Mechatronic Systems Engineering
School of Engineering Science
SIMON FRASER UNIVERSITY

ENSC 283 - Introduction to Fluid Mechanics
- Project Report -

Simulated study of effect of parameters such as velocity, aspect ration, and attack angle, on the drag and lift forces for an elliptical cylinder.

Student Name: Sohail S. Sangha
Student number: 301186636

Report due date: 21st, April, 2013

Introduction

In the study of fluid dynamics, one of the most popular implications is the study of lift and drag produced over an airfoil. Such studies are of vital importance to the aerospace, aircraft, hydro, automobile, wind turbine industries, and the list goes on even further. Often these shapes are very complex for hand calculations, thus they are transformed into a different domain, where an easier shape is used to evaluate them. Other method is the application of computer simulations to study the approximated behavior of the airfoil. This helps in pin pointing definite behaviors and characteristics which are important to the designer. Although the simulations have and do afford results very close to experimental values, still time to time the validity of simulations have come under scrutiny as theoretical, and experimental solutions show very different behaviors. Hence these simulations afford us a quick study of the behavior but never afford an exactly accurate solution.

In this project we study an elliptical object embedded in a 2-D flow, and study the affect of various variables like the aspect ratio, angle of attack of body, and the velocity of the body. Such a body can be seen as a cross section of an elliptical cylinder, hence the results of such a study can be applied or compared to various real world examples. Although the study is still questionable as we are certifying a laminar flow, which in a sense is never in question. The more important question is the performance of such airfoils under turbulent air. An effect which is achieved in wind tunnels.

We shall conduct this study, under the ideology that it will teach us further about how various parameters effect the lift and drag, by studying the various velocity and pressure fields for the object, and further evaluation the drag, and lift coefficients for the object.

Model

In this study our variables are:

$$\epsilon=\frac{a}{b}=\frac{minor\ radii\ of\ ellipse}{major\ radii\ of\ ellipse}$$

$$u=velocity\ of\ the\ flow[m/s]$$

$$\alpha=angle\ of\ attack\ of\ the\ ellipse\ on\ the\ flow[Degrees]$$

$$h=16\times a\ height\ of\ fluid\ flow\ domain[m]$$

$$w=4\times b\ width\ of\ fluid\ flow\ domain[m]$$

and the constant is

$$b=2.5\,cm$$

To solve this problem we use the software, ComSol, which will solve the Navier Stroke's equation for fluid flow, using linear approximation. We further evaluate the forces in horizontal, and vertical direction acting on the object, in order to evaluate the coefficient of drag, and lift for the given set of variables.
The values of drag, and lift coefficients is calculated using the following equations

$$C_D=\frac{2F_D}{\rho u^2 A},\ C_L=\frac{2F_L}{\rho u^2 A} \qquad \text{eq (1)}$$

where F_D and F_L are then net drag and lift forces on the object, where these forces are produced due to pressure difference, and frictional force from the flow past the body. These forces are obtained using ComSol by integrating the necessary parameters over the whole boundary of the body.

Projected Area and Net force considerations

In eq(1) we are supposed to calculate the drag and lift forces over small chunks of area, these areas are perpendicular to the area in which the model lies. Our model can be seen as a very thin slice of an actual cylinder, hence if our integral force for the boundary of thin slice is given by f_D for drag, and f_L for lift, then $F_D=\int f_D dz, F_L=\int f_L dz$, as we assume that these forces are independent of the z dimension. $F_D=H\times f_D, F_L=H\times f_L$, where H is the height of cylinder in z dimension. Secondly the area asked for can be considered as the wet area of the cylinder in contact with the flow. For an ellipse the perimeter is approximated by

$$Y\approx 2\pi\sqrt{\frac{a^2+b^2}{2}}$$

Therefore the net projected area $A=H\times Y$, and the modified equations for drag and lift coefficient become

