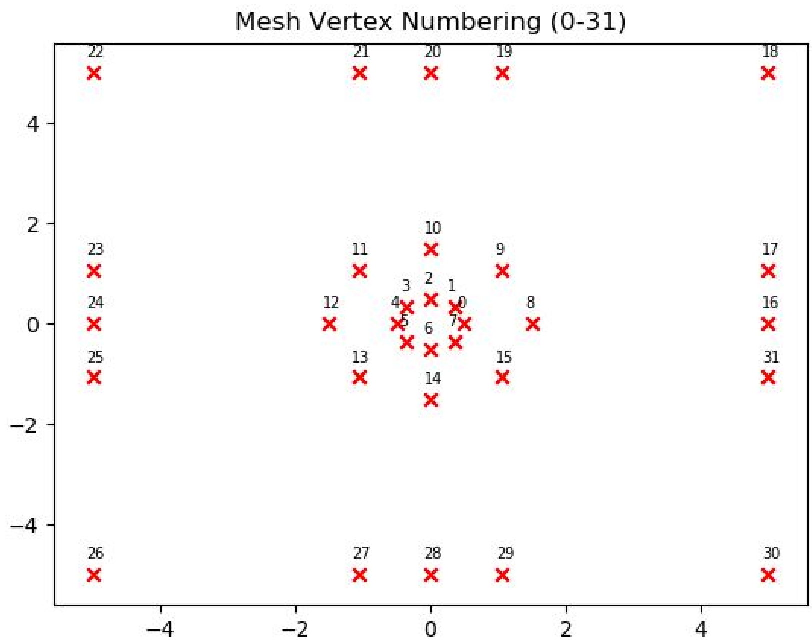
Henry Rossiter
Braeden Conrad
Daniel Fonseka
Gareth Meredith

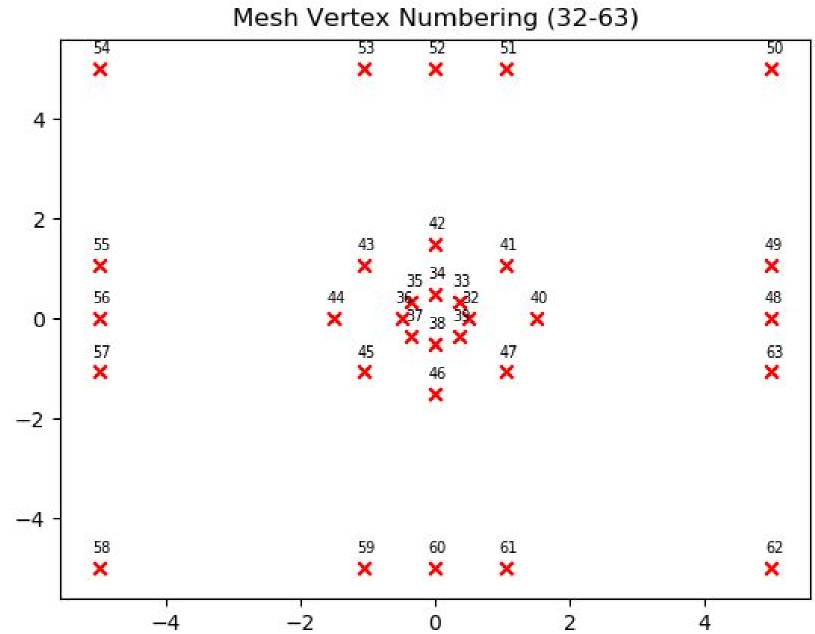
OpenFOAM Assignment 2

Assembly of preliminary meshes

We produced two preliminary meshes, A and B, using the OpenFoam utility blockMesh.
Mesh A was designed to simulate a steady flow with a low Reynolds number (Re=20). Mesh B was designed to simulate an unsteady flow with a higher Reynolds number (Re=110) and vortex shedding.

We developed a numbering scheme to use for our meshes.





Using our vertex numbering scheme, we developed a block numbering scheme. We used 20 blocks. The table below details the block numbering scheme.

Block Number	**Vertices**							
0	0	8	9	1	32	40	41	33
1	8	16	17	9	40	48	49	41
2	9	17	18	19	41	49	50	51
3	9	19	20	10	41	51	52	42
4	10	20	21	11	42	52	53	43
5	11	21	22	23	43	53	54	55
6	11	23	24	12	43	55	56	44
7	12	24	25	13	44	56	57	45
8	13	25	26	27	45	57	58	59
9	13	27	28	14	45	59	60	46
10	14	28	29	15	46	60	61	47
11	15	29	30	31	47	61	62	63
12	15	31	16	8	47	63	48	40
13	1	9	10	2	33	41	42	34

14	2	10	11	3	34	42	43	35
15	3	11	12	4	35	43	44	36
16	4	12	13	5	36	44	45	37
17	5	13	14	6	37	45	46	38
18	6	14	15	7	38	46	47	39
19	7	15	8	0	39	47	40	32

Using our numbering schemes, we developed a python program to generate a blockMeshDict file using our numbering scheme. Using the program, we established the domain sizes and shapes of Mesh A and Mesh B.

