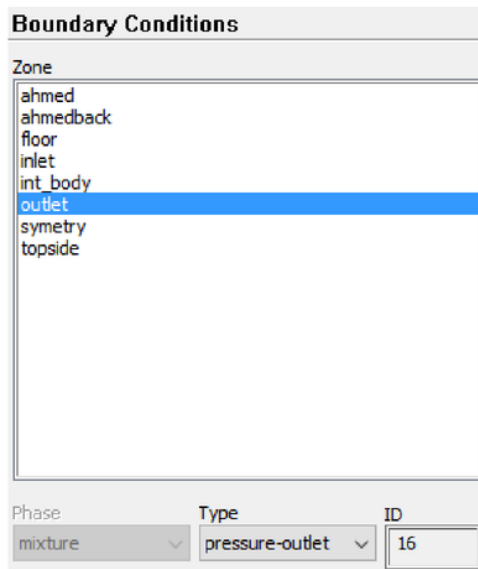


Fig.15

Then the reference values, we already set the materials, so we just need to put in the area of the x direction. It can be calculated with the reports function.

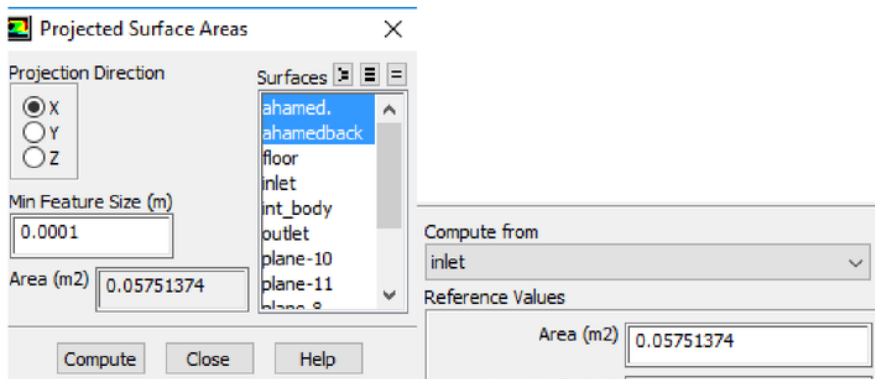


Fig.16

Fig.17

Coupled means you will calculate twice at the same time, it will accelerate the whole calculate time

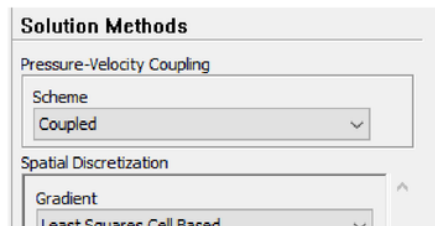


Fig.18

We monitor the draft coefficient and lift coefficient, just set the monitor on the whole Ahmed body in x and y direction

Monitors

Residuals, Statistic and Force Monitors

Residuals - Print, Plot

Statistic - Off

cd-1 - Print, Plot

cl-1 - Print, Plot

Fig.19

Then I initialize and run the calculation, after the continuity get down to $1e-03$, I stop the calculation and get the velocity contour. The lift coefficient is $3.9518e-01$ and the drag coefficient $5.7360e-01$, the drag coefficient is much higher than the experimental data.

The shape of the contour is very irregular which means the velocity distribution is very uneven, we need to get more fine mesh.

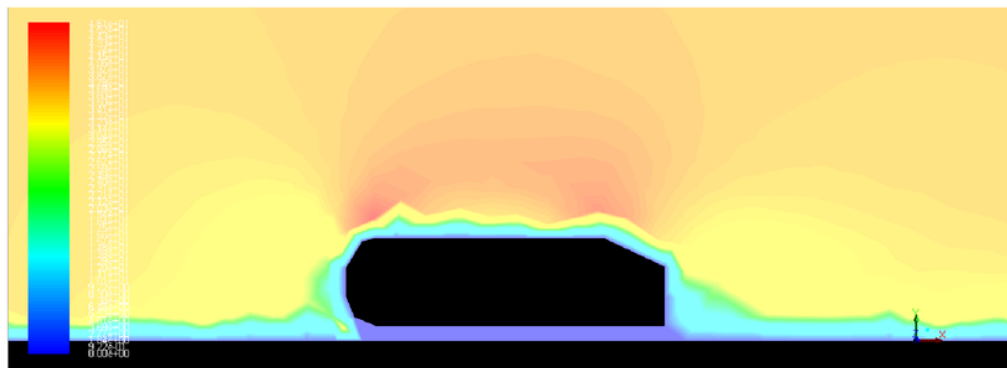


Fig.20

Second & third mesh

Setting meshing densities around the Ahmed body. I create a density box around the Ahmed body, which is three times as long as the Ahmed body in x direction. And in Y and Z direction, the box is half as long again as the Ahmed body. In order to calculate less, I set the default maximum element size to 0.2,