$$C_D=\frac{2f_D}{\rho u^2 Y}, C_L=\frac{2f_L}{\rho u^2 Y} \qquad \text{eq(2)}$$

Grid In-dependency Study

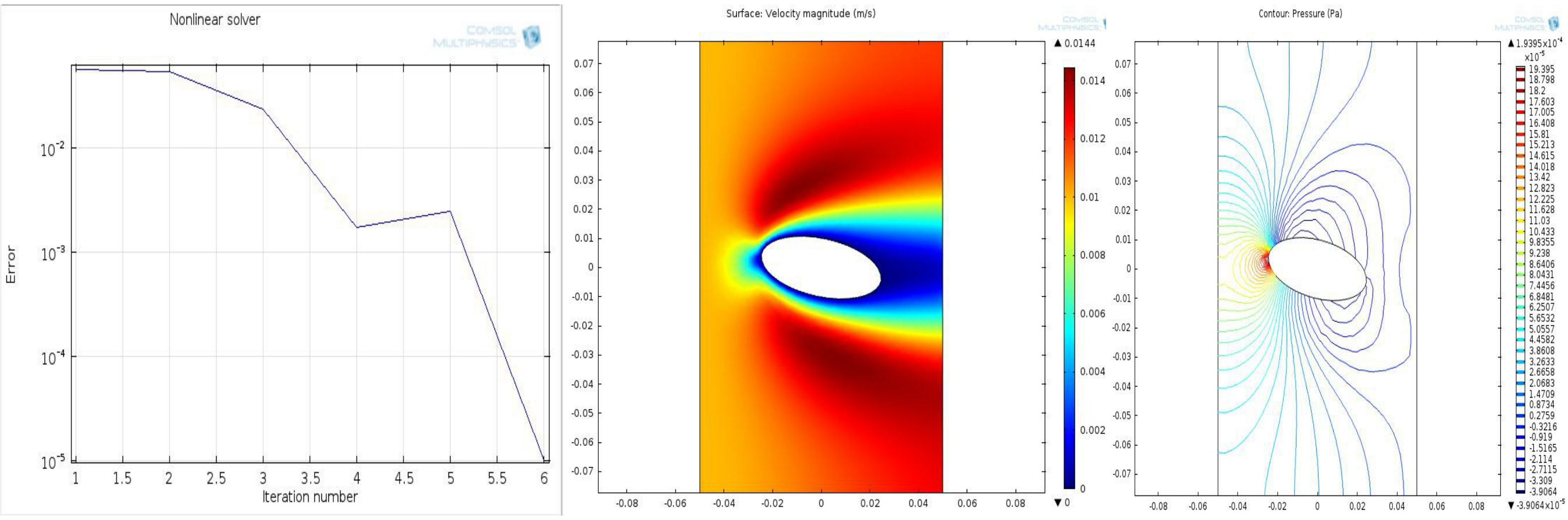
For this study, the selected grid type is triangular, as it is one of the best suited grids for fluid dynamics study. ComSol further offers the fineness of the grid. Higher fineness means that the system is evaluated at more points using the approximation to differential equations before progressing. By generality the accuracy of the solution should increase with the decrease in the grid size, although this is not true, as we are trying to approximate a highly complex differential equations using linear solutions, the equations have limitations in their accuracy which become more prominent as we try to make these linear approximations smaller and smaller. Hence we have to do a grid in-dependency study so as to certain that ComSol will be able to find a solution, while at the same time compare solutions in order to certify that we are not wasting computer memory and time by choosing smaller than useful grid size. Although the convergence rate of the solution can change very fast, depending on the domain and scatter of the triangle grid, by choosing one orientation to study, we can validate grid sizes with respect to one another.

For this study the variables were set as:

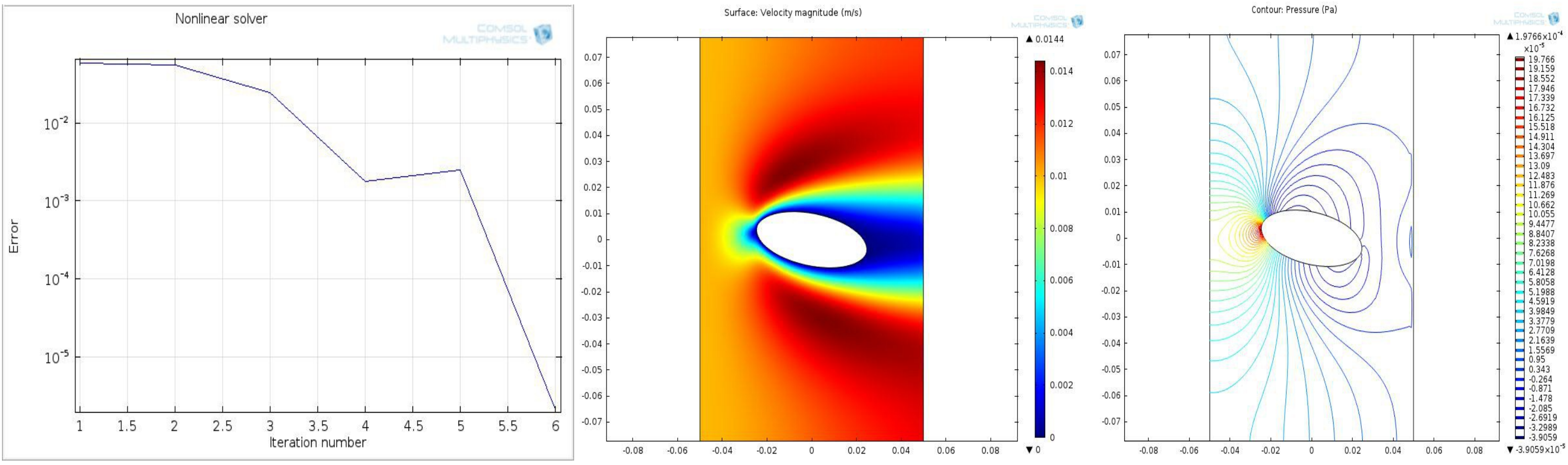
$$\epsilon=0.4$$
$$u=0.01\,m/s$$
$$\alpha=10^o$$

While the grid size was changed from normal to extremely fine.

