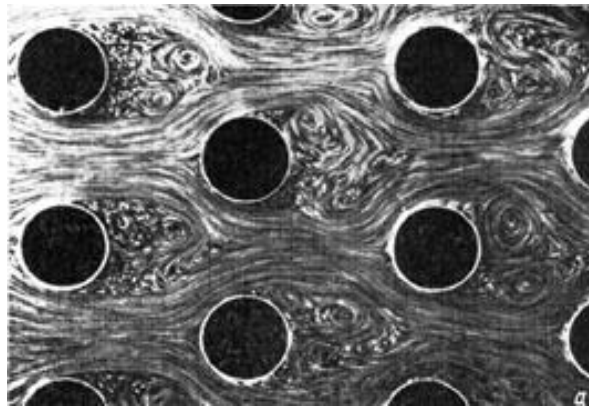


Turbulent flow and heat transfer in a tube bank



1. Purpose

The Purpose of CFD Lab 3 is to simulate a steady **turbulent** flow and heat transfer in a tube bank. A schematic of the problem is shown in Fig. 1. The bank consists of uniformly spaced tubes with a diameter of 9.7 mm, which are staggered across the cross-fluid flow. Their centers are separated by a distance S_2 of 20.3 mm in the x direction, and $S_1 = 24.8$ mm in the y direction. Because of the symmetry of the tube bank geometry, only a portion of the domain needs to be modeled.

Students will validate **Nusselt number and pressure loss coefficients** obtained by simulation using empirical data, analyze the differences between calculations and experimental results, and present results in CFD Lab report.

2. Simulation Design

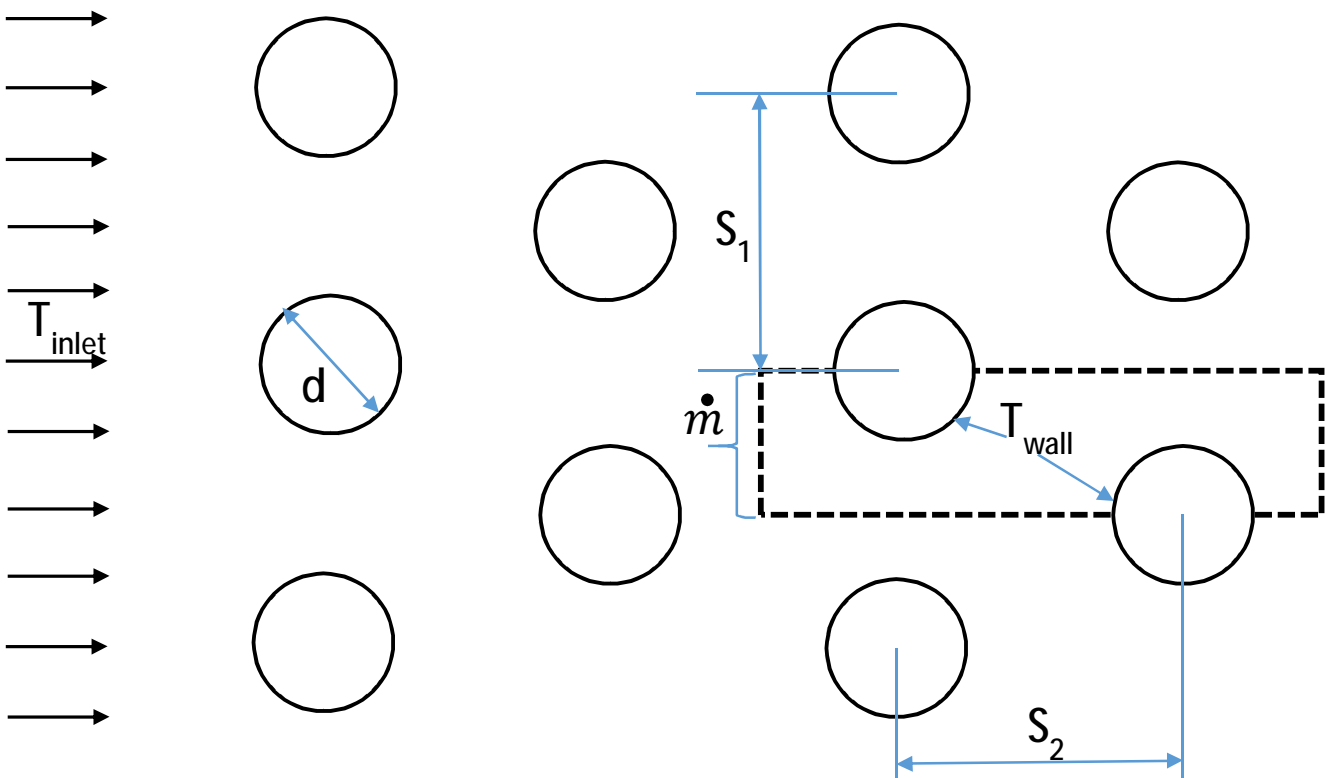


Fig. 1. 2D section of a tube bank

The problem to be solved is that of turbulent flow and heat transfer in a tube bank.

The inlet air mass flow rate is in the range of 0.08 to 0.8 kg/s. Three cases corresponding to different values of flow rate should be simulated. The values of mass flow rate are specified by teacher.