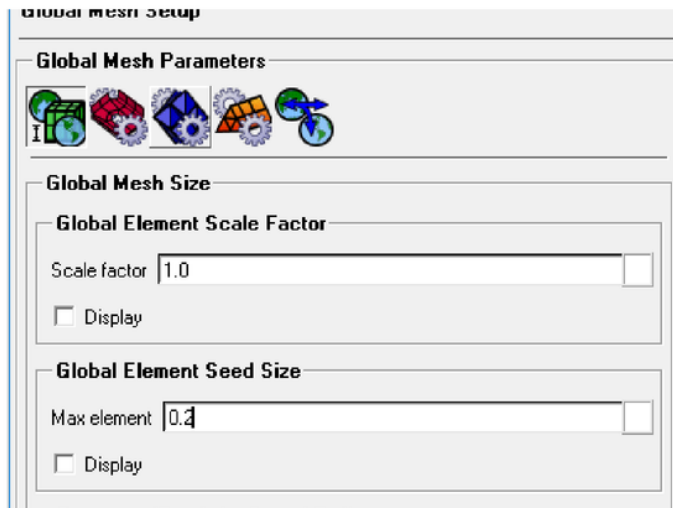


Fig.21

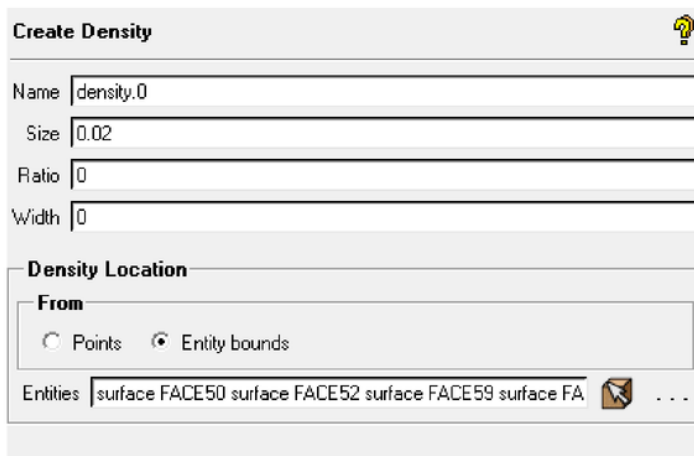


Fig.22

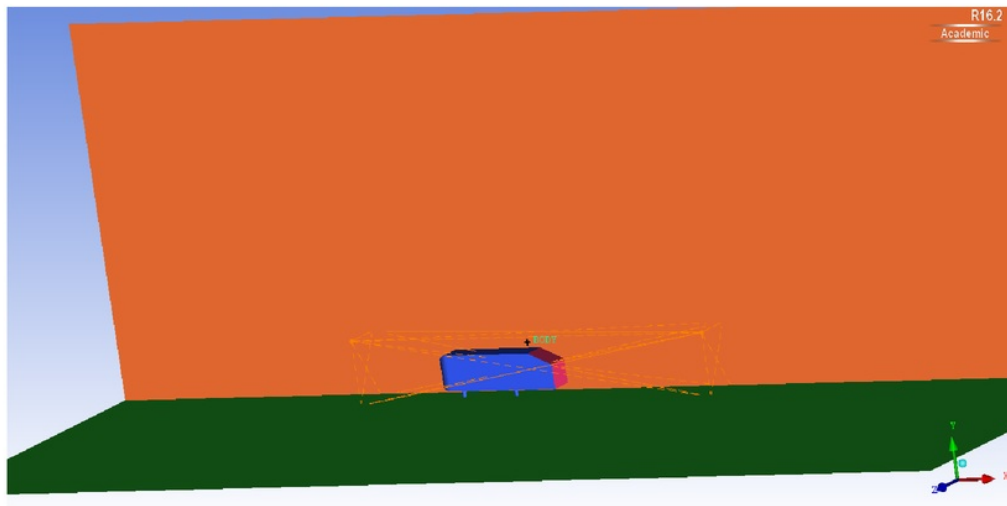


Fig.23

The solver setting is the same. Lift coefficient is $3.8054e-01$ and the drag coefficient is $3.1813e-01$.

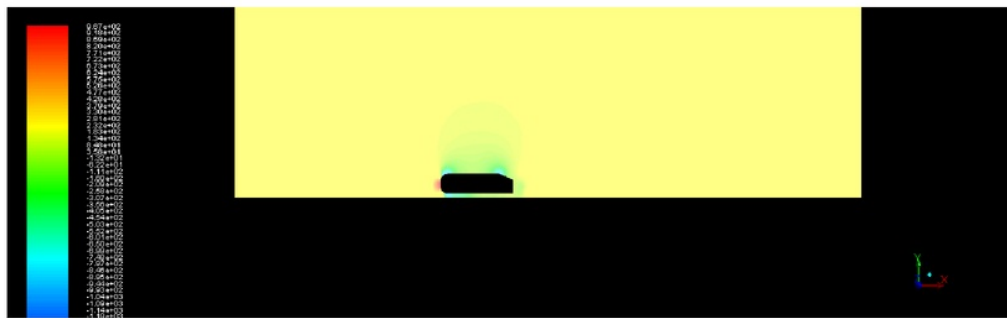


Fig.24

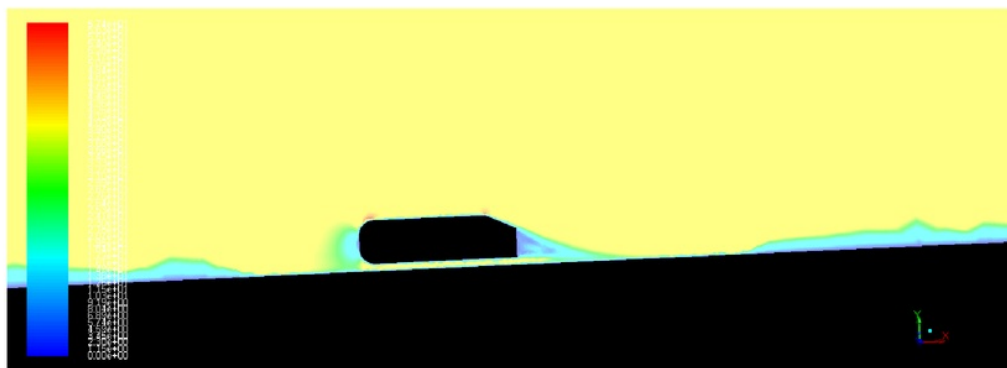


Fig.25



Fig.26

Fig.27

The elements are too big so the airflow around the Ahmed body are in a mass, so I apply the Curvature and Proximity settings (a kind of size setting), I adjust the elements in gap to 5. And then in surface mesh setup I size the element in surface to 10mm.

