Flow past over an Elliptical Cylinder

Introduction to problem:

This study of the flow around elliptical bodies examines the nominally two-dimensional. In particular, it is concerned with the study of elliptical cylinder compared with the study in reference paper for drag and flow over cylinder. Numerical simulation is an efficient and effective approach to simulate the transport of particles in a fluid. The shape of a particle plays an important role in its interaction with the surrounding fluid. An understanding and an accurate prediction of the drag coefficient of drill cuttings can help optimize drilling operations. In this study, a numerical solution with ANSYS FLUENT is proposed to investigate the settling of irregular-shaped particles in Newtonian fluid. The effects of the particle shapes on settling velocity are studied. The results of the study can provide valuable insights into drilling hydraulics optimization for more effective removal of cuttings during drilling operations. This can help rig personnel avoid drilling problems such as stuck pipe and low rate of penetration issues.

Problem Description and its application:

Flow past over elliptical cylinder is the fluid flow over an elliptical body in 2D frame of reference of an elliptical body. The effective removal of drill cuttings from a wellbore is indispensable, and the proper hole cleaning ensures the safety of drilling operations. Cuttings are broken bits of rock removed from a borehole, produced by an action of a drill bit during drilling. An understanding of the transport of these drill cuttings is important. Inefficient hole cleaning can cause drilling problems such as circulation loss and drill bit abrasion. However, the transport of these cuttings from the bottom to surface during drilling can be a challenging task due to so many wellbore complexities. These include variations of wall angles, curvature, edge and appendices difference.

While it is less complicated to model drill cuttings with a spherical model, it is practically inaccurate to assume that these cuttings are all spherical. To improve the accuracy of cuttings transport models, extra parameters are required. One such parameter is the drag coefficient which varies with the Reynolds number. The drag coefficient of a particle is a dimensionless property defined as the fraction of the kinetic energy of the settling velocity used in overcoming the force of drag on the particle. Many experimental works have investigated the settling of spherical particles. The drag coefficient is used to calculate the settling velocity of drill cuttings. This requires a good understanding of the settling behavior of bow-shaped (non-spherical) planner particles, which is the focus of this study. Although there are many empirical correlations for predicting the drag coefficient of non-spherical particles, the models have limitations. One limitation is the number of shapes that a particle can assume. The drag coefficient of non-spherical particles is affected by the particle shapes and settling orientations. Numerical simulation is an efficient and effective approach to simulate the transport of particles in a fluid. The shape of a particle plays an important role in its interaction with the surrounding fluid.

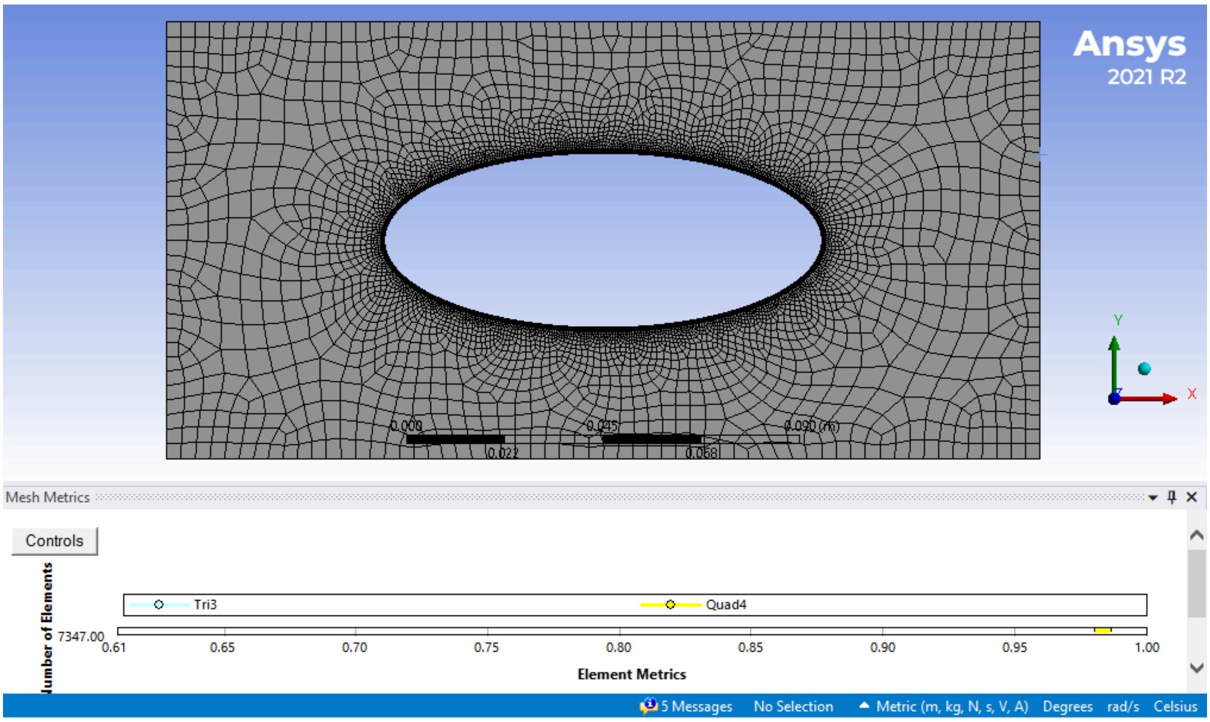
Model Description:

