Unsteady laminar flow around a cylinder

1. **Purpose**

The Purpose of CFD Lab 2 is to simulate unsteady **laminar air flow** around a cylinder. Students will validate **velocity a d vorticity fields** obtained by simulation using experimental data (PIV results), analyze the differences between calculated and experimental frequencies of vortex break down, and present results in CFD Lab report.

2. Simulation Design

The problem to be solved is that of **laminar flow around a cylinder**. The diameter of the cylinder is equal to 18 mm. The inlet air velocity and temperature are 0.42 m/s and 20 C correspondingly. The conditions correspond to a PIV experiment [1]. Since the flow coming to the cylinder was uniformed in a wind channel and the length of the cylinder is much longer than its diameter the flow can be treated as two-dimensional.
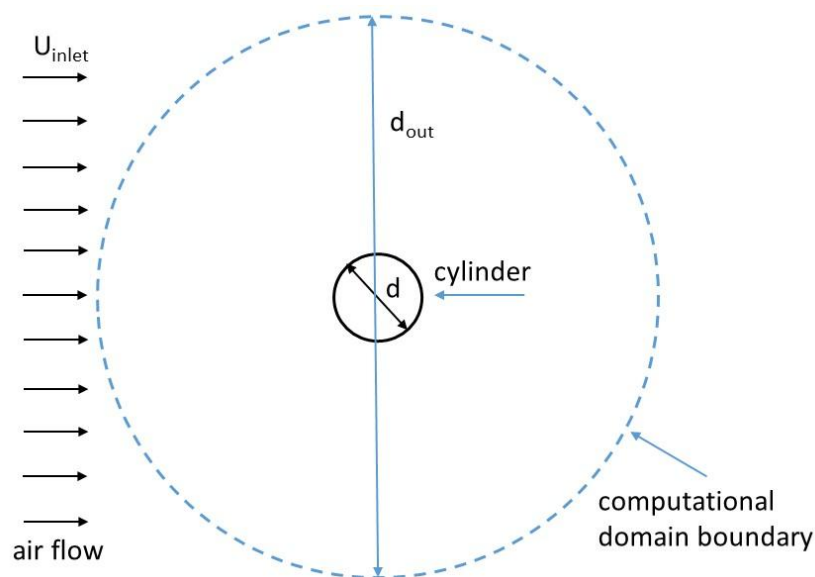

Figure 1. Flow around a cylinder: experimental conditions and computational domain.

Boundary conditions need to be specified include **inlet**, **outlet** and **wall**. Uniform flow is specified at inlet. No-slip boundary condition will be used on the wall and constant pressure for outlet.

The computations should be started using a steady option. Perform the calculations until a vortex-street after a cylinder appeared. This solution should be used as an initial field for the further transient calculations. The time step should not exceed to 0.01 s.

A non-uniform structured O-type mesh with the number of divisions equal to 100 in radial and 220 in circumferential direction correspondingly. The details of mesh generation are given below.

3. Project Schematic in Ansys Workbench

3.1. **Start ANSYS Workbench**

3.2. **Toolbox** -> **Component Systems**. Drag and drop **Geometry**, **Mesh** and **Fluent** components and create connections between components as per below.

**File** > **Save As**. Save the workbench file.

4. Geometry

4.1. Right click **Geometry** and select **New DesignModelerGeometry…**.

4.2. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.

4.3. Enable the auto constraints option to pick the exact point as below. Select **Sketching** > **Constraints** > **Auto Constraints >** make sure Cursor is selected**.

! The geometry of the computational domain is shown in Fig.1
It is recommended to created a circular domain around a cylinder.
The diameter of the domain should be equal to 20 diameters of the cylinder.

4.4. Select **Sketching** > **Draw** > **Circle**. Create two circles. The cursor will show "P" when it is on the origin point.
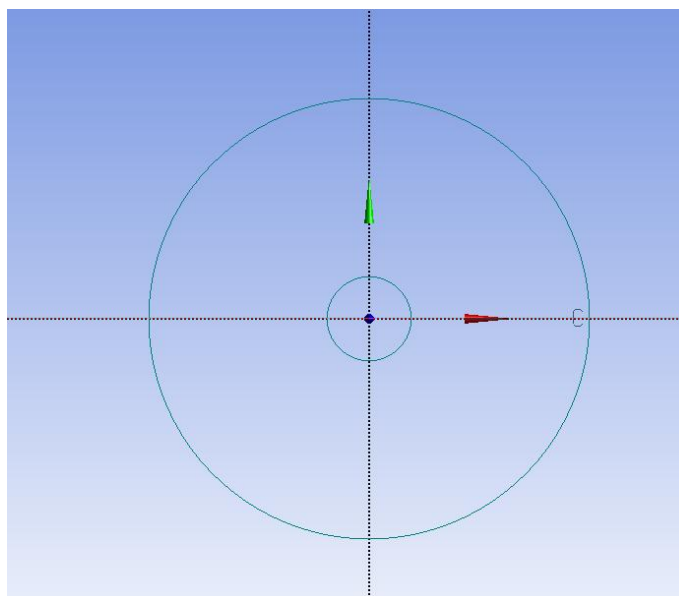
4.5. Select **Sketching** > **Dimensions** > **General**. Set the dimensions.
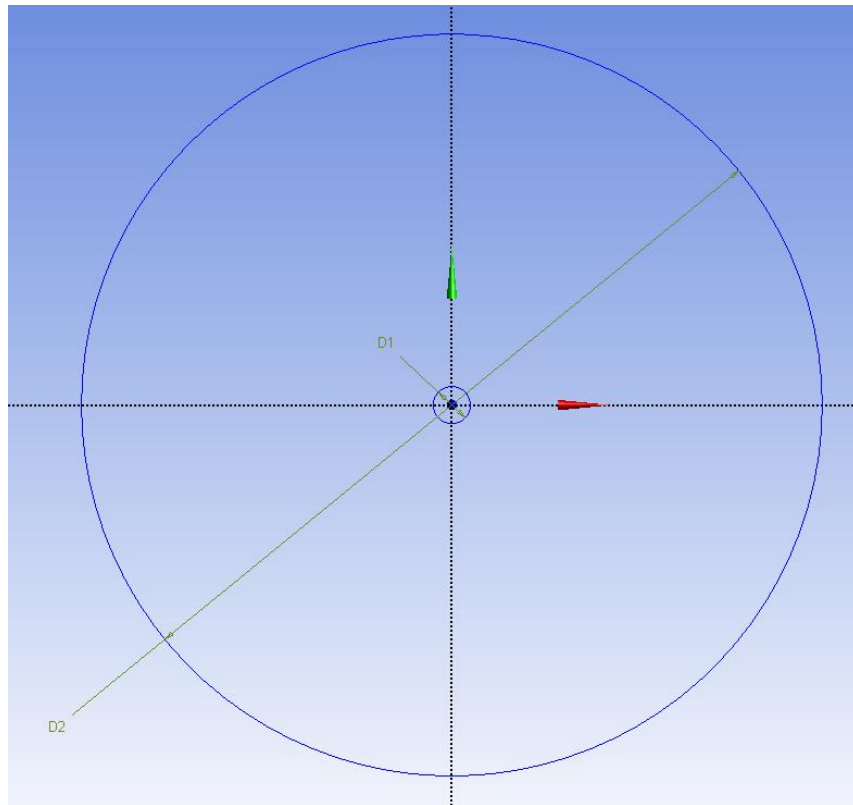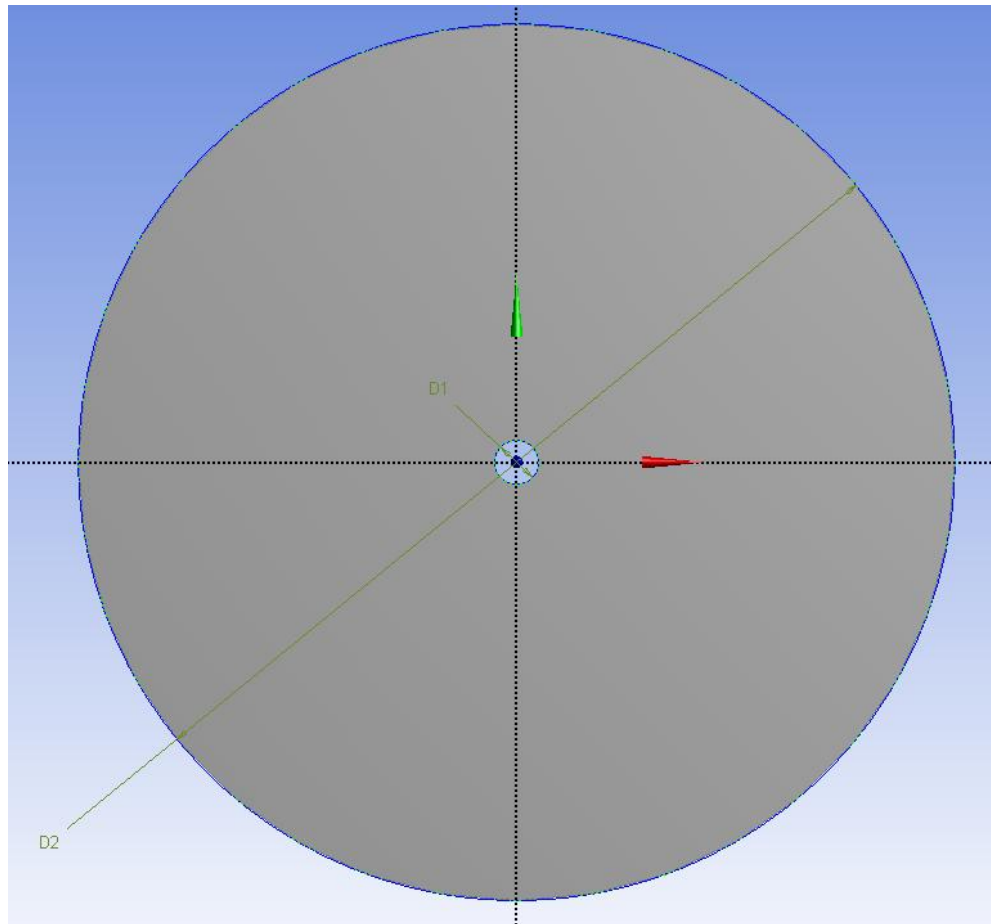

4.6. Click on **D1** and **D2** under **Details View** and change them to *0.018 m* and *0.36 m* correspondingly.

4.7. **Concept** > **Surfaces From Sketches** and select **Sketch1** from the **Tree Outline** and hit **Apply** on **Base Objects** under **Details view**.

