



School of Engineering

CFD Ahmed Body

Xilong Song

1904076

Automotive Engineering BEng

22/01/2020

Advanced Computational Methods

M6X00408

Contents

1. Introduction.....	3
1. The Fluid Geometry Domain create.....	4
2. The mesh and appraise quality strategies.....	7
3. The fluent setup.....	10
4. The results of speed graph.....	17
5. Conclusion.....	19
6. Reference.....	19

Introduction

In this study, author demonstrate the methods and techniques that have learned to review, consolidate, extend and apply knowledge and understanding, and to initiate and carry out projects. (Geoffrey M. Le Good, 2004) In automotive aerodynamics much use has been made of generic reference models for research and correlation. Research work has been conducted mostly on small-scale versions of the models to investigate flow regimes and aerodynamic force and moment characteristics while correlation tests have made use of full-scale models to compare results between wind tunnels. More recently reference geometries have also been used as test cases in the validation of computational techniques. Author do the finite element analysis of the Ahmed Body by geometry, model, fluent and results with ANSYS 16.0. The software mainly includes three parts: pre-processing module, analysis calculation module and post-processing module. The pre-processing module provides a powerful solid modelling and meshing tool, and we can conveniently construct finite element analysis. Analysis and calculation module include structural

analysis (linear analysis, nonlinear analysis, and highly nonlinear analysis), fluid dynamics, piezoelectric analysis, and multi-physics coupling analysis, which can simulate a variety of physical media interaction with sensitivity analysis and optimization analysis capabilities. The post-processing module can display the calculation results in colour contour display, gradient display, vector display, particle trace display, three-dimensional slice display, transparent trace display which can see the inside of the structure and other graphical methods. The calculation results are displayed or output in the form of graphs and curves.

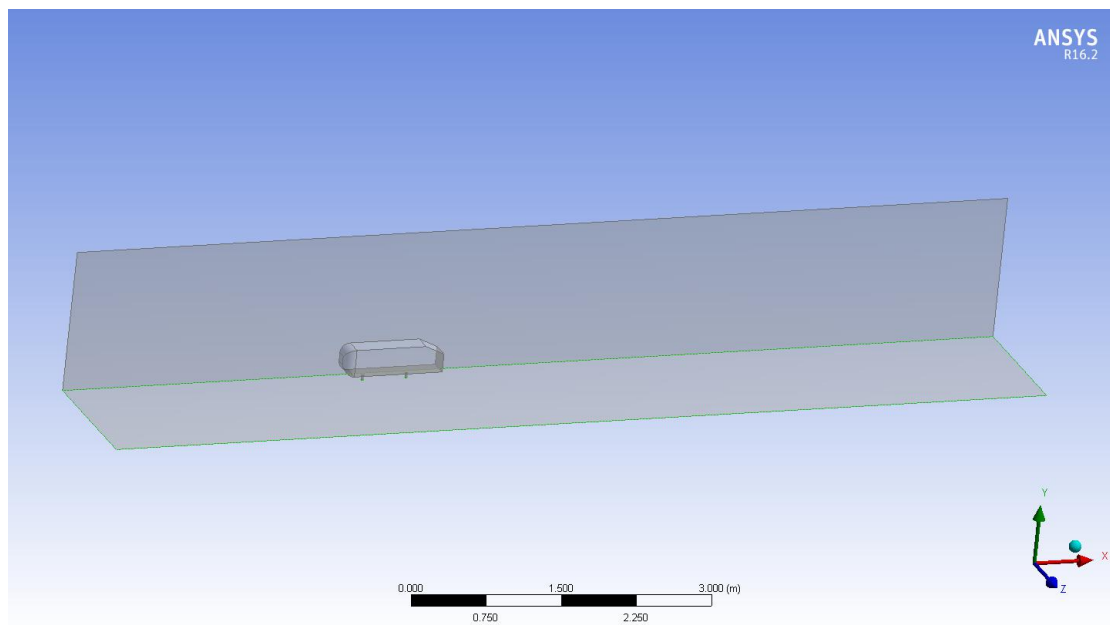
1. The Fluid Geometry Domain create

Flow field means that in a flow field, speed, pressure and so on, which will change. In the case of flight, it is caused by the movement of the aircraft, in the wind tunnel experiment, it is caused by the model being placed in a uniform linear airflow, and the model disturbs the airflow. It is a general term for fluid particle motion described by Erler's method, whose velocity, pressure and other functions are defined in the time and space point coordinate field. Spatial distribution of air movement at a certain moment. Here author creates a geometry domain different from the Ahmed base body and compares them.

[-] Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
[-] Dimensions: 3	
☐ H1	10 m
☐ H4	4 m
☐ V2	1.5 m
[-] Edges: 4	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10

The dimension from the front of Ahmed body to the X axis is 4 m and the total dimension length of the Ahmed body is 10 m.

