

University of Wales Trinity Saint David

School of Engineering

CFD Ahmed Body

Advanced Computational Methods

M6X00408

Zhangwei Guo

1805078

25/01/2020

目录

1	Abstract.....	1
2	Introduction	2
3	Conduct Research	3
3.1	General CFD Process	3
3.2	Governing equations.....	3
4	Create Fluid Domain Geometry	5
5	Create Mesh and Appraise Quality	6
6	Solve and Analyze Results.....	10
6.1	25 Degrees Ahmed Geometry	10
6.2	35 degrees Ahmed Geometry.....	14
7	Conclusion.....	16
8	Reference.....	17

1 Abstract

This paper focuses on create a suitable simulation model for an Ahmed body analysis and investigate CFD methodologies (Tudor, 2020). First of all, research on the whole process of CFD and the governing equations of it are finished to get an understanding of CFD. After that, the geometry is imported into ANSYS and a fluid domain been established as well to finish the analysis of the aerodynamics around the Ahmed. The choice of geometry domain is made at the same time considering the flow structures and mesh strategy. Thirdly, a mesh of the domain is created as the mesh density been applied using the mesh control in where the important flow occurs. Finally, the solver is set up and the solution been got as the results could be checked by plots and coefficients. Different set of domain sizes, mesh strategies, turbulence models and solver settings been used to find the effect they have on the results.

Keywords

ANSYS, CFD (Computational fluid dynamics), mesh strategies, solver settings

2 Introduction

As everyone knows, computational fluid dynamics (CFD) has been used to analysis the interaction of the aerodynamic devices and the aerodynamics around the vehicles which means all the simulations have to be 3D as the highly turbulent flow regimes are 3D phenomena.

Using computational fluid dynamics (CFD) to investigate the external and internal aerodynamics is widely been used to redesign some parts and vehicles trying to get a better aerodynamic performance. Computational fluid dynamics (CFD) has been applied to develop a 3D simulation model and find the lift and drag coefficients at different settings on two geometries with different slant angles.

The major target of this paper is to investigate the effects of different domain sizes, mesh strategies, turbulence models and solver settings on lift and drag coefficients of the geometry, trying to find the best control solution for each geometry as well. This paper has been organized as follows. At the beginning, an outline of the whole CFD process and several control equations needs to be established after researching. After that, import the geometry into ANSYS and create a fluid domain preparing for the investigate of the external aerodynamics around the geometry. Thirdly, using mesh controls to apply mesh density to get a suitable 3D mesh of the fluid domain. Finally, the solver needs to be set so the initial solutions can be obtained and then the results can be analyzed, the best control solutions can be found out as well.

3 Conduct Research

First of all, there are two computational fluid dynamics analysis methods could be used to predict the aerodynamics around the vehicles which is solid modeling method and shell modeling method (Ashok Patidar - Mahindra & Mahindra Limited, 2020). For solid modeling method, the Ahmed geometry is modeled as solid and modeled as shell element with defined thickness for shell modeling method.

3.1 General CFD Process

The step of both of these two CFD analyses methods are as follows. First of all, the geometry should be established and imported into ANSYS. A suitable fluid domain should be created around the geometry after that, so a 3D mesh of the fluid domain could be created as well. And for some regions in which significant flow regimes occurs, mesh controls could be used to create mesh density. Before set up the solver, the mesh quality should be appraised from skew, orthogonality and aspect ratio. Finally, all the results can be obtained from contour plots and reporting coefficients.

3.2 Governing equations

The governing equations are shown as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

Figure 1. Continuity equation

$$\rho \frac{Du}{Dt} = \frac{\partial(-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx}$$

Figure 2. Momentum equation

Rate of increase of energy of fluid particle	=	Net rate of heat added to fluid particle	+	Net rate of work done on fluid particle
$\rho \frac{DE}{Dt}$		$\text{div}(k \text{ grad } T)$		$-\text{div}(\rho \mathbf{u}) + \Phi$

Figure 6. Energy equation

$$\frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu_t + \mu) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\mu_t + \mu) \frac{\partial u_j}{\partial x_i} \delta_{ij} \right] + \rho g_i$$

$$\begin{aligned} \frac{\partial}{\partial x_j} (\rho u_i u_j) = & -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu + \mu_t) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\mu + \mu_t) \frac{\partial u_i}{\partial x_i} \delta_{ij} \right. \\ & \left. - \frac{2}{3} \rho k \delta_{ij} \right] + F \end{aligned}$$

$$\frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \bar{u}_i}{\partial x_i} \delta_{ij} - \rho \overline{u_i' u_j'} \right),$$

$$\frac{\partial}{\partial t} (\rho_m v_m) + \nabla \cdot (\rho_m v_m v_m) = -\nabla p_m + \nabla \cdot (\tau_m + \tau_{Tm}) + S_m$$

Figure 4. RANS momentum equation

$$\frac{\partial}{\partial t} (\rho k) + \text{div} (\rho \mathbf{u} k) = \text{div} (\Gamma_k \text{ grad } k) + G - \rho \epsilon$$

Figure 5. K equation for turbulent energy

$$\frac{\partial(\rho\phi)}{\partial t} + \text{div}(\rho\phi\mathbf{u}) = \text{div}(\Gamma \text{ grad } \phi) + S_\phi$$

Figure 6. General form of K equation for turbulent energy

4 Create Fluid Domain Geometry

First of all, import the external geometry file to get the geometry in the design modeler. Domain and all analyses only needed to be performed on half of the geometry on account of it is symmetric. For the size of domain, from the results of some experiments as shown in figure 7 it can be defined as 10 meters long with 6 meters behind the geometry, 1.5 meters high from the ground and 3 meters wide with 1.5 meters each side.