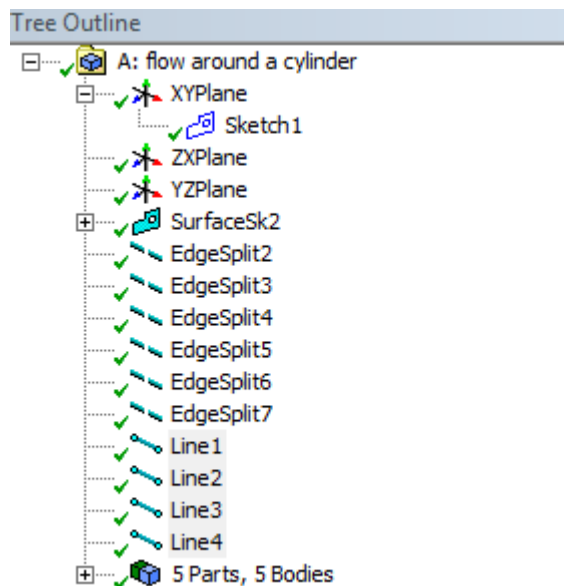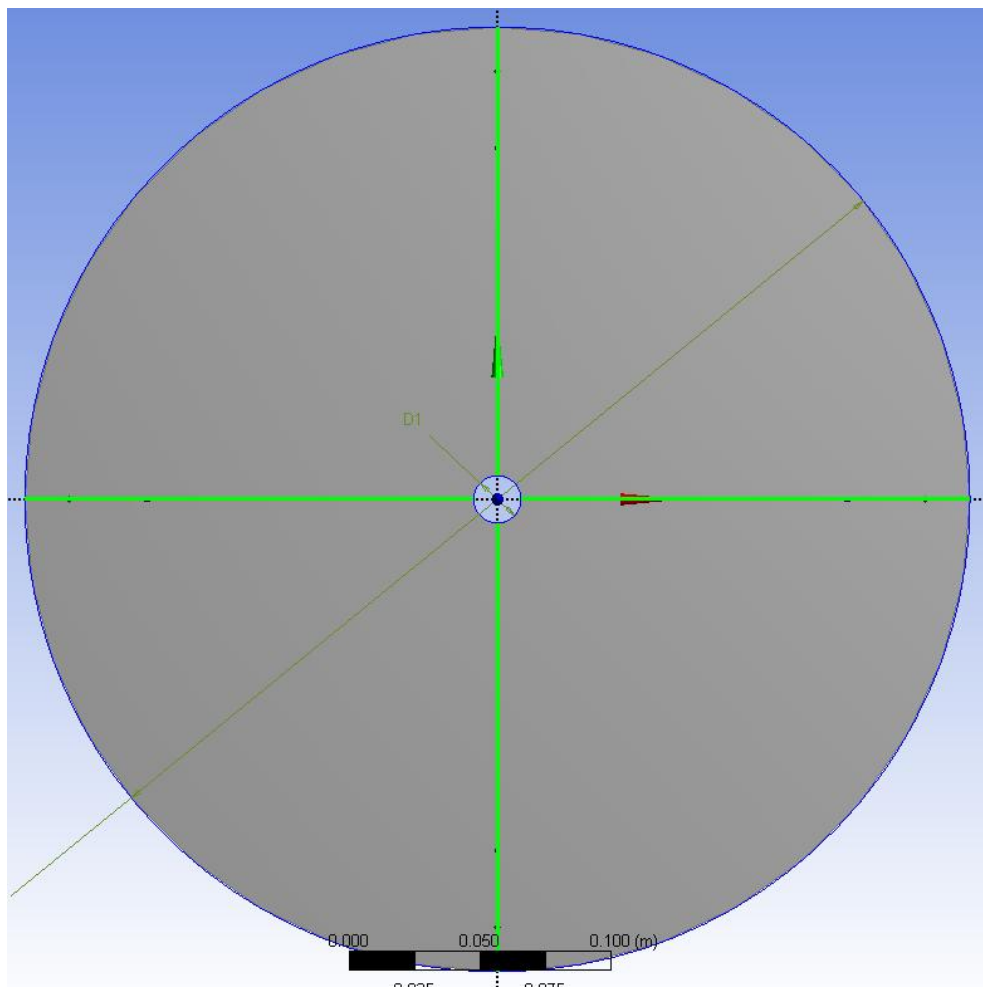4.8. Click **Generate**. This will create a surface.

4.9. **Concept > Split Edges**. Select the outer circle and click Apply. Make sure the Fraction is set to 0.5. This splits the edge in half. Click **Generate**.

4.10.   Repeat this action for the upper arc and outer arc.

4.11.   Repeat the same procedure for the inner circle. So each of the inner and the outer circle will be divided into four arcs.

4.12.   **Concept** > **Lines From Points** draw lines from the domain perimeter to the perimeter of the cylinder always starting from the outer circle domain and ending at the cylinder. Do this by selecting the point on the domain, hold **Ctrl** and select the point on the cylinder. Click **Apply** and then **Generate**. Repeat this process to create all the lines shown below.
        NOTE: If you do not create your lines starting from domain and ending in cylinder you will need to use a different bias type in the mesh generation section.

4.13. **Tools > Projection. Select all the lines you just created by holding Ctrl while selecting them** and then click **Apply**. For the **Target** select the surface of the domain and click **Apply.** Click **Generate.** This splits the domain into four sections as seen below.
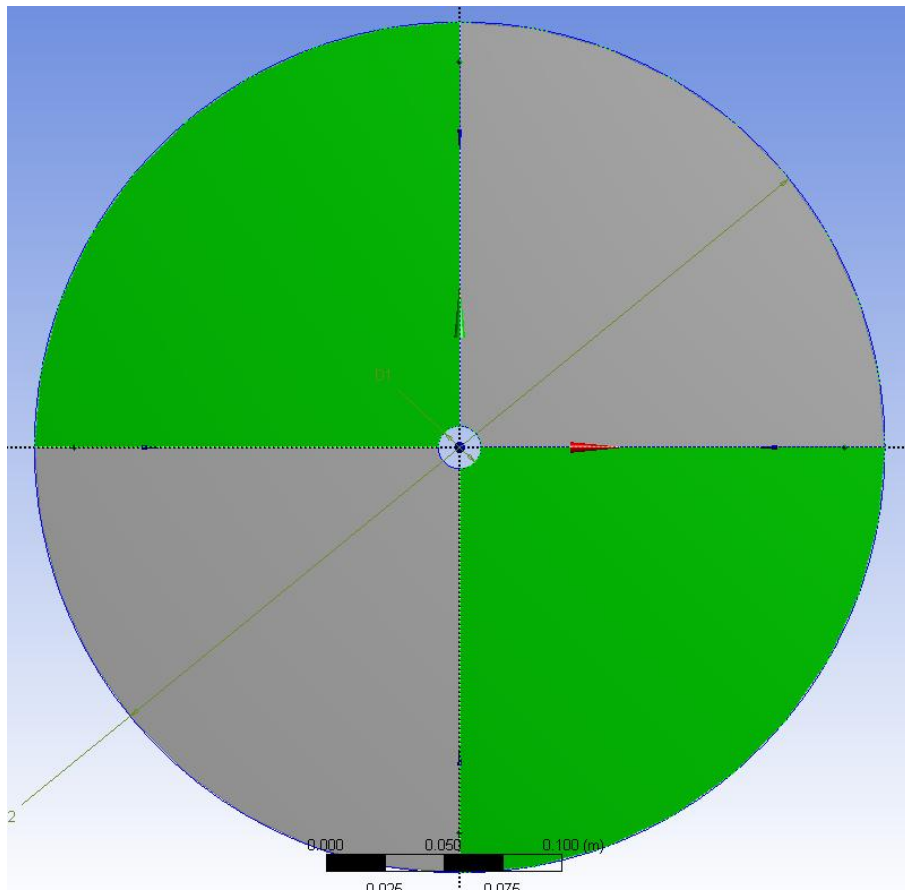
4.13.   **File > Save Project.** Close window.

5. Meshing

Create a nonuniform structured O-type mesh. The grid nodes should be concentrated near the cylinder walls. A step-by-steps description of a mesh generation is given below.