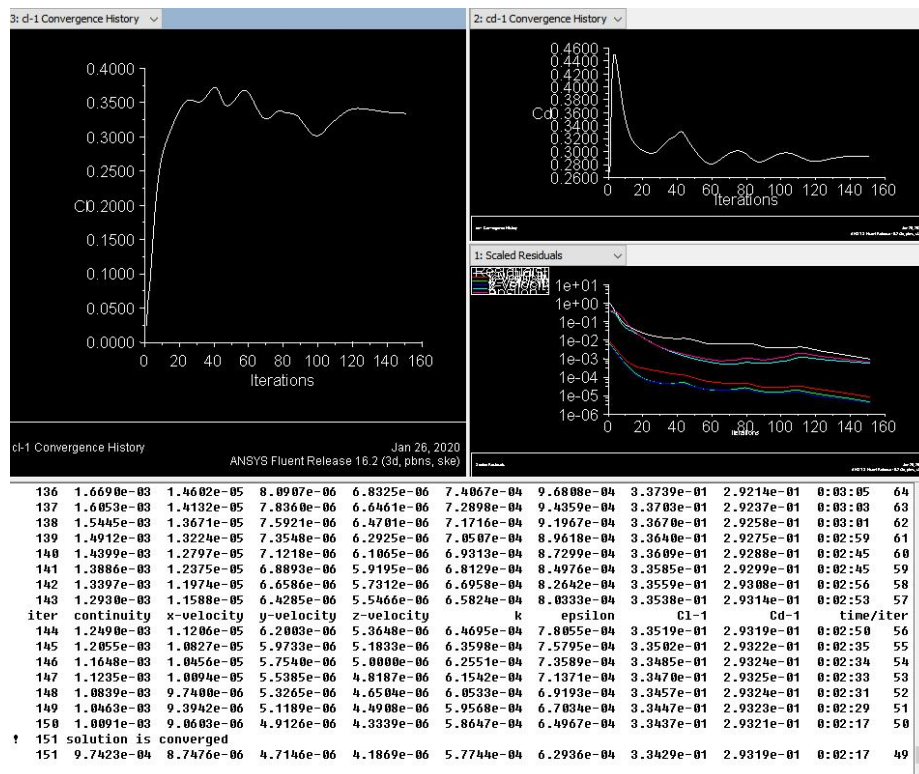
The vertical dimension of the domain is 1.5m. and the extrude of the domain is 1.5m. Then rename the domain surface with topside, symmetry, inlet, outlet and floor. Hiding the topside, inlet, outlet and symmetry, the domain as the graph show:



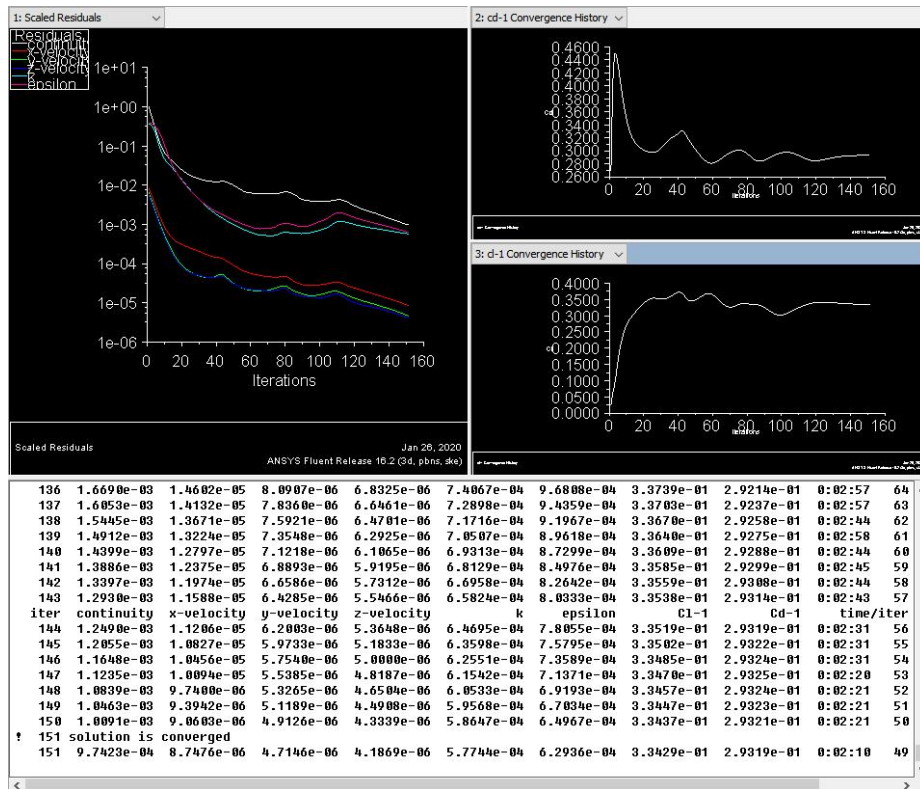
It is the Ahmed body domain and the different domain author creates with different size as the graph show below:

Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
☐ H3	5 m
☐ H4	5 m
☐ V1	0.02 m
☐ V2	2 m
Edges: 4	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10

And the results from different domain size as the graph show



The domain size different from the Ahmed body base.



The result of Ahmed body base domain size shows no difference from the base one. Then the domain size cannot make difference to the value of C_l and C_d .

