Validation Study for Flow over a Sphere using CEL Method in Abaqus

Ahmad Firdaus Ismail

Mechanical & Human Factor Development,
MIMOS Berhad, Malaysia

Abstract: The study of fluid dynamics has evolved so much from mathematical modeling to numerical calculations. Recent advancement in numerical calculations introduced by Abaqus for fluid analysis is Coupled Eulerian-Lagrangian (CEL) method. The new method provides quicker solution for a Fluid-Solid Interaction (FSI) analysis. However, when a new method introduced, it is important for the users to know the accuracy, efficiency or even the limitation of the method. The objective of this study is to validate the famous flow over a sphere using CEL method in comparison to the experimental data.

Keywords: CFD Coupling, Coupled Analysis, Experimental Verification, Fluid Mechanics

1. Introduction

1.1 History of Computational Fluid Dynamics

Although aerodynamics is the fundamental of an aircraft design nowadays, the study of fluid mechanics is not something new. Informally, the knowledge of fluid mechanics had been practiced since ancient civilization to build arrows, spears and boats (Garbrecht, 1987). In 250 BC, Archimedes discovers the fundamental principle of hydrostatics in which he develops the law of buoyancy. The knowledge continuously developed through time by noticeable physicists such as Blaise Pascal, Isaac Newton, Daniel Bernoulli, Claude-Louis Navier and George Gabriel Stokes. Since the event near the Kill Devil Hills in 1903, supported by the First World War, the knowledge of aerodynamics has rapidly developed. The demand of a flying machine grew up for military application as well as for commercial. Since Archimedes, it takes over 2 centuries for human kind to develop an aircraft, but it only takes 50 years to send a spacecraft into orbit since the first successful flight by the Wright Brothers.

Nowadays, with the advancement of knowledge in mathematics and computing power, fluid mechanics analysis becomes faster and faster. In the 18^{th} Century, a mathematical model of a fluid flow has been developed by Navier and Stokes which is the Navier-Stokes Equation. This equation describe the velocity at each calculated point in 3-dimensional. Because of lack of computing power, the equation is not widely used until the invention of computer and the development of Computational Fluid Dynamics (CFD) in 1930. CFD is a numerical formulation based on the Navier-Stokes Equation.

As per Moore's Law, the computational power has increased 15 thousand times from 1971 until 2010. Previously, there was no way a full model aircraft can be simulated, but nowadays it can be done within hours. In addition to the hardware advancement, mathematicians and computer programmers worked together to produce efficient way of simulation. Since Abaqus 6.8, Coupled Eulerian-Lagrangian (CEL) method has been introduced. This method enables analyst to simulate large deformation including fluid deformation. CEL method differs from CFD method in terms of:

i. CEL calculates solid and fluid at the same time whereas CFD only calculates fluid flow using Finite Volume Method

ii. CEL provide direct fluid-solid interaction in a single solver Abaqus/Explicit whereas Abaqus/CFD calculates fluid flow and must be coupled with Abaqus/Standard or Abaqus/Explicit

iii. CEL provide faster solution for fluid-solid interaction with some limitation that will be discussed in this paper

1.2 Motivation

When a project comes in to analyze the strength of a mounting of an outdoor product against wind speed, the fluid-solid interaction simulation must be carried out. Due to its complex geometry, it is difficult to be modeled in CFD Co-Simulation. Thus, the CEL method must be explored. However, before running a full scale simulation on the actual product, the accuracy of the CEL method must be firstly analyzed. The most common CFD problems that can be used to study the CEL method is fluid flow over sphere since the experimental data of this problem is vastly available in literatures.

This paper will focus on the following study using fluid flow of sphere as the model:

i. The correlation of CEL method with experimental data

ii. The advantages and disadvantages of CEL method

2. Flow over Sphere Model

2.1 Coefficient of Drag Formula

Since the early days of fluid mechanics study, flow over sphere model is one of the famous problems that physicists tried to solve. There are 2 important forces in aerodynamics namely lift and drag. In this paper, drag forces for a sphere model is obtained from Abaqus CEL simulation is compared to an experimental data.