The temperature of the tube wall (T_{wall}) is equal to 400 K and the bulk temperature of the cross flow water (T_1) is 300 K.

An unstructured mesh should be created.

The mesh should have concentrated quadrilateral cells near wall surfaces to provide better resolution of the viscous gradients. After calculations the nondimensional distance to the wall should be controlled.

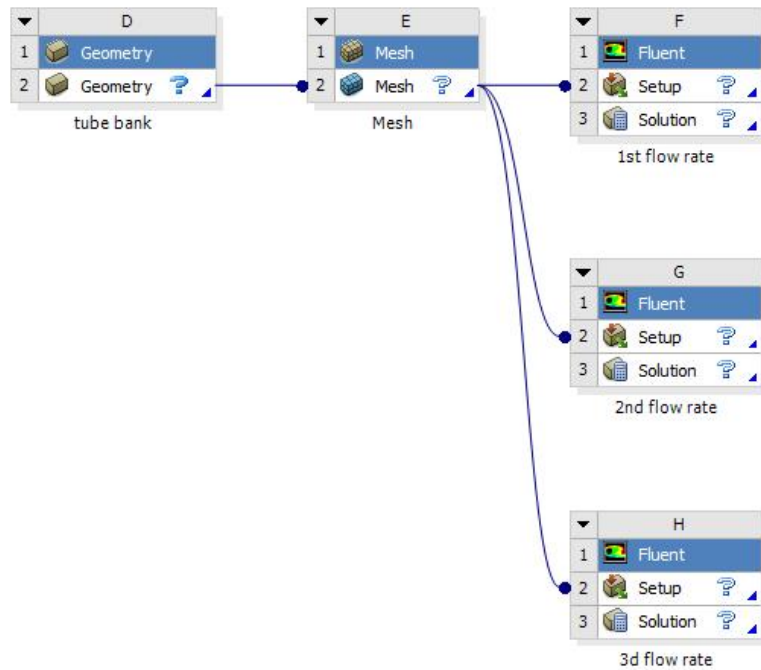
The mesh should be improved if necessary.

3. Project Schematic in Ansys Workbench

3.1. Start ANSYS Workbench

3.2. **Toolbox -> Component Systems.** Create a scheme of your project in Workbench Project Schematic Window.

Drag and drop **Geometry**, **Mesh** and **Fluent** components and create connections between components as per below.

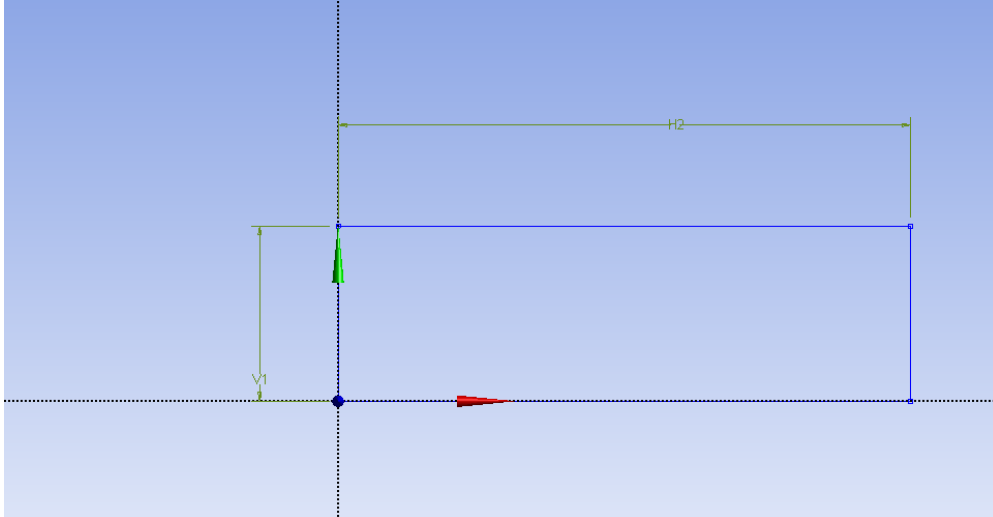


3.3. **File > Save As.** Save the workbench file.