5.1. Right click on **Mesh** and select **Edit**. Start **Meshing**.

5.2. Right click on **Mesh** > **Insert** > **Face Meshing**. Select all four surfaces while holding Ctrl and click **Apply** in the yellow Geometry selection box**.**
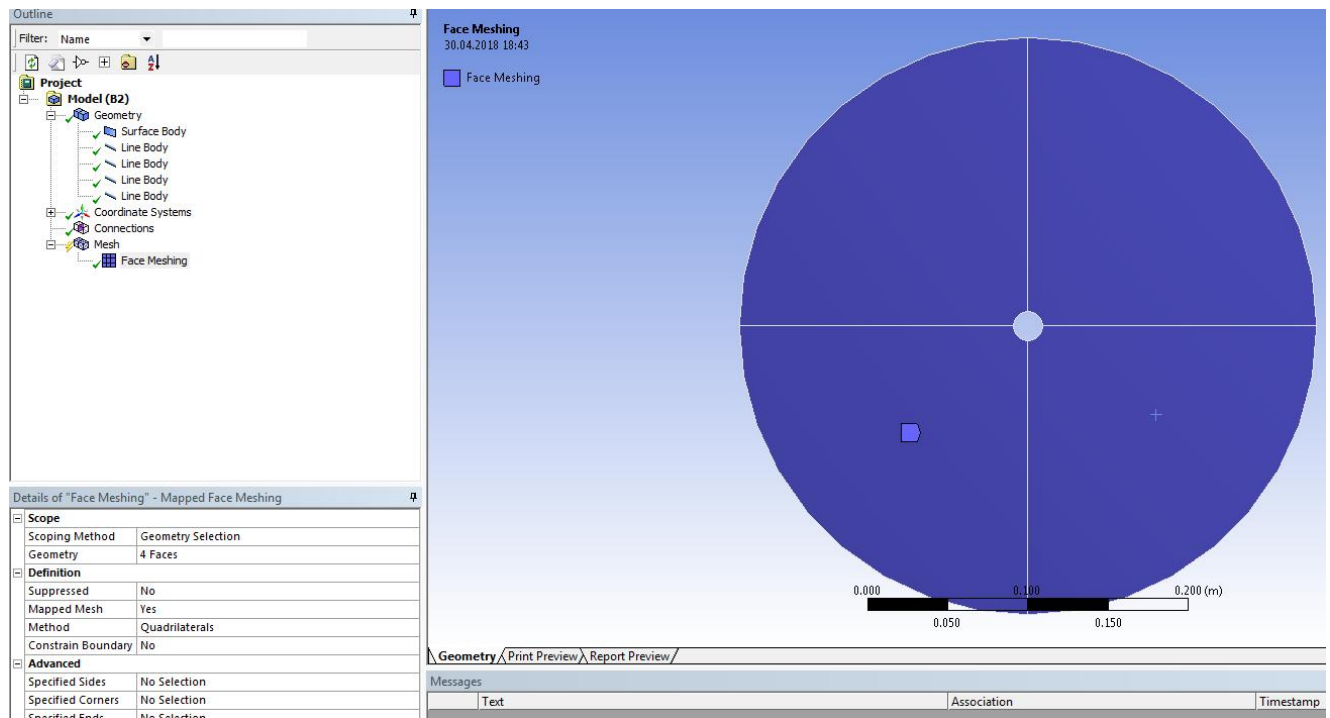

5.3. Select the edge button.

5.4. Right click on **Mesh > Insert > Sizing.** Select lines below and click **Apply.** Change parameters as per below.

NOTE: The divisions must be finer toward the cylinder. If they are not fine toward the cylinder you may need to change bias direction by changing bias type.

| Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	100
Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	____ ___ _ _
Bias Option	Bias Factor
Bias Factor	35.
Reverse Bias	No Selection
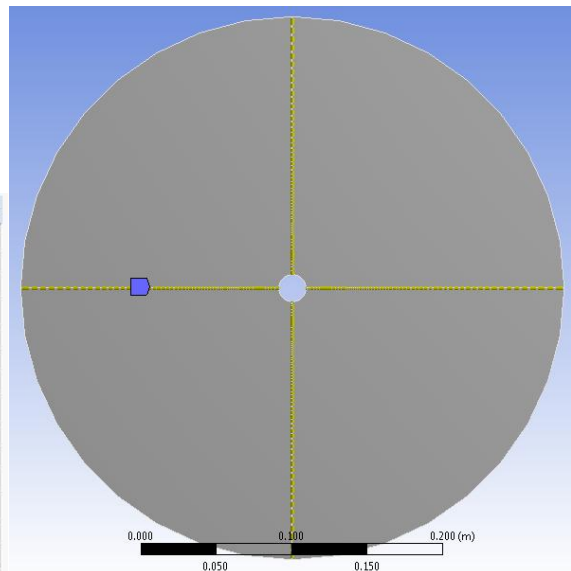
5.5. Right click on **Mesh > Insert > Sizing.** Select four left arcs shown below and click **Apply.** Change parameters as per below.
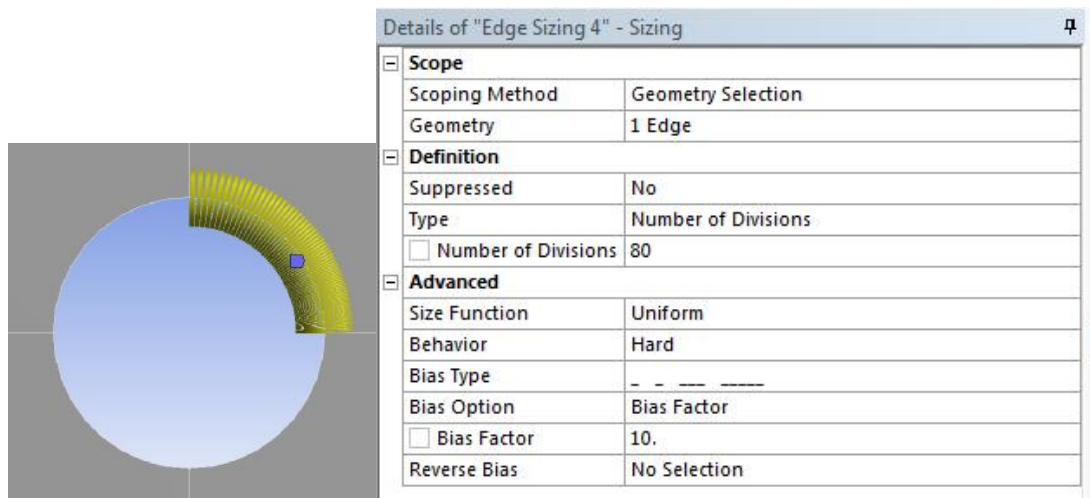

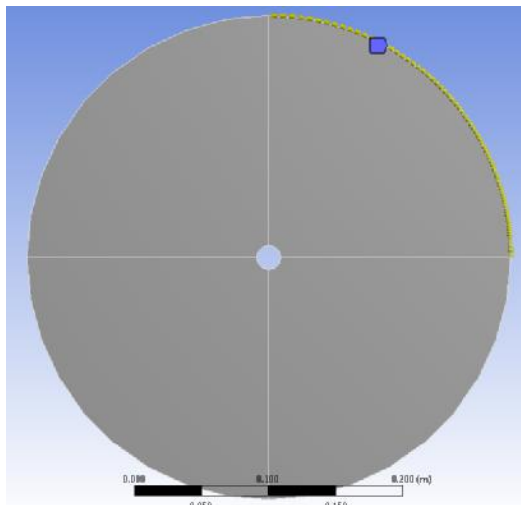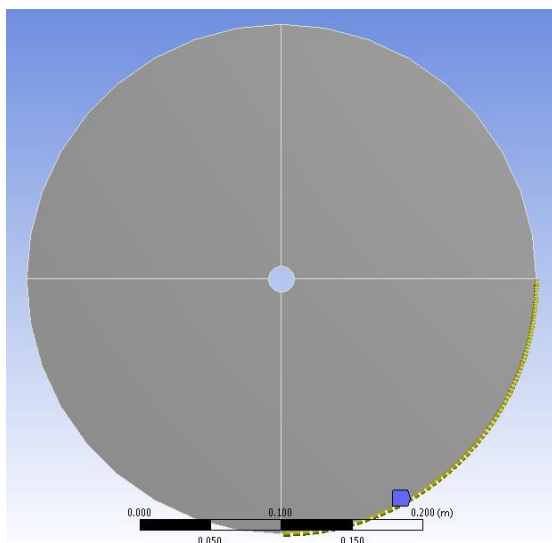| Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	30
Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	No Bias
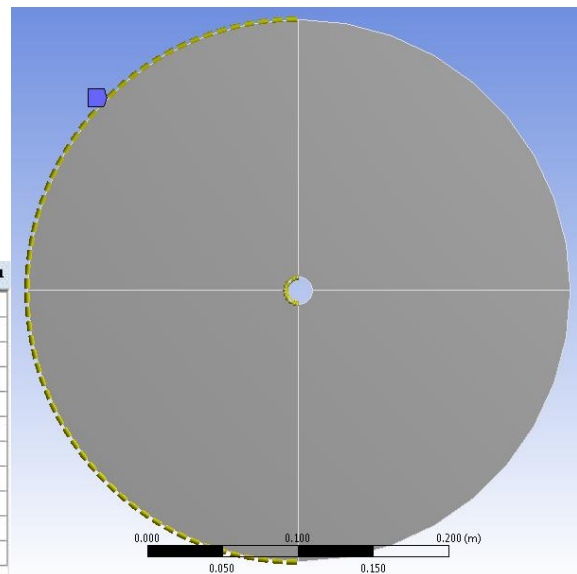
5.6. Right click on **Mesh > Insert > Sizing.** Select right inner arc shown below and click **Apply.** Change parameters as per below.