Drag Force, F_D is defined as (Theodore, 1975):

$$\boldsymbol{F_D = \frac{1}{2}\rho v^2 S c_d} \qquad (1)$$

where

F_D is the drag force

ρ is the air density

v is the air velocity

S is the projected area in the flow direction

c_d is the coefficient of drag

Since c_d is a dimensionless term, aerodynamicists always concluded an experiment by plotting the coefficient of drag graph. Thus, rearranging the Equation 1, yield Coefficient of Drag, c_d:

$$c_d = \frac{2F_D}{\rho v^2 S} \qquad (2)$$

For a sphere, the projected area, S, can be defined as:

$$S = \pi r^2 = \pi \left(\frac{d}{2}\right)^2 = \pi \frac{d^2}{4} \qquad (3)$$

where

r is the radius of the sphere

d is the diameter of the sphere

Upon inserting Equation 3 into Equation 2, yield:

$$c_d = \frac{2F_D}{\rho v^2 \pi \frac{d^2}{4}} \qquad (4)$$

2.2 Abaqus/CAE Setup

2.2.1 Model and Property

The model consists of 2 parts, a sphere and a large block.

The sphere is assumed to be a rigid body and the block is the fluid domain at which in this case with the property of air. Table 1 denotes the detailed of the property values:

Table 1. Property of air used in this paper.

Material Behaviors	**Value**
Density, ρ	1.2047×10^{-12}
Viscosity, μ	1.8205×10^{-10}
Eos	c0 = 343210 (Us – Up)

Please take note that all units in this setup are in accordance to the SI (mm) standard as noted in Table 2 below:

Table 2. Unit Conventions in accordance to SI (mm) Standard.

Quantity	SI (mm)
Length	millimeter (mm)
Force	Newton (N)
Mass	tonne
Time	Second (s)
Stress	Mega Pascal (MPa)
Energy	mili Joule (mJ)
Density	tonne/mm^3

2.2.2 Meshing

Figure 1 shows the overall mesh for the model assembly.