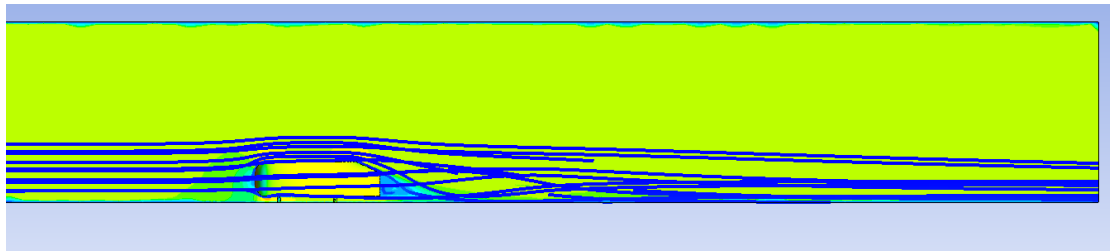


Figure 7. Results of Experiment

After using the feature of Boolean to get the solid body needed for the mesh, name each face is necessary as well.

The fluid domain is ready for mesh as follows:

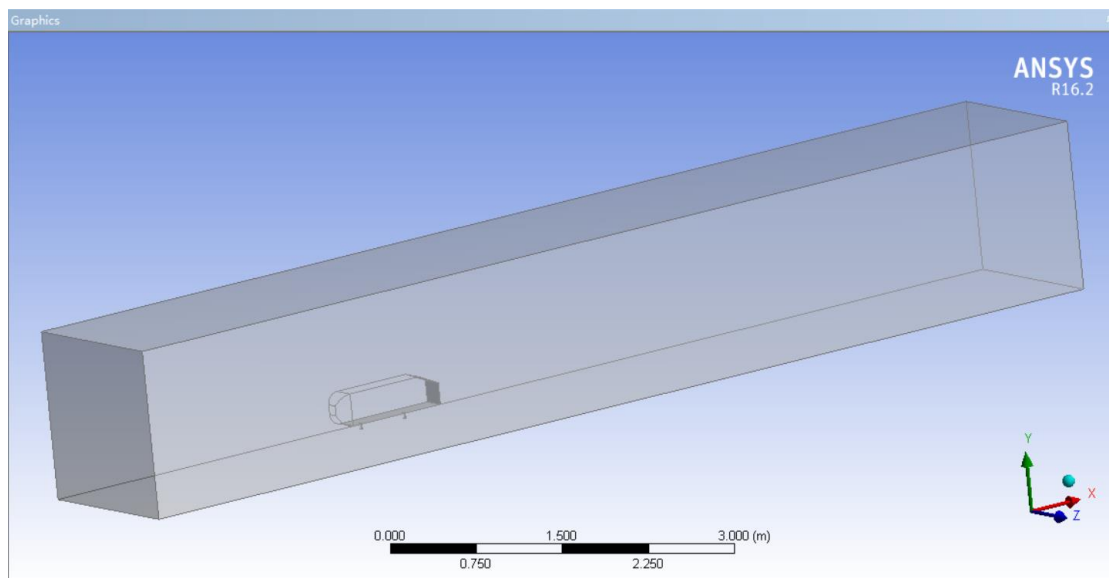


Figure 8. Fluid Domain Geometry

5 Create Mesh and Appraise Quality

To create the mesh, first of all, body of domain has to be created as figure 9 shows.

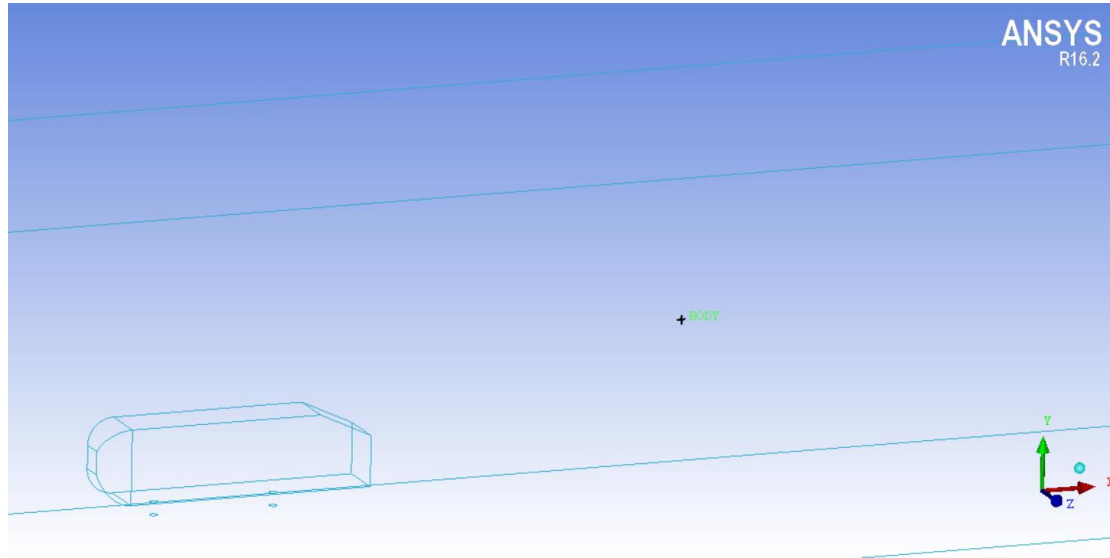


Figure 9. Body of Domain

The work of mesh could be started when the body has been done. The first quick mesh could be computed when global mesh setups like element scale factor, element seed size and global mesh parameters been set.

But obviously, as figure 10 shows it is not suitable for the analysis on account of the accuracy. The feature of part mesh setup could be used to improve it as it can define the maximum size of each part as shown in figure 11. But both of the accuracy and quality of the mesh are still not enough because of the Boundary - layer theory.

