

MEC E 539

# Assignment 4: Laminar flow Around a Square Object

by: Alex Diep

Date: March 28, 2023

CCID: abdiep  
Student ID: 1664334

# Contents

<b>List of Figures</b>	<b>ii</b>
<b>List of Tables</b>	<b>ii</b>
<b>1 Problem Statement</b>	<b>1</b>
1.1 Objective . . . . .	1
1.2 Mathematical Model . . . . .	1
1.2.1 Flow Description . . . . .	1
1.2.2 Boundary Conditions . . . . .	1
1.3 Computational Model Parameters . . . . .	2
1.4 Geometry and Mesh . . . . .	2
1.4.1 Boundary Conditions . . . . .	3
1.4.2 Solver Details . . . . .	3
<b>2 Results analyses</b>	<b>4</b>
2.1 Convergence of Simulations . . . . .	4
2.2 Analyses of the Velocity Fields . . . . .	7
2.3 Drag Coefficient and Vortex Length . . . . .	8
2.3.1 Vortex Length and Drag Coefficient . . . . .	8
2.3.2 Recirculation Length on All Grids . . . . .	9
2.3.3 Mesh Independence . . . . .	9
2.4 Estimation of the Order of the Method . . . . .	10
2.5 Validation of the Results . . . . .	11
<b>3 Conclusions</b>	<b>11</b>
<b>4 References</b>	<b>12</b>

## List of Figures

1	Geometry and Mesh for Mesh 1 . . . . .	2
2	Residuals for Grid 1 . . . . .	4
3	Drag Coefficient for Grid 1 . . . . .	4
4	Residuals for Grid 2 . . . . .	5
5	Drag Coefficient for Grid 2 . . . . .	5
6	Residuals for Grid 3 . . . . .	5
7	Drag Coefficient for Grid 3 . . . . .	5
8	Residuals for Grid 4 . . . . .	5
9	Drag Coefficient for Grid 4 . . . . .	5
10	Residuals for Grid 5 . . . . .	5
11	Drag Coefficient for Grid 5 . . . . .	5
12	Velocity Profile for Grid 1 . . . . .	6
13	Velocity Profile for Grid 2 . . . . .	6
14	Velocity Profile for Grid 3 . . . . .	6
15	Velocity Profile for Grid 4 . . . . .	6
16	Velocity Profile for Grid 5 . . . . .	6
17	Velocity Contour for Grid 1 . . . . .	7
18	Velocity Contour for Grid 5 . . . . .	7
19	Streamlines for Grid 1 . . . . .	7
20	Streamlines for Grid 3 . . . . .	7
21	Streamlines for Grid 5 . . . . .	8
22	Velocity Profile for All Grids . . . . .	9
23	Drag Coefficient vs. Cells . . . . .	9
24	Recirculation Length vs. Cells . . . . .	9
25	Error in Drag Coefficient vs. Element Size . . . . .	11

## List of Tables

1	Boundary Conditions . . . . .	2
2	Mesh Sizes . . . . .	3
3	General Solver Details . . . . .	3
4	Discretization Scheme . . . . .	4
5	Grid Convergence . . . . .	4
6	Vortex Length and Drag Coefficient . . . . .	8
7	Error in Vortex Length and Drag Coefficient with Respect to Grid 5 . . . . .	10

# 1 Problem Statement

## 1.1 Objective

The problem objective is to calculate the drag coefficient  $C_d$  of a square object at a Reynolds number of 20 and estimate the length of a vortex that forms behind the object.

Secondary objectives include learning about mesh refinement and mesh convergence. The problem will be solved numerically in ANSYS Fluent for five systematically refined meshes. The first (coarsest) mesh should contain in the order of 2000-3000 cells or control volumes with a cell size of 5.0 mm. The rest of the meshes are generated by reducing the size of each cell by two in both directions, i.e. increasing the total number of cells by four. The topology of the mesh remains the same as the mesh is refined. During the simulation, it is necessary to monitor the drag force  $F_d$  as a function of iteration.