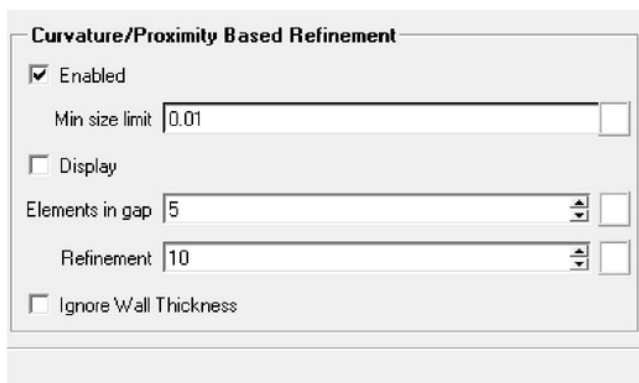


Fig.28

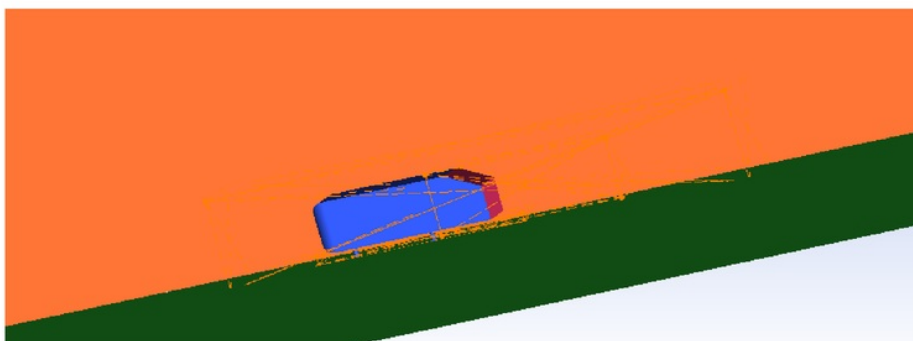


Fig.29

I also apply 2 more density boxes to get a more precise calculation in the rear part and the gap between Ahmed body and road, the maximum element size is set to 0.01.

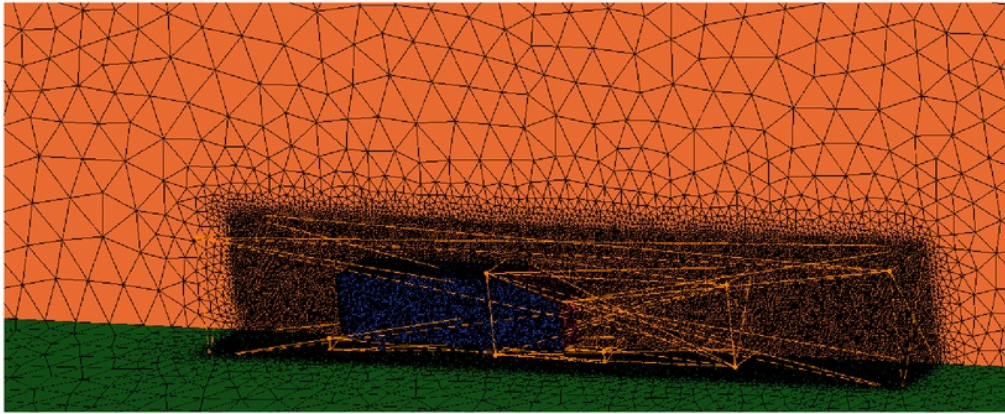


Fig.30

Lift coefficient is 3.2115×10^{-1} and the drag coefficient is 2.9015×10^{-1} . But I can still find some obvious jagged part in the cross section in the x-axis direction.