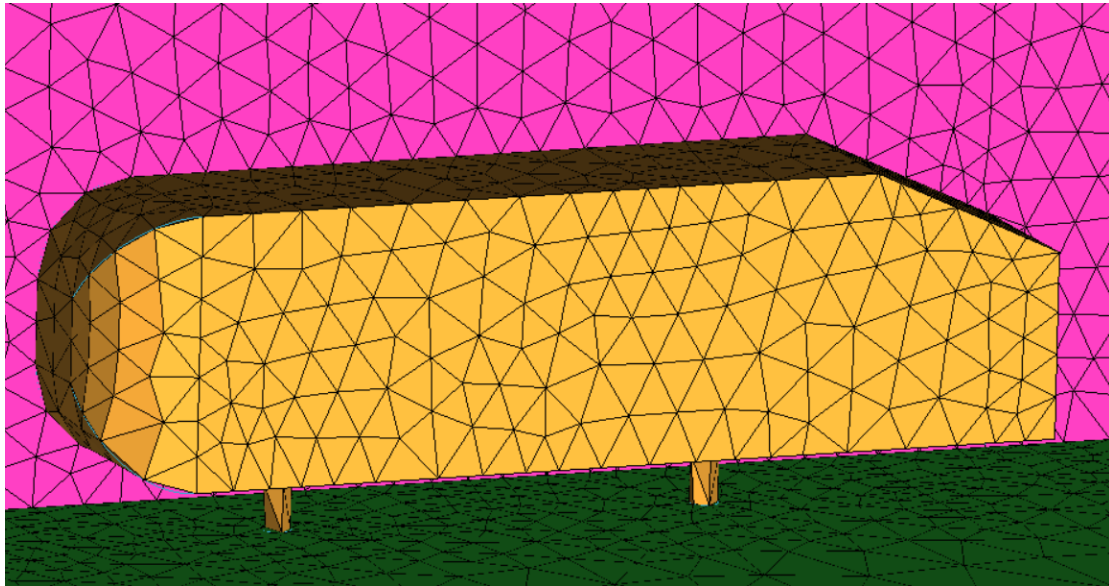


Figure 10. First Quick Mesh

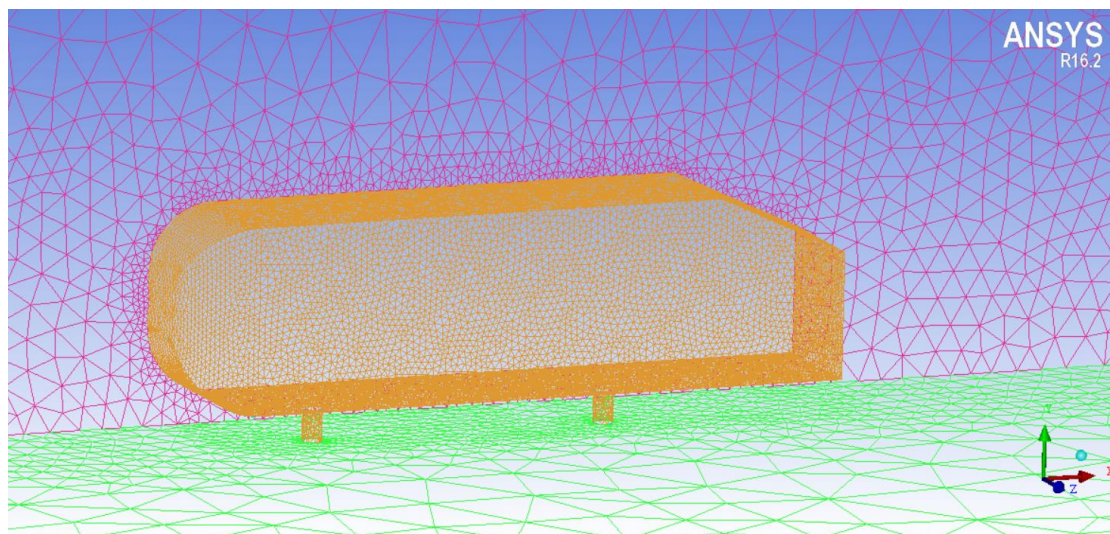


Figure 11. Part Mesh Setup

The boundary – layer theory means when fluid is next to a solid surface, there is no slip of the fluid relative to the surface, so if the surface is stationary, then the fluid immediately adjacent to the surface must have zero velocity. But fluid any distance from the surface can move (vanderain, 2020). This principle states that there is a boundary layer near the solid surface. The fluid in the layer considers a large velocity gradient, and the velocity of the layer that is in contact with the solid surface is always equal to the solid surface speed which is zero when the solid is fixed. However, the boundary layer is very thin, and all fluids with speeds <99% of the mainstream velocity are considered to be within the boundary layer. So apply the

feature of Prism around the geometry and the ground is necessary for the analysis. Mesh quality of domain gone much better in figure 12 after applying the Prism.