Normal



Extra Fine



Extremely Fine
In this case the solution did not converge and ComSol was not able to evaluate the system. Thus confirming our supposition regarding the performance of simulators with respect to allotted grid sizes.

Conclusion
As given in the convergence graphs above, the solution achieves the required accuracy in nearly the same number of iterations for all selected grid sizes, except extremely fine. But going for a larger grid size will mean discarding the fineness of the result. As it might be evident in the velocity graphs. The solution becomes more coarse with increase in grid size. The end product i.e. coefficients of drag, and lift can be guaranteed the same accuracy from the evaluations, thus making the choice of a finer grid more of an aesthetic choice for our particular study. Still keeping in mind that the results can be highly affected by the parameters of the study, I decided to continue the study with finer size grid, as it makes sure that I am not overusing the computer, or will run into non converging solutions.

Parametric Study

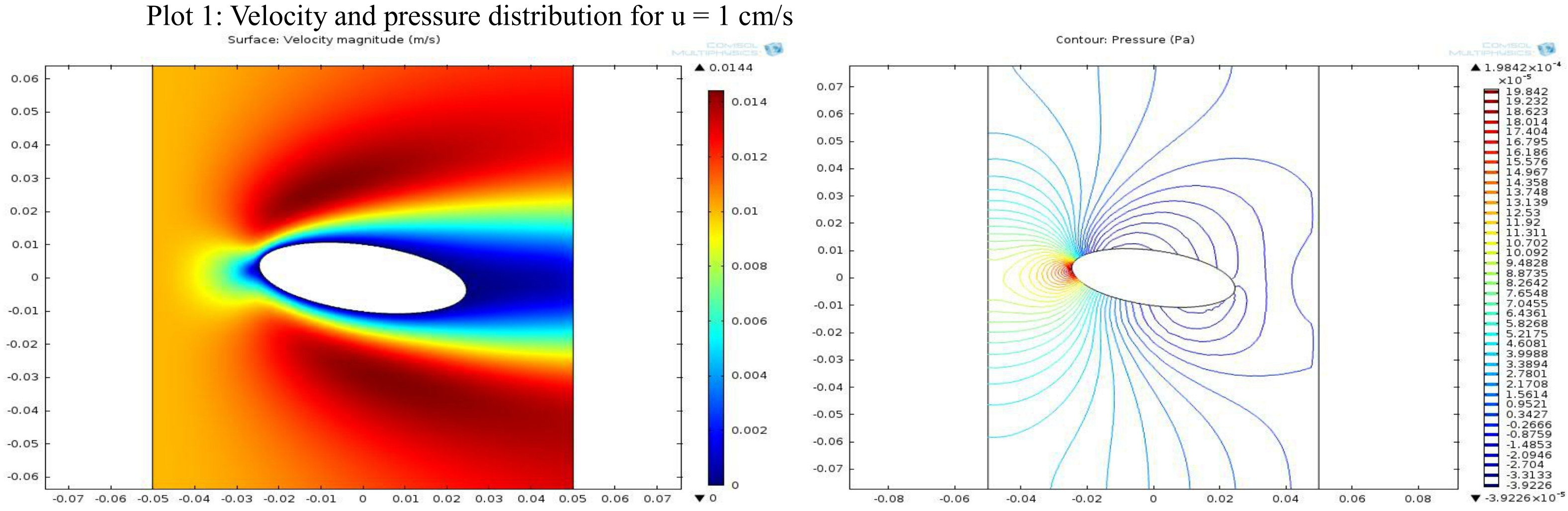
1. Dependence on velocity

Constant parameters

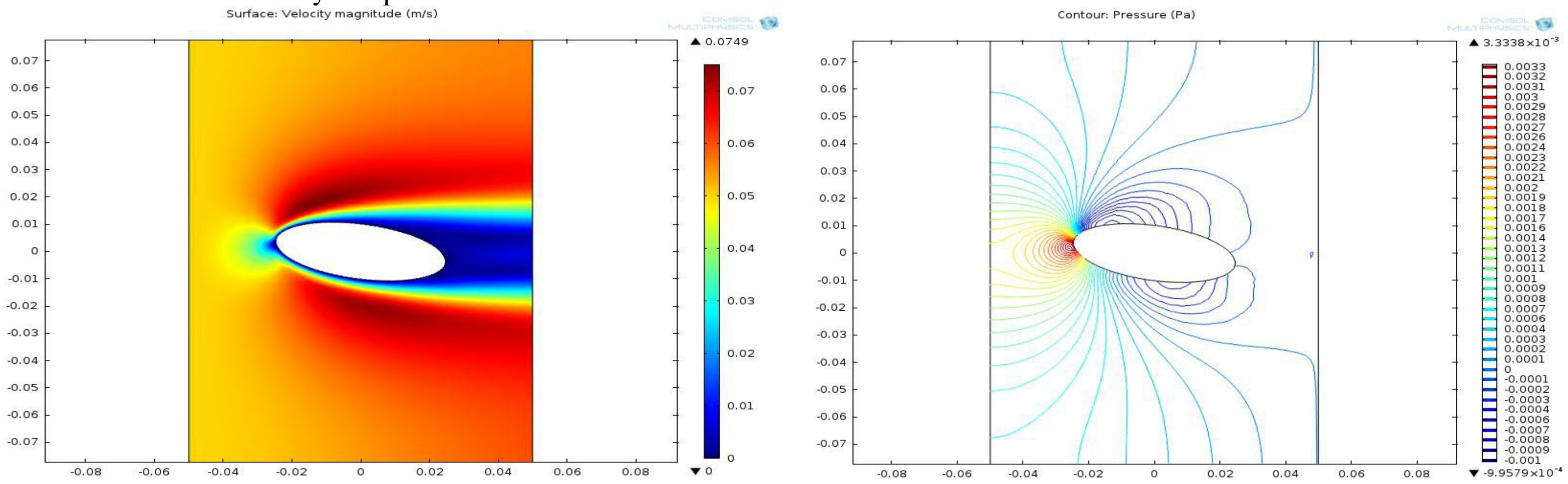