Details of "Edge Sizing 3" - Sizing		ᴘ
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Edge	
Definition		
Suppressed	No	
Type	Number of Divisions	
☐ Number of Divisions	80	
Advanced		
Size Function	Uniform	
Behavior	Hard	
Bias Type	_____ ___ _ _	
Bias Option	Bias Factor	
☐ Bias Factor	10.	
Reverse Bias	No Selection	

5.7. Right click on **Mesh > Insert > Sizing.** Select right inner arc shown below and click **Apply.** Change parameters as per below.

| Details of "Edge Sizing 4" - Sizing		ᴘ
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Edge	
Definition		
Suppressed	No	
Type	Number of Divisions	
☐ Number of Divisions	80	
Advanced		
Size Function	Uniform	
Behavior	Hard	
Bias Type	_ _ ___ _____	
Bias Option	Bias Factor	
☐ Bias Factor	10.	
Reverse Bias	No Selection	
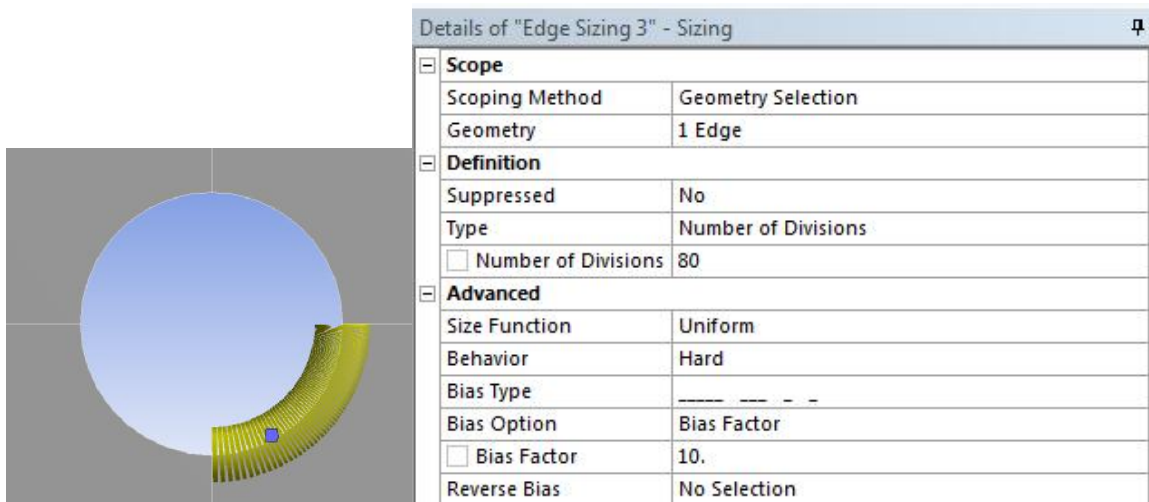
5.8. Right click on **Mesh > Insert > Sizing.** Select right outer arc shown below and click **Apply.** Change parameters as per below.


Details of "Edge Sizing 5" - Sizing		⊟
⊟ **Scope**		
Scoping Method	Geometry Selection	
Geometry	1 Edge	
⊟ **Definition**		
Suppressed	No	
Type	Number of Divisions	
☐ Number of Divisions	80	
⊟ **Advanced**		
Size Function	Uniform	
Behavior	Hard	
Bias Type	_ _ ___ _____	
Bias Option	Bias Factor	
☐ Bias Factor	5.	
Reverse Bias	No Selection	

5.9. Right click on **Mesh > Insert > Sizing.** Select right outer arc shown below and click **Apply.** Change parameters as per below.

Details of "Edge Sizing 6" - Sizing		⊟
⊟ **Scope**		
Scoping Method	Geometry Selection	
Geometry	1 Edge	
⊟ **Definition**		
Suppressed	No	
Type	Number of Divisions	
☐ Number of Divisions	80	
⊟ **Advanced**		
Size Function	Uniform	
Behavior	Hard	
Bias Type	_____ ___ _ _	
Bias Option	Bias Factor	
☐ Bias Factor	5.	
Reverse Bias	No Selection	

5.10.   Click on **Generate Mesh** button and click **Mesh** under **Outline** to show mesh.
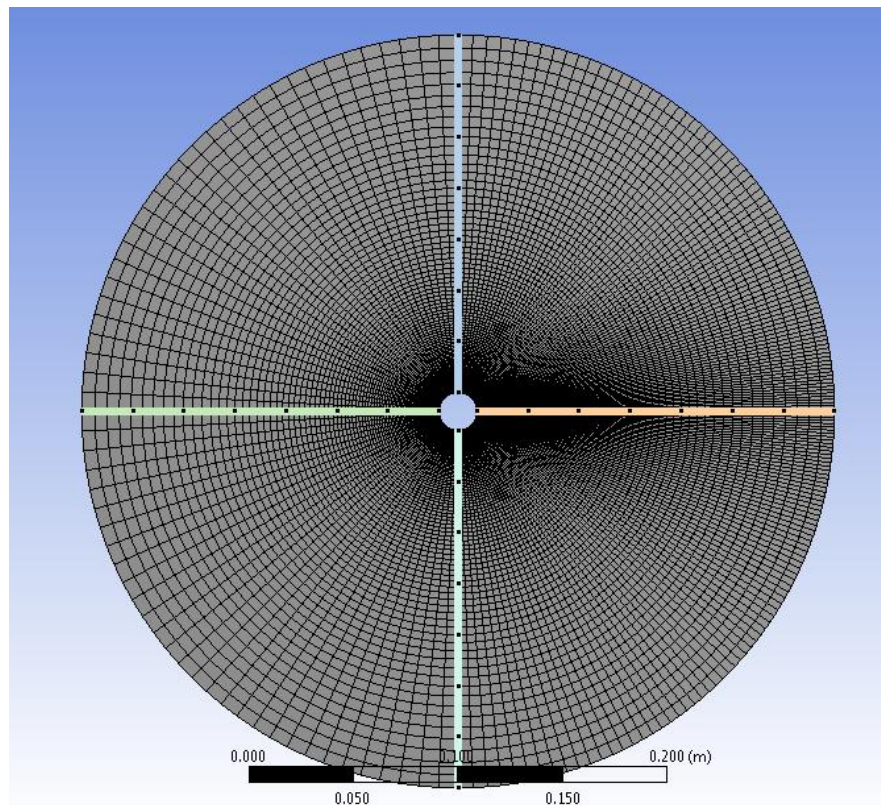
5.11.   Change the edge names by right clicking on the edge and selecting **Create Named Se-
lection**. Name left outer arcs as inlet, right outer arcs as outlet and inner arcs as wall. Your
outline should look same as the figure below.
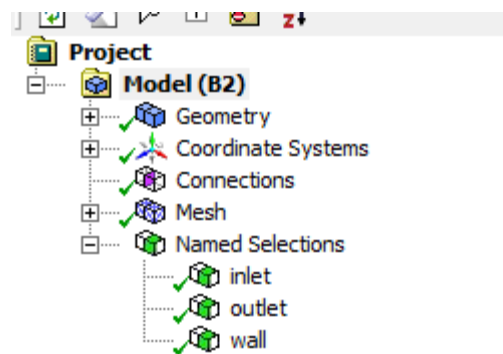
5.12.   **File** > **Save Project**. Save the project and close the window. Update **Mesh** on Work-
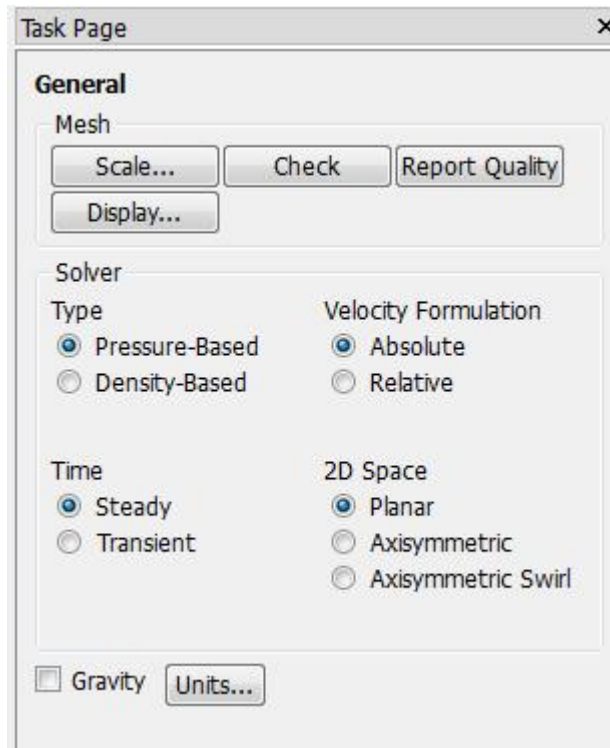bench if necessary.

6. Solving in Fluent

The computations should be started using a steady option. Perform the calculations until a vortex-street after a cylinder appeared (around 1000 iterations).

6.1. Right click **Setup** and select **Edit**.
6.2. Under options check **Double Precision** and click **OK**.
6.3. **Setup** > **General** > **Solver**. Choose options shown below.
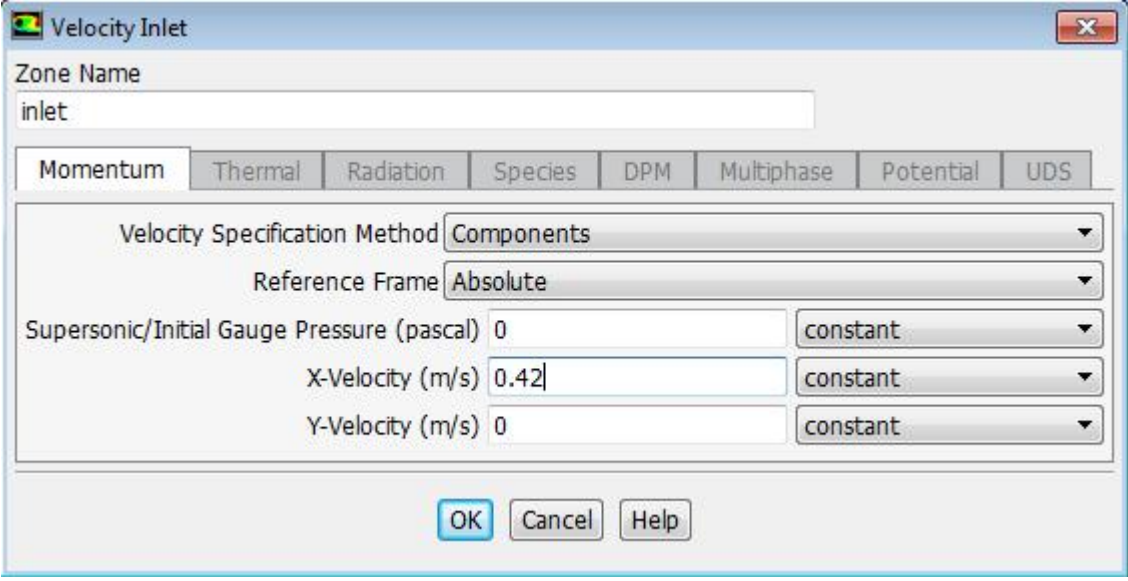
6.4. **Setup** > **Models** >**Viscous (Laminar)** (double click). Select parameters as per below and click OK.