2. The mesh and appraise quality strategies

Mesh quality refers to the rationality of mesh geometry. The quality will affect the calculation accuracy. Poor quality meshes can even abort calculations. Intuitively, the edges or interior angles of the mesh are not much different, the mesh surface is not excessively distorted, and the mesh quality of the edge nodes near the boundary equal points is better. Mesh quality can be measured by slenderness ratio, taper ratio, internal angel, warpage amount, stretch value, edge node position deviation and

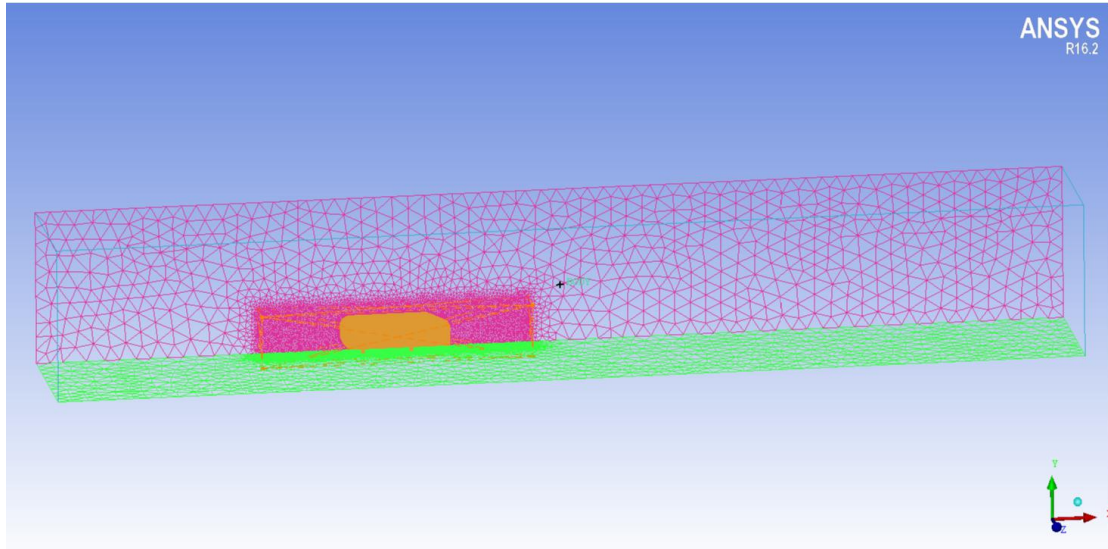
other indicators. When dividing a grid, it is necessary to ensure that high quality meshes are divided, and even individual meshes with poor quality can cause large local errors. And in the secondary parts of the structure, the mesh quality can be appropriately reduced. When there is a poorly quality mesh in the model (called a deformed mesh), the calculation process cannot be performed. For general CFD programs, structured grids require better orthogonality and smoothness, but only for software such as FULENT based on structured grids. Based on the non-mechanical grid's finite volume method, there may be calculation errors in the flux calculation of neighbouring nodes when calculating the flux, so the accuracy is always limited. Here author give two comparative tests about the Ahmed body with different mesh strategies.

a) The Ahmed base body ICEM CFD model

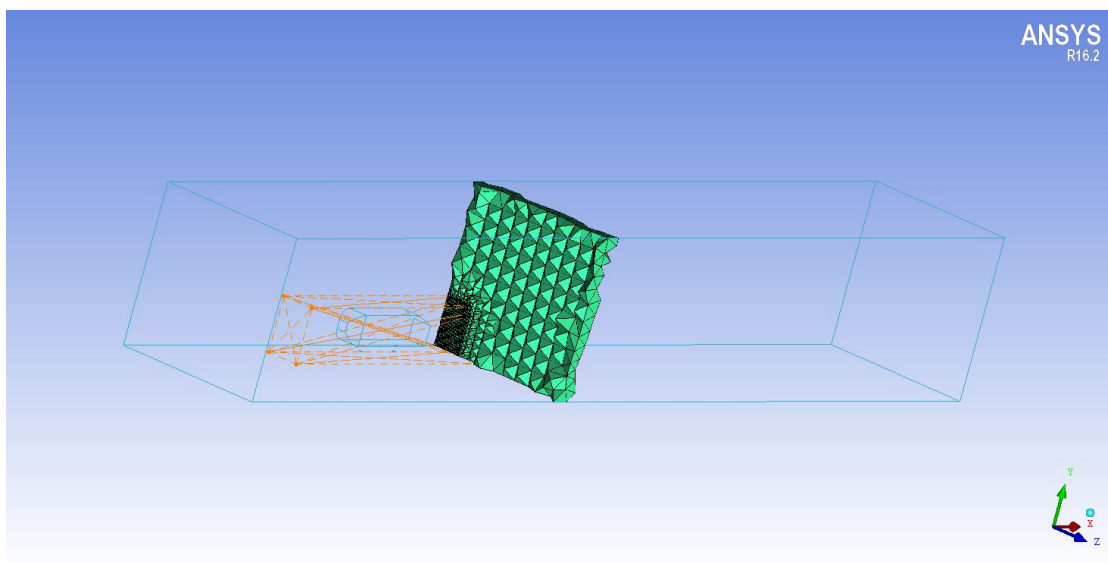
After generating the geometry of the body, model is loading to create the mesh. First check the geometry quality, and set the global mesh. All tri is selected in mesh type and patch independent in mesh method. After doing that, making the global element scale factor 0.5 and global element seed size is 0.5.