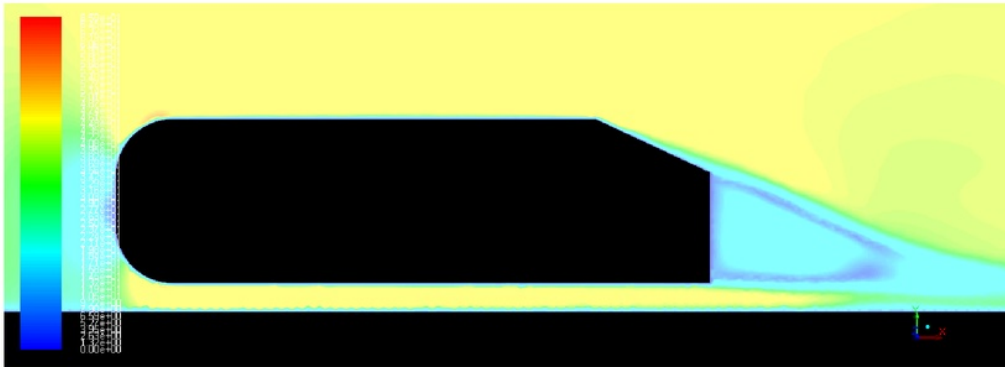


Fig.31

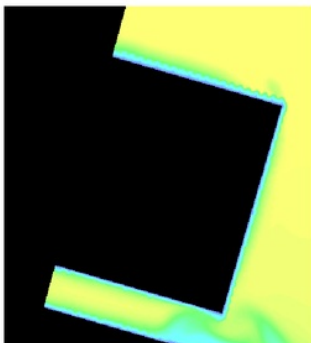


Fig.32

Fourth and Fifth mesh

I need to add the prism layers to simulate the airflow near the surface

	Prism	Height	Height ratio	Prism layers	Prism height min ratio
IMED	☒	0.02	1.2	8	0.2
IMEDBACK	☒	0.02	1.2	8	0.2
JDY	☐				
JDR	☒	0.02	1.2	8	0.2
ILET	☐	0	0	0	0
UTLET	☐	0	0	0	0
SOLID 1 1	☐				n

Fig.33

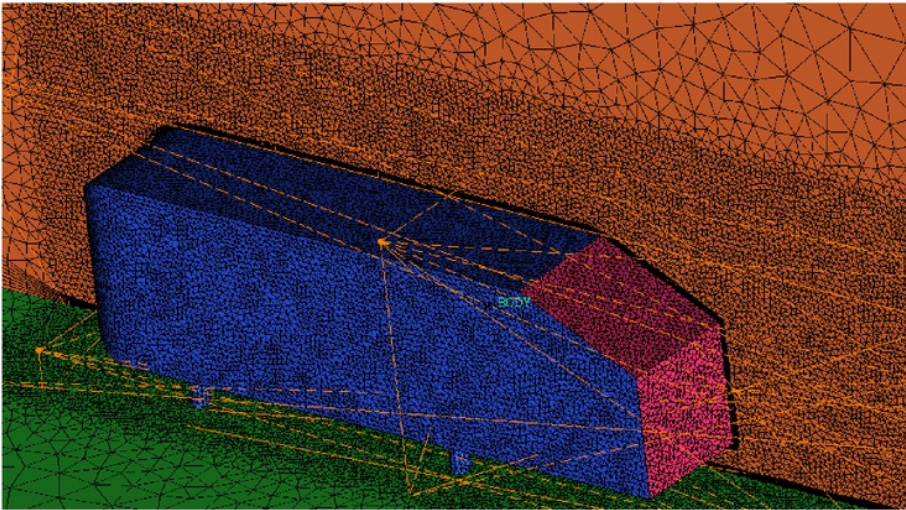


Fig.34

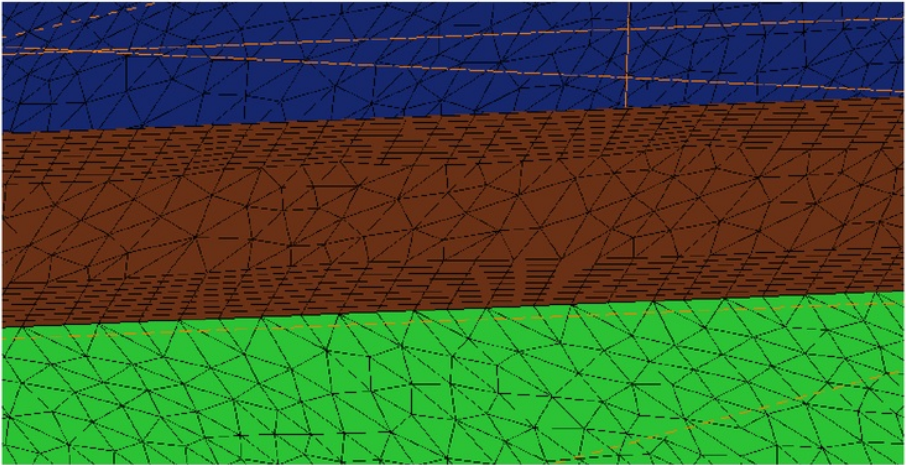


Fig.04

But after the prism layers, the lift coefficient dropped to $7.0574e-02$. The drag force is still $2.8165e-01$. After checking the velocity contour, I find out that the airflow is seriously dispersed by the upper edge of the rear part, because the prism layer here are too big.