## 1.2 Mathematical Model

### 1.2.1 Flow Description

For the mathematical description of fluids, the Navier-Stokes equations are an appropriate description of the flow. The continuity equation is given by

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = 0$$

the momentum equation is

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \frac{1}{3}\mu \nabla(\nabla \cdot \vec{v}) + \mu \nabla^2 \vec{v} + \rho \vec{b}$$

There are some simplifications that can be made to the Navier-Stokes equations. Noting that the Reynolds number is low ( $Re \ll 100$ ), the unsteady and convective terms can be neglected. The flow is also incompressible, so the density is constant. The continuity equation simplifies to

$$\boxed{\nabla \cdot \vec{v} = 0}$$

The momentum equation simplifies to

$$\boxed{\mu \nabla^2 \vec{v} - \nabla p + \rho \vec{g} = 0}$$

### 1.2.2 Boundary Conditions

The boundary conditions for the problem are listed in Table 1.

Table 1: Boundary Conditions

Boundary	Condition
Inlet	$u _{\text{inlet}} = U_{\infty} = 0.04 \text{ m/s}$
Outlet	$\left. \frac{\partial u}{\partial x} \right _{\text{outlet}} = 0$
Top and Bottom Walls	$\mu \left. \frac{\partial u}{\partial y} \right _{\pm 10D} = 0$
Square Wall	$\vec{v} _{\text{square}} = 0$

### 1.3 Computational Model Parameters

### 1.4 Geometry and Mesh

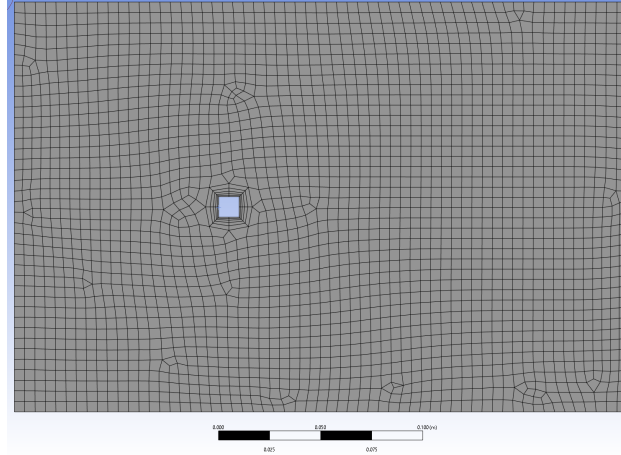


Figure 1: Geometry and Mesh for Mesh 1

The geometry of the simulation domain is a square object with side length  $D = 0.01 \text{ m}$ . The domain is  $0.2 \text{ m}$  long and  $0.1 \text{ m}$  tall.

Face sizing was used with 5 inflation layers around the square with a growth rate of 1.2 and a default transition ratio of 0.272. The number of elements in each mesh is shown in Table 2. The mesh for the first mesh is shown in Figure 1.

Table 2: Mesh Sizes

Mesh	Element Size (mm)	Number of Elements
1	5.0	2390
2	2.5	9390
3	1.25	37870
4	0.625	152044
5	0.3125	613378

### 1.4.1 Boundary Conditions

First  $U_\infty$  needs to be determined. The Reynolds number is given from the problem statement as  $Re = 20$ . Then,

$$Re = \frac{U_\infty \rho D}{\mu}$$

$$\Rightarrow U_\infty = \frac{Re \mu}{\rho D}$$

The properties of air are given as  $\rho = 1.0 \text{ kg/m}^3$  and  $\mu = 2.0 \times 10^{-5} \text{ kg/m}\cdot\text{s}$ . The side length of the square object is  $D = 0.01 \text{ m}$ . Thus,

$$U_\infty = \frac{20 \times 2.0 \times 10^{-5}}{1.0 \times 0.01} = 0.04 \text{ m/s}$$

The inlet velocity was specified as  $U_\infty = 0.04 \text{ m/s}$ . The outlet boundary condition was specified as zero pressure gradient. The top and bottom walls were specified as no-shear walls. The square wall was specified as a no-slip wall.