Then a fluent body is created by selecting two point of the domain. Author selects the whole Ahmed body to create a density box and select the wake box to scale- 2.5 in X factor, 1.5 in Y factor and 1.5 in Z factor. Before computing the mesh, part mesh setup is defined, Ahmed body's maximum size is 0.1 and the number layers are defined as 5. The body mesh as the picture below show:



After accomplishing the mesh, the mesh cut plane in X direction as the graph show:



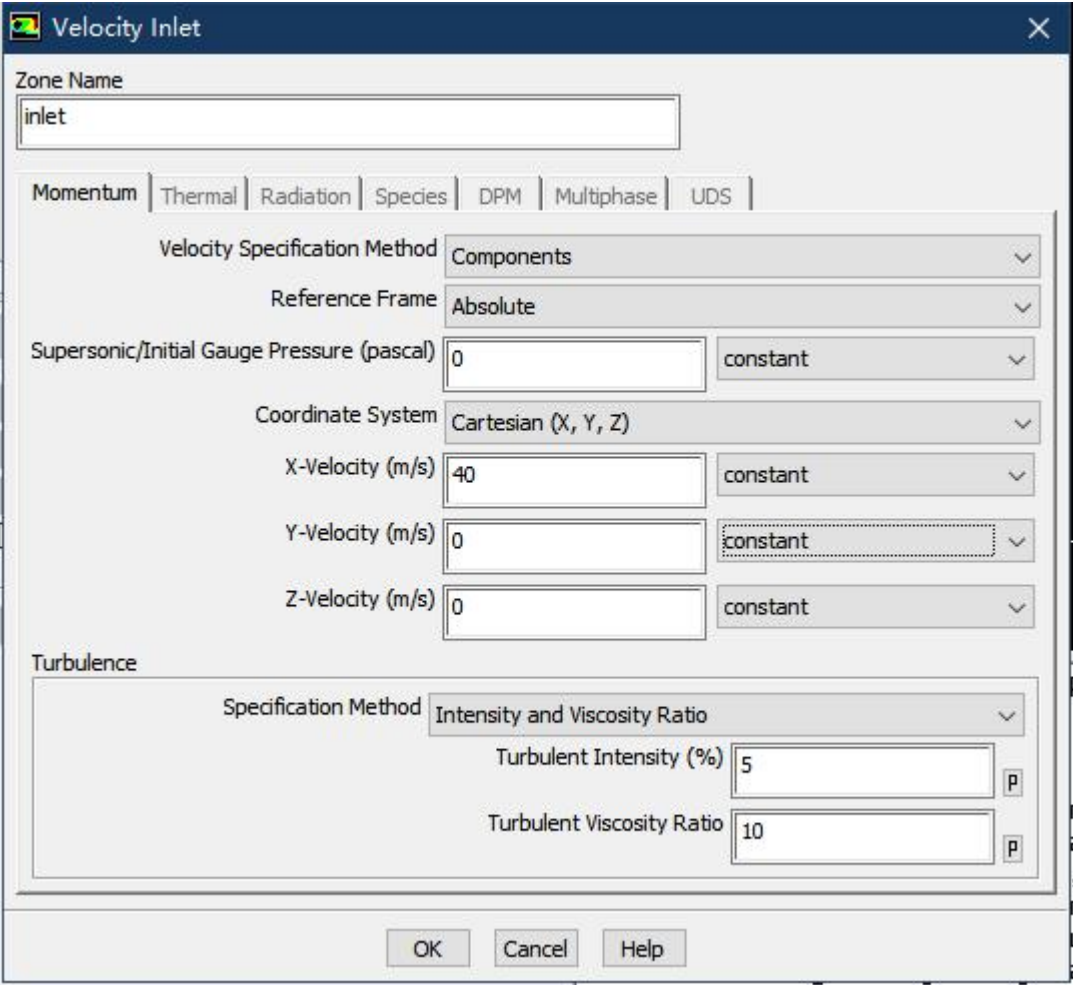
3. The fluent setup

The resistance coefficient is often expressed as a dimensionless quantity in fluid mechanics, which is used to present the resistance of an object in a fluid. The drag coefficient appears in the equation. A smaller drag coefficient indicates that the object

is less subject to wind or fluid resistance. The drag coefficient is related to the shape of the object and its surface characteristics. And the C_l is a dimensionless quantity refers to the ratio of the lift force experienced by an object to the product of the airflow pressure and the reference area. After accomplishing the mesh operation, the first step in fluent setup is checking the mesh and the mesh quality as the graph show:

```
272969 nodes, binary.  
1445652 tetrahedral cells, zone 10, binary.  
2838621 triangular interior faces, zone 11, binary.  
6930 triangular wall faces, zone 12, binary.  
415 triangular velocity-inlet faces, zone 13, binary.  
432 triangular pressure-outlet faces, zone 14, binary.  
9143 triangular symmetry faces, zone 15, binary.  
3825 triangular wall faces, zone 16, binary.  
84621 triangular wall faces, zone 17, binary.
```

Then converting the mesh domain to polyhedral. The inlet boundary conditions velocity inlet and edit the X-velocity with 40 m per second



Velocity Inlet

Zone Name: inlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Components

Reference Frame: Absolute

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Coordinate System: Cartesian (X, Y, Z)

X-Velocity (m/s): 40 constant

Y-Velocity (m/s): 0 constant

Z-Velocity (m/s): 0 constant

Turbulence

Specification Method: Intensity and Viscosity Ratio

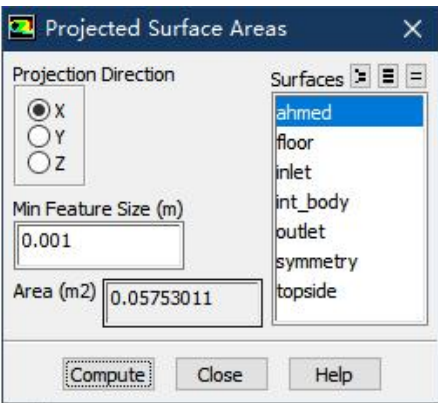
Turbulent Intensity (%): 5 P

Turbulent Viscosity Ratio: 10 P

OK Cancel Help

Ensuring the outlet is pressure outlet and gauge pressure is 0.