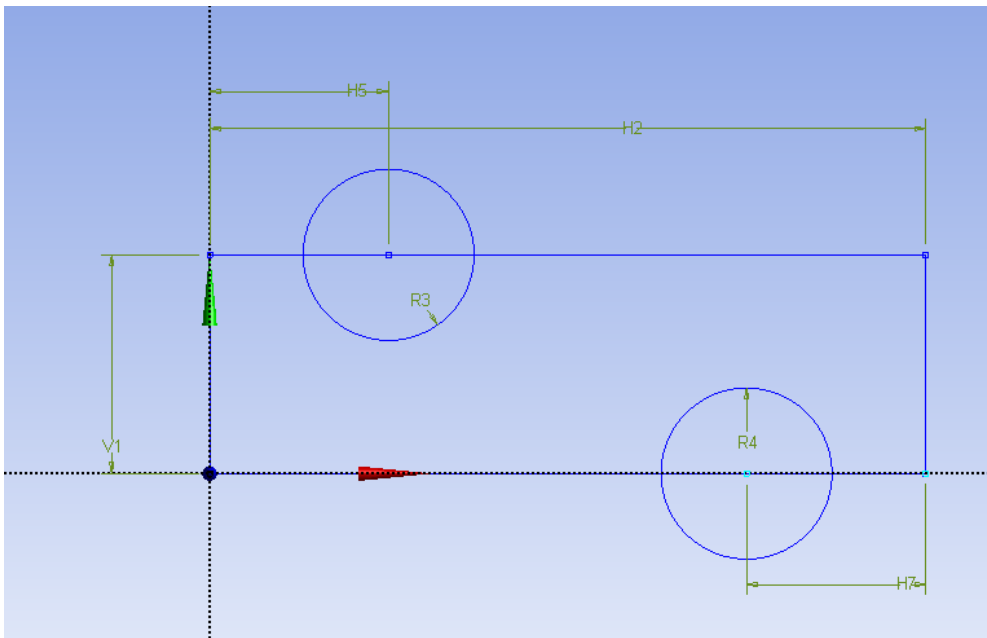
4. Geometry

4.1. Right click **Geometry** and select **New DesignModelerGeometry....** Start **Design Modeler** to create the computational area.

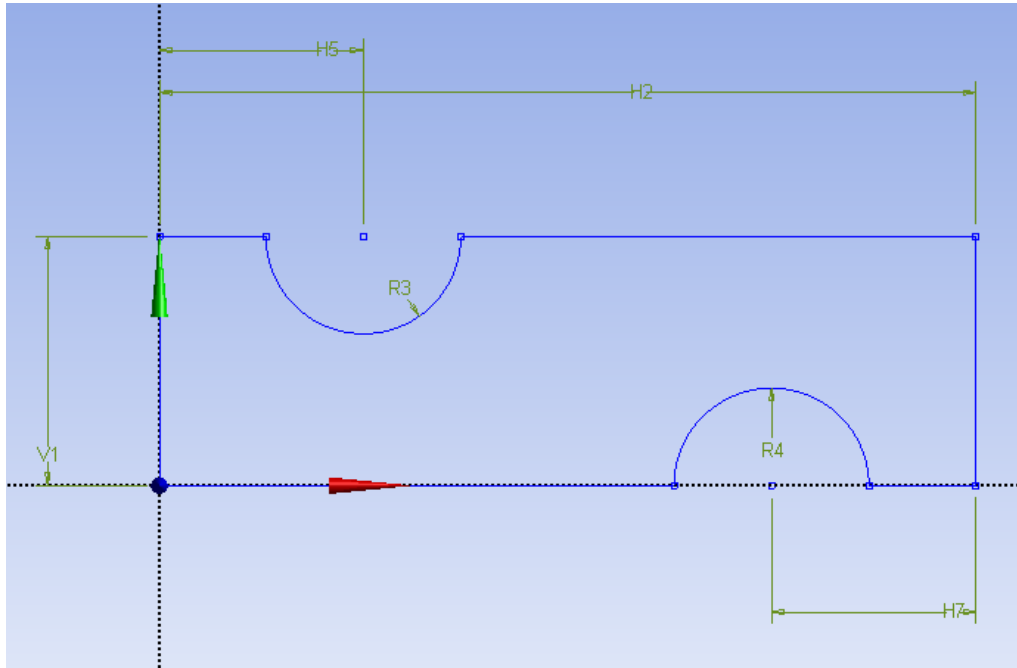
4.2. Create a rectangular area of given dimensions.



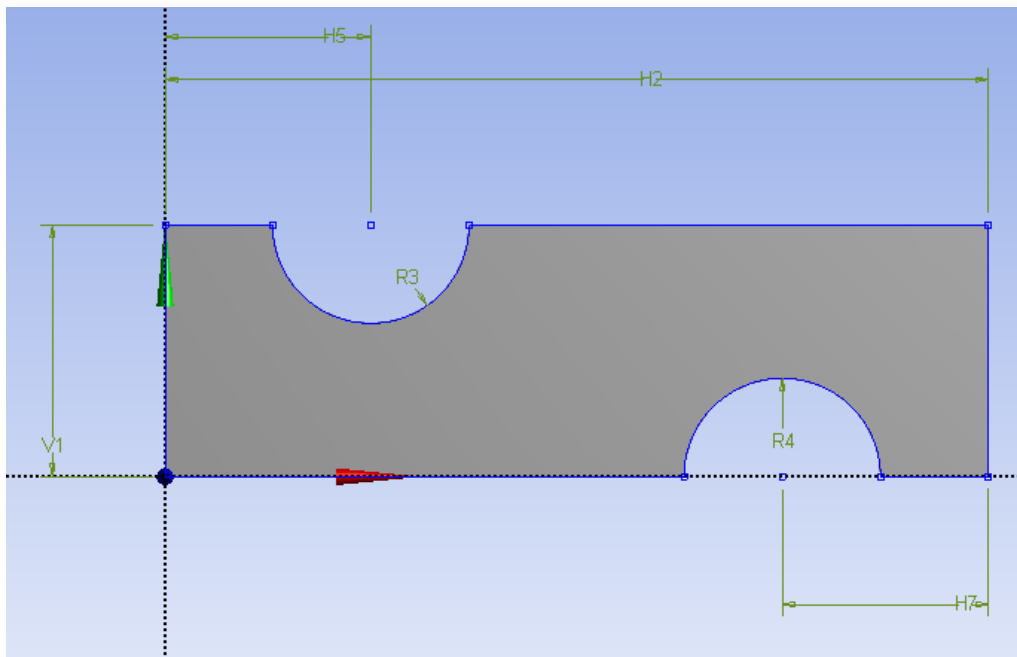
4.3. Add two circles of given radius.