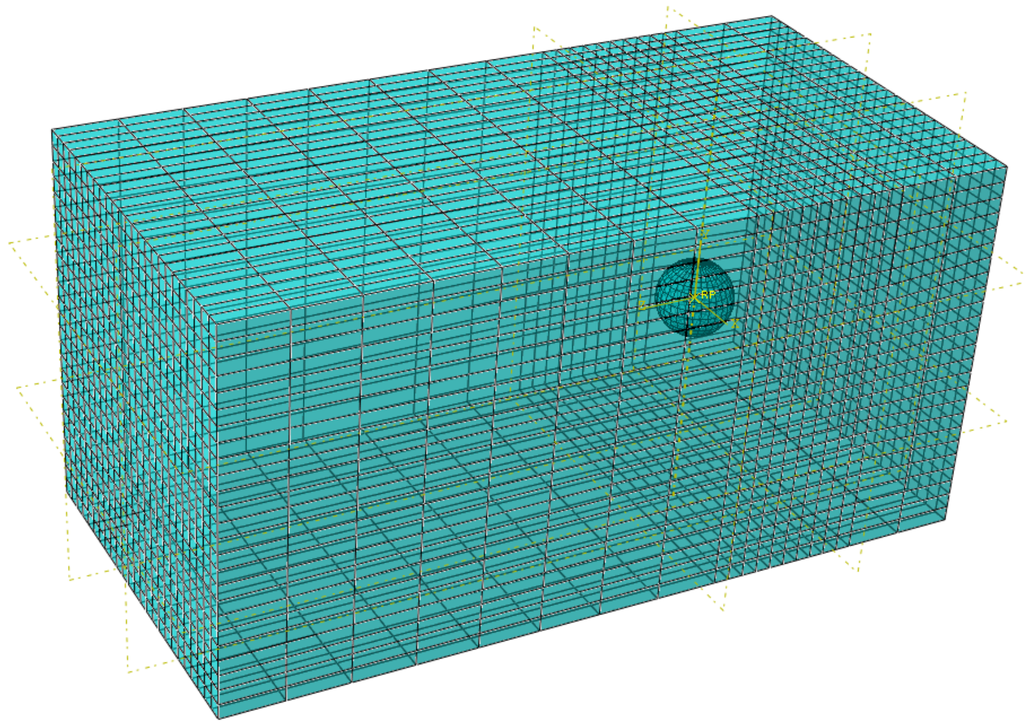

Figure 1. Mesh view in the Assembly Setup

Mesh detailed report for sphere and the fluid domain are shown in Table 3.

Table 3. Mesh Detailed Report

Part Name	Element Type	No of Elements
Sphere	C3D8R	1054
	C3D6	279
Fluid	EC3D8R	12500

2.2.3 Boundary Conditions and Initial Conditions

For the sphere, the boundary condition is fixed in all degree of freedom (encastre).

Whereas for the fluid domain, the boundary condition is the inlet velocity at which the flow will come from. Again in this case, aerodynamicists typically would generalize velocity into Reynolds number, Re (Theodore, 1975).

$$Re = \frac{\rho v L}{\mu} = \frac{\rho v d}{\mu} \qquad (5)$$

where

Re is Reynolds Number

ρ is the air density

v is the air velocity

μ is the air viscosity

L is the characteristic length (which in this case diameter, d)

d is the sphere diameter

Since the input into Abaqus/CAE will be in velocity, $v\left(\frac{mm}{s}\right)$, Equation 5 is rearranged as below:

$$v = \frac{\mu Re}{\rho d} \qquad (6)$$

Since viscosity, density and diameter is fixed, the inlet velocity is determined by the Reynolds number identified from the experimental data that will be compared. Table 4 shows all the Reynolds number in focus for this paper.

Table 4. Inlet velocity calculated from Reynolds Number for boundary conditions input.

Reynolds Number, Re	**Velocity, $v\left(\frac{mm}{s}\right)$**
1	0.151
10	1.511
100	15.112
500	75.558
1,000	151.116
3,000	453.349
5,000	755.582
6,617	999.938
7,000	1057.815
9,000	1360.048

Reynolds Number, Re	**Velocity, $v\ \left(\frac{mm}{s}\right)$**
10,000	1511.165
25,000	3777.912
50,000	7555.823
75,000	11333.735
100,000	15111.646

2.2.4 Interactions

When more than 1 part involved in a simulation, interaction properties need to be defined. In this case, the interaction property has been defined as frictionless tangential behavior. This property has been applied to `All* with self` in Abaqus/CAE.