Then calculating the project surfaces of the Ahmed body area in the setup reports. Making the minimum feature size be 0.001m and compute the area to 0.05753011 square meter.



Projected Surface Areas

Projection Direction: ☒ X ☐ Y ☐ Z

Min Feature Size (m): 0.001

Area (m2): 0.05753011

Surfaces: ahmed, floor, inlet, int_body, outlet, symmetry, topside

Compute Close Help

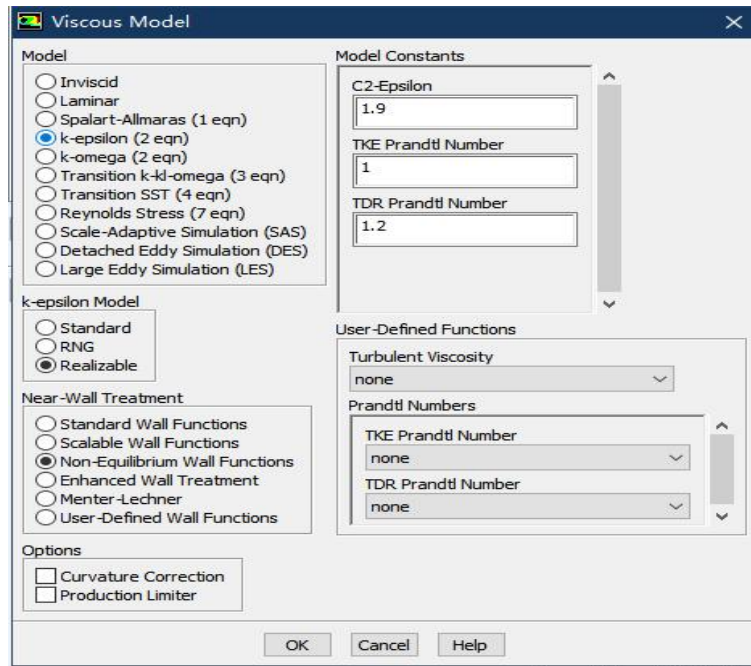
Applying the data to the reference value area and selecting coupled in pressure-velocity coupling. The data in the solution controls as the graph show:

The image shows a 'Solution Controls' dialog box with the following settings:

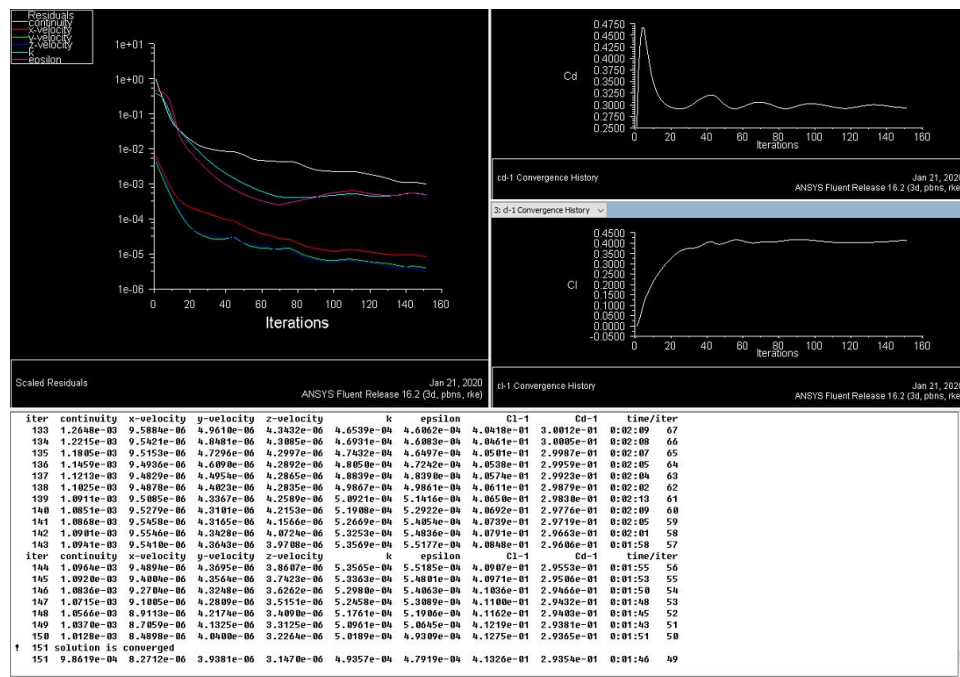
- Flow Courant Number:** 50
- Explicit Relaxation Factors:**
 - Momentum: 0.25
 - Pressure: 0.25
- Under-Relaxation Factors:**
 - Density: 1
 - Body Forces: 1
 - Turbulent Kinetic Energy: 0.5
 - Turbulent Dissipation Rate: 0.5
 - Turbulent Viscosity: 1

At the bottom, there are buttons for 'Default', 'Equations...', 'Limits...', 'Advanced...', and 'Help'.