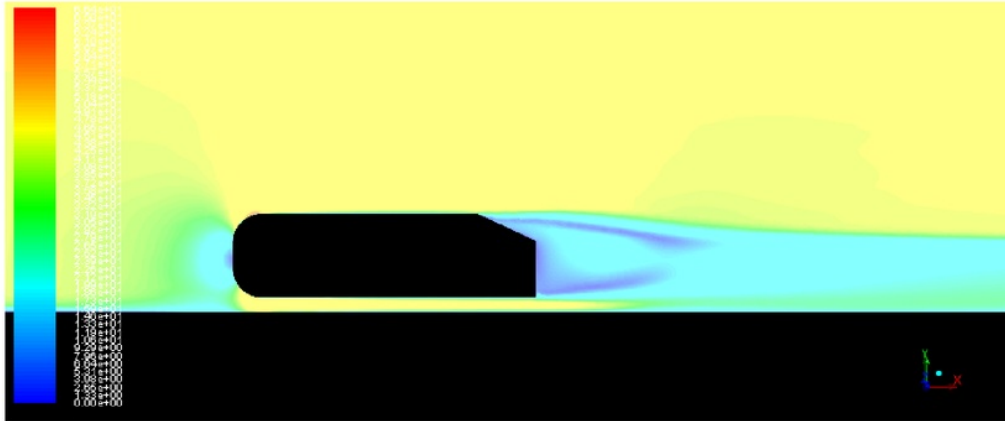


Fig.36

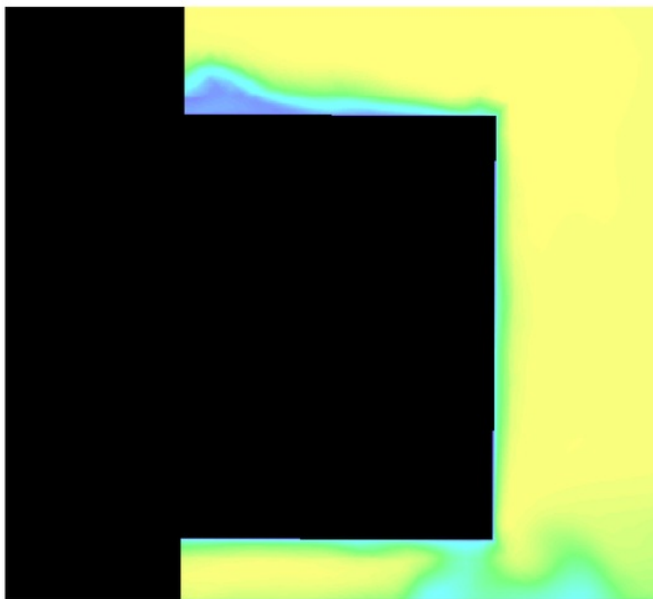


Fig.37

So, I try to reduce the element size on the place where the airflow dispersed. I set the maximum element size on the edges in Fig.38 and the face in Fig.39.

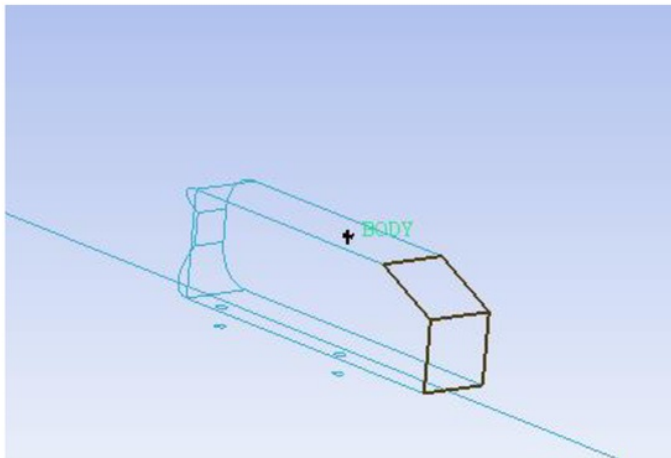


Fig.38

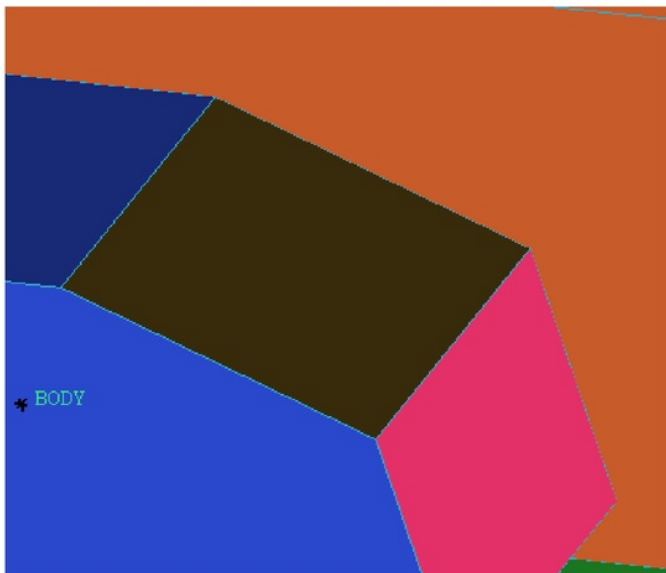


Fig.39

Then the air emerges behind the rear part. And the air around the Ahmed body is smoother. The lift coefficient is $3.8424e-01$ and the drag coefficient is $2.8784e-01$. We can see that the rear of the Ahmed body has two distinct wakes, it will drag the car, its harmful.