3. Result and Observation

3.1.1 Flow Observation

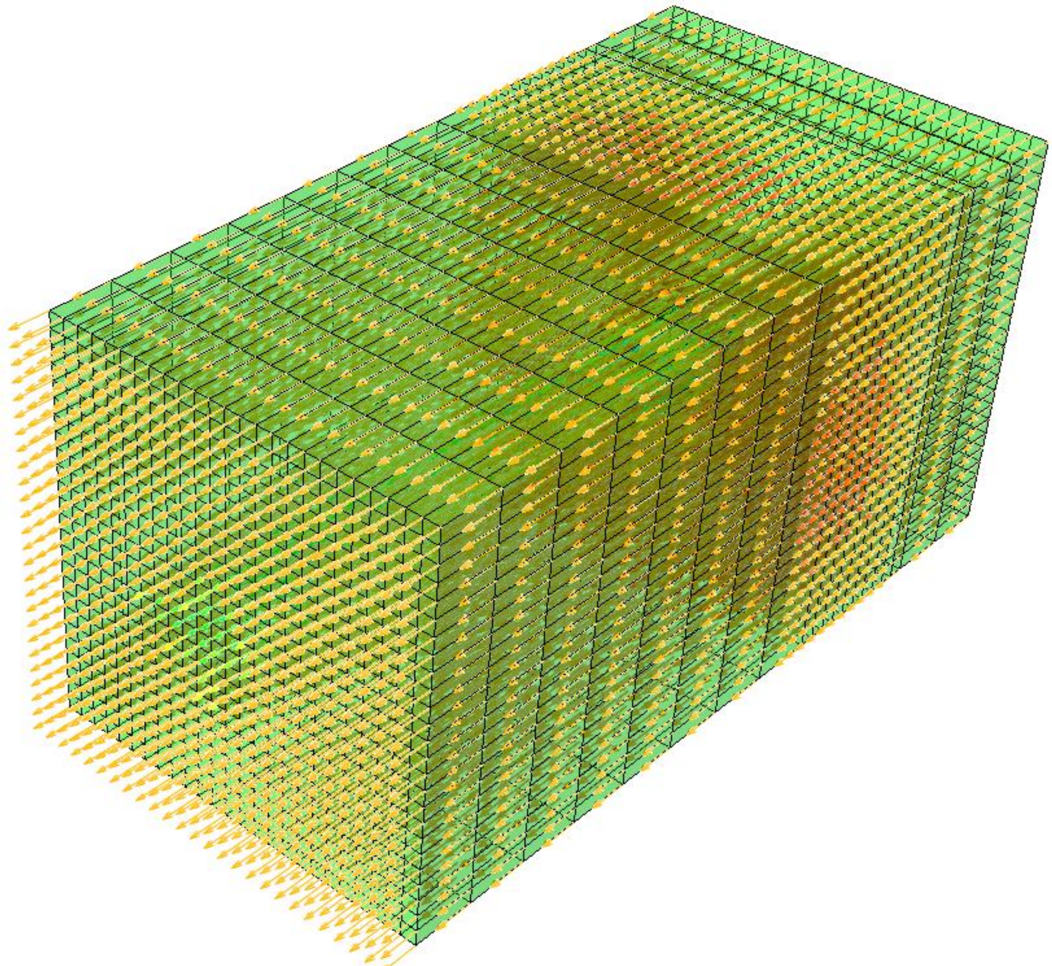

Figure 2. Velocity vector plot, Re = 10,000, Steady-state.

Figure 2 above shows the overall velocity vector plot for Re = 10,000 after achieving steady-state flow. The following figures show the cross-section view for various Reynolds number.