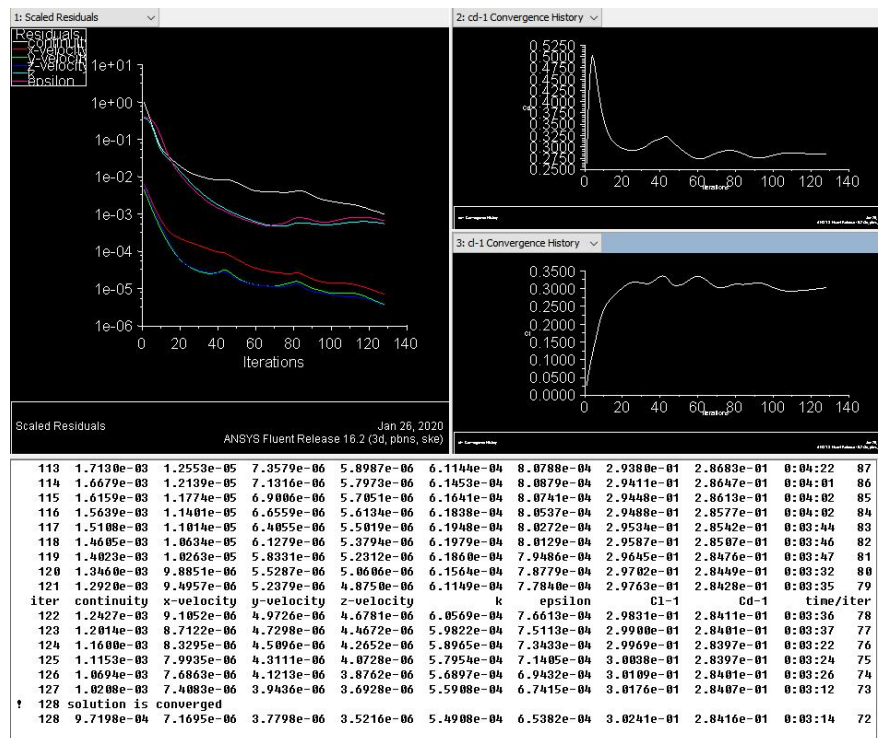
The type of it is pressure-based and absolute velocity formulation. The viscous model is realizable k-epsilon and non-equilibrium wall function.



The viscous model also make difference to the results about the values of C_l and C_d . Making 200 iterations and running the calculating. Here are the two results about the realizable k-epsilon model and standard k-epsilon model.



The realizable k-epsilon model

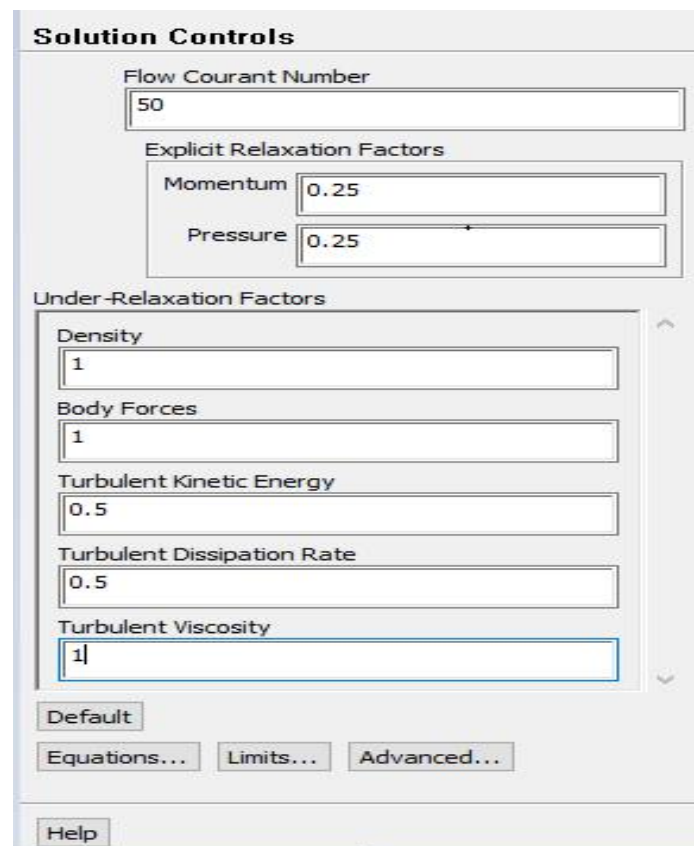


The standard k-epsilon model

And here is the mainly difference between realizable k-epsilon and standard k-epsilon. New turbulent velocity formula used in realizable k-epsilon model equation is derived from the exact operation equation of the mean square root of the vortex disturbance. The realizable k-epsilon model satisfies the constraints on Reynolds stress. This is not possible with standard k-epsilon model and RNG k-epsilon model. The advantage of this feature in the calculation is that it can more accurately simulate the diffusion velocity of the plane and the prototype jet. At the same time, in the calculation of rotational flow, the calculation of boundary layer with direction pressure gradient and the calculation of separated flow, the calculation results are more in line with real

situation. The best turbulence model in the realizable k-epsilon model. Then the realizable k-epsilon model is closer to the real situation.