### 1.4.2 Solver Details

The solver details are shown in Table 3.

Table 3: General Solver Details

Type	Pressure Based
Velocity Formation	Absolute
Time	Steady
2D Space	Planar

The convergence criteria for continuity, x-velocity, and y-velocity were set to  $10^{-5}$ . The spatial discretization scheme is shown in Table 4. A coupled solver was used. The maximum number of iterations was set to 1000.

Table 4: Discretization Scheme

Variable	Scheme
Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind

## 2 Results analyses

### 2.1 Convergence of Simulations

The residuals had a convergence criteria of  $10^{-5}$ . For all grids, the residuals converged before the maximum number of iterations (1000) was reached. The drag coefficient reached a steady value in all grids. Figures 2 to 16 show the residuals, drag coefficient, and velocity profile for each grid.

Table 5: Grid Convergence

Grid	Iterations	Drag Force
1	57	$2.51 \times 10^{-5}$
2	53	$2.18 \times 10^{-5}$
3	51	$2.08 \times 10^{-5}$
4	51	$2.05 \times 10^{-5}$
5	52	$2.03 \times 10^{-5}$

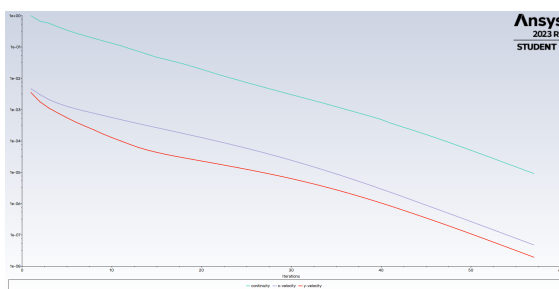


Figure 2: Residuals for Grid 1

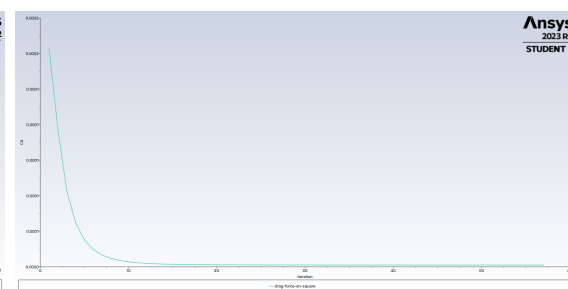


Figure 3: Drag Coefficient for Grid 1

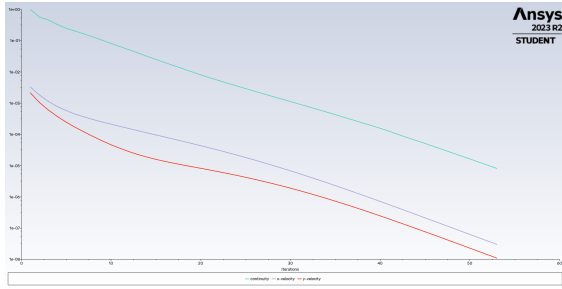


Figure 4: Residuals for Grid 2

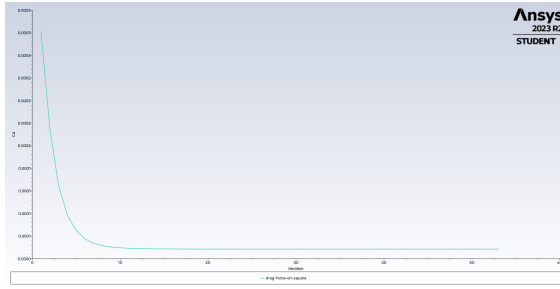


Figure 5: Drag Coefficient for Grid 2

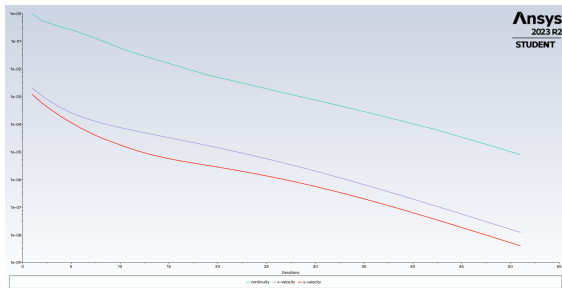


Figure 6: Residuals for Grid 3

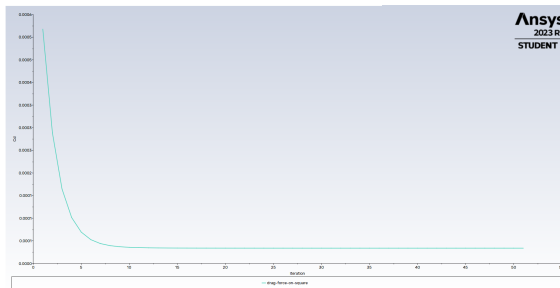


Figure 7: Drag Coefficient for Grid 3

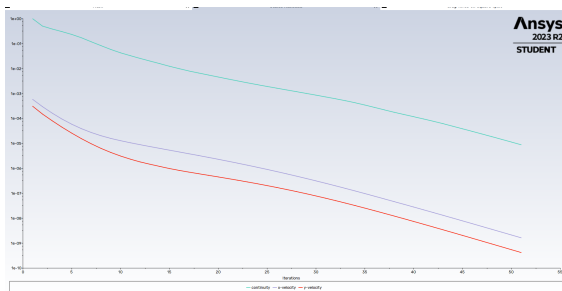


Figure 8: Residuals for Grid 4

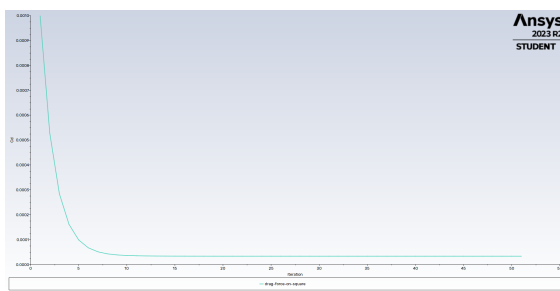


Figure 9: Drag Coefficient for Grid 4

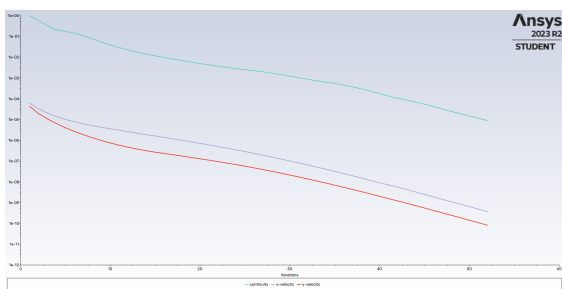


Figure 10: Residuals for Grid 5

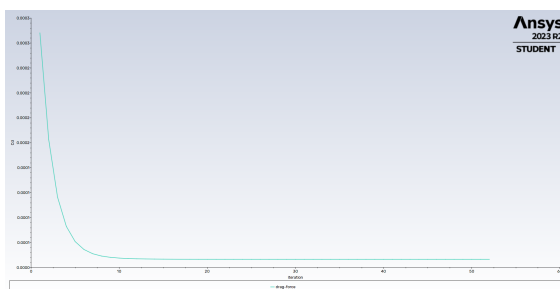


Figure 11: Drag Coefficient for Grid 5