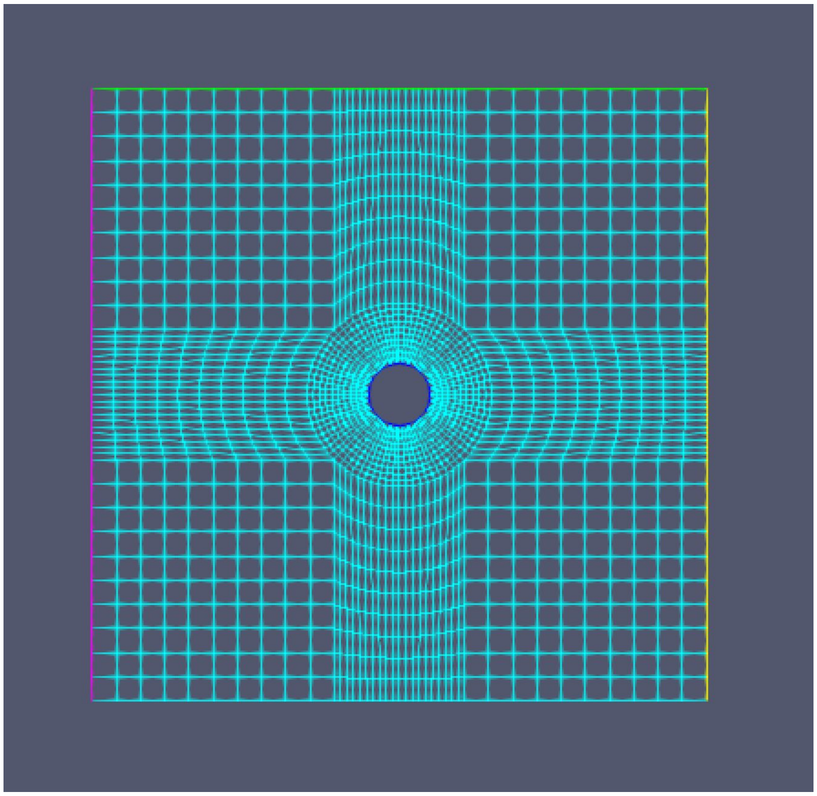
<u>Mesh A</u>

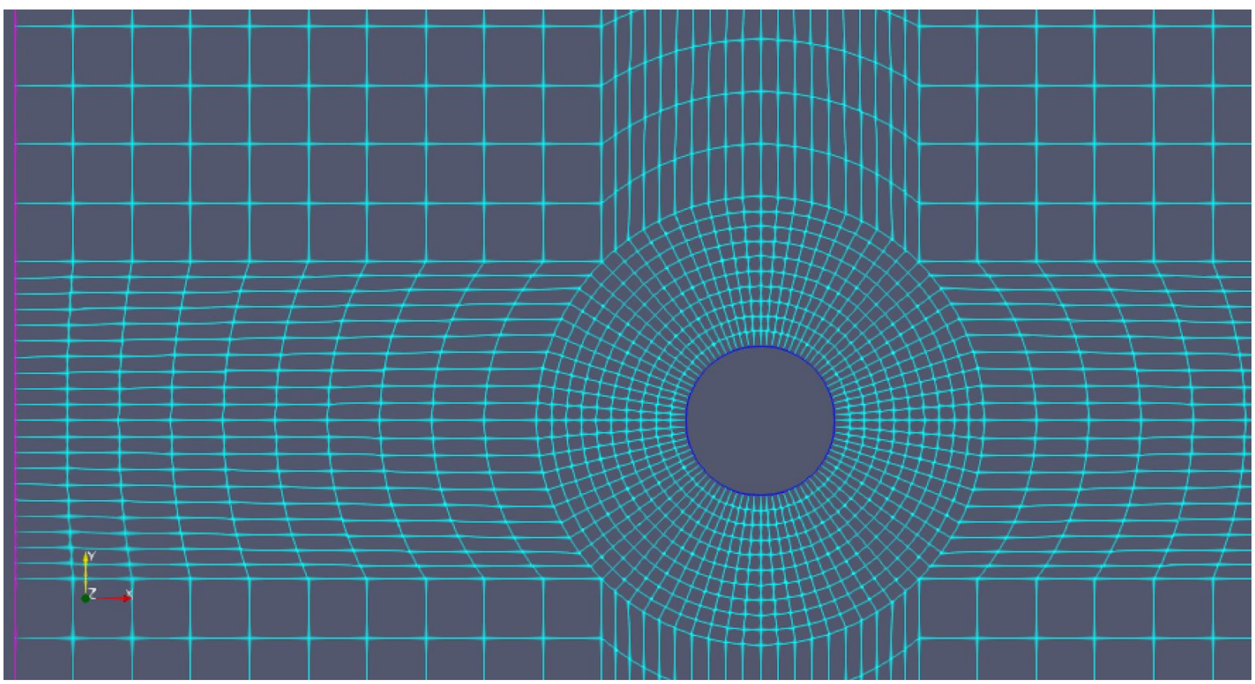
Mesh information:

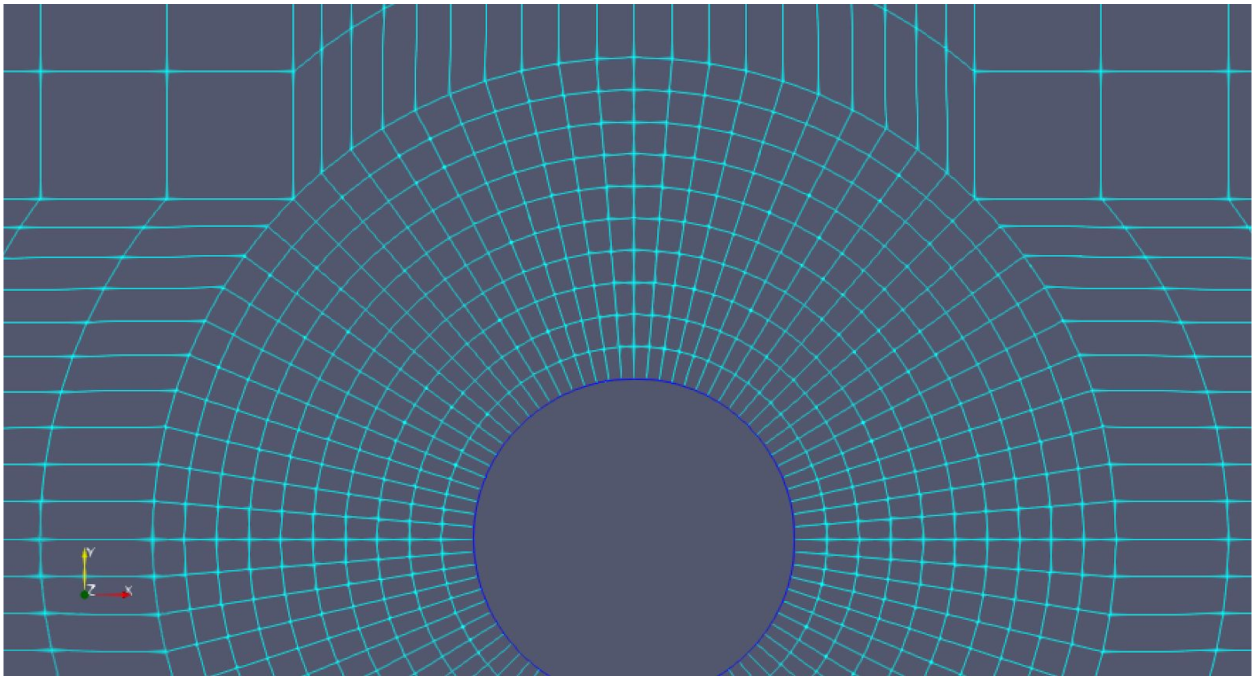
Domain: ($-5 \leq x \leq 5$, $-5 \leq y \leq 5$, $0 \leq z \leq 1$)

# of Points: 4240
# of Cells: 2000
# of Faces: 8120
# of Internal Faces: 3880
# of Faces per Cell: 6

Images of Mesh A:







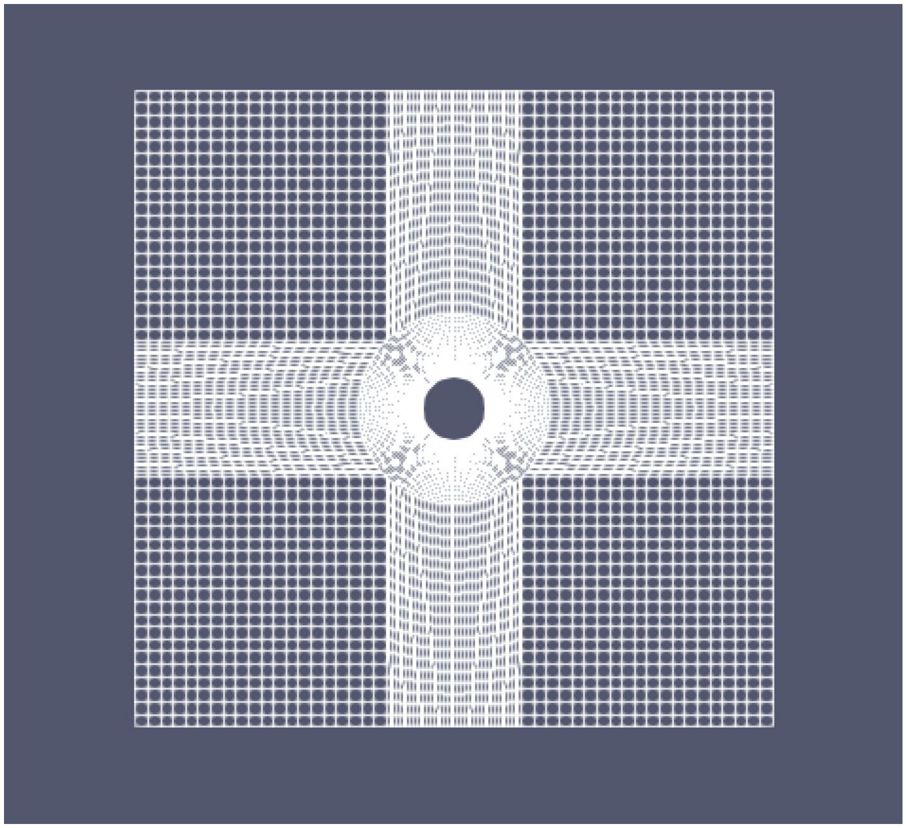
<u>Mesh B</u>

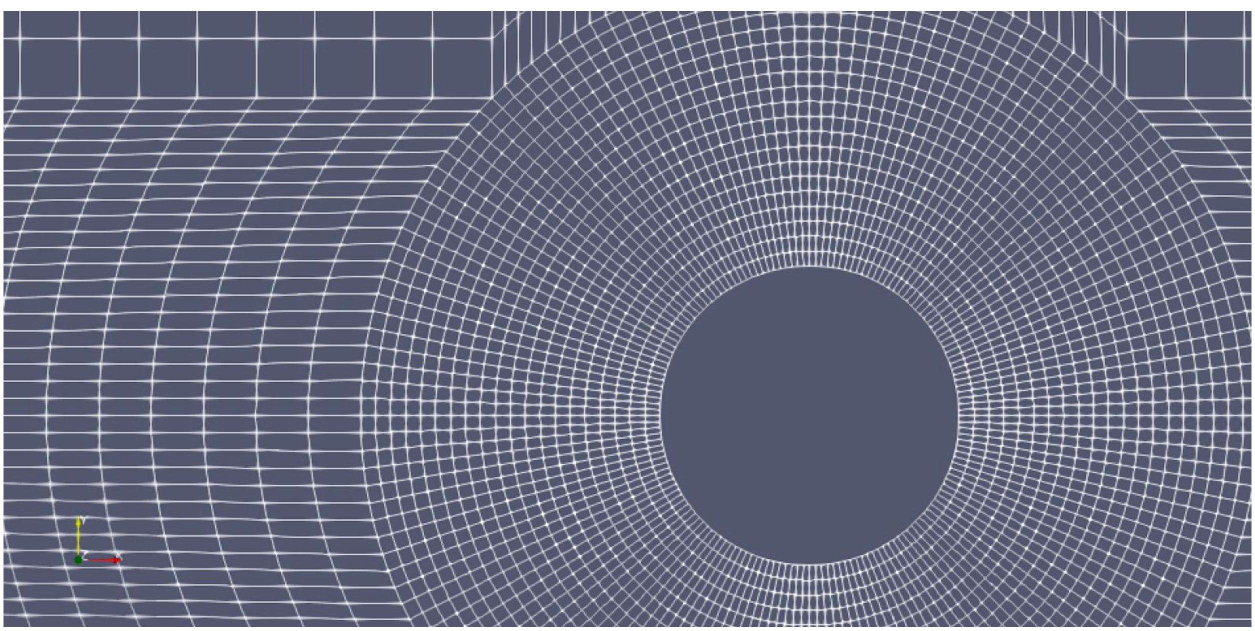
Mesh Information:

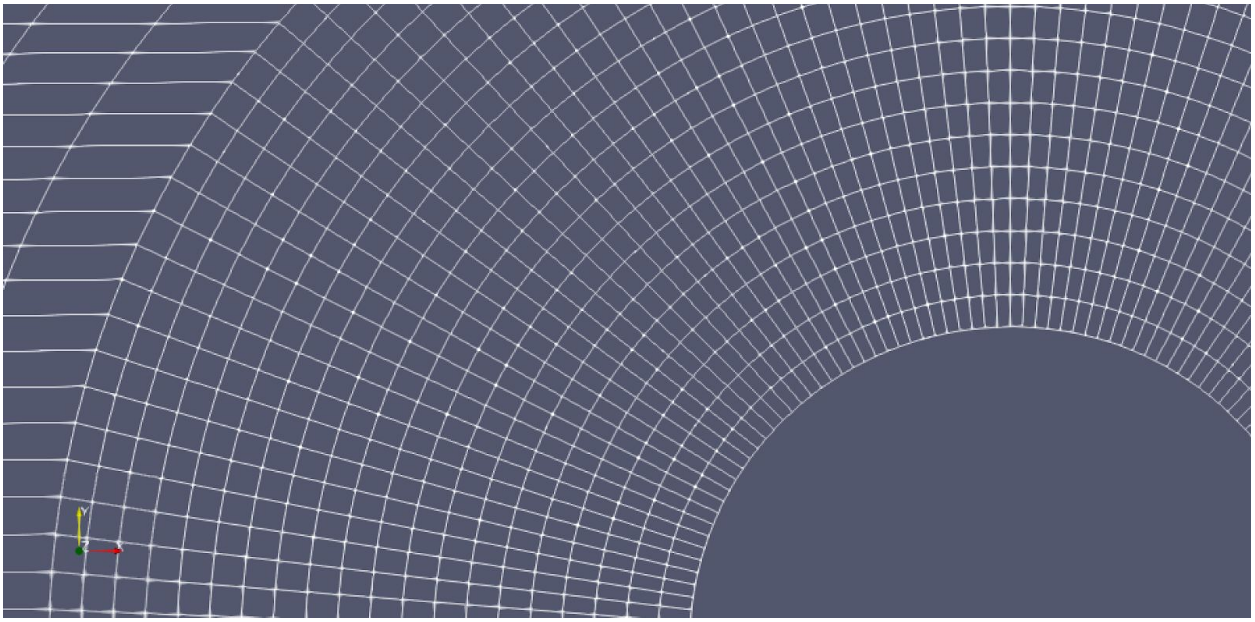
Domain: (-5 ≤ x ≤ 5, -5 ≤ y ≤ 5, 0 ≤ z ≤ 1)

\# of Points: 16480
\# of Cells: 8000
\# of Faces: 32240
\# of Internal Faces: 15760
\# of Faces per Cell: 6

Images of Mesh B:







To summarize, the details of our preliminary meshes are provided below:

	Mesh A	Mesh B
Grid boundaries	x=[-5, 5], y=[-5, 5]	x=[-5, 5], y=[-5, 5]
Cylinder location	[0, 0]	[0, 0]
Block Density	(10 10 1)	(20 20 1)
Total blocks	2e3	4e3