4.4. Use **Modify>Trim** option to create a desired shape.



4.5. **Concept> Surfaces from Sketches**. Choose the sketch. Click **Generate**.



Comment

*The Sketch of the area can be also in a different way. You can use **arc** and **line** tools. The third way is to use circle and rectangular tools and **Boolean>Subtract** Option.*

4.6. Close **Design Modeler**.

5. Meshing

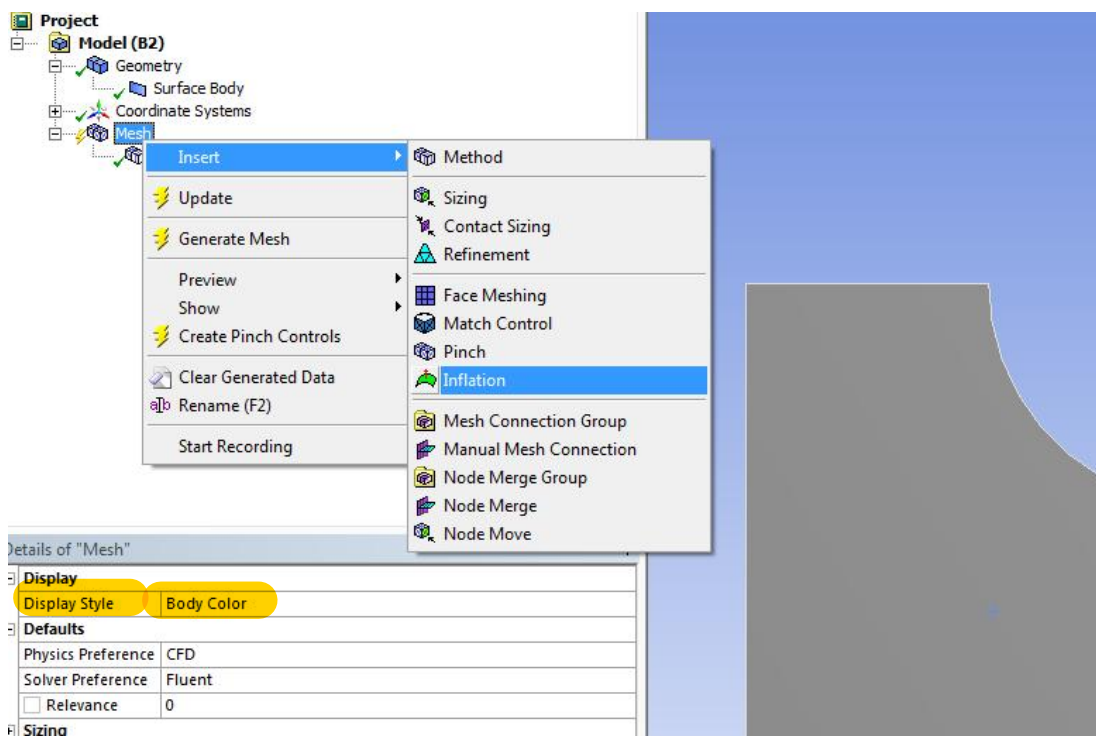
A nonuniform unstructured mesh using **Meshing** software should be created.

The grid nodes should be concentrated near the tube walls. A uniform distribution of grid nodes around the cylinder is recommended. *Quadrilateral cells can be used in the regions surrounding the tube walls and triangular cells can be used for the rest of the domain, resulting in a hybrid mesh. The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain can be filled with triangular cells for the sake of convenience.*

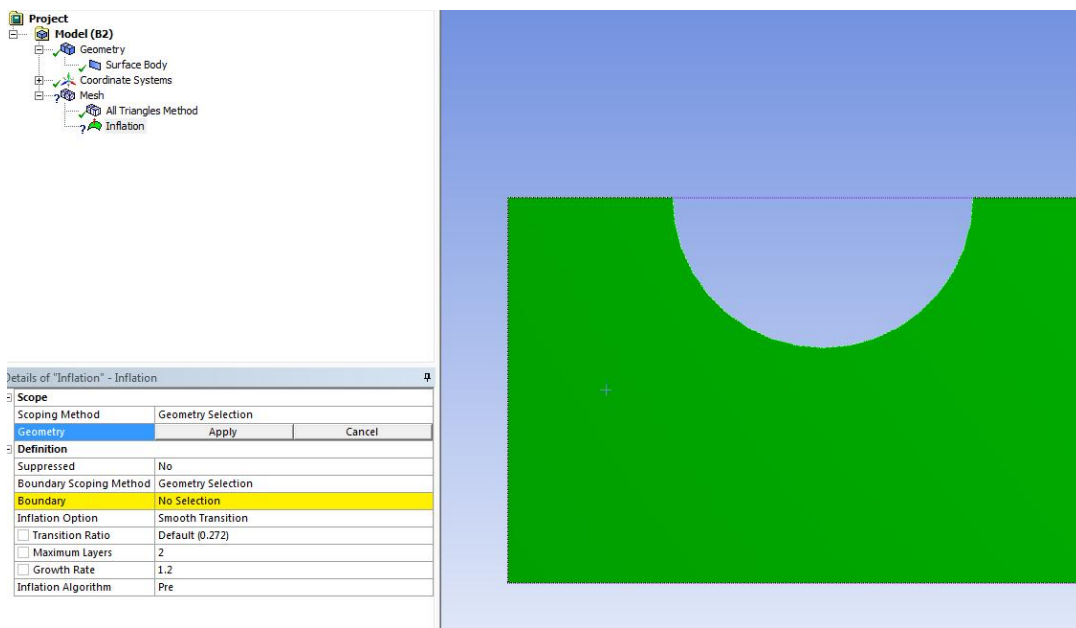
5.1. Right click on **Mesh** and select **Edit**. Start **Meshing**.