Setup > **Materials** > **Fluid** > **air** (double click). Define the **Density** and **Viscosity** for air at 20°C and click **Change/Create**. Close the dialog box when finished.

Boundary conditions will be "inlet", "outlet", and "wall". Uniform flow is specified at inlet. The x-component of the velocity at the inlet of the computational domain should be set equal to 0.42 m/s. The y-velocity is equal zero. For outlet, zero pressure is defined. No-slip boundary condition will be used on the "wall".
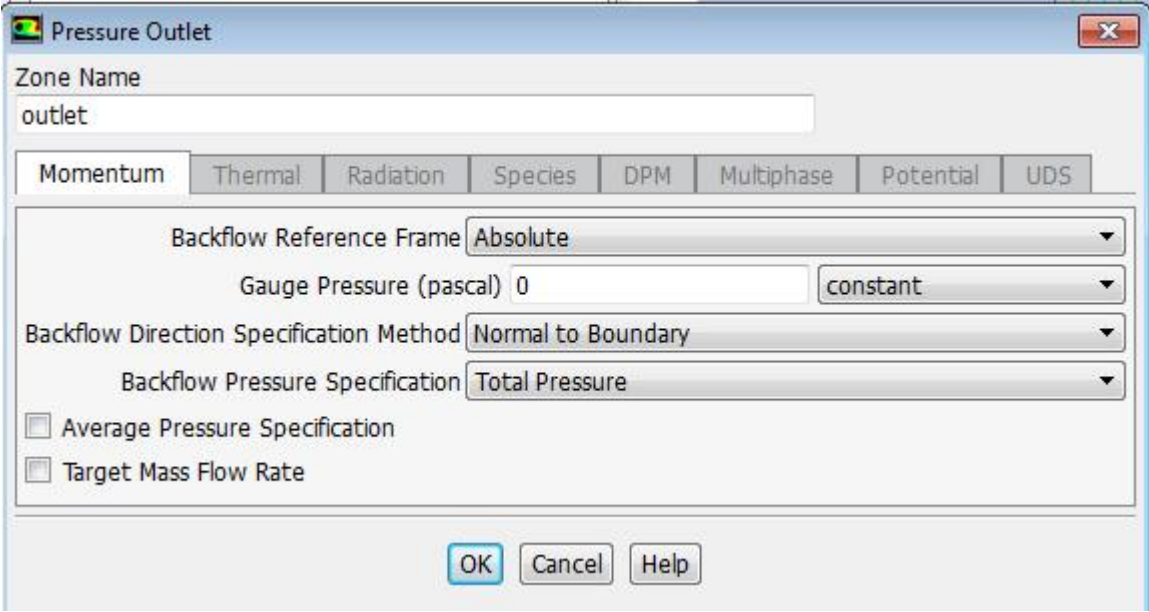
6.5. **Setup** > **Boundary Conditions** > **inlet** (double click). Change parameters as per below and click **OK**.

6.6. **Setup** > **Boundary Conditions** > **outlet** (double click) or click **Edit…**. Change parameters as per below and click **OK**. **Laminar flow**

6.7. **Setup** > **Boundary Conditions** > **wall** (double Click) Change parameters as per below and click **OK**. No need to change for laminar cases.

6.8. **Setup** > **Boundary Conditions** > **Operating Condition…**. Check the operating pressure (it should be equal to 1atmosphere) and click **OK**.

6.9. **Solution** > **Solution Methods**. Check the parameters are set as on the figure below.

6.10.　**Solution** > **Monitors** > **Residual**. Change convergence criterion to 1e-6 for all three equations click **OK**.

6.11.　**Solution** > **Solution Initialization**. Change parameters as per below and click **Initialize**.

6.12.　**Solution** > **Run calculation**. Change number of iterations to 1000 and click **Calculate**.

The steady solution should be used as an initial field for the further transient calculations.

6.13.　**Setup** > **General** > **Solver**. Choose **Transient** option as shown below.
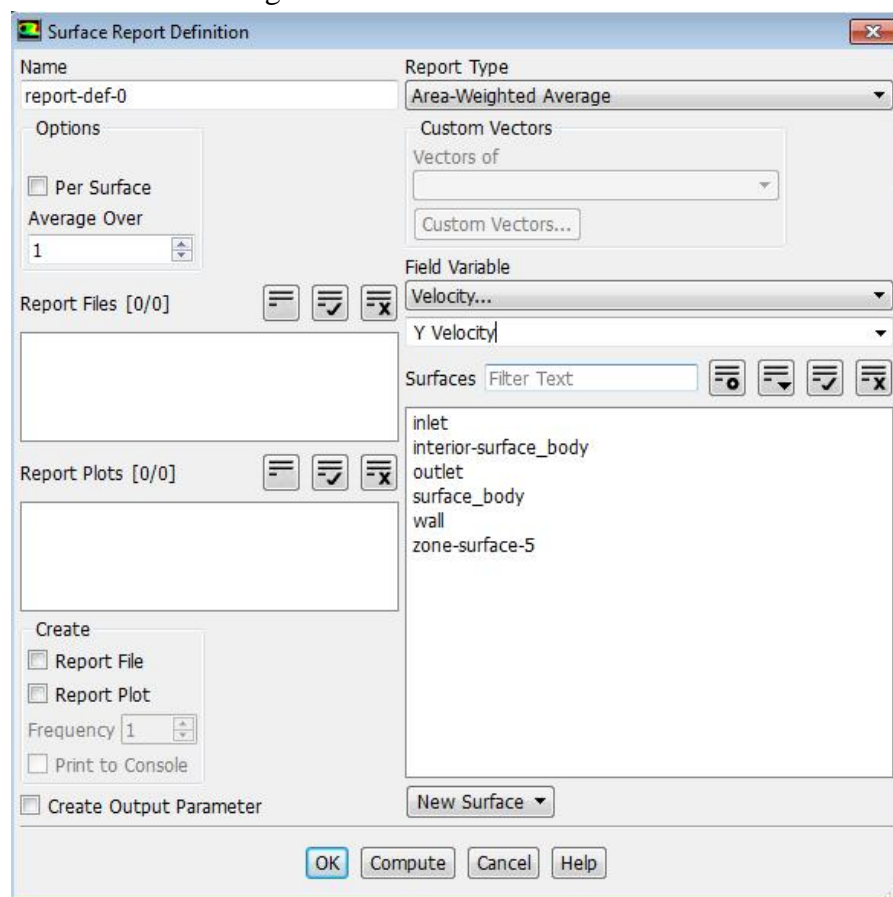
Define the monitoring points at the distance of 1÷5 caliber downstream of the cylinder.

6.14. **Monitors**>**Report Plots**. Click **New**. Change the parameters equal to those are set on the figure below. Click **New**.
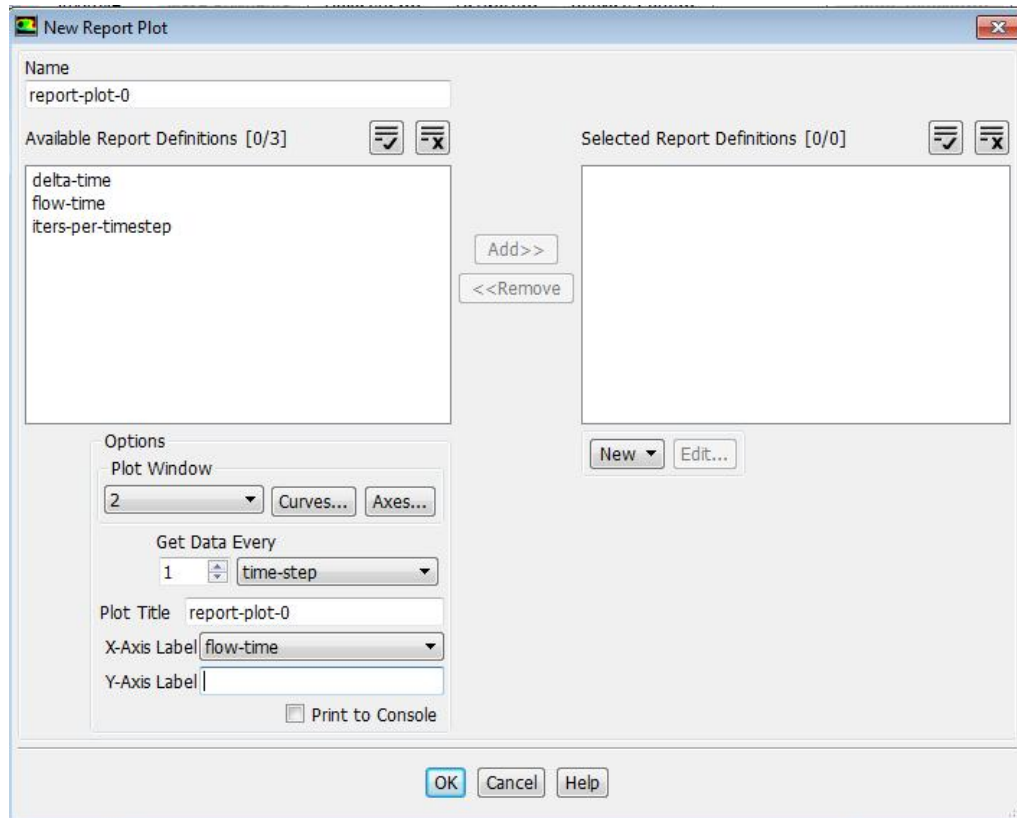
6.15. **New**>**Surface Report**>**Area-Weighted Average**. Click **New**. Change the parameters equal to those are set on the figure below.