The non-spherical shape considered in this work is an elliptic cylinder.

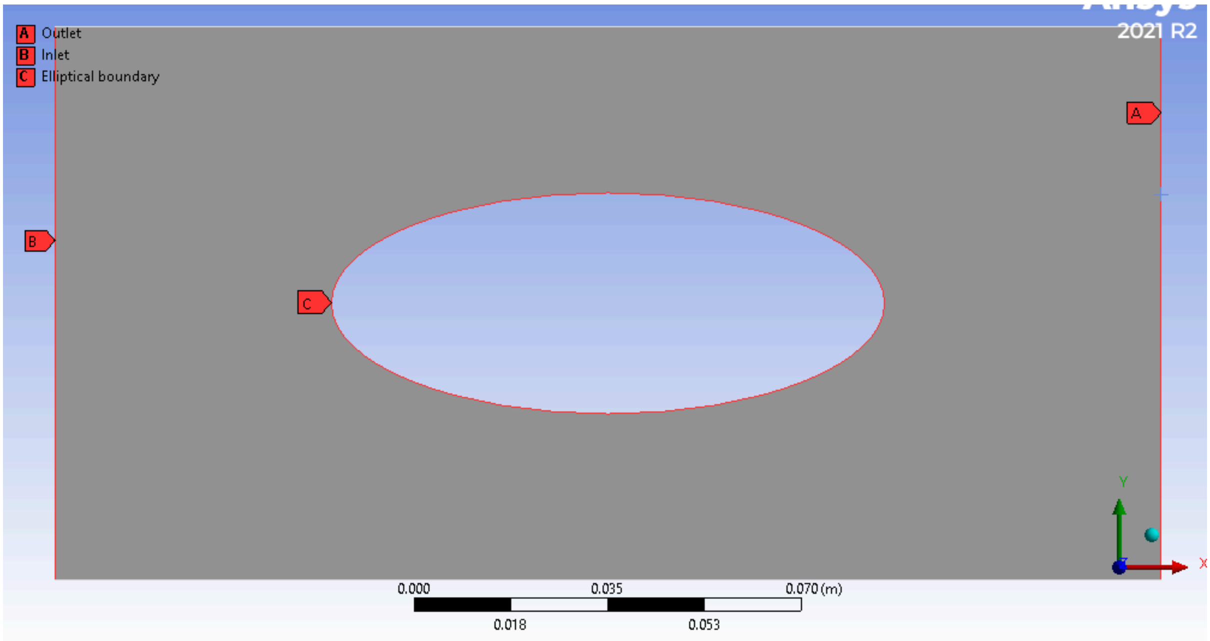
Geometrical Parameter	Magnitude (m)
Major axis, a	0.002
Minor axis, b	0.001
Height, t	0.0005