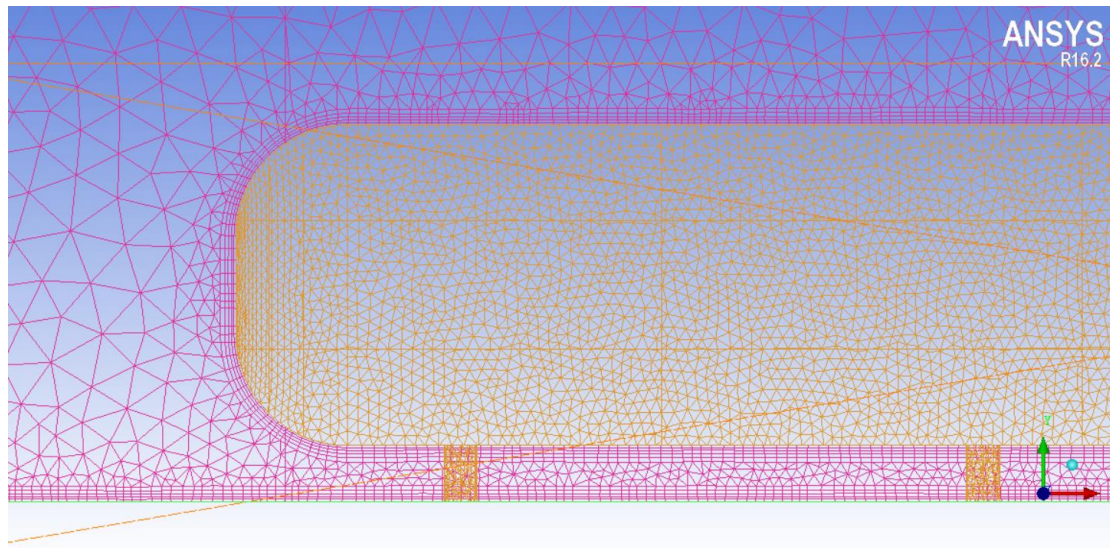


Figure 12. Prism

When vehicle is running at 40 meters per second as the boundary conditions, the airflow flows through and vortex will be generated at the parts where the local airflow speed changes sharply. So the first mesh density around the whole geometry is been created. But from the results of several experiments shown in figure 13, there have significant flow regimes occurs at the rear of the vehicle and the air flow separates from the connecting point of the top and rear faces. So another mesh density especially for the rear part of the geometry is been established like figure 14 and the results comes a huge difference in figure 15.