6.16.	Click **New Surface** and set a monitoring point in a trace after a cylinder. The distance should be in the range of 1-5 caliber.

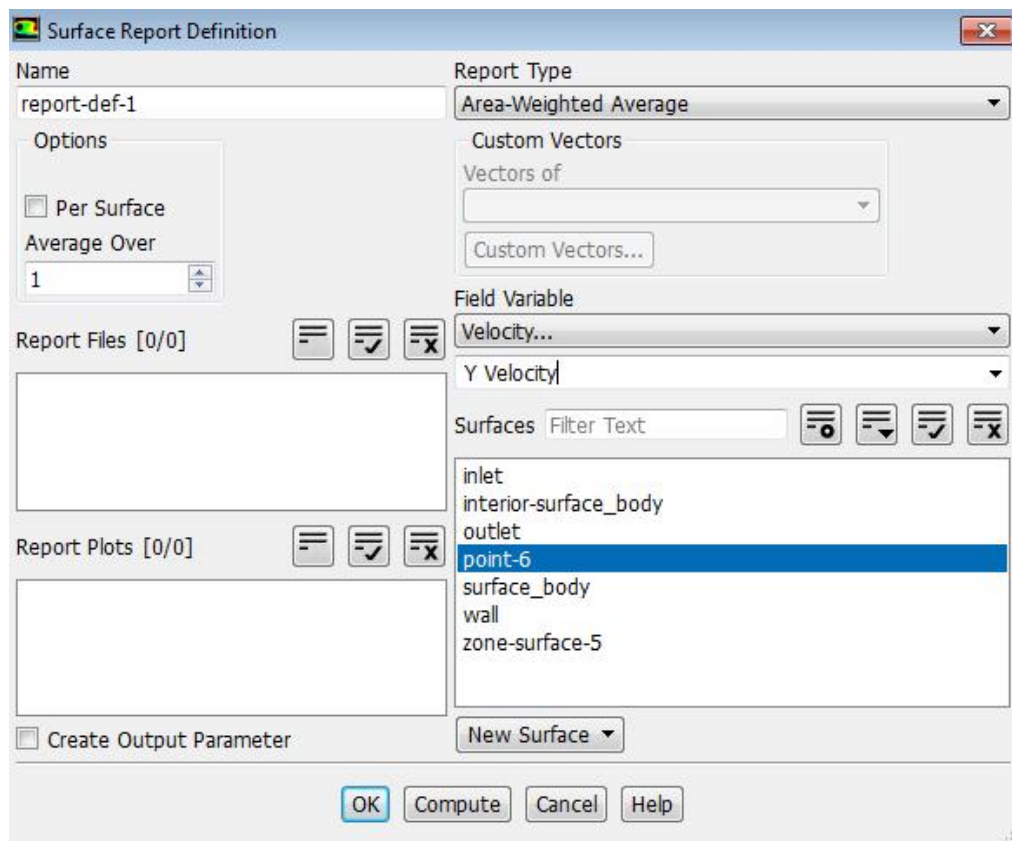6.17.	Choose **New Surface> Point> Select Point with Mouse**. Choose a Mouse-Probe Button. Then click **Create**.
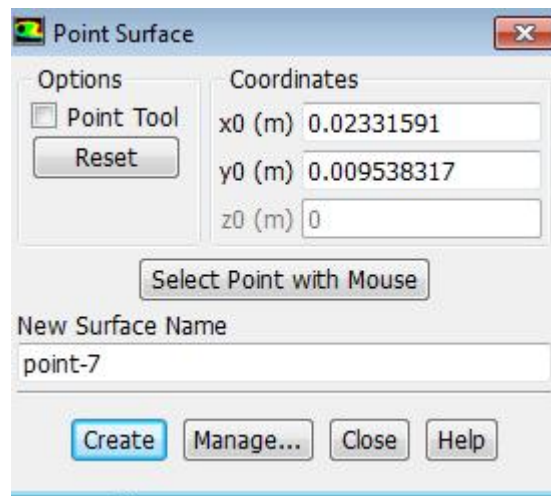
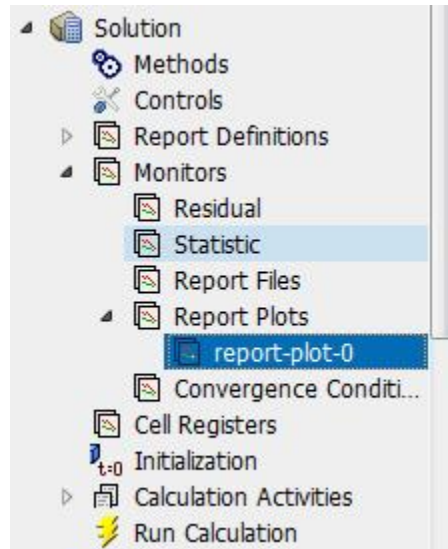Note! The coordinates in your case may differ from the example.

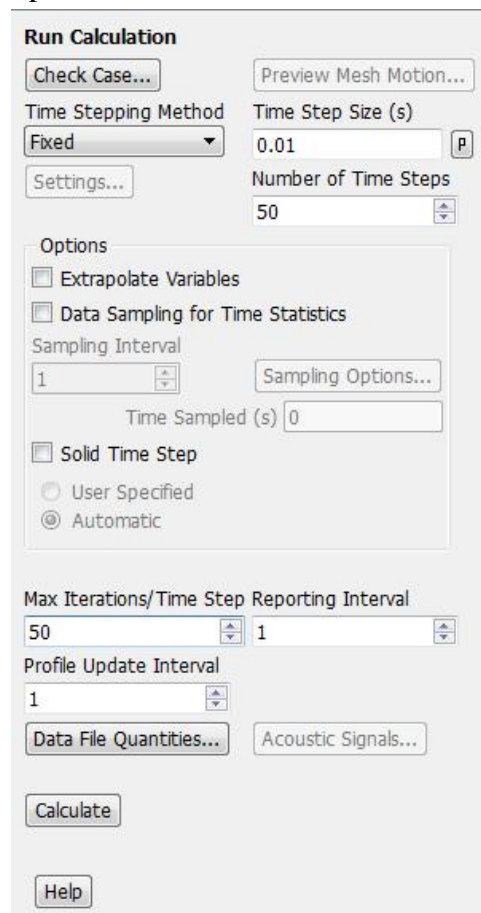The MOUSE-PROBE Button is the following

6.18.	A new point will appear in **Surface Report Definition.** Choose it and click **OK.**

6.19.   It will appear in **Report Plots**

6.20.   **Solution** > **Run calculation**. Change the time step to 0.01 s, number of step to 50 and number of iterations per step to 50 and click **Calculate**.

Examine the monitoring point graph and decide whether more time steps are requaired.

Note: Finally, you should obtain a periodic curve with a stabilized amplitude and period.

After that, you can stop the calculations.

6.21.   **File** > **Save** > **Picture**. Save the graph you obtained in a monitoring point.

7. Results post-processing

- Save a movie of transient vortex breakdown. (the instructions for animation are given below)

Pictures to be presented:
- Convergence history (Residuals)
- Instant flow fields after the cylinder.
- Instant vorticity fields after the cylinder.

Data to be reported:
- Time period and Strouhal number of vortex break down.

Questions to be answered:
- Compare the velocity fields obtained by numerical simulation and PIV (See Fig. 2). What is similar and what differ? Try to fine the reasons.
- Compare the vorticity fields obtained by numerical simulation and PIV(See Fig. 3). Instant flow fields after the cylinder.
- Compare Strouhal numbers obtained by numerical simulation and by experimental study (See Figure 3).