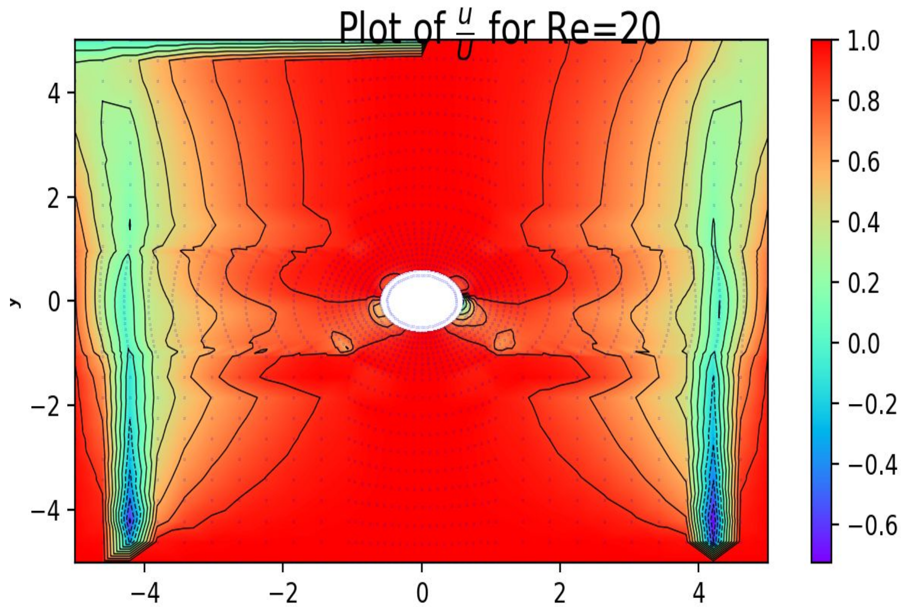
Obtain a solution for Re=20 and Re=110

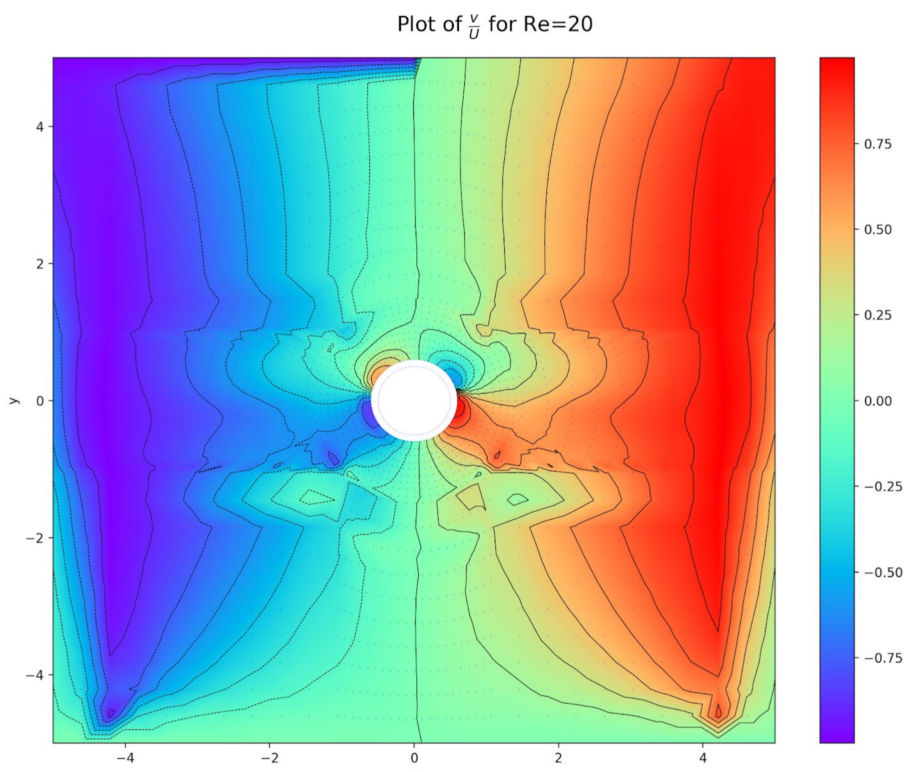
We obtained solutions for two values of Reynold's numbers:

1. Re=20 (v=0.05)
2. Re=110 (v=0.0090909)

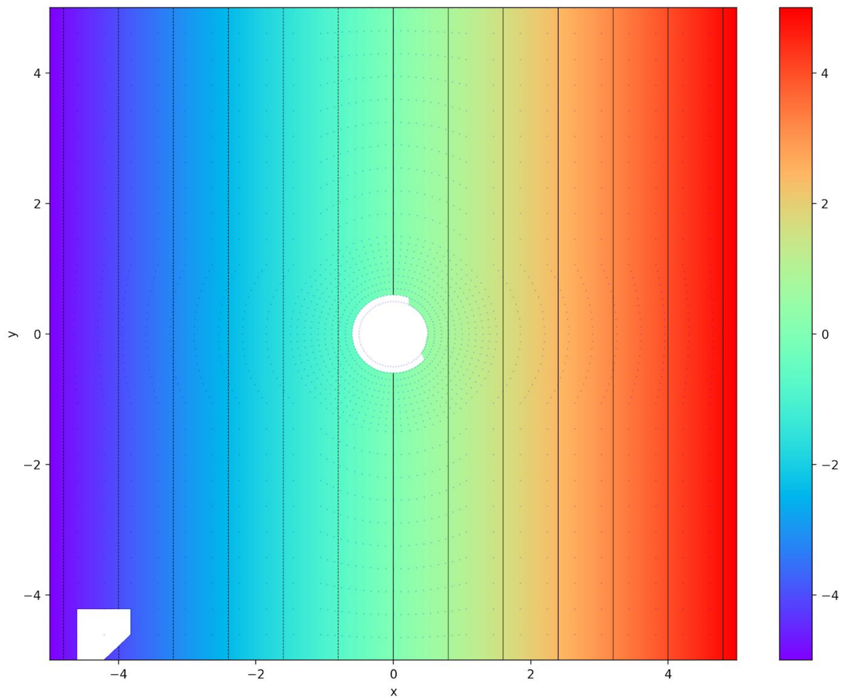
Mesh A was used for the trial with Re=20. Mesh B was used for the trial with Re=110. For both trials, we used a timestep of 0.01s and a total duration of 10s.

For both simulations (Re= 20 and Re=110), we have created contour plots to illustrate the following quantities: u/U, v/U, and $p/(\rho U^2)$ (where ρ assumed to equal 1).