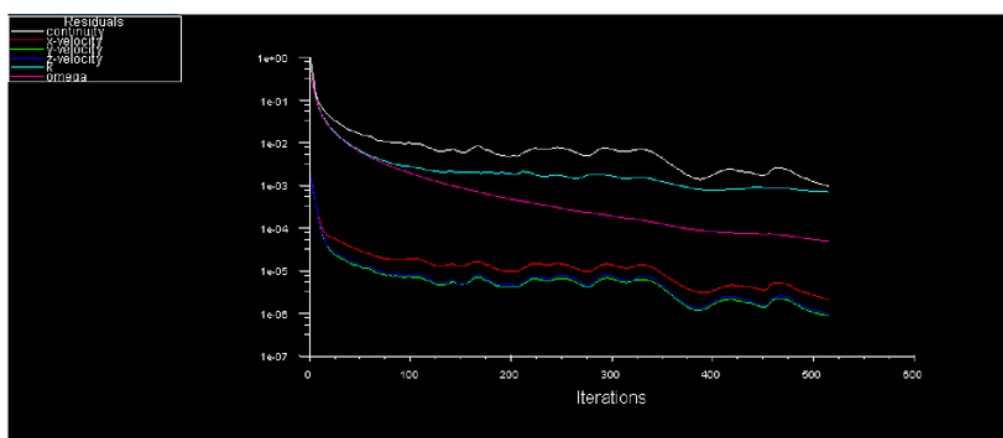


Fig.40

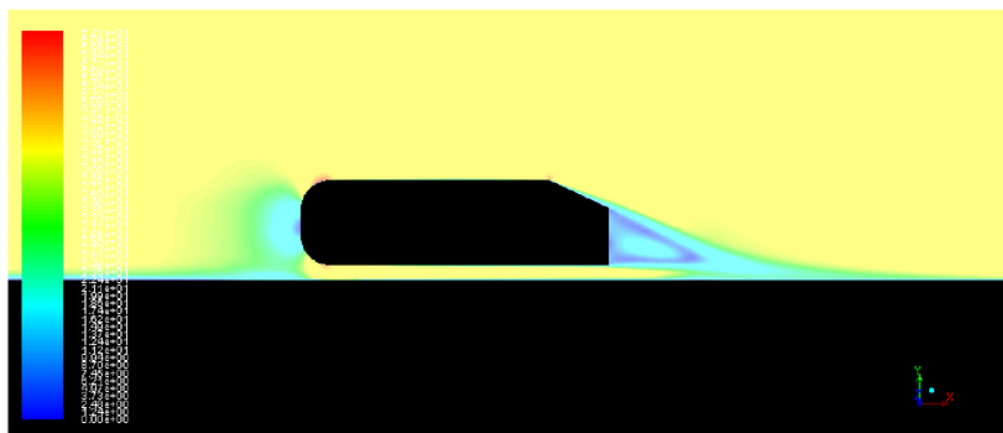


Fig.41

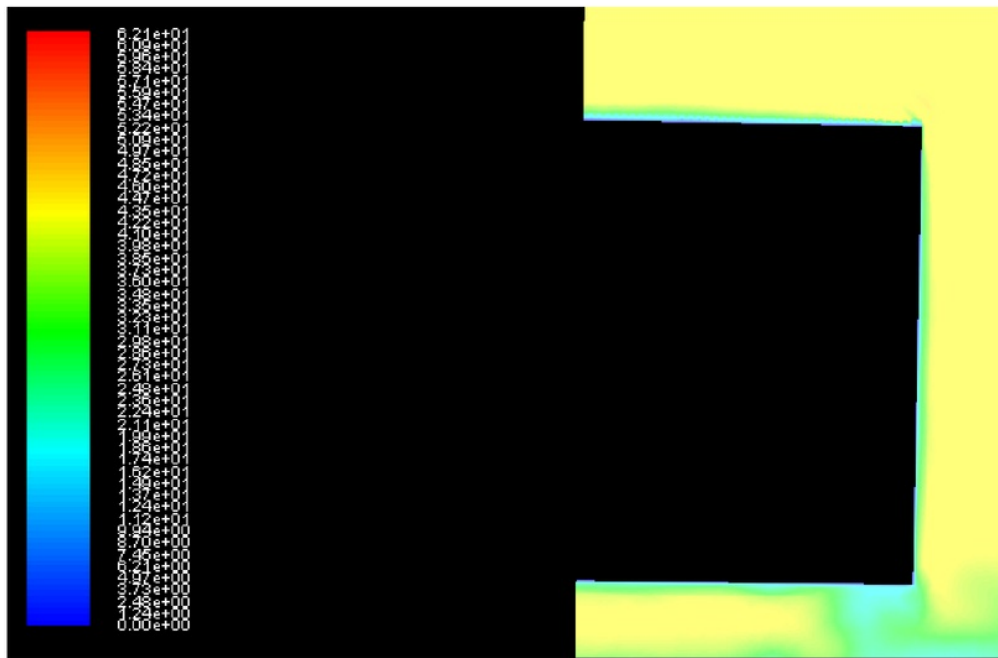


Fig.42

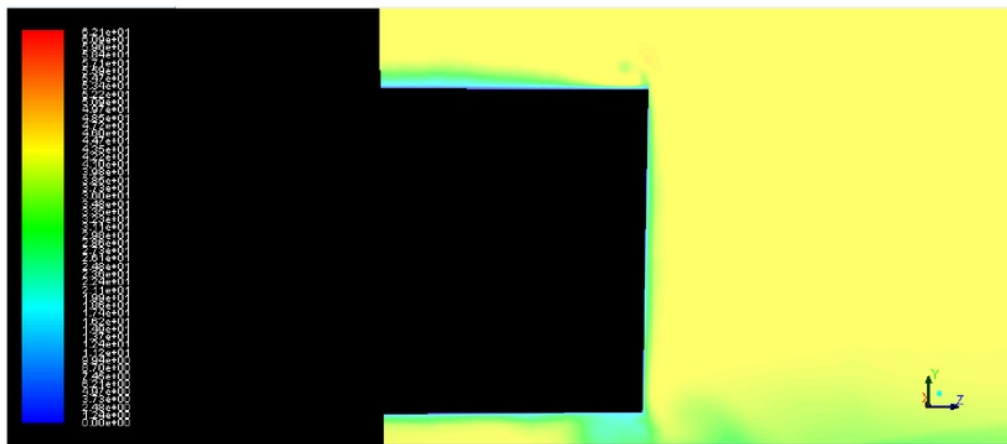


Fig.43

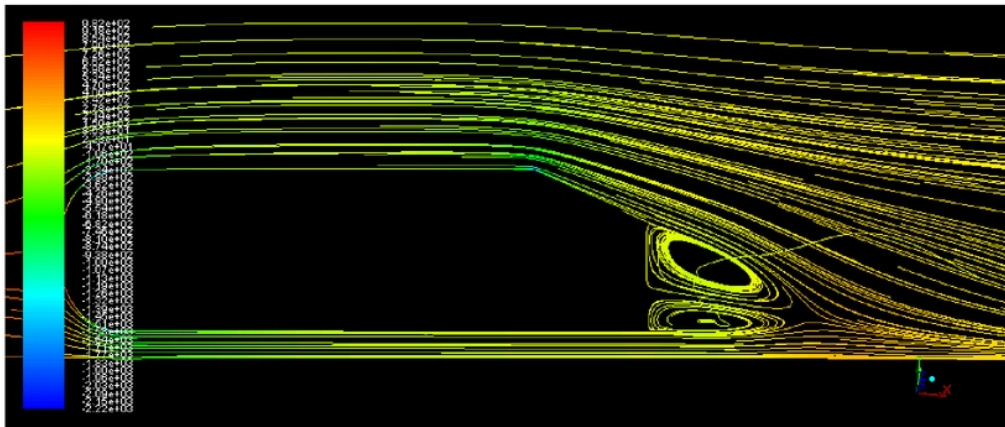