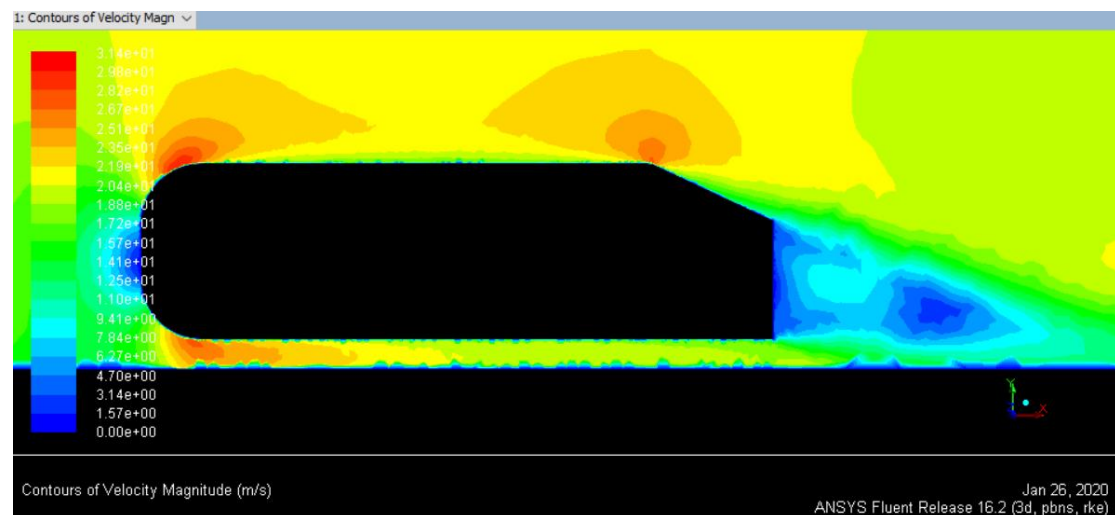


Figure 13. Vortex at Rear Part

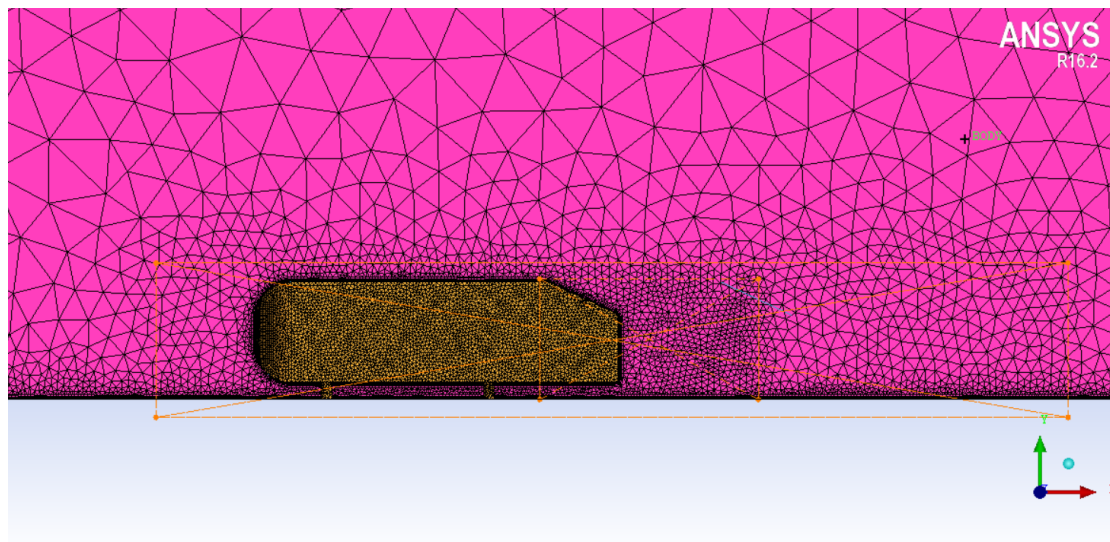


Figure 14. Second Mesh Density

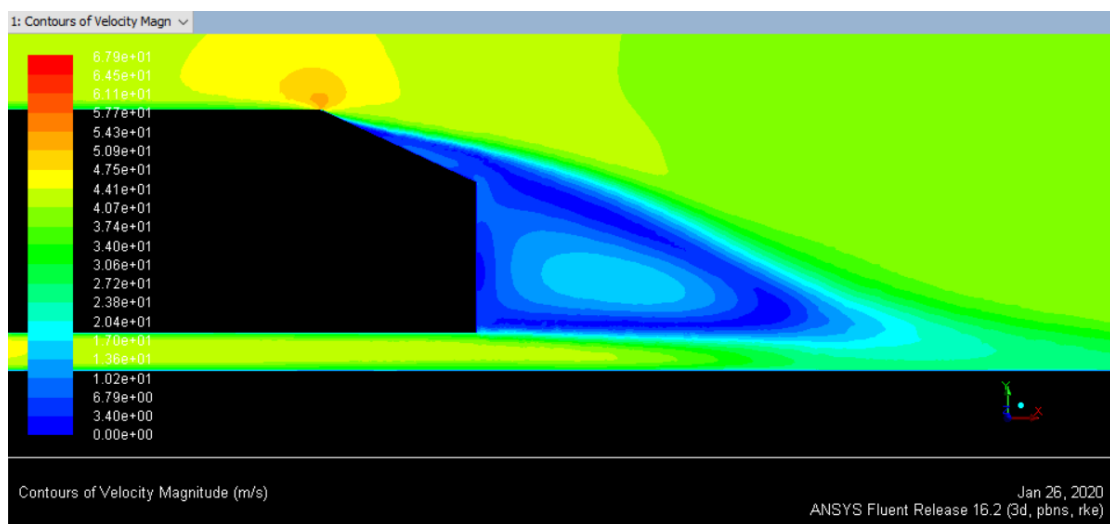


Figure 15. Result of Velocity

6 Solve and Analyze Results

6.1 25 Degrees Ahmed Geometry

As to the solving setup, report quality of mesh from minimum orthogonal quality, maximum orthogonal skew and maximum aspect ratio like figure 16 shows is important so that the quality of mesh could be obtained and recorded in Excel shown in figure 17.

```
Mesh Quality:
Minimum Orthogonal Quality = 2.74682e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.25318e-01
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 1.18590e+01
```

Figure 16. Report Quality

	Report Quality		
	Min Ortho Quality	Max Ortho Skew	Max Aspect Ratio
2	0.274682	0.725318	11.859
3	0.106505	0.893495	29.1955
4	0.0842995	0.9157	31.7134
5	0.104093	0.895907	22.6291
6	0.128521	0.871479	23.6017
7	0.104093	0.895907	22.6291
8	0.07444	0.92556	31.8721

Figure 16. Report Quality in Excel

After the quality check, come back to the solving setup. First of all, the boundary conditions should be set with inlet velocity of 40 meters per second and reference values used to be compute from inlet. As to the models, viscous model with setup of k-epsilon (2 eqn), non-Equilibrium Wall Functions is set. For the k-epsilon Model, both of the Standard and Realizable model are used to calculate the coefficients separately as figure 17 and 18 follows.