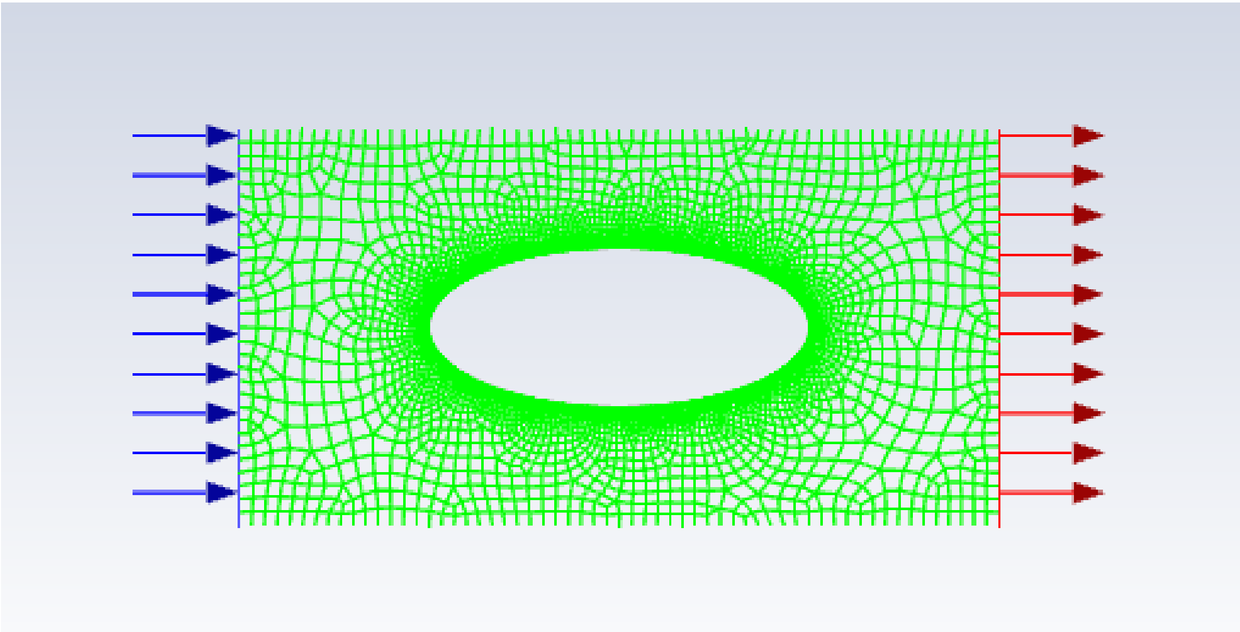
Mesh/Grid:

A standard mesh is created by an element size of 0.001m on linear pattern with mesh generation in form of quadra, also sizing is added to edges for 30 and 60 simultaneously. For smooth mesh generation at the surface of elliptical body an inflammation of 10 numbers is added to make a fine and proper meshing which is shown in image below.



After mesh generation all the boundaries are named by using named selection from the bar as given in image below.





Results and its Comparison with reference:

For the result generation we have taken boundary conditions and the inlet conditions set by the reference paper as given below,

Table 2. Boundary Conditions.

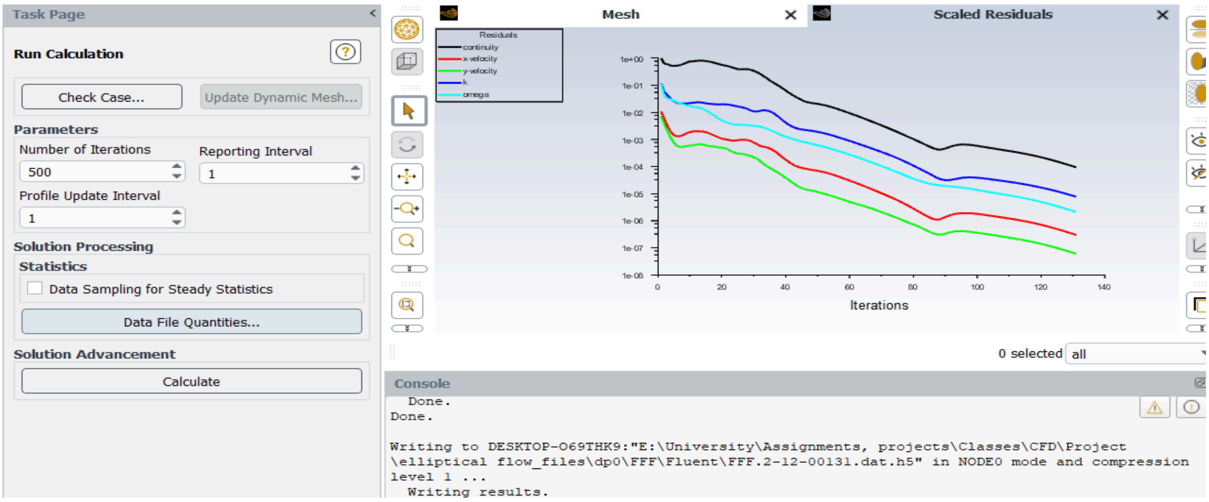
Conditions	
Inlet Boundary	
Viscosity ratio	10%
Fluid inlet velocities	0.1 m/s, 0.5 m/s, 1.0 m/s
Initial gauge pressure	0 Pa
Velocity specification method	Magnitude, Normal to Boundary
Outlet boundary	
Viscosity ratio	10%
Pressure	0 Pa
Backflow pressure specification	Total pressure
Wall	
Roughness model	Standard
Road	No-slip