Fig.44

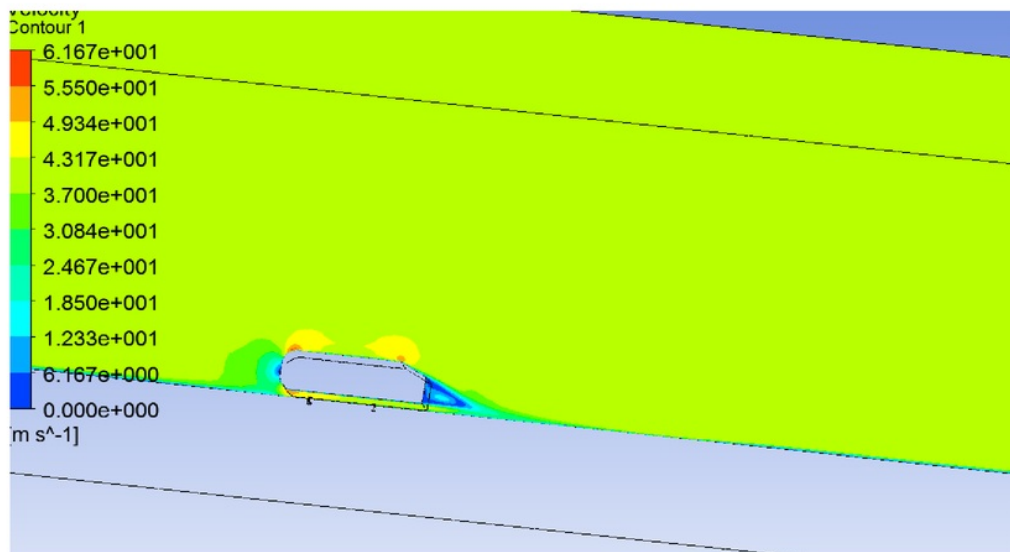


Fig.45

Use another turbulence model: Realizable k-epsilon with enhanced wall treatment.

The lift coefficient is 3.9981e-01 and the drag coefficient is 3.0004e-01.