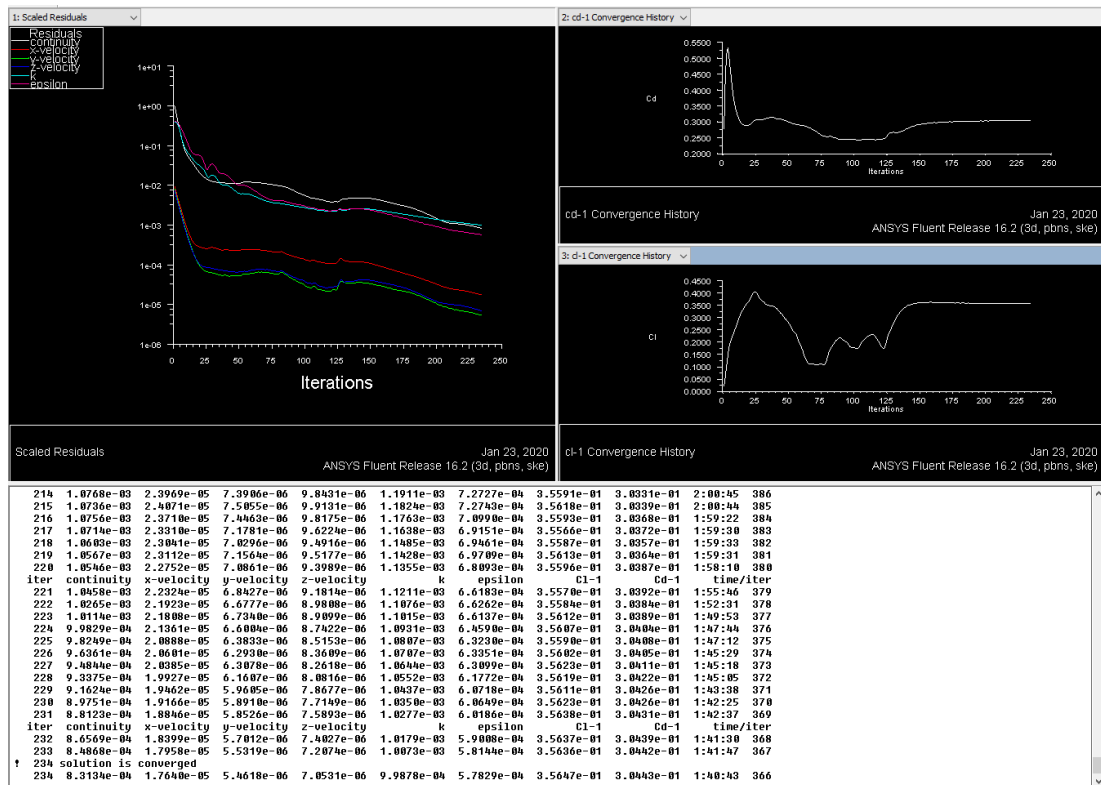


Figure 17. *k-epsilon-standard*

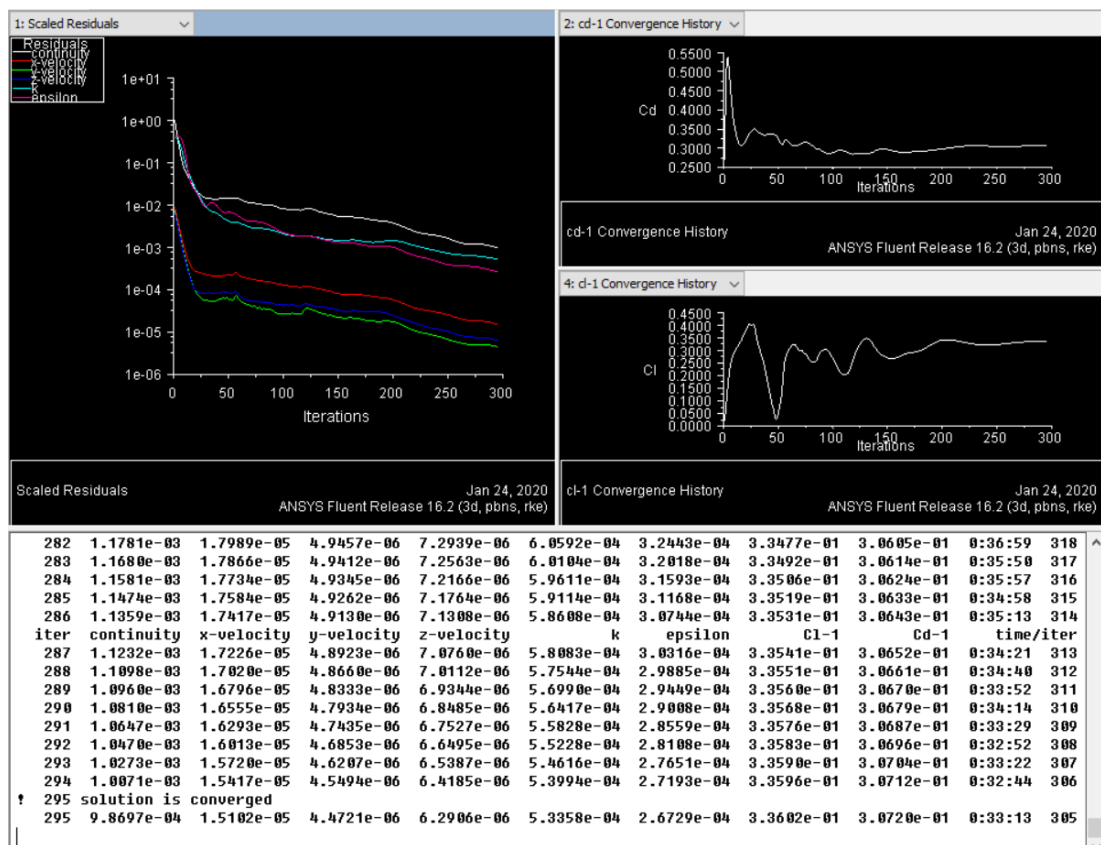


Figure 18. *k-epsilon-realizable*

For the Ahmed geometry with slant angle of 25 degrees, seven calculations are finished with all the results recorded in Excel as follows.

Table 1- Results of Slant Angle 25Degrees with Standard Model

	Slant Angle-25-STAND model			
	CL		CD	
	Experimental (-01)	Tested (-01)	Experimental (-01)	Tested (-01)
1	3.45	3.6451	2.99	3.2747
2	3.45	3.5016	2.99	3.2115
3	3.45	3.7000	2.99	3.0620
4	3.45	3.5617	2.99	3.2384
5	3.45	3.5647	2.99	3.0443
6	3.45	3.5617	2.99	3.2384
7	3.45	3.4166	2.99	3.2857

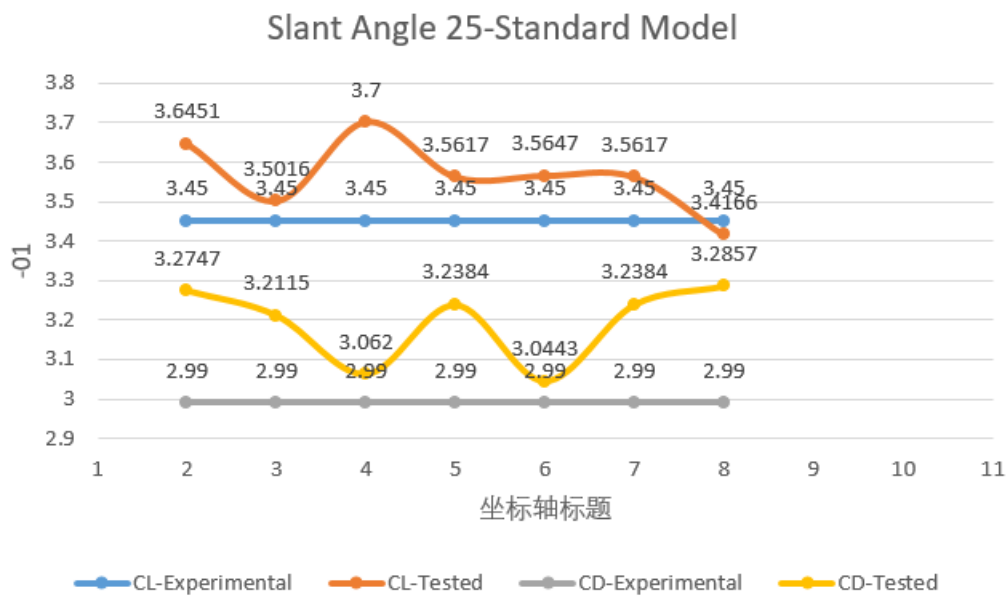


Figure 19. 25 Degrees Standard Model

Table 2-Results of Slant Angle 25 Degrees with Realizable Model

	Slant Angle-25-REA model			
	CL		CD	
	Experimental (-01)	Tested (-01)	Experimental (-01)	Tested (-01)
1	3.45	3.6202	2.99	3.2941
2	3.45	3.2972	2.99	3.0579
3	3.45	3.4018	2.99	2.8470
4	3.45	3.3602	2.99	3.0720
5	3.45	2.7407	2.99	2.6261
6	3.45	3.3602	2.99	3.0720
7	3.45	3.1465	2.99	3.1283