This section shows how to create a movie and prepare vorticity field figures.
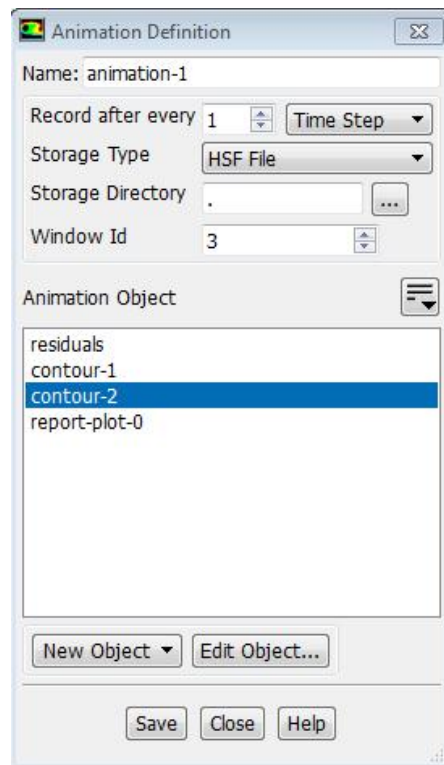
7.1. Vorticity field
7.1.1. **Graphs > Contours.** Choose the options and change the range as suggested below.
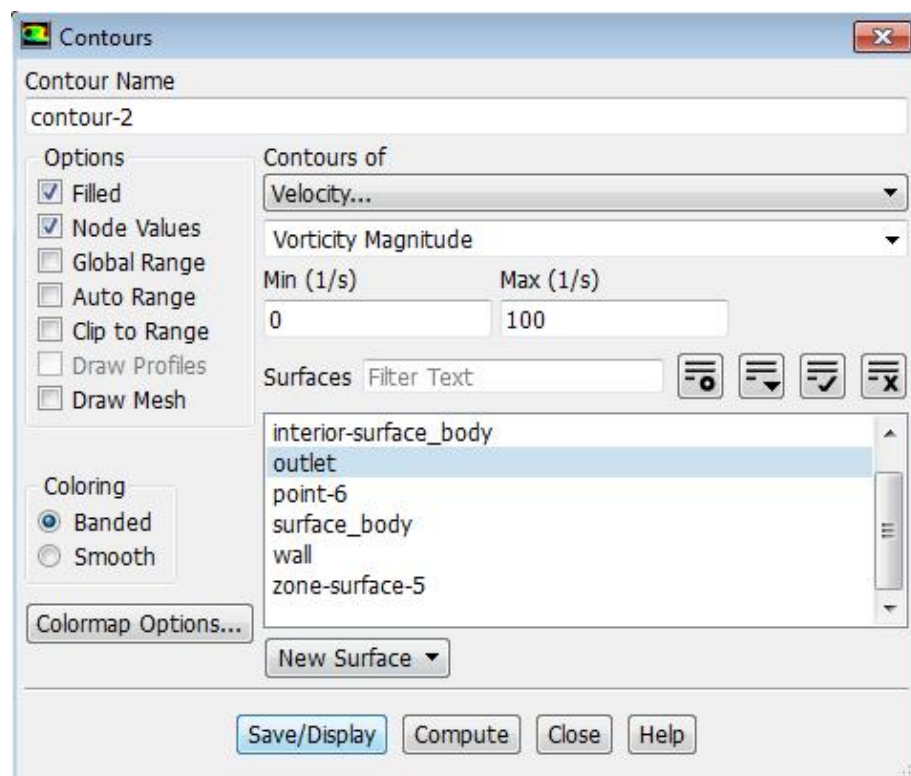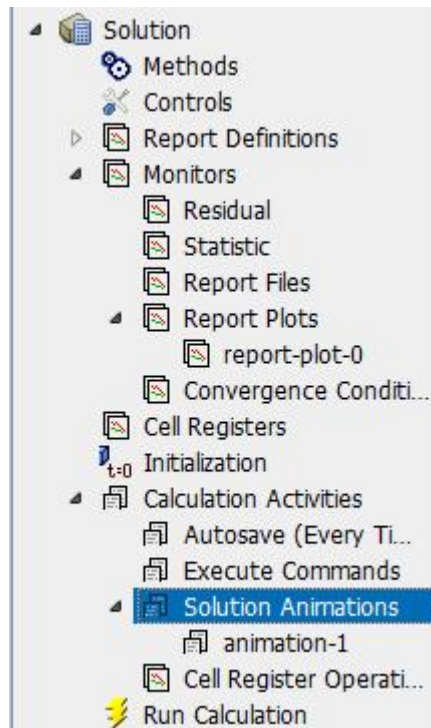
7.2. Animation instructions

7.2.1. Solution > Calculation Activities>Solution Animation. Change the options as per below. Check that you choose a contour that corresponds to vorticity field. Click **Close**.
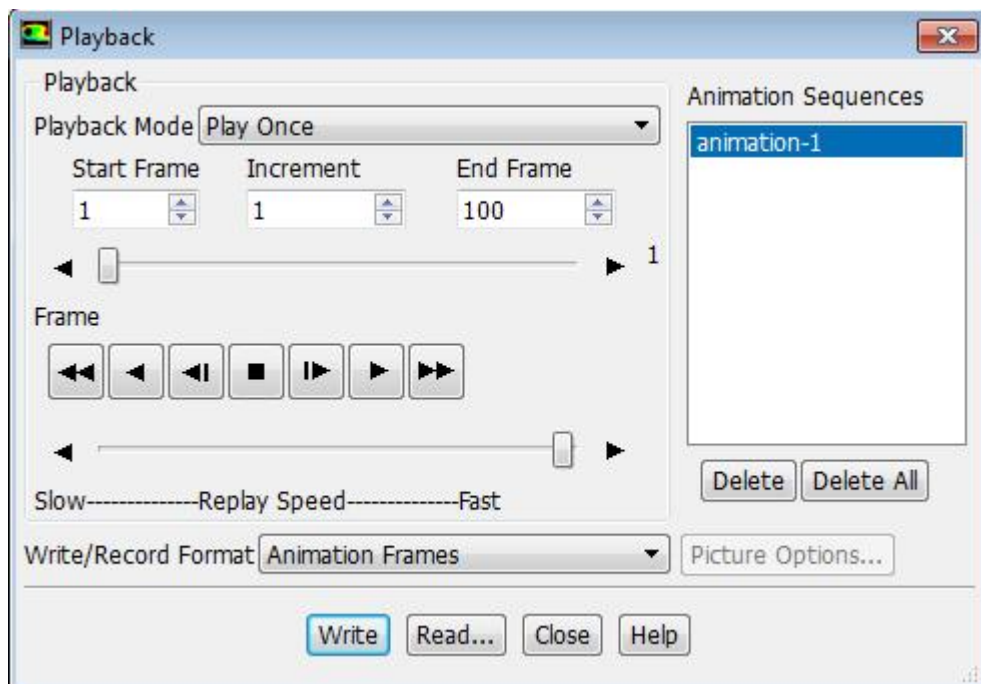

7.2.2. Now you can see your animation setiings in **Solution Animation** list.
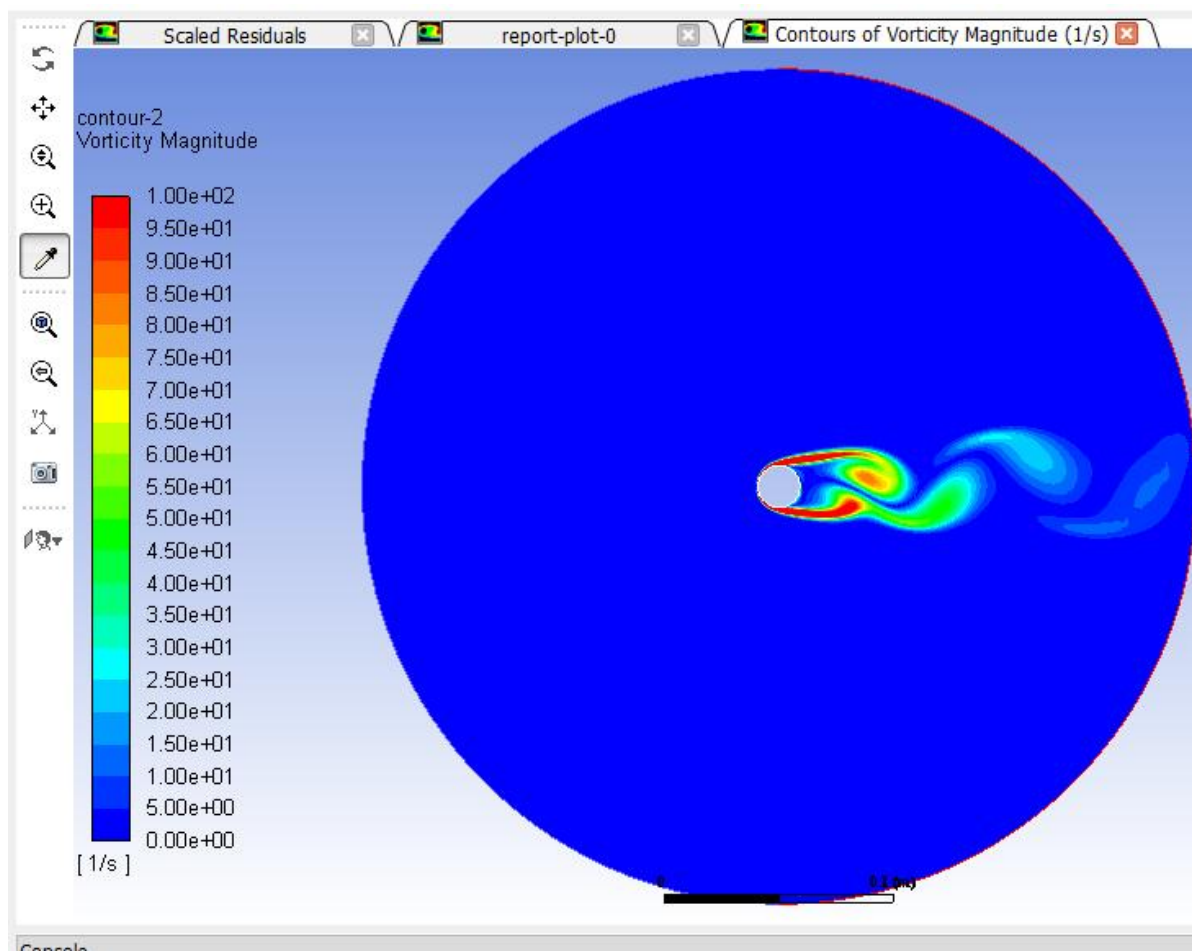
7.2.3. **Solution** > **Run calculation**. Set the number of steps and click **Calculate**.