5.2. Details of “Mesh”-> **Physic Reference->CFD**

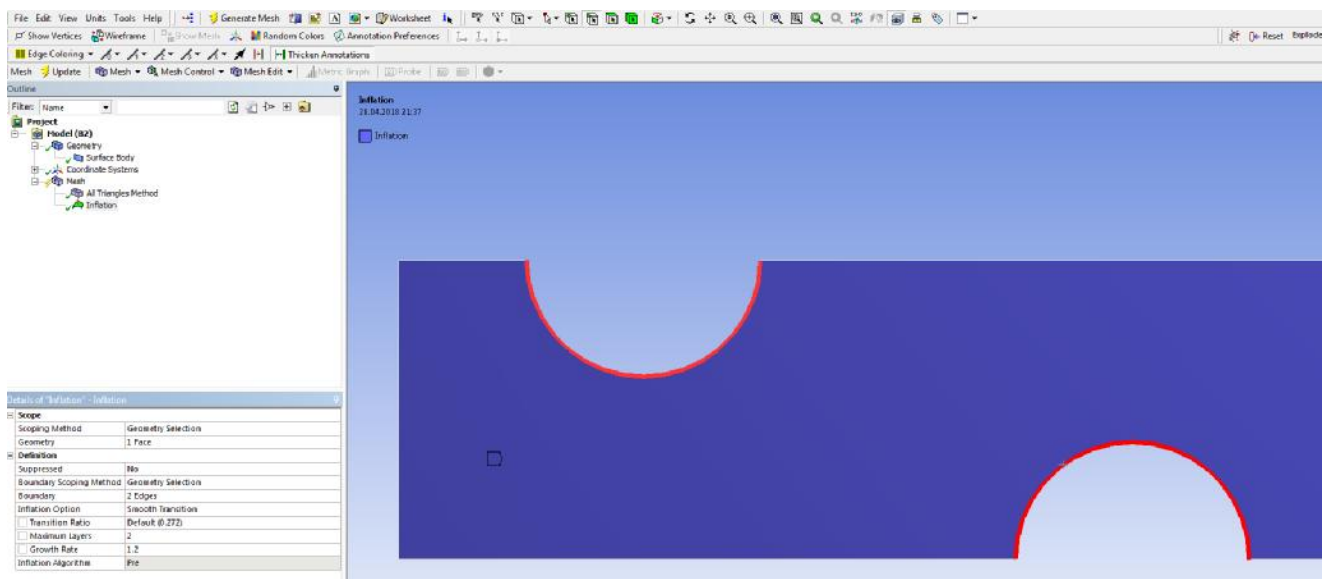
5.3. **Mesh > Insert > Inflation**. Use Inflation Tool to create quadrilateral cells near the walls.



5.4. Detail of Inflation>Geometry. Select the computational Domain as a **Target Body**:



5.5. Detail of Inflation>Boundary. Select boundaries (tube walls) to create quadrilateral cells near them:



5.6. Set the **Maximum Layers** equal to 15 (you can try to set different values, but not less than 10) in **Inflation**. This will generate quadrilateral cells in the regions surrounding the tube walls.

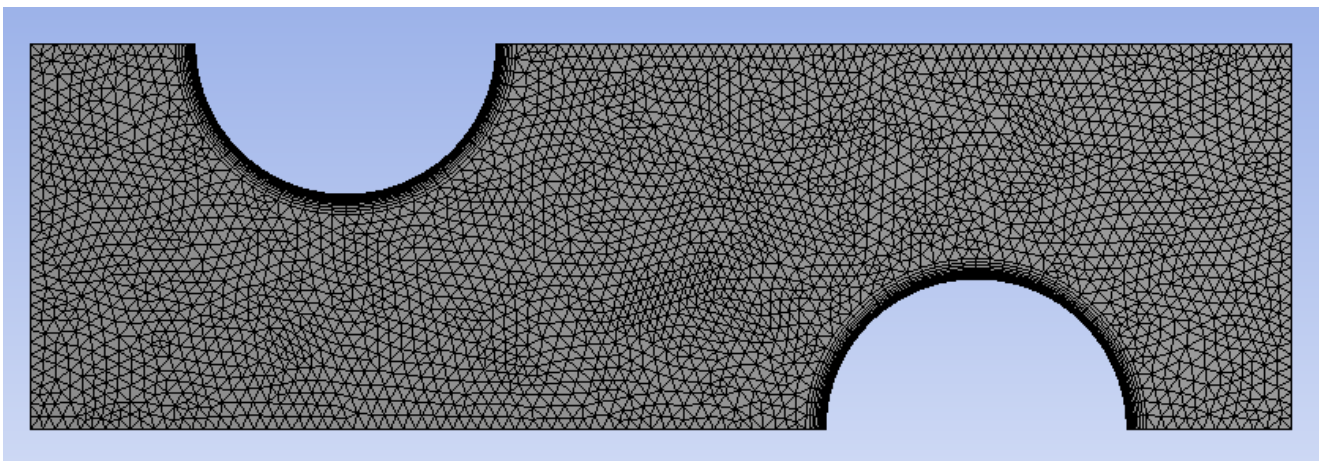
5.7. Insert the method of discretization for the rest of the domain. **Mesh > Insert > Method> Triangles**