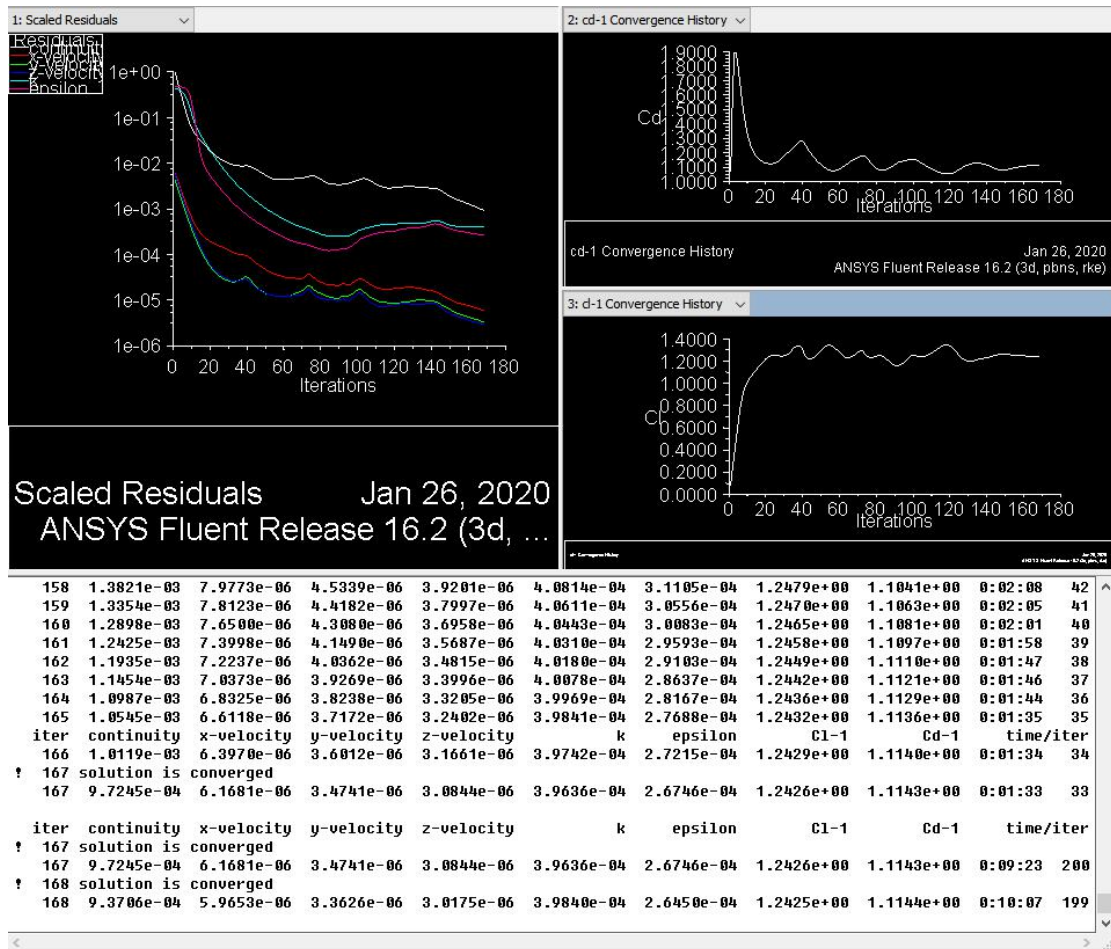
By changing the data of solution control and the data as the graph show:



The image shows the 'Solution Controls' dialog box in ANSYS Fluent. It contains several input fields for numerical parameters. The 'Flow Courant Number' is set to 50. Under 'Explicit Relaxation Factors', both 'Momentum' and 'Pressure' are set to 0.25. Under 'Under-Relaxation Factors', 'Density' is 1, 'Body Forces' is 1, 'Turbulent Kinetic Energy' is 0.5, 'Turbulent Dissipation Rate' is 0.5, and 'Turbulent Viscosity' is 1. At the bottom, there are buttons for 'Default', 'Equations...', 'Limits...', 'Advanced...', and 'Help'.

Parameter	Value
Flow Courant Number	50
Momentum (Explicit Relaxation Factor)	0.25
Pressure (Explicit Relaxation Factor)	0.25
Density (Under-Relaxation Factor)	1
Body Forces (Under-Relaxation Factor)	1
Turbulent Kinetic Energy (Under-Relaxation Factor)	0.5
Turbulent Dissipation Rate (Under-Relaxation Factor)	0.5
Turbulent Viscosity (Under-Relaxation Factor)	1