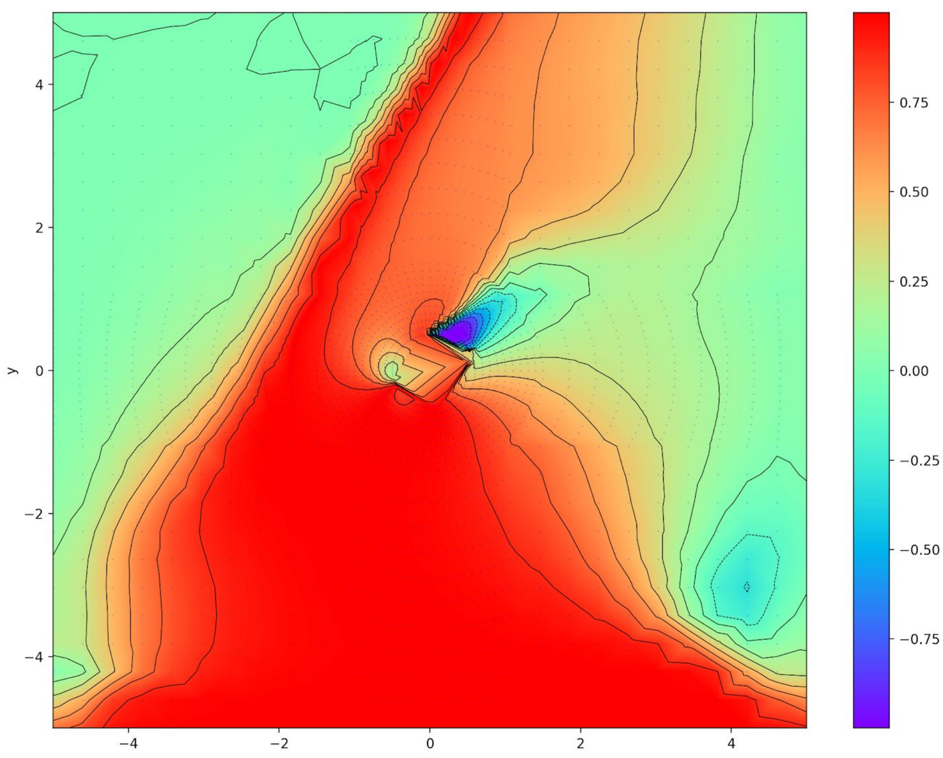




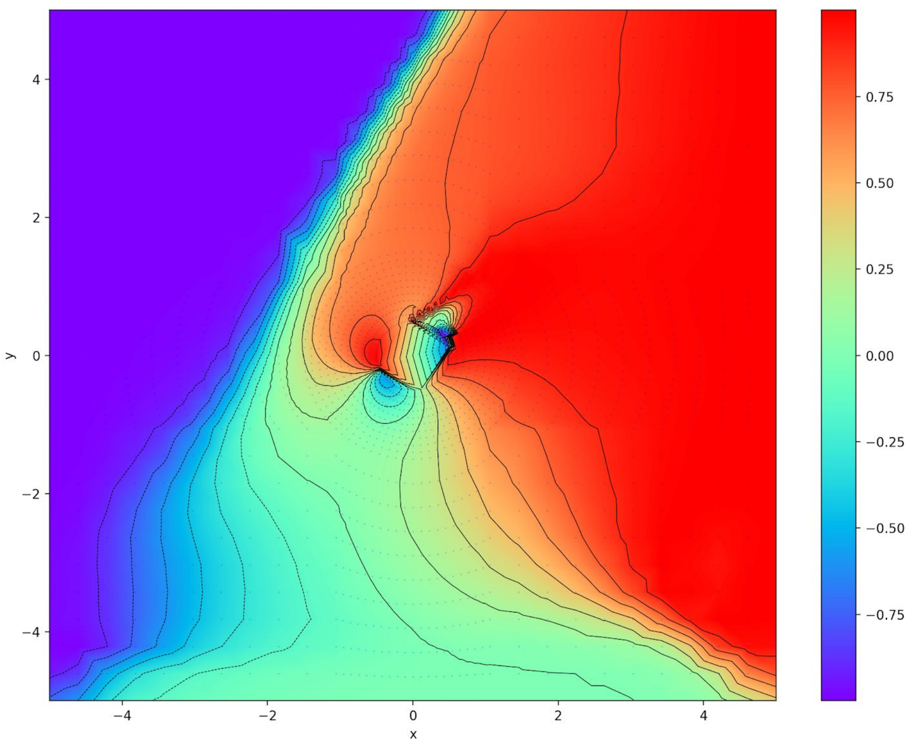
Plot of $\frac{p}{U^2}$ for Re=20



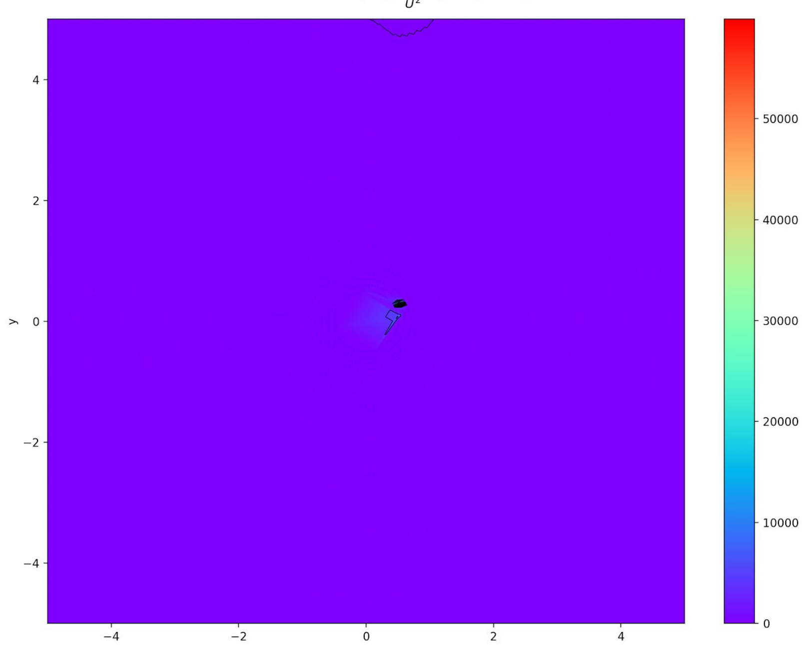
Plot of $\frac{u}{U}$ for Re=110



Plot of $\frac{v}{U}$ for Re=110



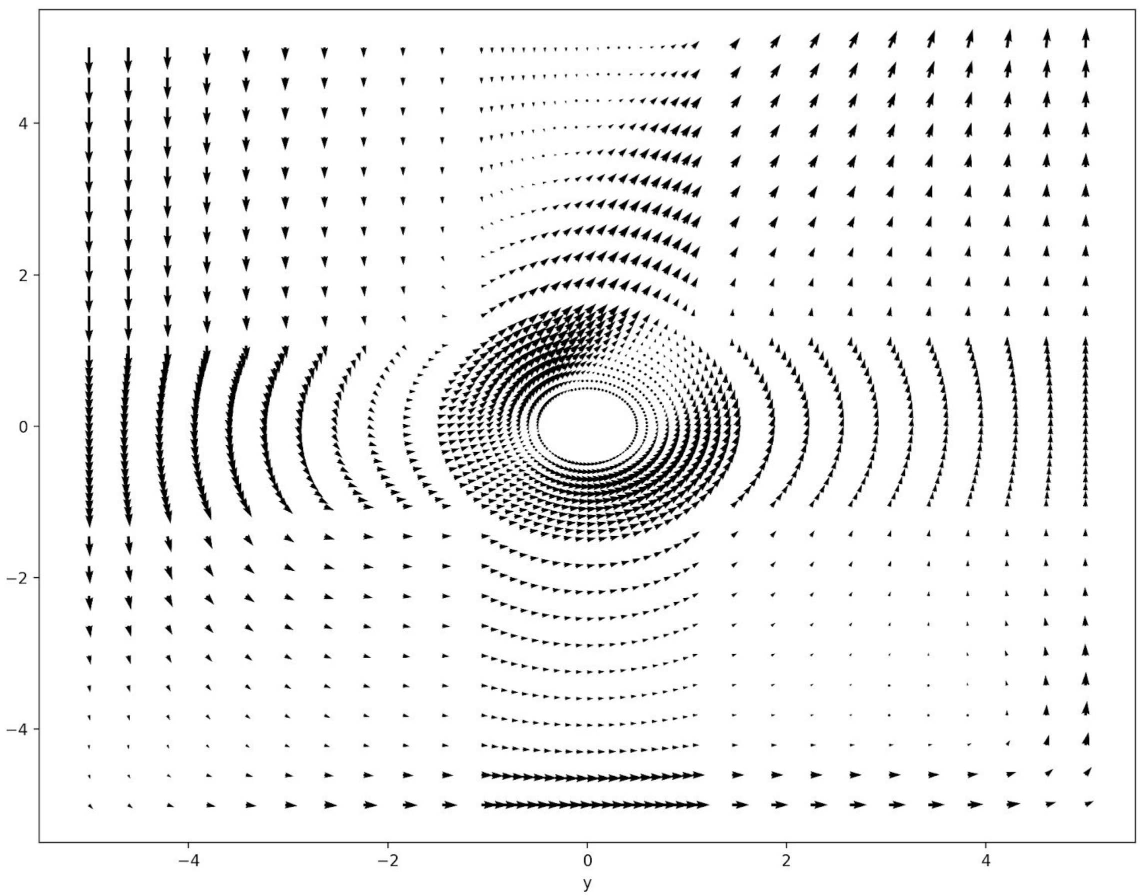
Plot of $\frac{p}{U^2}$ for Re=110



One can see that for the image above, there are areas of such high pressure, that the majority of the area looks low pressure by comparison due to the large scale.