5.8. Click **Generate**

5.9. Improve the grid quality. It is possible to use the following options to improve the grid (to decrease the cell sizes). Both are available in **Details of “Mesh”**.

- Increase **Relevance** value up to maximum possible.
- Decrease **Max Size** in **Sizing**. Try different values of Sizing

Details of "Mesh"	
[-] Display	
Display Style	Body Color
[-] Defaults	
Physics Preference	CFD
Solver Preference	Fluent
<input type="checkbox"/> Relevance	100
[-] Sizing	
Use Advanced Si...	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Span Angle Center	Fine
<input type="checkbox"/> Curvature Nor...	Default (12.0 °)
<input type="checkbox"/> Min Size	Default (1.3266e-005 m)
<input type="checkbox"/> Max Face Size	2.e-004 m
<input type="checkbox"/> Max Size	2.e-004 m
<input type="checkbox"/> Growth Rate	Default (1.10)
Minimum Edge L...	5.3e-003 m
[+] Inflation	
[+] Assembly Meshing	
[+] Patch Conforming Options	
[+] Patch Independent Options	
[+] Advanced	



5.10. Create the following **Named Selections**: “inlet”, “outlet”, “wall” and “symmetry”. The inlet and outlet boundaries will be redefined as periodic using the text user interface later (in **Fluent**).

5.11. Close **Meshing**.

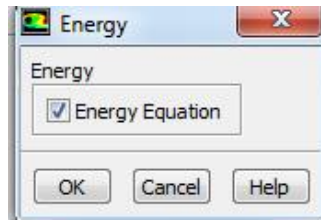
5.12. **File > Save Project**. Save the project and close the window. **Update Mesh** on Workbench if necessary.

6. Solving in Fluent

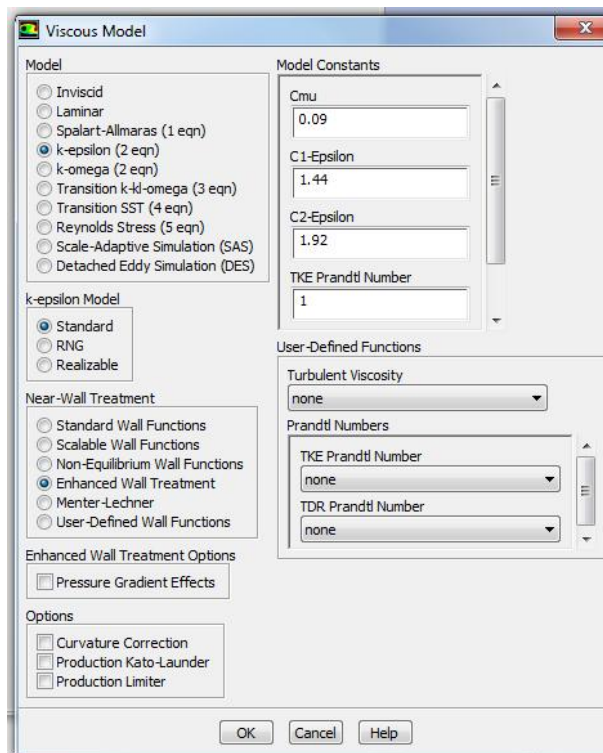
6.1. Run **Fluent**.

Be sure that the following options are chosen in **Setup**.

6.2. **Energy > On**



6.3. **Models->Viscous Model->k-epsilon with Enhanced Wall Treatment**



Boundary Conditions:

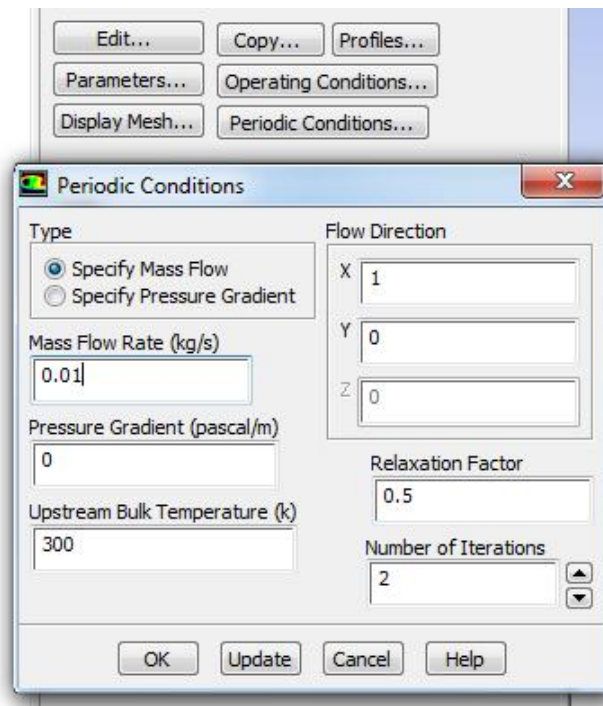
Create the **periodic zone**.