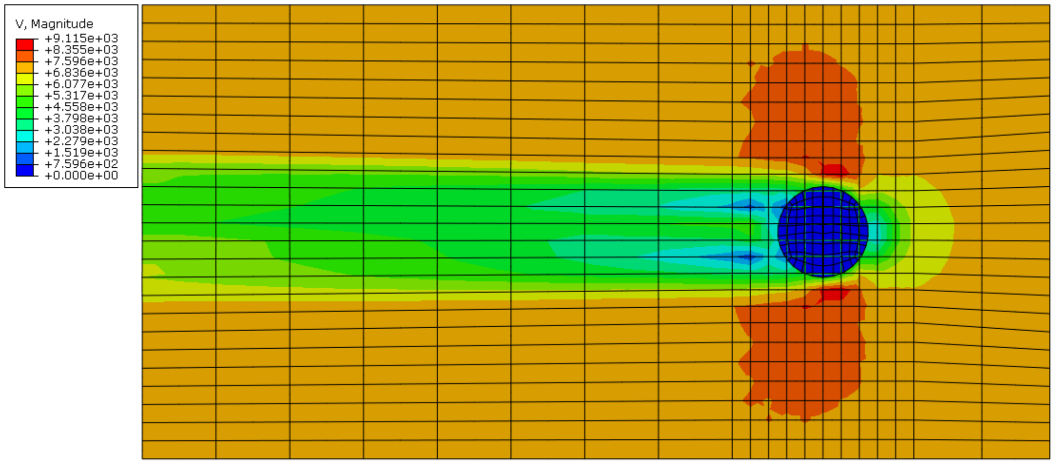

Figure 3. Re = 50,000, Steady-state

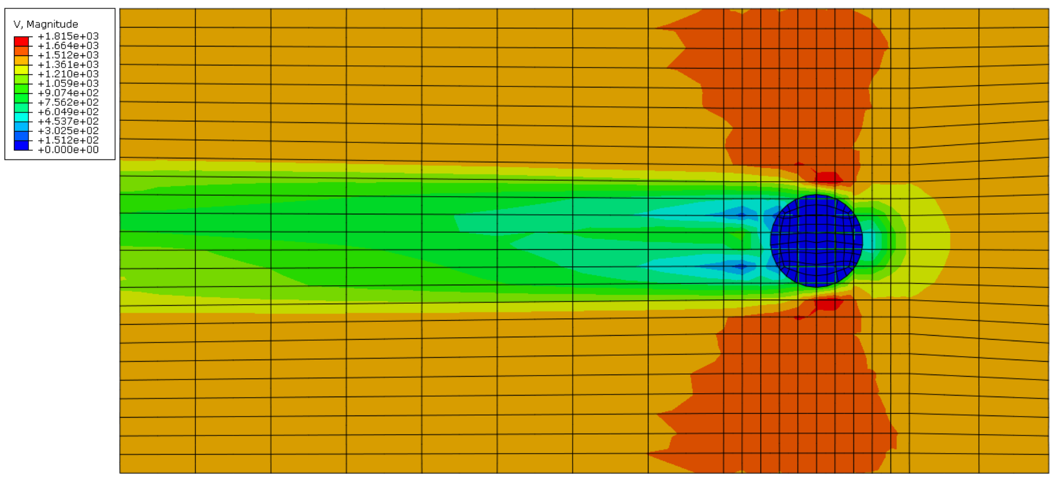

Figure 4. Re = 10,000, Steady-state

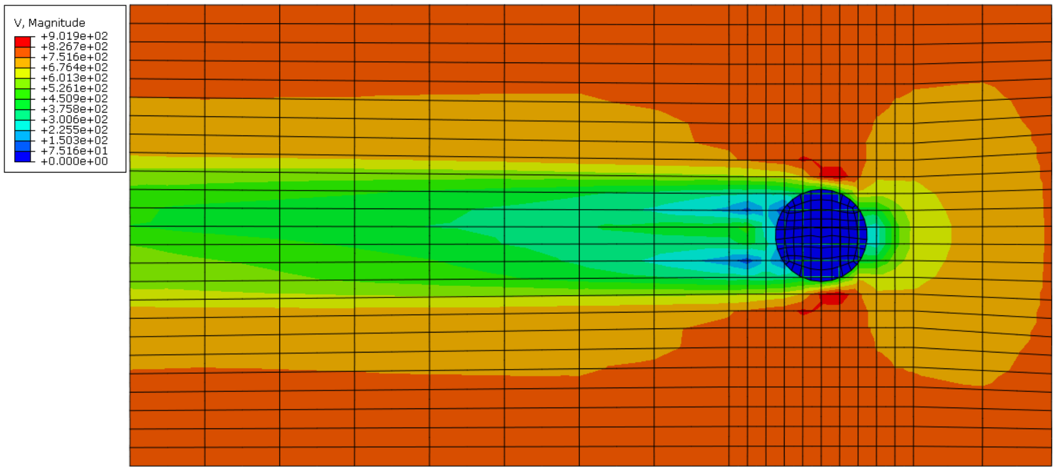

Figure 5. Re = 5,000, Steady-state

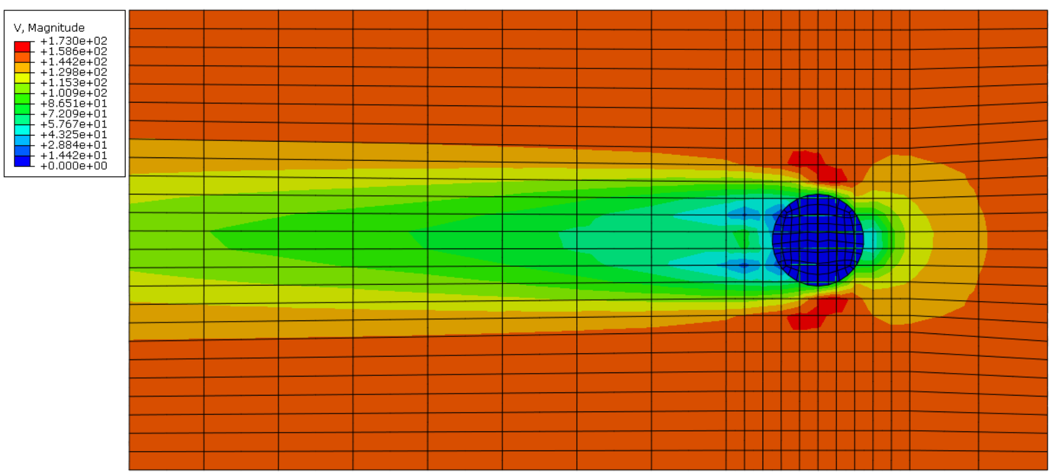

Figure 6. Re = 1,000, Steady-state

Observation from the velocity plots above, there is no significant difference between flow pattern for Re = 50,000 and Re = 10,000. But the flow wake is thinner when Re = 5,000 and getting thinner when Re = 1,000.

3.1.2 Result Extraction