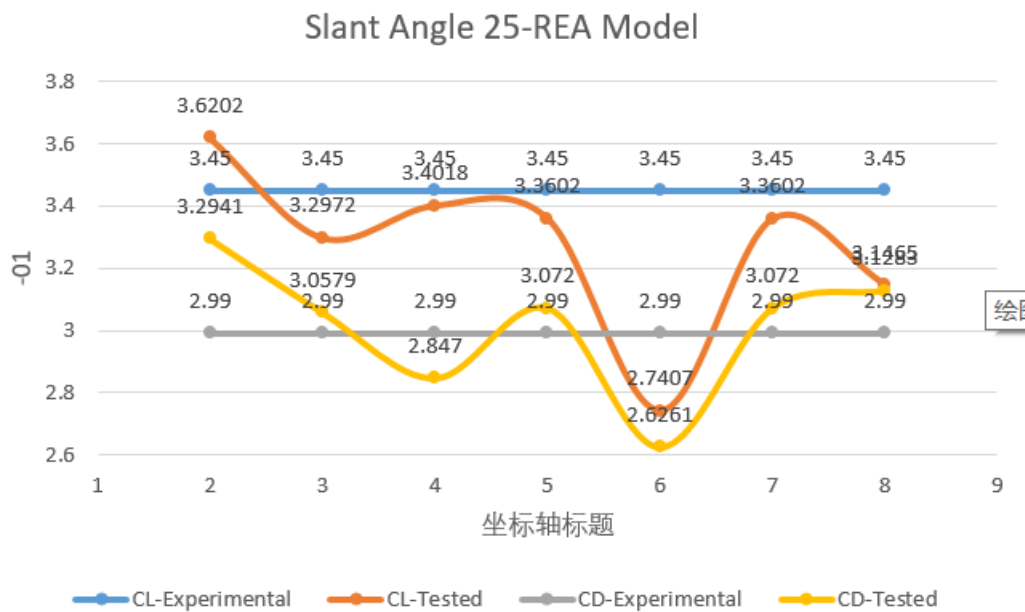


Figure 20. 25 Degrees Realizable Model

From the overall calculation results of both of these two models, the addition of the second mesh density has greatly improved the accuracy of the calculation results.

For the standard model, the deviation is successfully reduced from 5.35% to 1.49%, while the calculation deviation of the real model is also reduced from 10.17% to 2.27%. Six calculations are performed by changing various parameters including domain sizes, mesh strategies, turbulence models and solver settings. In the sixth calculation, there is a large difference between the standard model and the realizable model, especially the calculation results of the realizable model can be discarded from consideration and the analysis due to the large deviation. In the process of reducing the unit size in the mesh step by step, increasing the number of layers of prism, reducing the thickness of prism, and changing the quality of mesh density, it is obviously that the change trends of the calculation results of the two coefficients by the two models are generally consistent. The accuracy of the calculation results shows an overall trend of increasing first and then decreasing. It can be seen that both the standard model and the realizable model can reflect the influence of the changes of various parameters on the calculation results to a certain extent. At the same time, the results are not always better if the settings are more precise. But it is obviously that the finer the setting of each parameter, the longer it takes for the processes of both mesh and solve, so it is more demanding that we can find a balance between the accuracy of calculation and the time spent.

6.2 35 degrees Ahmed Geometry

For the geometry with slant angle of 35 degrees, using the settings of test sixth and the quality and results are as follows:

```
Setting inlet (mixture) ... Done.
Setting mira (mixture) ... Done.
Setting symmetry (mixture) ... Done.
Setting topside (mixture) ... Done.
Setting wheel (mixture) ... Done.
parallel,
Done.
Preparing mesh for display...
Done.
Setting Post Processing and Surfaces information ... Done.
Mesh Quality:
Minimum Orthogonal Quality = 3.87012e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 5.65965e-01
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 1.04943e+01
```