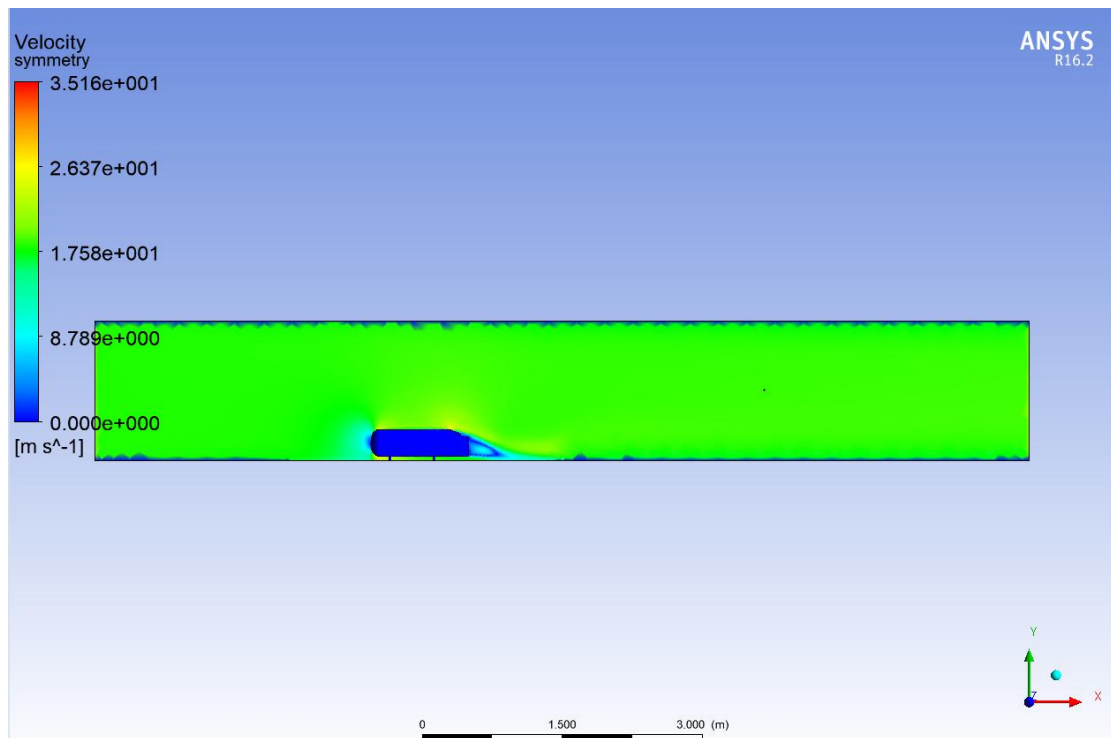
The flow courant number is increasing to 200 and both the momentum explicit relaxation factor and the pressure explicit relaxation factor are increasing to 0.5. The results changing to:



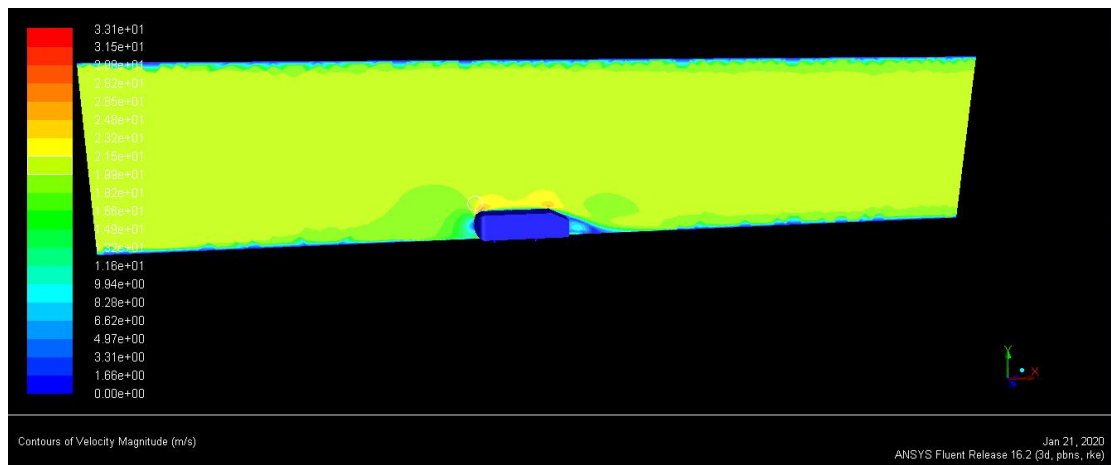
In the realizable k-epsilon model, the values of Cl and Cd become smaller.

4. The results of speed graph

After getting the value of Cd and Cl by numbers of iterations, the velocity graph of Ahmed body base can be carried out by selecting the field contours about the Ahmed body and symmetry in the results.



And the different domain size velocity graph as the graph show:



Which shows different mesh qualities make difference in the fluent velocity. Generally, the mesh quality cannot make differences in speed graph, which means poor quality of mesh strategies make inaccurate results.

5. Conclusion

This report includes geometry domain creating, mesh quality strategies and fluent analysis. Author demonstrate the process of the analysis and compare with the different conditions of them.

(Fluids, 1996) CFD has come a long way in influencing the design of automotive components due to continuing advances in computer hardware and software as well as advances in the numerical techniques to solve the equations of fluid flow. The automotive industry's interest in CFD applications stems from its ability to improve automotive design and to reduce product cost and cycle time. Author also compare them with few comparative tests and analyse them.

6. Reference

Fluids, J. (1996, 12 01). CFD Applications in the Automotive Industry. *Journar of Fluied Engineering*.

Geoffrey M. Le Good, K. P. (2004, 01). On the Use of Reference Models in Automotive Aerodynamics. *SAE Papers*, p. 28.