$\epsilon = 0.4$

$\alpha = 10^o$

Plot 1: Velocity and pressure distribution for u = 1 cm/s



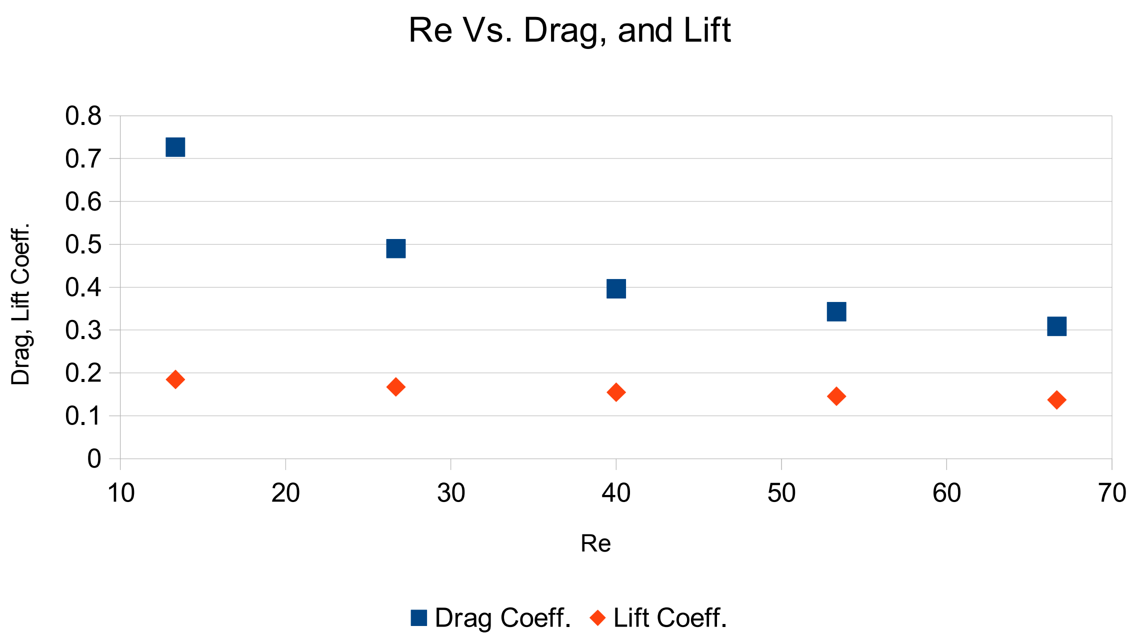
Plot 2: Velocity and pressure distribution for u = 5 cm/s



Comparing plots 1, and 2, we can see that with increase of velocity the envelope of u = 0 around the object becomes smaller, which directly effects the distribution of pressure around the object. In both cases we can see that the frictional forces will be more prominent on the attack end as the velocity changes drastically there. While the back side is impacted by the pressure, as the pressure at back end increases with increase in velocity, although the effect of pressure is debatable, as the values are quite low. Still if we observe the Drag and Lift forces in Table & plot 1, we see an explicit downward trend in the drag, while lift stays almost the same, The possible explanation for this is the separation angle of the flow from the body, the higher the separation angle, longer the flow exerts frictional forces on the objects, hence resulting in higher drag. But with increase in velocity the separation angle decreases, thus larger area of the body is contained in the trail created by the attack end.