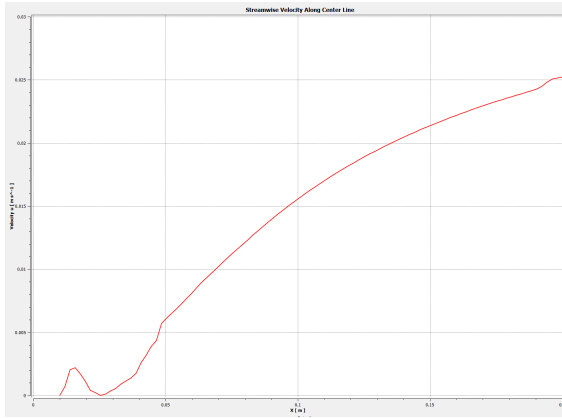


Figure 12: Velocity Profile for Grid 1

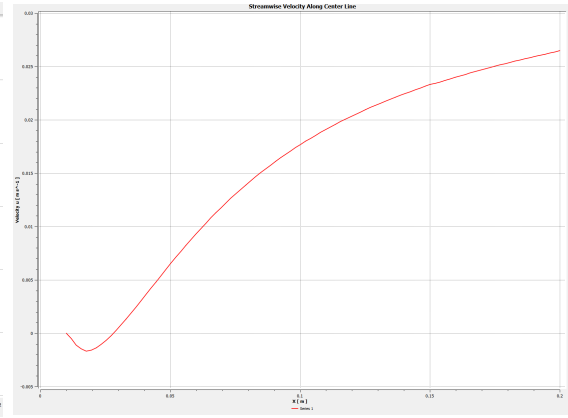


Figure 13: Velocity Profile for Grid 2

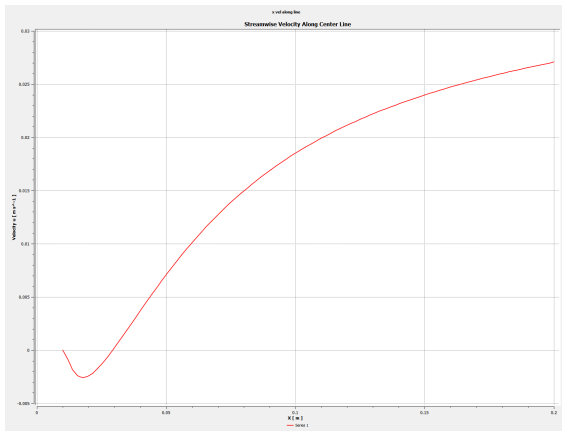


Figure 14: Velocity Profile for Grid 3

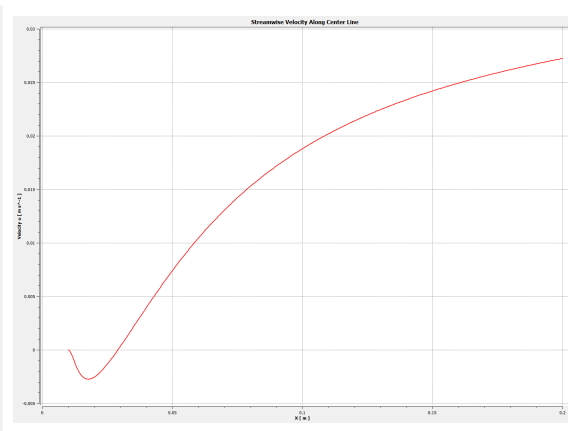


Figure 15: Velocity Profile for Grid 4

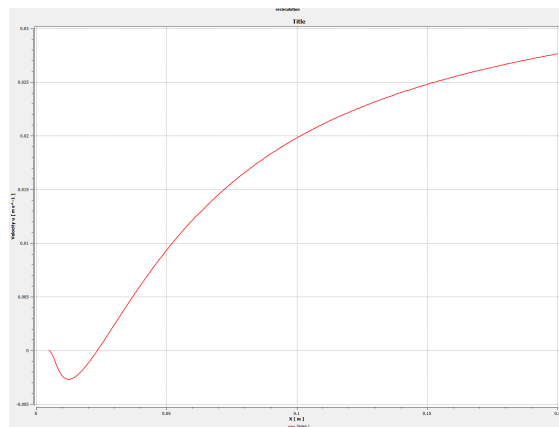


Figure 16: Velocity Profile for Grid 5

## 2.2 Analyses of the Velocity Fields

The contours are shown in Figures 17 and 18. The flow is symmetric about the centerline. The flow behind the square object is observable in grid 1, but much more refined in grid 5. In grid 5, two flanges of high velocity flow can be seen to the side of the square object. In comparison, the whole region to the side of the square object is a high velocity region in grid 1, which makes less physical sense. Grid 5 should be used since the results are more physical.