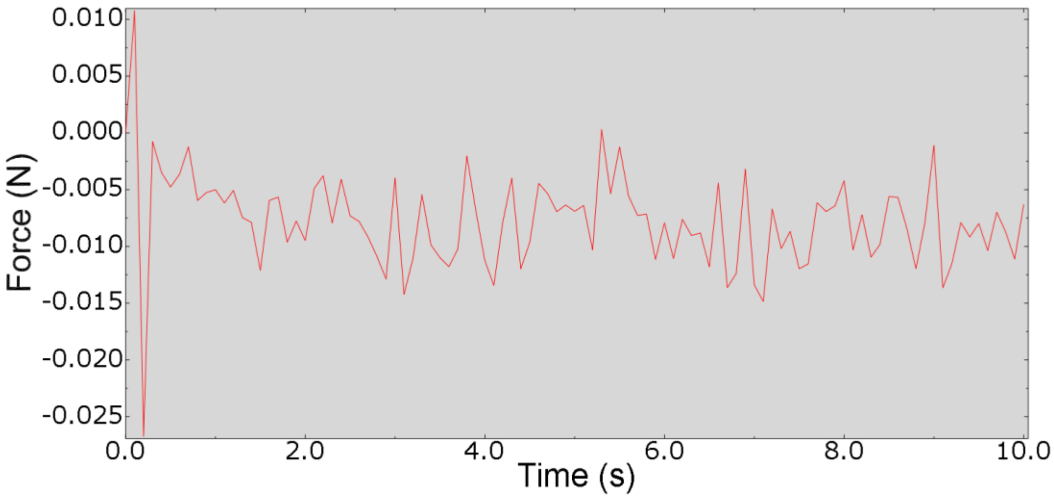

Figure 7. Drag Force vs Time plot, Re = 10,000

From Figure 7, it is observed that the Drag Force is fluctuating in large amplitude at the beginning but tend to stabilize as the time passes. However, during the steady-state, the fluctuations still exist in small amplitudes. This fluctuation is due to the wake flow pulse behind the sphere. Thus, in order to analyze the fluctuation and come out with a proper value of Drag Force at each Reynolds Number, statistical method has been used.