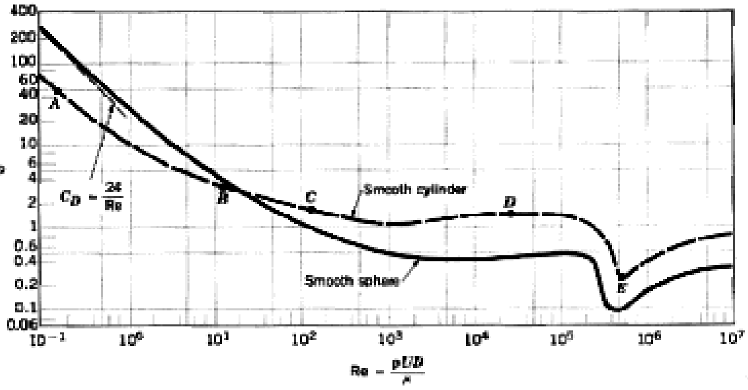
Table & plot 1: dependence of lift and drag on velocity of flow

u [cm/s]	1	2	3	4	5
Reynolds No	13.33	26.67	40	53.33	66.67
Drag Coeff	0.73	0.49	0.4	0.34	0.31
Lift Coeff	0.18	0.17	0.15	0.15	0.14



Discussion

The following plot was obtained form http://www.princeton.edu/~asmits/Bicycle_web/blunt.html . Which plots the experimental values of the drag forces for a smooth cylinder. By comparing the behavior of this plot with Re Vs. Drag obtained from the simulation we see similar results. Also the comparison of range of drag coefficient for given range in Reynolds number is as:



	ComSol	Exp.
Re	13-67	10-100
Drag	0.73-0.31	3.8-2

Given these similarities and disregarding the differences (which arise due to factors such as different geometries, different approximation for Reynolds Number, experimental errors, simulation error), the behavior is visually agreeable.