The streamlines are shown in Figures 19, 20, and 21. Again, the flow is symmetric about the centerline. A small vortex region can be seen from behind the square object in grids 3 and 5. The flow aspects are difficult to discern in grid 1. Considering that meshing grid 5 took roughly 20 minutes to mesh on my desktop computer, grid 3 is a good compromise between accuracy and computational cost.

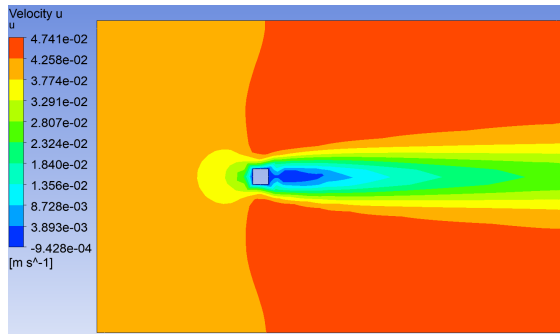


Figure 17: Velocity Contour for Grid 1

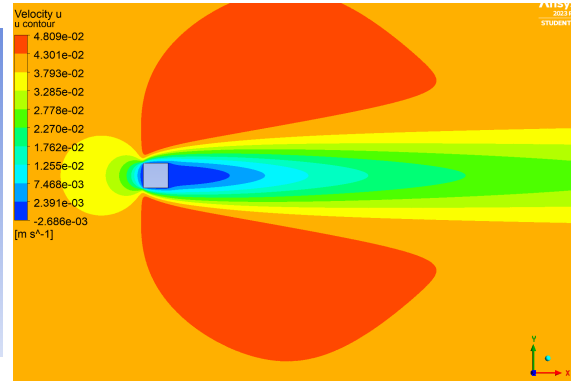


Figure 18: Velocity Contour for Grid 5

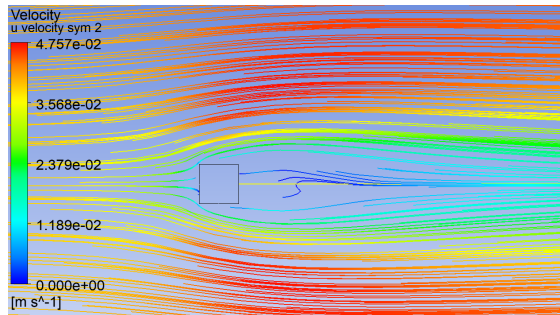


Figure 19: Streamlines for Grid 1

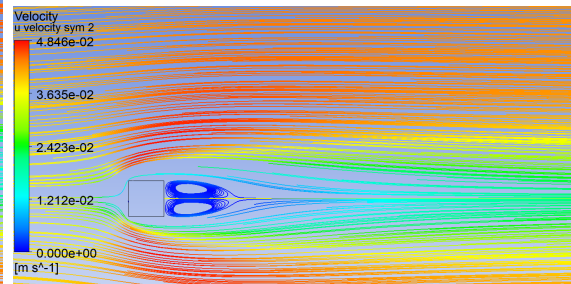


Figure 20: Streamlines for Grid 3

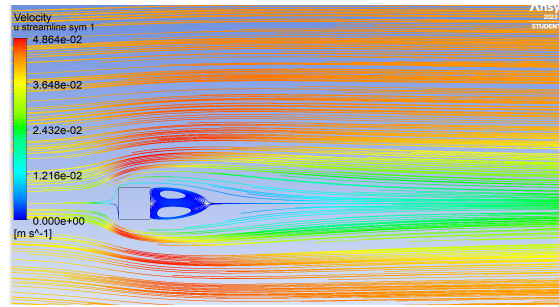


Figure 21: Streamlines for Grid 5

## 2.3 Drag Coefficient and Vortex Length

### 2.3.1 Vortex Length and Drag Coefficient

The vortex length and drag coefficient are summarized in Table 6.