Figure 21. Quality Report of Mesh

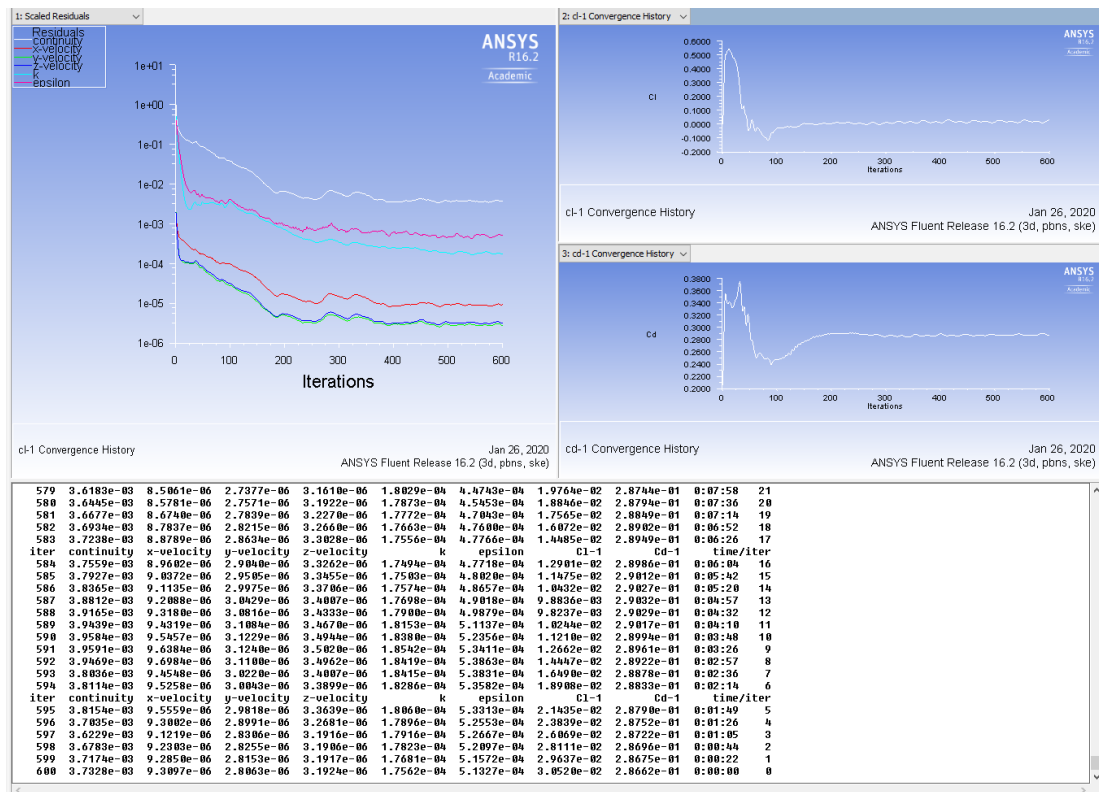


Figure 22. Result of Coefficients

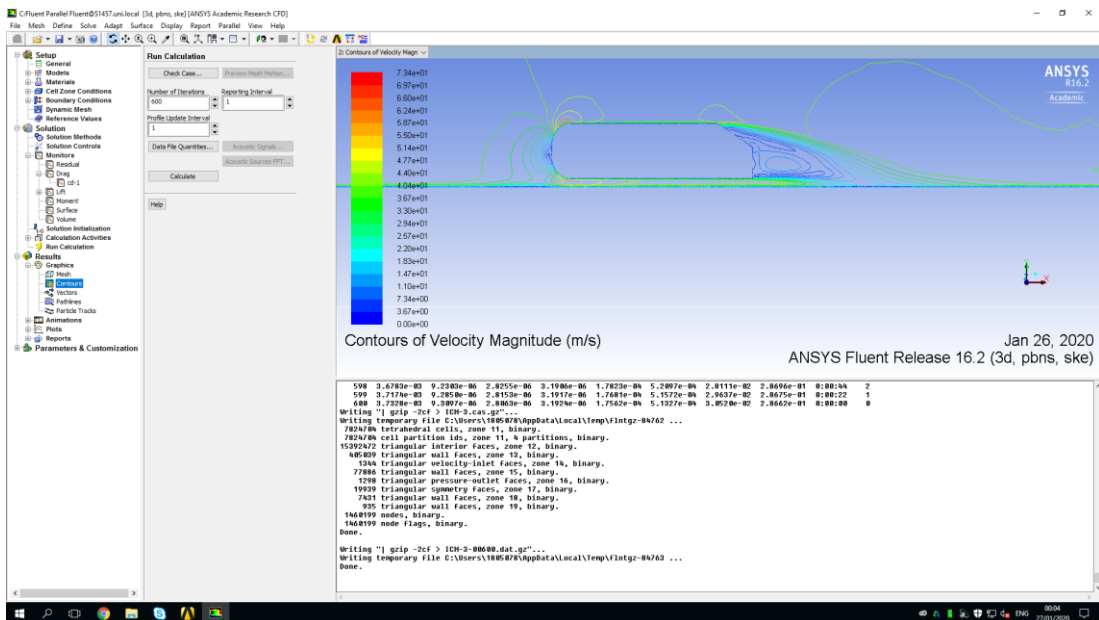


Figure 23. Velocity Result

It is obviously that the best settings been found for geometry with slant angle of 25 degrees is not suitable for the geometry with slant angle of 35 degrees as both of the coefficients have a huge difference from the experimental data.

7 Conclusion

This paper covers the entire process of CFD (Computational fluid dynamics), from the external model import to the establishment of domain, from the creation of the mesh to the setting of the solver and the analysis of calculated results.

In the process of investigating the influence of the setting of each parameter on the final calculation results, it is obviously that the change trends of the calculation results of the two coefficients by the two models are generally consistent. At the same time, the accuracy of the calculation results shows an overall trend of increasing first and then decreasing. From the differences in calculation results and changes in parameter settings, both of the standard model and the realizable model can reflect the influence of the changes of various parameters on the calculation results to a certain extent. However, in this process, it is also found that the accuracy of the calculation results and the precision of the parameter settings are not always linear which means the results are not always better if the settings are more precise. It can be also concluded that each geometry has a different best setting for the calculations of coefficients.

After that, there is another important point, whether for mesh or solve, more precise the parameter settings, longer it will take. When a test been found that the expected calculation time is more than 200 hours, been stopped immediately because it does not make any sense. In addition to the accuracy of the calculation results, it is significant to find the balance between the accuracy of calculation and the time spent.

8 Reference

- Ashok Patidar - Mahindra & Mahindra Limited. (2020 年 01 月 23 日). Comparison of CFD Analysis Methods for Simulating De-Icing Pattern over Automotive Windshield and Windows. 检 索 来 源 : SAE MOBILUS: <https://saemobilus-sae-org.ezproxy.uwtsd.ac.uk/content/2010-01-0555/#authors>
- TudorTim. (2020 年 01 月 23 日). Course Advanced Computational Methods. 检索来源: UWTSD Moodle 2019/2020: https://moodle.uwtsd.ac.uk/pluginfile.php/282728/mod_resource/content/0/M6X00408_Cwk1_CFD.doc
- vanderain. (2020 年 01 月 23 日). On the no-slip condition in fluid mechanics. 检索来源: Baidu Wenku.
- Varma Pakalapati, V. K. (2011, 04 12). CAE based 'Multi Objective Optimization approach for Spot Weld Connections Layout' in Automotive Structure. *SAE INTERNATIONAL*, pp. 50-62.
- Vlahopoulos, N. (2011, 05 17). Modeling of Stiffened Panels Using the Energy Finite Element Analysis. *SAE INTERNATIONAL*, pp. 60-28.
- Yang LeiLi, Chuxuan Wang and Yunqing ZhangQiang. (2017 年 03 月 28 日). Loads Analysis and Optimization of FSAE Race Car Frame. *SAE INTERNATIONAL*, 页 12-18.