In this case, statistical software called Minitab is used to facilitate in analyzing the data for this study. Using Abaqus/CAE, the Reaction Force (RF) in the direction of flow is extracted using “Create XYData from ODB Field Output” command. The XYData is then exported to Minitab to produce a graphical summary as shown in Figure 8.

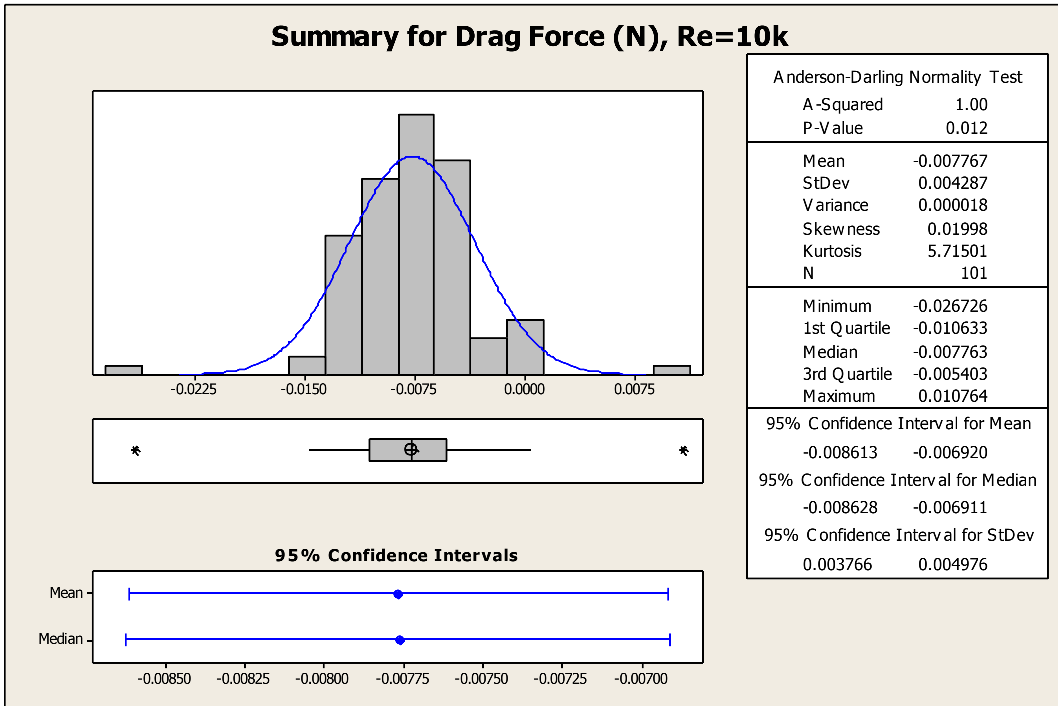

Figure 8. Graphical Summary plot for Drag Force, Re = 10,000. (Produced in Minitab)

From Figure 8, the value for mean and median is very close to each other. This shows that the data is not skewed, thus either mean or median value is a valid value for the Drag Force.

Taking $F_D = 0.00777$ into Equation 4 yield:

$$c_d = \frac{2F_D}{\rho v^2 \pi \frac{d^2}{4}} = \frac{2 \cdot 0.00777}{1.2047\text{x}10^{-12} \cdot 1511.165^2 \cdot 3.142 \cdot \frac{100^2}{4}} = 0.7191$$

Thus, $c_d = 0.7191$ at Re = 10,000.

This process is carried out for each Drag Force extracted from Abaqus Simulation Results at each Reynolds Number listed in Table 4.