Table 6: Vortex Length and Drag Coefficient

Grid	Recirculation Length (m)	Drag Coefficient
1	0.0115	3.1413
2	0.0181	2.7280
3	0.0186	2.5984
4	0.0183	2.5630
5	0.01801	2.5363

Sample calculations for grid 1 will be shown. The recirculation length is the distance between the two points  $(x_1, x_2)$  where the velocity is zero. Then

$$\begin{aligned}
 L_r &= x_2 - x_1 \\
 &= 0.0234 - 0.0119 \\
 &= 0.0115 \text{ m}
 \end{aligned}$$

The drag coefficient is calculated as

$$\begin{aligned}
 C_d &= \frac{2F_d}{\rho U_\infty^2 D} \\
 &= \frac{2 \times 2.51 \times 10^{-5}}{1.0 \times 0.04^2 \times 0.01} \\
 &= 3.1413
 \end{aligned}$$

### 2.3.2 Recirculation Length on All Grids

The combined plot of the velocity profile for all grids is shown in Figure 22. Note, there is an issue with the coordinates of the line in grid 5 because the origin was defined slightly differently than grids 1-4 (by accident).

Grid 1 does not really have a recirculation length, while grid 2 shows some recirculation. In Grids 3-5, the concave shape develops and becomes steady around grid 4. Grid 3 seems to capture the details of grids 4 and 5 for  $\sim 16x$  fewer cells.

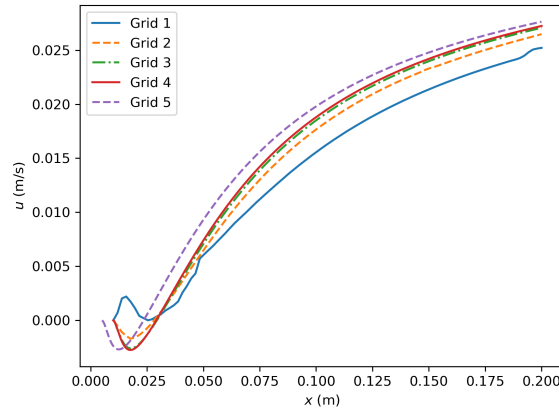


Figure 22: Velocity Profile for All Grids

### 2.3.3 Mesh Independence

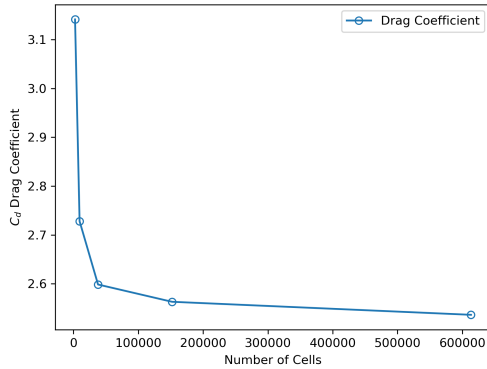


Figure 23: Drag Coefficient vs. Cells

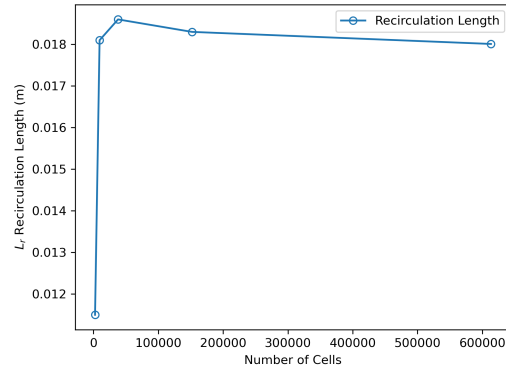


Figure 24: Recirculation Length vs. Cells

Table 7: Error in Vortex Length and Drag Coefficient with Respect to Grid 5

Grid	Vortex Length Error (%)	Drag Coefficient Error (%)
1	36.15	23.85
2	0.50	7.56
3	3.28	2.45
4	1.61	1.05
5	-	-

The drag coefficient and recirculation length are plotted against the total number of cells in Figures 23 and 24. The drag coefficient and recirculation length become independent around mesh 3, with it's results being within 3% of grid 5. The error in the vortex length and drag coefficient with respect to grid 5 is shown in Table 7.