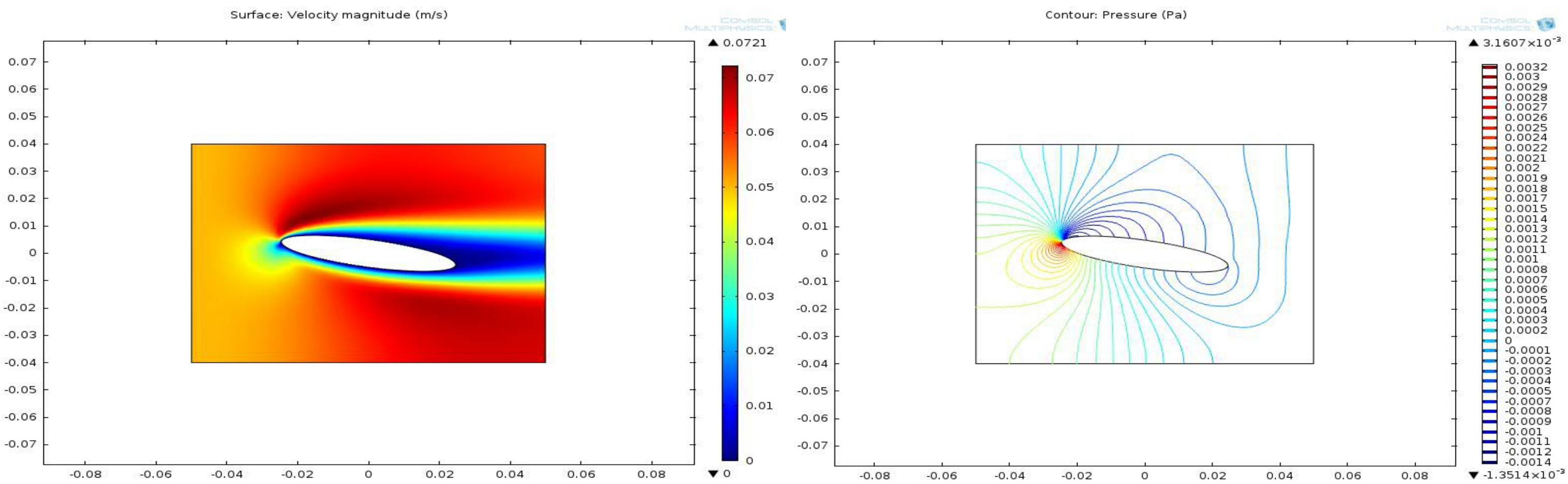
2. Dependence on Aspect Ratio

Constant Parameters

$u = 0.05\, m/s$

$\alpha = 10^o$

Plot 3: Velocity and pressure distribution for Aspect Ratio = 0.2 cm/cm



Plot 4: Velocity and pressure distribution for Aspect Ratio = 1 cm/cm

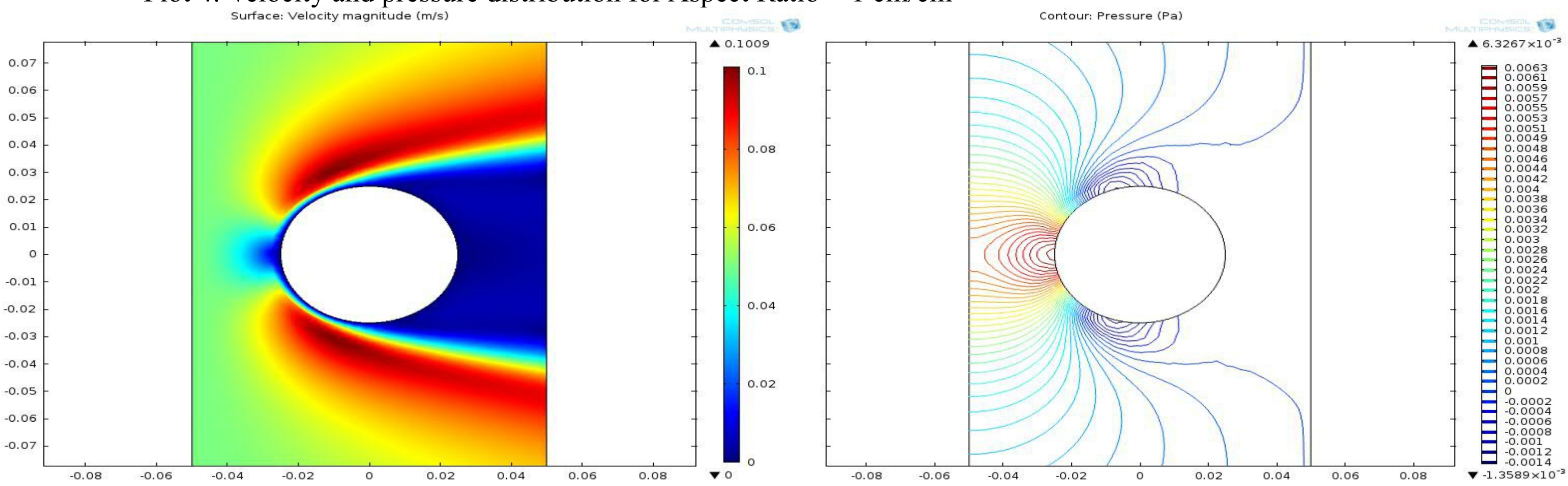
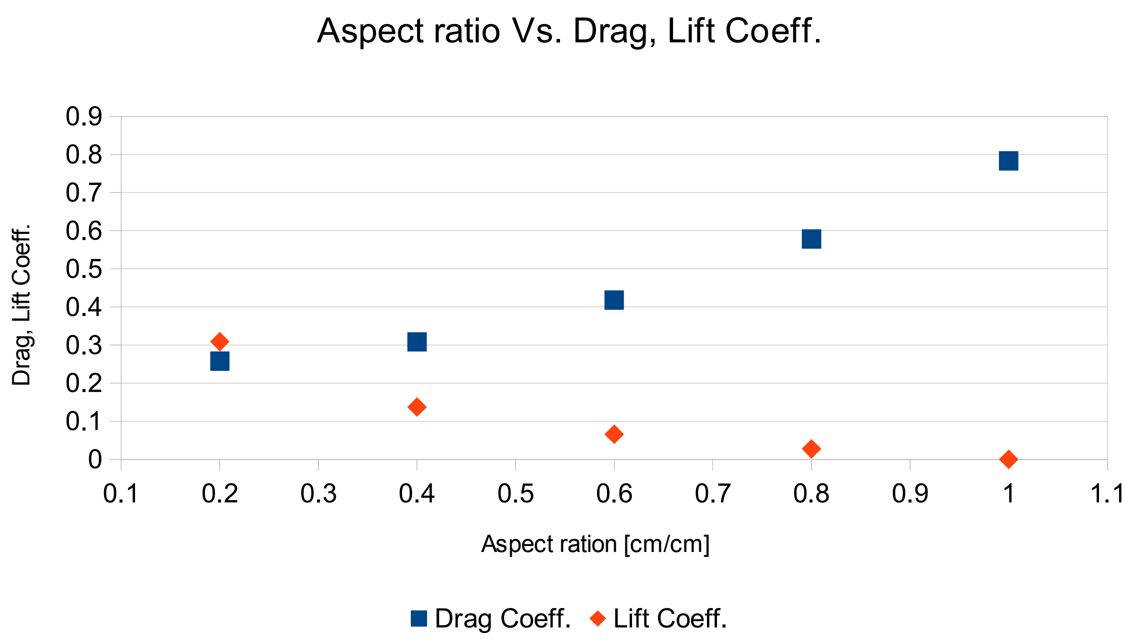


Table & plot 2: dependence of lift and drag on Aspect Ratio of Body

aspect ratio	0.2	0.4	0.6	0.8	1
Drag Coeff.	0.26	0.31	0.42	0.58	0.78
Lift Coeff.	0.31	0.14	0.07	0.03	0



Discussion

As the aspect ratio for the object increases the area seen by the flow changes, thus increasing the frictional drag forces on the object, also the pressure distribution change to increase on attack end, thus the net drag on the object increases. On the other hand as the aspect ratio approaches a cylinder, the pressure distribution above and below the geometry start becoming identical, thus canceling the effect from each other, hence decreasing the lift.