3.1.3 Summary Plot

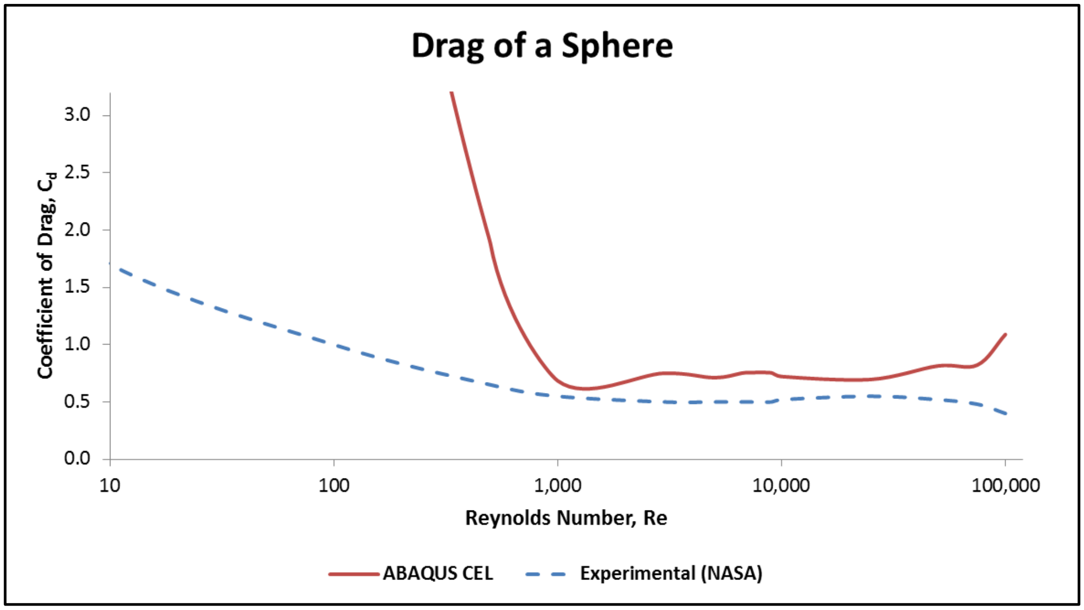

Figure 9. Coefficient of Drag of a Sphere vs Reynolds Number. Comparing between the simulation results and experimental data.

Figure 9 above shows the summary plot for all coefficient of drag obtain by simulation versus experimental result obtained from NASA's website (NASA, January 2013). From the plot above some important observations are:

i. The value of drag coefficient is usable from Re > 1,000 to Re < 100,000.

ii. When Re < 1,000 the drag coefficient goes up exponentially.

iii. When Re > 100,000 the drag coefficient has tendency to go up as well.

4. Conclusion

There are advantages and disadvantages of using Coupled Eulerian-Lagrangian (CEL) method to simulate flow over sphere. The advantage is fluid-solid interaction can be simulated easily and quickly with lower computing power. However, the disadvantage of this method is the accuracy limitation for the speed or velocity which is in this case represented as Reynolds Number. At low speed (Re < 1,000) and very high speed (Re > 100,000) the method will not provide good accuracy. Another disadvantage is unable to show accurate flow pattern for pure fluid flow study such as flow vortices and flow separation. These phenomena only can be observed when using CFD method.

In conclusion, Coupled Eulerian-Lagrangian (CEL) method yields sufficient result for those who want to run a quick fluid-solid interaction study especially for the project that has been mentioned in the introduction above. However, for a deeper fluid-solid interaction simulation, coupled FEA and CFD simulation should be carried out.

5. Future Work

This paper only covers air flow over sphere. There are many common shapes that aerodynamicist study for their drag and lift coefficient. Continuation of this paper would be to study those common shapes by comparing CEL method simulation result versus published experimental data.

6. References

1. G. Garbrecht, "Hydrologic and hydraulic concepts in antiquity in Hydraulics and Hydraulic Research: A Historical Review", A.A. Balkema, 1987.
2. Theodore A. Talay, "Introduction to the Aerodynamics of Flight", NASA, 1975.
3. Abaqus User's Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI.
4. "Coefficient of Drag", http://www.grc.nasa.gov/WWW/k-12/airplane/dragsphere.html, NASA, January 2013.