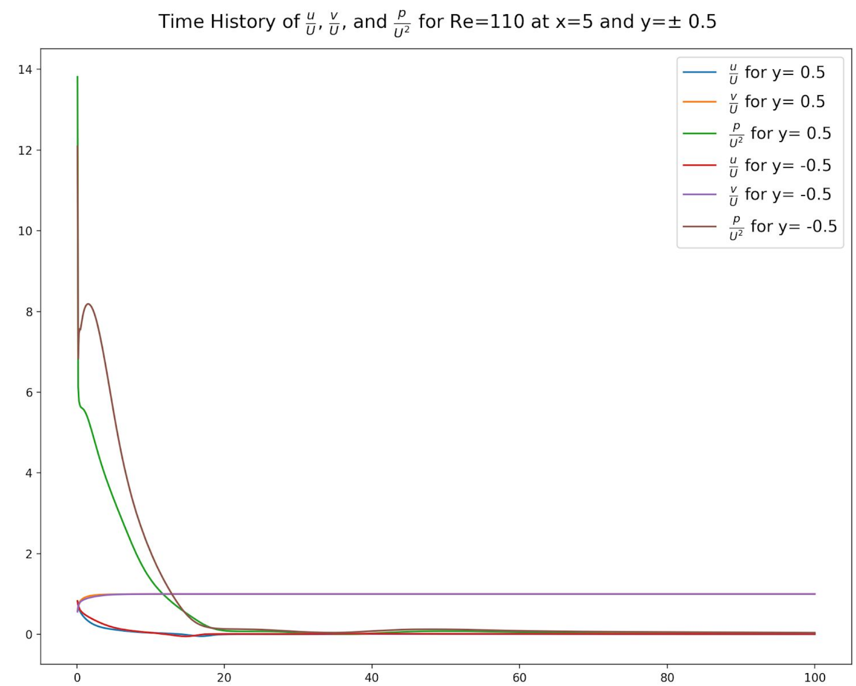
For the steady case (Re=20), we plotted the streamlines to provide a qualitative overview of the flow.

Streamplot for Re=20.



It appears that the velocity is not symmetric about the x axis as may have been expected. This asymmetry likely accounts for some of the asymmetry in the contour plots above.

For the unsteady case (Re=110), we sampled the quantities u/U, v/U, and $p/(\rho U^2)$ at points (5,0.5) and (5,-0.5) and plotted those quantities over time.



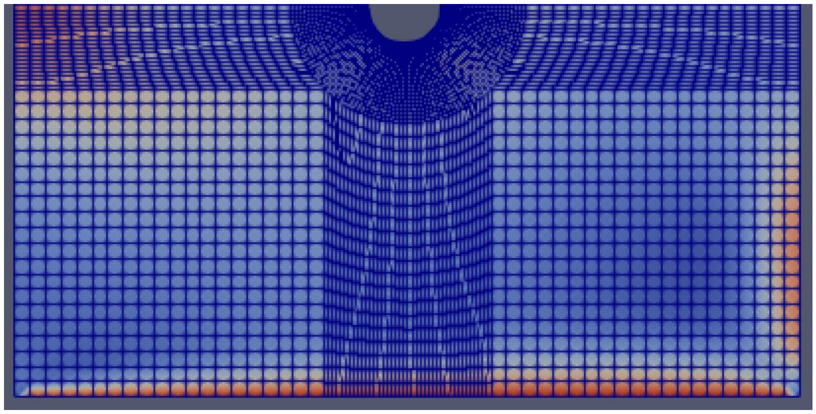
Improving the mesh

Using the two preliminary solutions, we decided how to modify mesh A and mesh B to improve our results. Specifically, we modified the meshes with respect to the following properties:

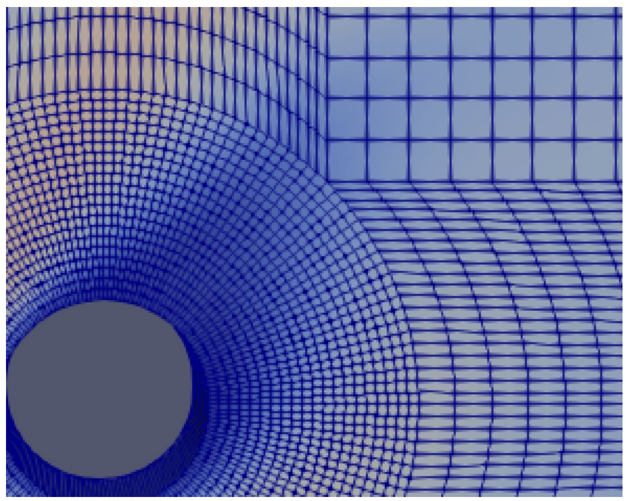
- Resolution
- Domain size

The considerations we made after inspecting the preliminary solutions are listed below:

- In the case with Re=110, the incoming flow was disturbed on the edge of the grid, where the flow was just entering the frame. As a result, we decided to increase the width of the grid for future trials.
- In both trials, the recirculation region was almost entirely visible inside our domain. However, we did have trouble identifying vortices shed from the cylinder. To better capture the recirculation region and vortices, we decided to further extend the rightmost boundary of the domain for future trials. Additionally, we decided to increase the density of the mesh for future trials to better capture vortex shedding.
- In both trials, we noticed irregular flow patterns at the top and bottom of our grids. We concluded that the flow at the top and bottom of the grid was different from a free stream flow. Since the presence of the wall was significantly perturbing the flow, we decided to increase the height of the grid for future trials. In the visualization below, we can see the irregular flow near the bottom wall.

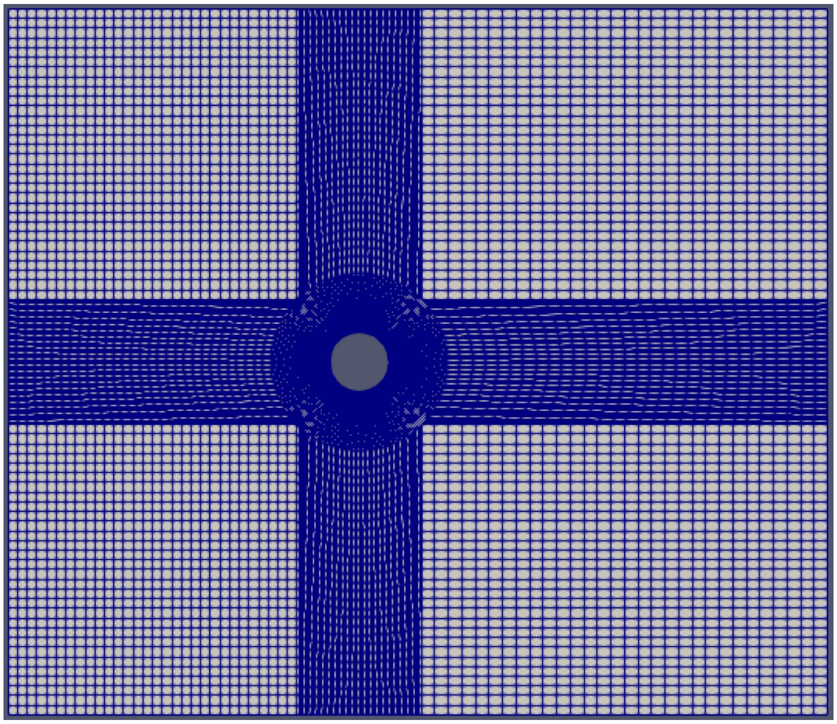


- In both trials, we felt our choice for the inner-cylinder's mesh was adequate. In the visualization below, we can see that the area of interest is almost entirely captured by the inner-cylinder section of our mesh.