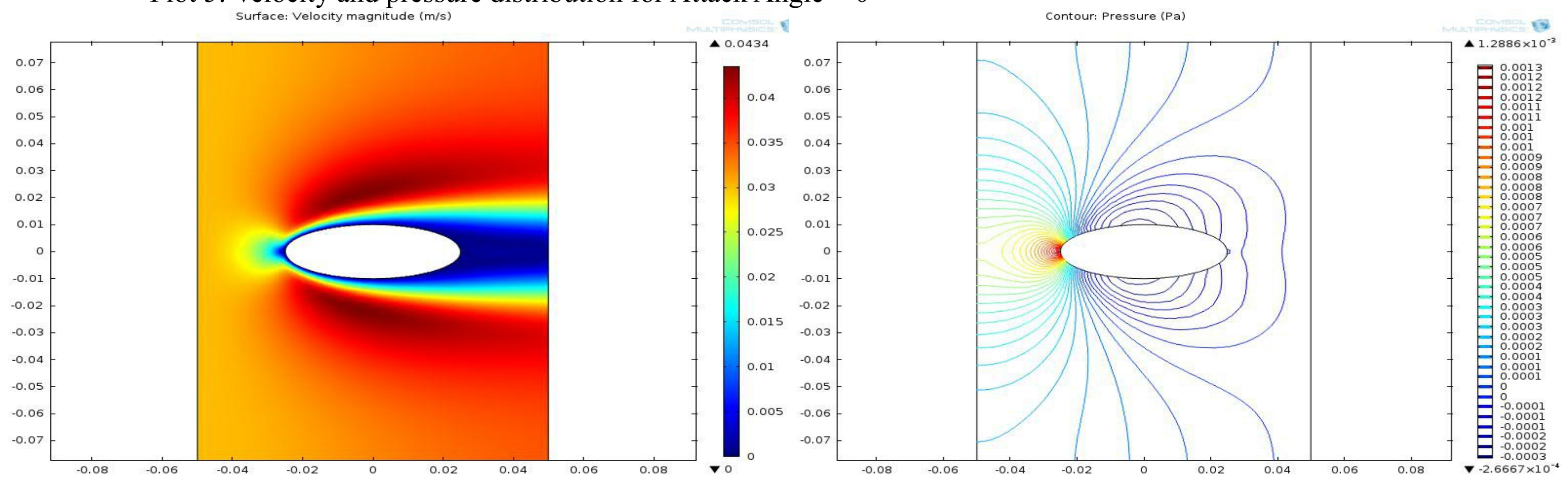
3. Dependence on Attack Angle

Constant parameters

$\epsilon = 0.4$

$u = 0.03\, m/s$

Plot 5: Velocity and pressure distribution for Attack Angle = 0º



Plot 6: Velocity and pressure distribution for Attack Angle = 25°

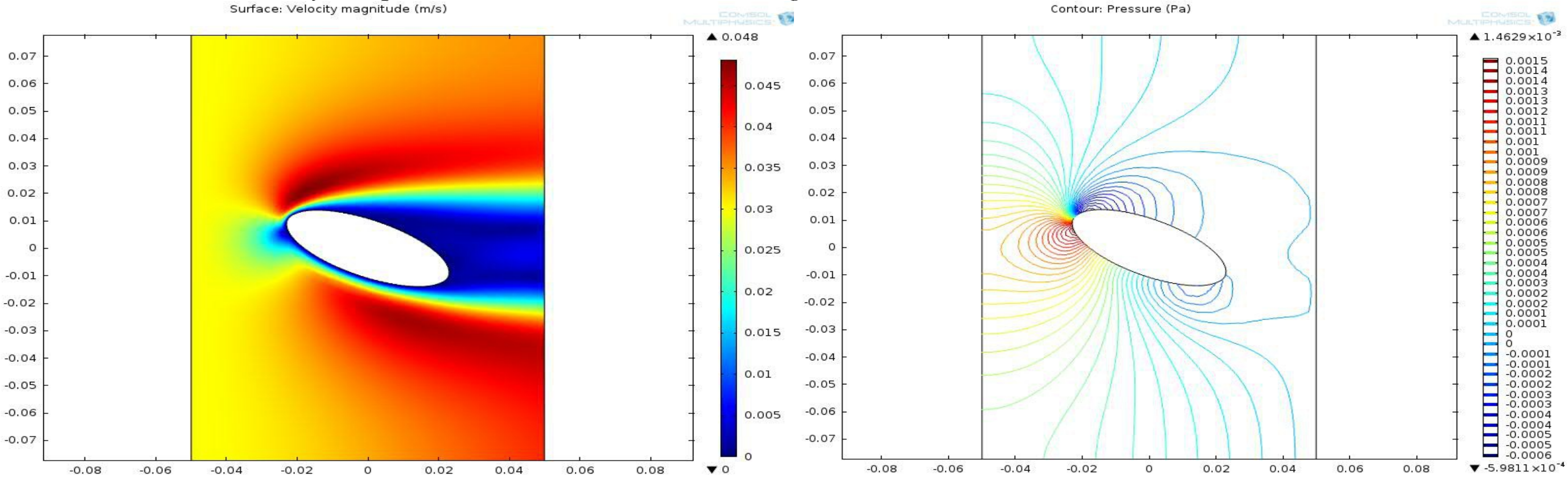
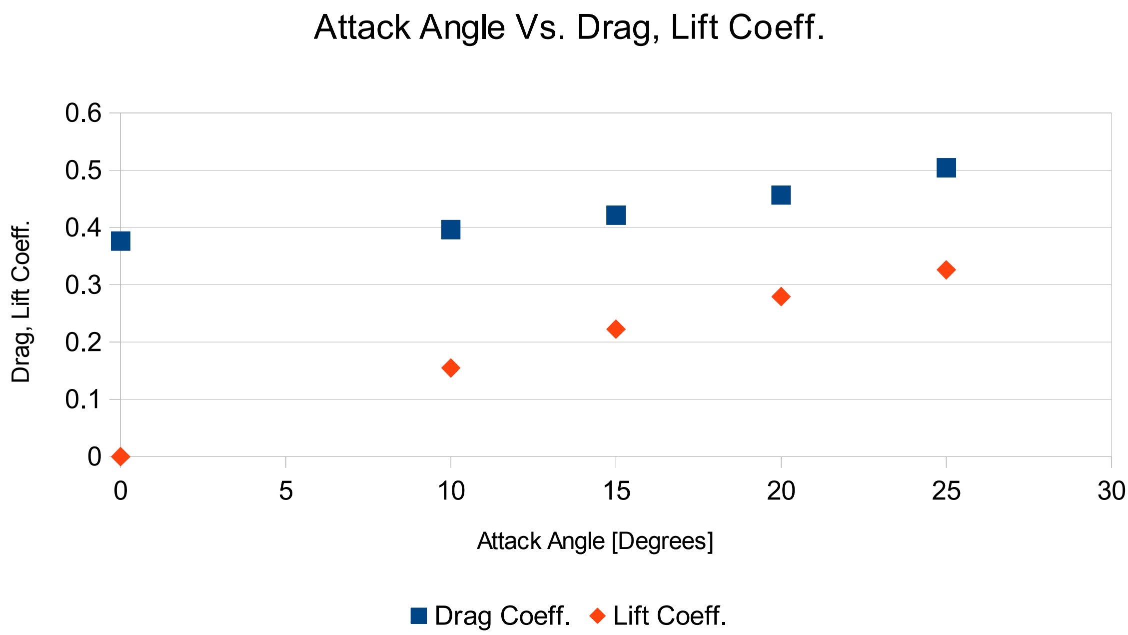


Table & plot 3: dependence of lift and drag on Attack Angle of Body

attack angle	0	10	15	20	25
Drag Coeff.	0.38	0.4	0.42	0.46	0.5
Lift Coeff.	0	0.15	0.22	0.28	0.33



Discussion

As the attack angle changes more area comes in direct contact of the flow, rather than lying in the wake, thus increasing the drag. On the other hand the reason in increase of the lift can directly be attributed to pressure distribution changes, as the pressure at the bottom of the geometry is becoming higher than the top, thus providing an upward force.

Conclusion

In this report I conducted the study of the effect of various parameters on the lift and drag for a cylindrical cross section. By comparing the results of an elliptical cylinder, to a circular cylinder, we can say that the behaviour of data is visually agreeable. The data compared is as such:

	ComSol	Exp.
Re	13-67	10-100
Drag	0.73-0.31	3.8-2

Effect of Flow Velocity
The drag decreases with increase in velocity, which might be due to the decrease in effective boundary in contact with the flow. Or in other words increase in the boundary enclosed in trail from attack end. The lift of the body shows very slight change, largely due to the fact that the pressure distributions above and below the geometry do not change much

Effect of Aspect Ratio
Due to increase in wetting area, the net frictional forces on object increase, thus increasing the drag. The lift shows a drastically decreasing trend, which is attributes to the change in pressure distribution. The pressure distribution above and below the geometry start becoming similar to each other with increase in aspect ratio, thus effectively canceling any lift.

Effect of Attack Angle
The drag forces show an increase due to the increase in effective area in contact with the flow, while the lift forces change due to drastic change in the pressure distribution. With increasing attack angle, the pressure distribution below the geometry becomes higher than the one above, thus creating a net upward force.