6.4. In **Fluent Setup** the inlet boundary will be redefined as a **translationally periodic zone** and outlet as a **periodic shadow** of inlet.

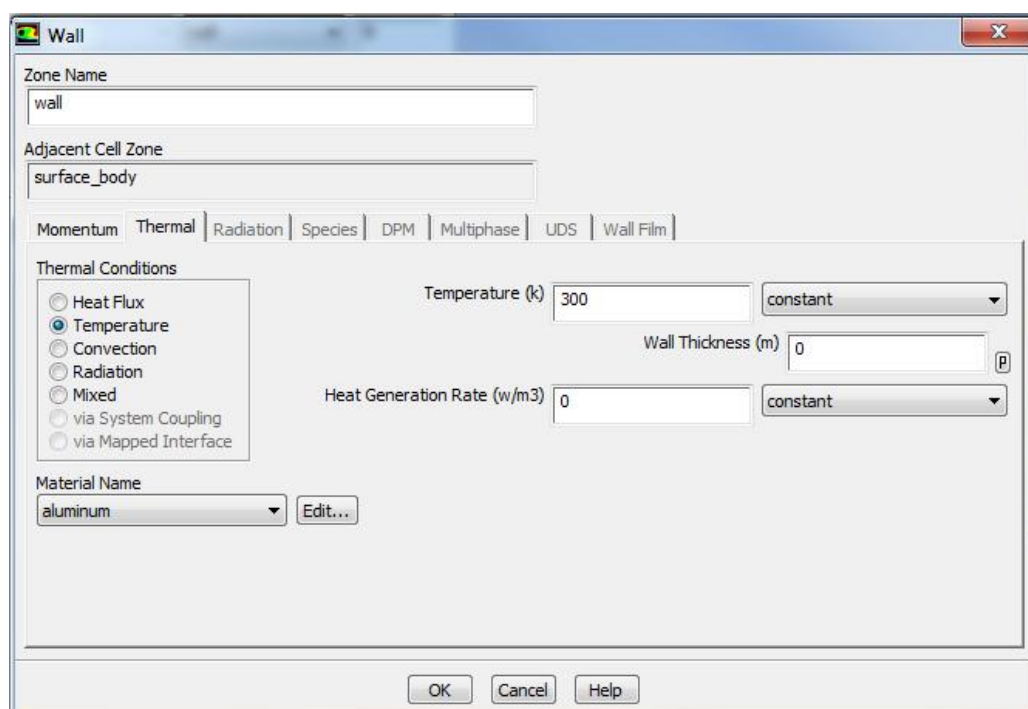
- Press <Enter> in the console to get the command prompt (>).
- Enter the text command and input responses outlined in boxes as shown:

```
> mesh/modify-zones/make-periodic
Periodic zone [()] inlet
Shadow zone [()] outlet
Rotational periodic? (if no, translational) [yes] no
Create periodic zones? [yes] yes
Auto detect translation vector? [yes] yes
```

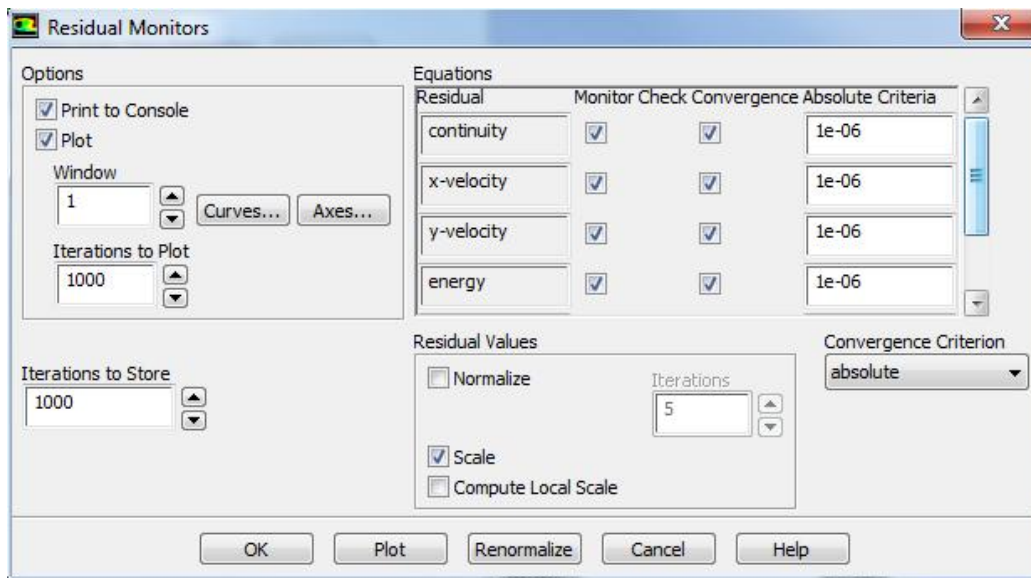
6.5. **Boundary Conditions>Periodic Conditions.** Define the mass flow rate for periodic conditions



6.6. **Boundary Conditions>wall.** Set the wall temperature equal to 400 K.