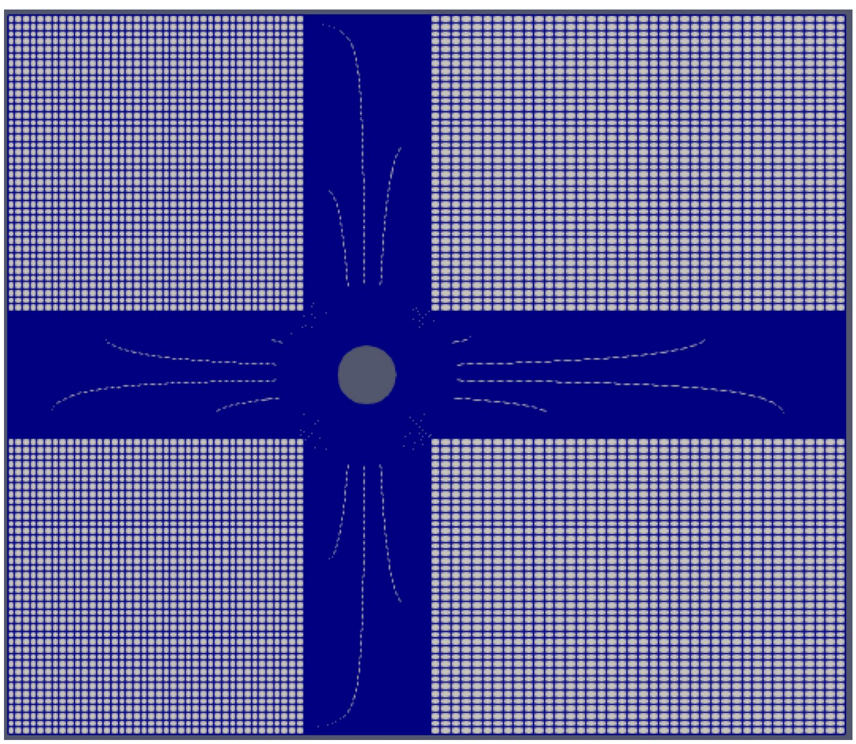


Using our observations, we devised 2 more mesh schemes. The two improved meshes, mesh C and mesh D, are described in the table below.

	Mesh A	Mesh B	Mesh C	Mesh D
Grid boundaries	x=[-5, 5], y=[-5, 5]	x=[-5, 5], y=[-5, 5]	x=[-6, 8], y=[-6, 6]	x=[-6, 8], y=[-6, 6]
Cylinder location	[0, 0]	[0, 0]	[0, 0]	[0, 0]
Block Density	(10 10 1)	(20 20 1)	(30 30 1)	(40 40 1)
Total blocks	2e3	4e3	1.8e4	3.2e4



Mesh C



Mesh D

Unsteady flow: Strouhal number of the vortex shedding

From the literature, we can see that for Re = 110 that the Strouhal number is about .16.

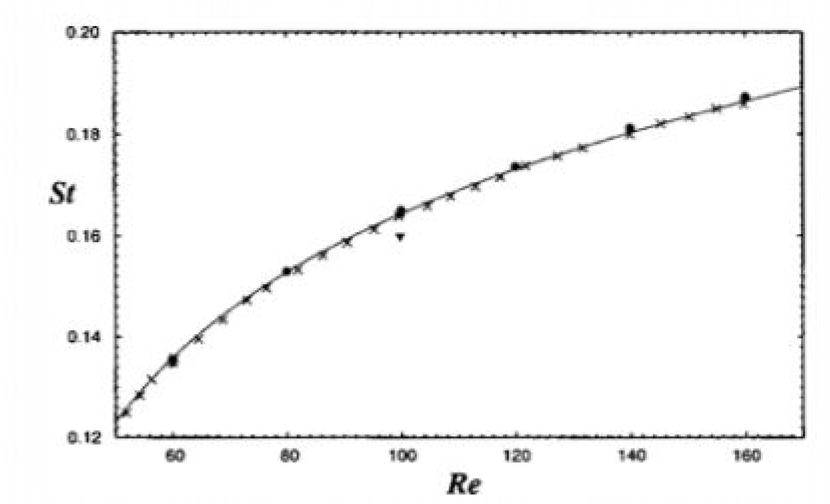


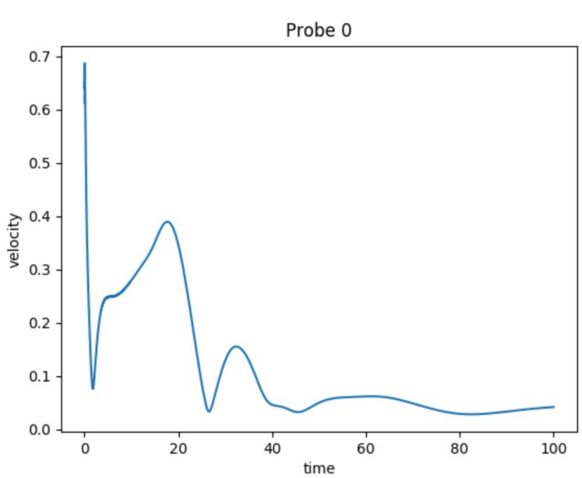
Fig. 2 Strouhal number vs. Reynolds number: ●, present study; ———, $St = -3.3265/Re + 0.1816 + 1.6 \times 10^{-4} Re$ (Williamson 1989); ×, Norberg (1994); ▼, Braza *et al.* (1986); ▲, Park *et al.* (1994).

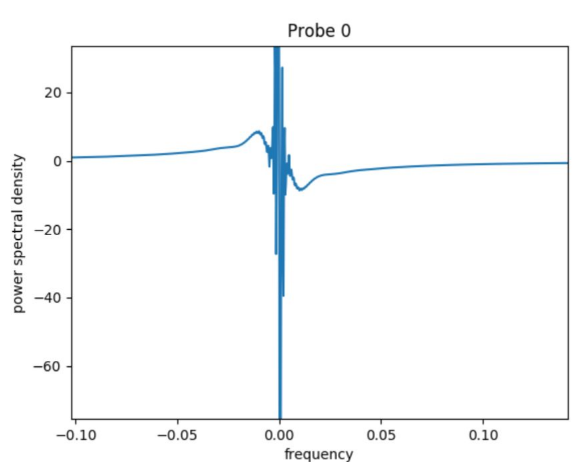
In order to attempt to calculate the Strouhal number from our simulations, we placed probes at various locations throughout the wake. At those probes, we measure the velocity at each time point. We apply a discrete Fourier transform to the velocity data, and from there we can plot the transform. The frequency at which the graph has a peak will be the frequency of the vortex shedding.

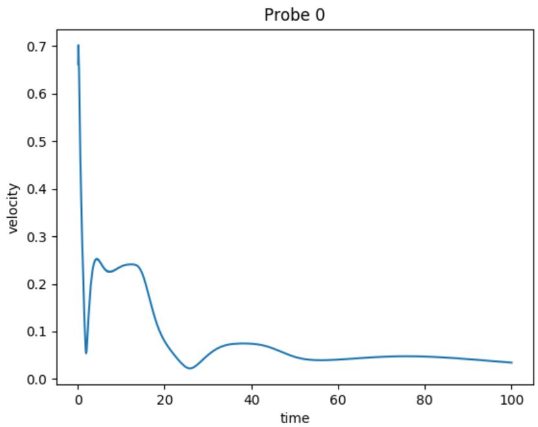
Below are the results for different meshes and different timesteps on Probe 0 located at (1.0 0.0, 0.5):