Also, for the scheme used and method adopted are given as,

Parameters	
Density of water	997 kg/m^3
Viscosity of water	0.001 Pa·s
Turbulence model	K-Omega (2-equation)
Solver type	Pressure based
Solution method	Pressure velocity coupling, 2nd order upwind iterations
Solution initialization type	Hybrid

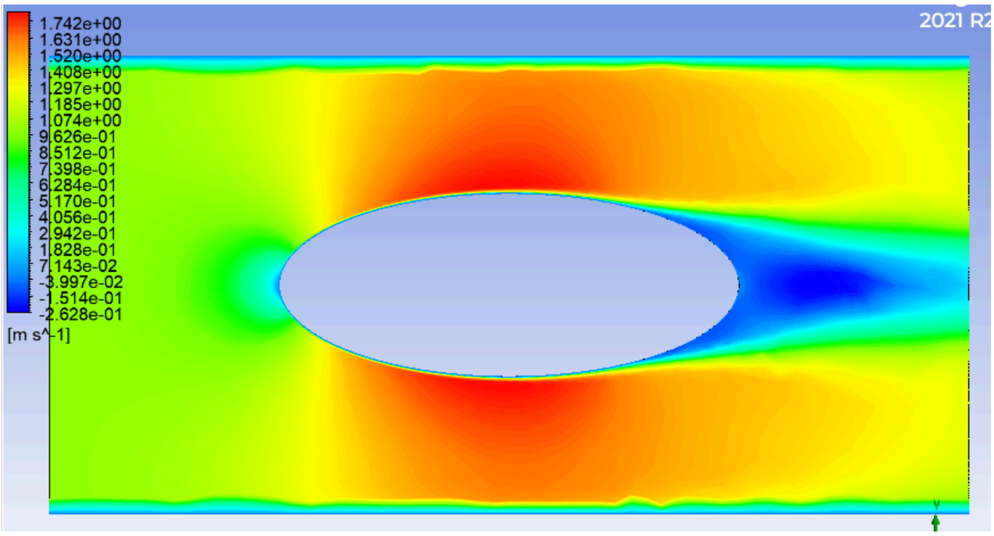
‘

By applying all these conditions and parameters we have started our calculations and got converged solution at iterations of 162 with residuals of 0.0001m.

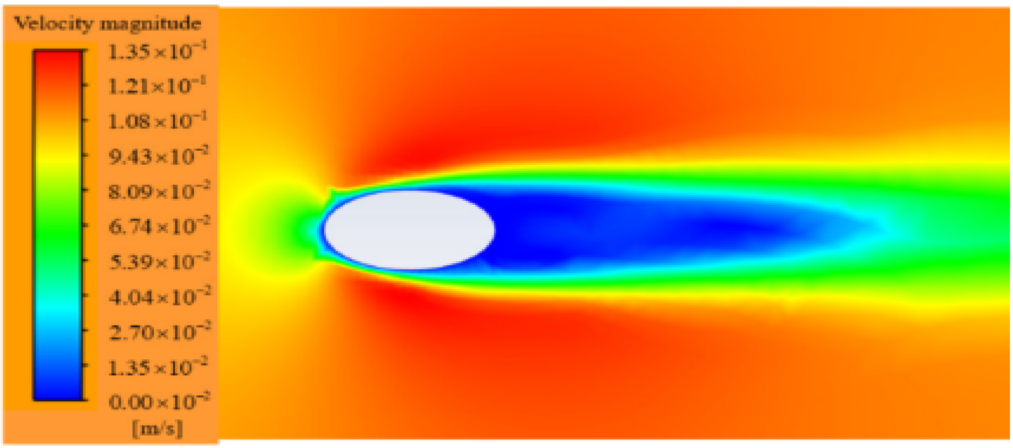


As compared to the solution in reference paper velocity contour can be given as,

Simulated:



Reference:



Reference Paper:

Drag Coefficients of Irregularly Shaped Particles in Newtonian Fluids by Owolabi Akanni ,Chunkai Fu and Boyun Guo Department of Petroleum Engineering, University of Louisiana at Lafayette, Lafayette, LA 70504, USA [Online Available] https://www.mdpi.com/2071-1050/13/14/7517/htm