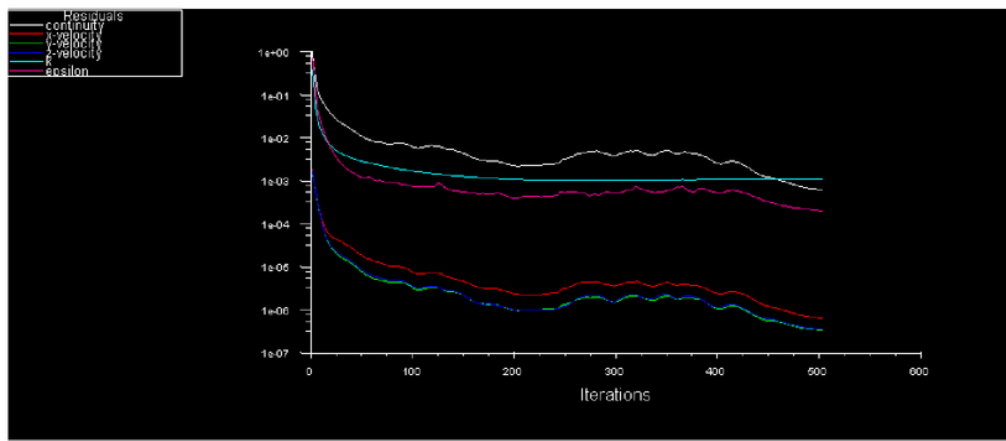


Fig.46

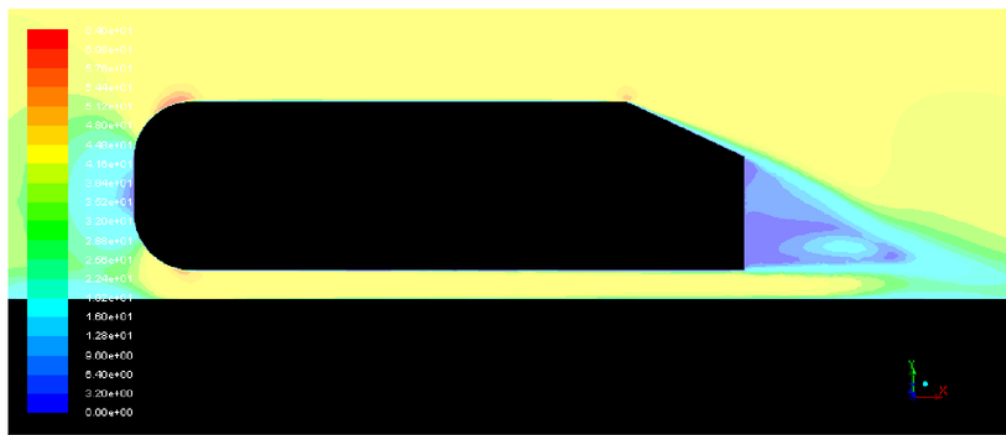


Fig.47

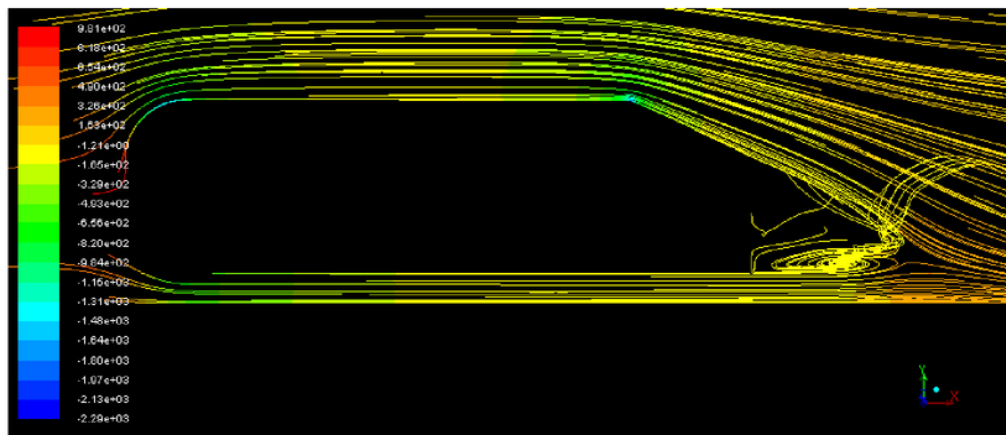


Fig.48

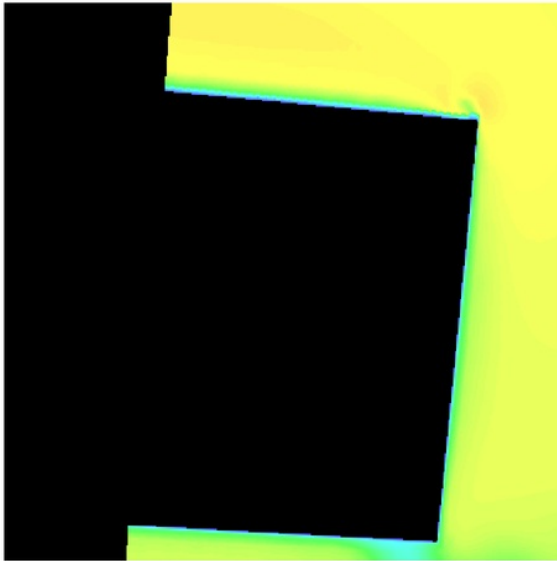


Fig.49

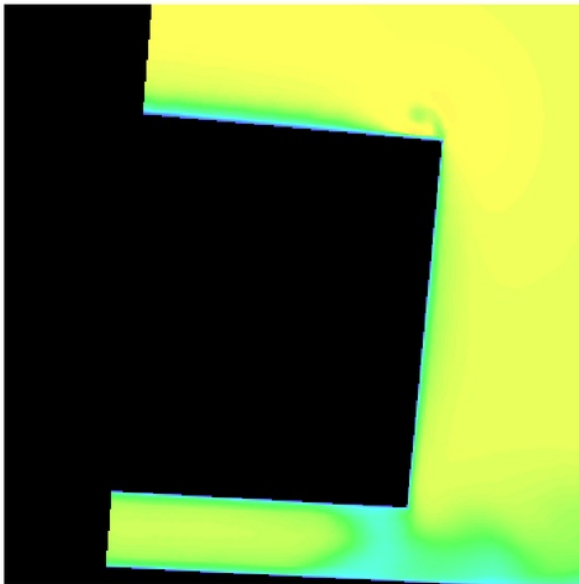


Fig.50

We can see the vortex near the surface more clearly with k-epsilon method than the SST method. But the k-epsilon can't calculate the wake behind the Ahmed body in this test, even calculated more than 400 times.

Bibliography

Andersson, B., 2012. *Computational fluid dynamics for engineers*. Cambridge: Cambridge University Press.

Eymard, R., Gallouet, T. & Herbin, R., 2003. Finite Volume Methods. P. Ciarlet & J. Lions, editor *Handbook of Numerical Analysis*. Marseille: Ecole Nationale des Ponts et Chaussees, pp. 713-1020.

Hanjalic, K. & Launder, B., 1972. A Reynolds stress model of turbulence and its application to thin shear flows. *Journal of Fluid Mechanics*.

Harlow, H. F., 2004. Fluid dynamics in Group T-3 Los Alamos National Laboratory: (LA-UR-03-3852). *Journal of Computational Physics*, 10 4, pp. 414-433.

Wilcox, D. C., 2008. Formulation of the $k-\omega$ Turbulence Model Revisited. *AIAA Journal*, p. 2823-2838.