## 2.4 Estimation of the Order of the Method

First calculate the order of the method. The order is calculated as

$$\begin{aligned}
 p &\approx \frac{\log \left( \frac{F_{\Delta x_4} - F_{\Delta x_3}}{F_{\Delta x_5} - F_{\Delta x_4}} \right)}{\log a} \\
 &= \frac{\log \left( \frac{2.05 - 2.08}{2.03 - 2.05} \right)}{\log 2} \\
 &= \boxed{0.407}
 \end{aligned}$$

This is not the expected value of 2. Grid independence has not been reached then.

The Richardson extrapolation is not valid since grid independence has not been reached. Regardless, calculating the Richardson extrapolation for grid 5 gives

$$\begin{aligned}
 \epsilon &= \frac{C_{\Delta x_5} - C_{\Delta x_4}}{a^p - 1} \\
 &= -0.0818
 \end{aligned}$$

then,

$$\begin{aligned}
 C_{\text{exact}} &\approx C_{\Delta x_5} + \epsilon \\
 &= 2.5363 - 0.0818 \\
 &= 2.4546
 \end{aligned}$$

The error in the drag coefficient against the element size is shown in Figure 25.

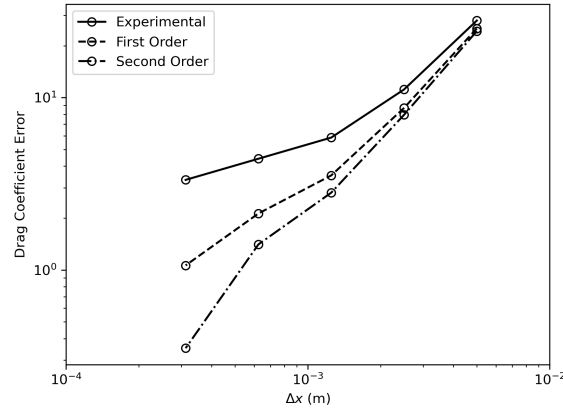


Figure 25: Error in Drag Coefficient vs. Element Size

The error graph is higher than the first and second order error lines. This was expected since  $p \approx 0.407$ , which suggests low order convergence.

It would be expected that  $1 < p_{\text{exp}} < 2$  since a second order scheme was used (not exactly 2 due to meshing impurities). For unknown reasons, this was not achieved. Further work would be needed to determine why the expected order was not achieved.

## 2.5 Validation of the Results

From another study, the drag coefficient was found to be 1.5657 [1] at a Reynolds number of 60. The drag coefficient was found to be 2.5363 at a Reynolds number of 20, which is the same order of magnitude. The relative error is then

$$\begin{aligned} \text{Relative Error} &= \frac{2.5363 - 1.5657}{1.5657} \times 100\% \\ &= \boxed{61.99\%} \end{aligned}$$

## 3 Conclusions

did mesh refinement by guessing an initial element size and halving it 5 times. then drag coefficient was calculated for each mesh as well as contours and streamlines to view the flow characteristics. mesh independence was performed. then convergence was calculated and tested to see the order of convergence which should've been second order since second order methods were used but that was not the case

## 4 References

- [1] A. Ostapenko and G. Bulanchuk, “Calculations of the drag coefficient of circular, square and rectangular cylinders using the lattice Boltzmann method with variable lattice speed of sound,” *Afrika Matematika*, vol. 29, no. 1-2, pp. 137–147, Mar. 2018. [Online]. Available: <http://link.springer.com/10.1007/s13370-017-0531-7>