7.2.4. After calculation complete go to **Results** > **Animations>Solution Animation Playback**.
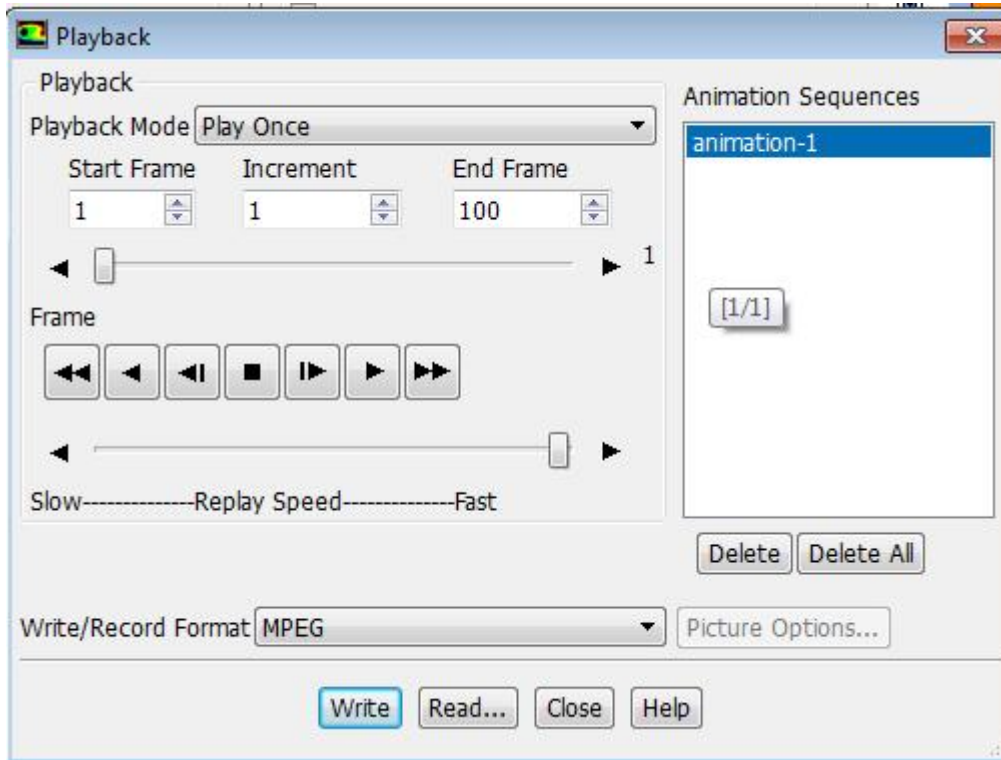

7.2.5. Choose Contours of Vorticity Magnitude

7.2.6. Click Play button in Playback Window

7.2.7. Change the **Write/Record Format** to MPEG. Click **Write** to Save animation file.

7.2.8. Don't forget to save the monitoring point graph! **File>Save Picture.Save**
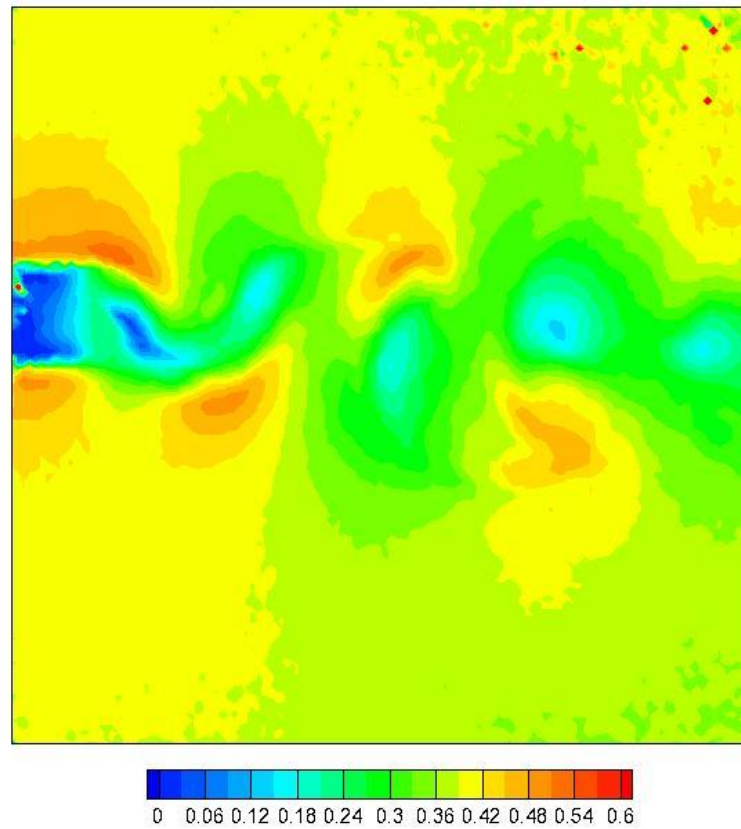
The experimental data for comparison.

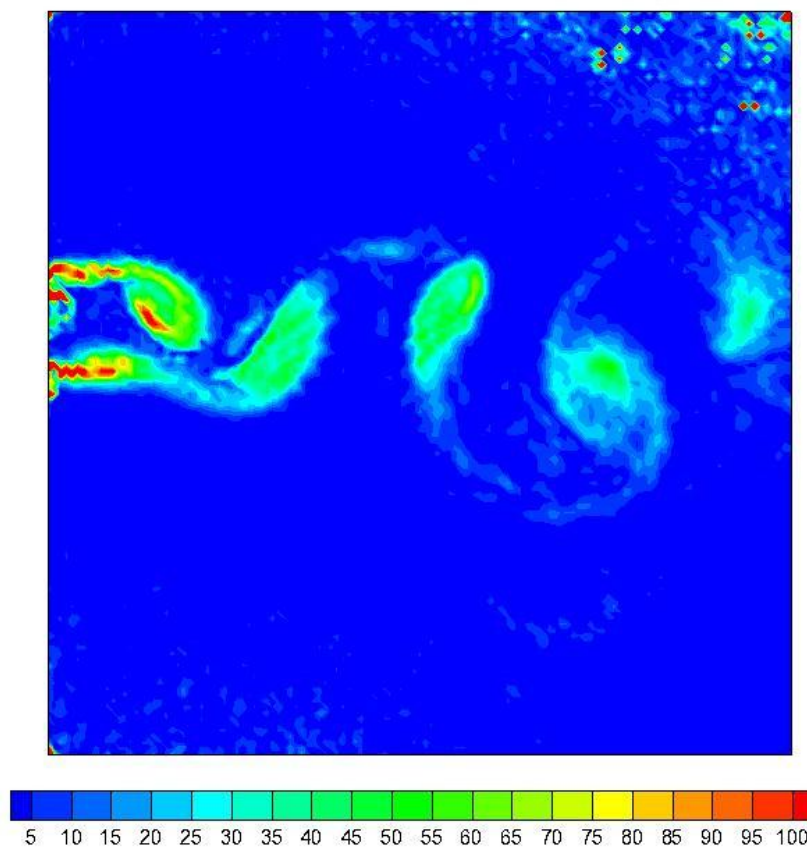Figure 2. An instantaneous field of velocity. Experimental (PIV) data.
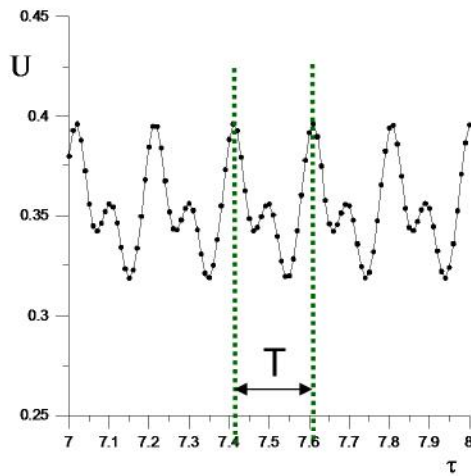

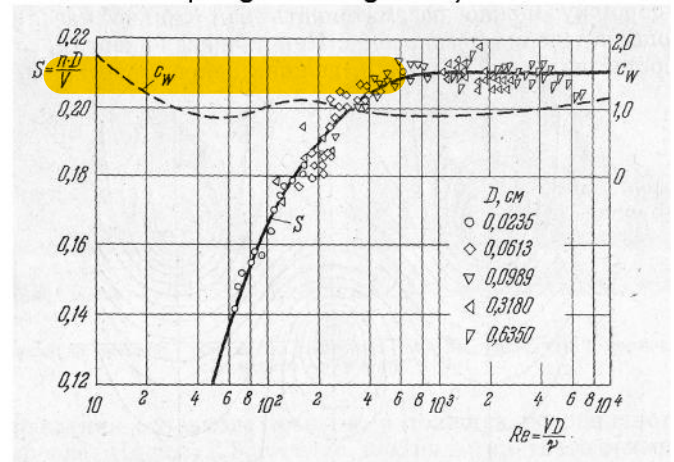Figure 3. An instantaneous field of vorticity. Experimental (PIV) data.

Velocity in a monitoring point
(numerical simulation results)

Experimental results
(Schlichting, Boundary Layer Theory,
8th ed. Springer-Verlag 2004)

$$S = d/(T \cdot U)$$

?

S, corresponding to
$Re = Ud/v$

Figure 4. Evaluation of Strouhal number.

References

1.	Phoenix News. Educational Use of PHOENICS in St Petersburg by Ekaterina E. Kitanina, PhD.
2.	POLIS description / Institute of Thermophysics SB RAS
http://www.itp.nsc.ru/piv/piv.htm
3.	Schlichting G. Boundary layer theory. M.: Nauka, – 712 c.
4.	P.K. Chang. Separation of Flow. – M.: Mir, 1972.– 300 c.