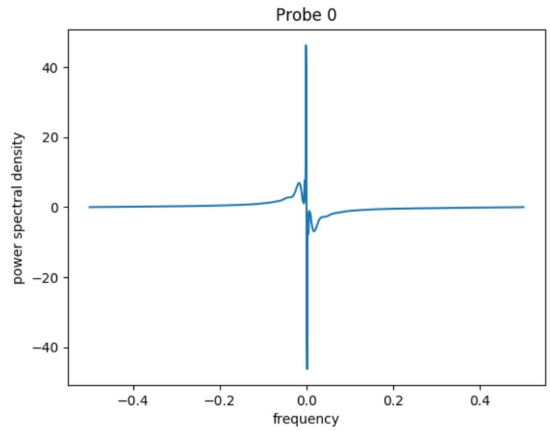
Mesh D, dt = 0.03

Mesh D, dt = 0.05

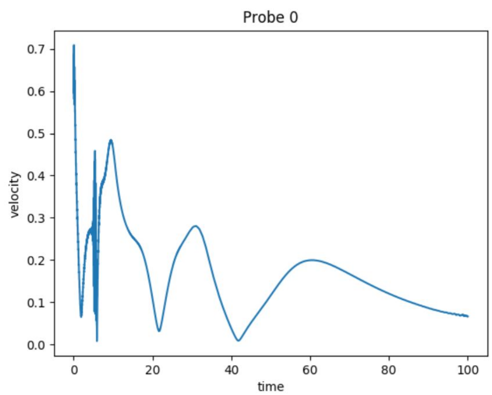


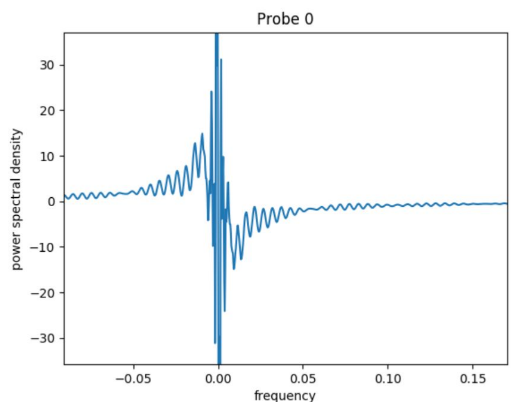




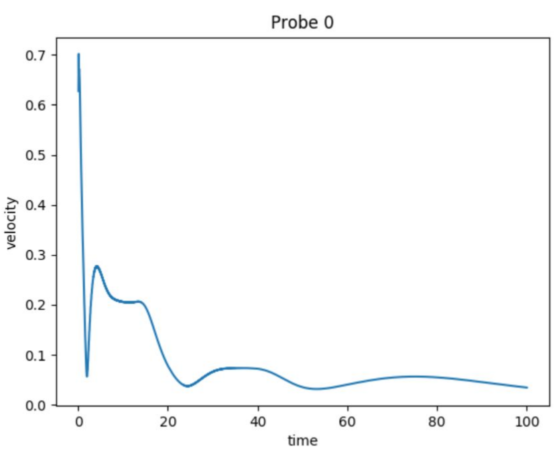


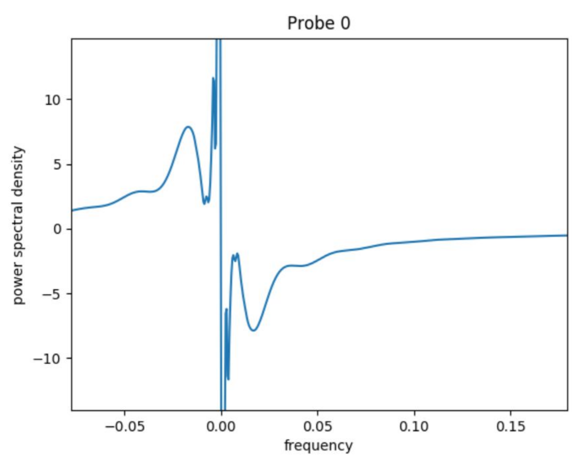
Mesh C, dt = 0.03





Mesh C, dt = 0.05

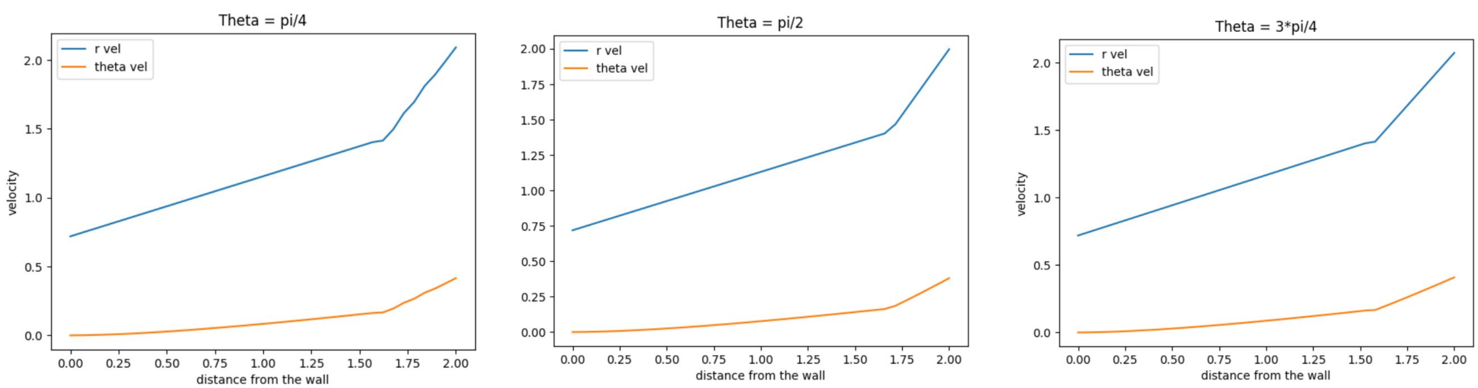




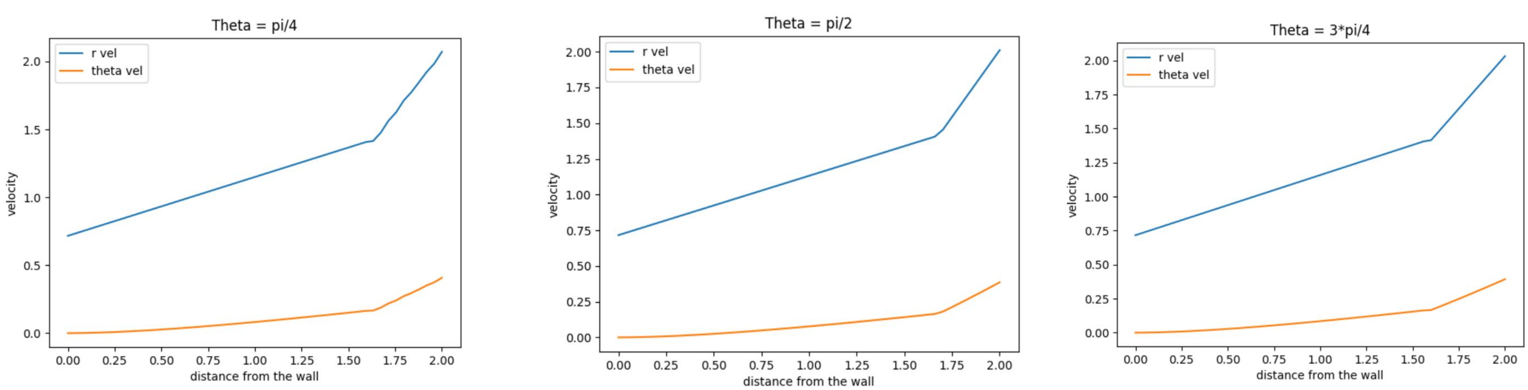
Steady flow: velocity in the boundary layer, separation length, and rate of strain at the cylinder's wall

The velocity was sampled along three lines, all with r=2.0 and different angles. The results on Mesh C and Mesh D at the 9 second mark are shown below:

Mesh C



Mesh D



Values for rate of strain tensor

Mesh & theta	e_r_r	e_r_theta
Mesh C theta = pi/4	0.4362646107164879	
Mesh C theta = pi/2	0.4124748309025966	
Mesh C theta = 3pi/4	0.44805554614125787	
Mesh D theta = pi/4	0.4333107479954011	

Mesh D theta = pi/2	0.41542523394709585	
Mesh D theta = 3pi/4	0.4421538244851031	