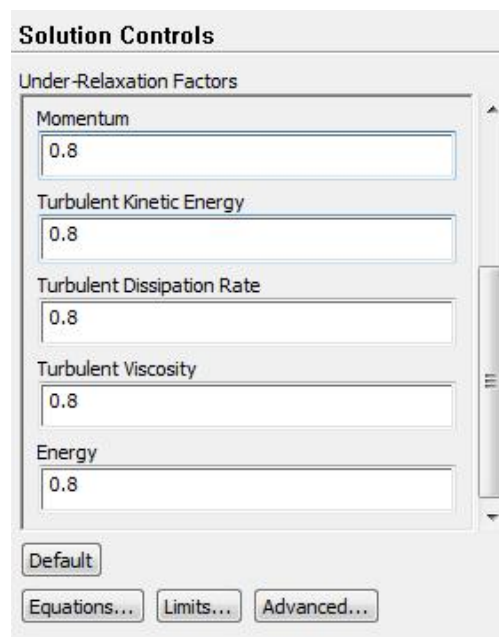
6.7. **Solution > Monitors > Residual.** Decrease the Residuals Criteria to $1e-6$ for all equations.



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

Comment

*In case of bad convergence decrease the **Under-Relaxation Factors**.*

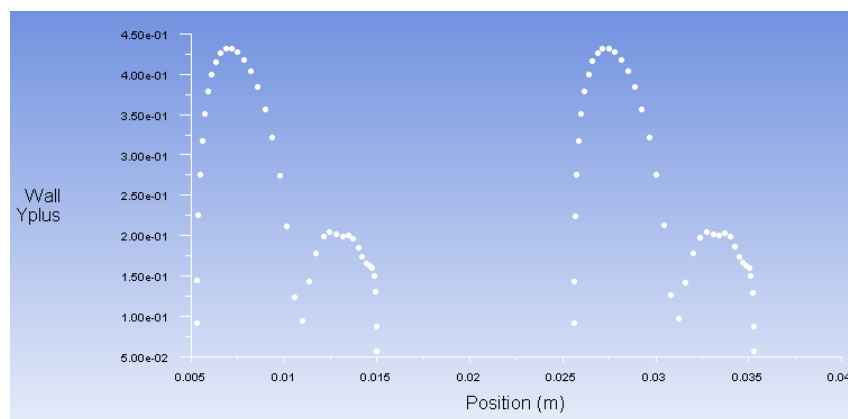
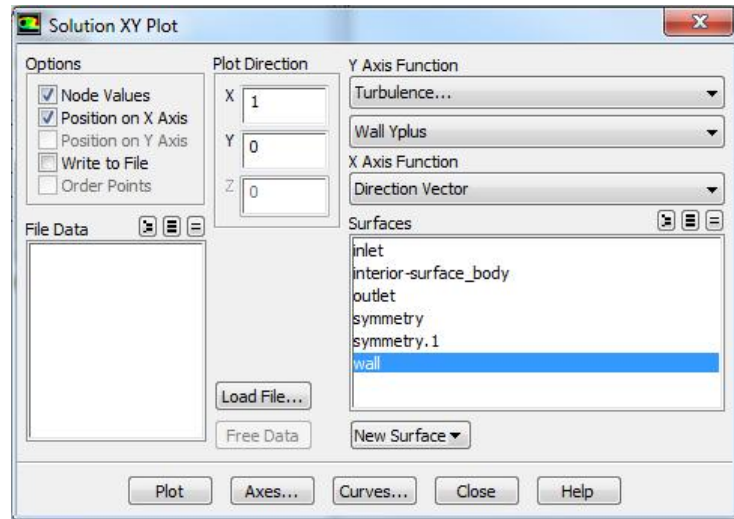


7. Results post-processing

! Before proceeding with Post-processing check the value of Y^+ . It should be less than 1. This condition corresponds to Enhanced Wall Treatment Option.

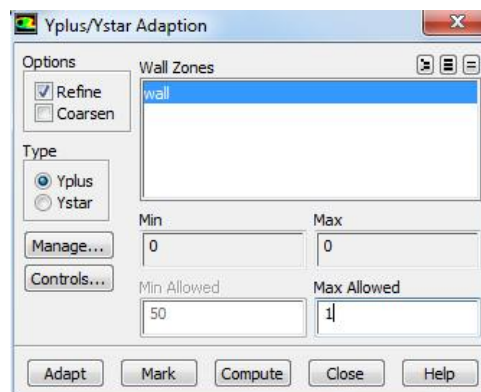
To check the Y^+ value do the following

7.1.1. Results> Plots > XY Plot. Choose the options as suggested below.



If the value of Y^+ is much more than 1

- You can go back and improve the grid. You should create smaller cells near the walls. For example, you can increase the number of layers in **Inflation**.
- You can also go to **Adapt>Yplus/Ystar** and change the options as below.



Pictures to be created

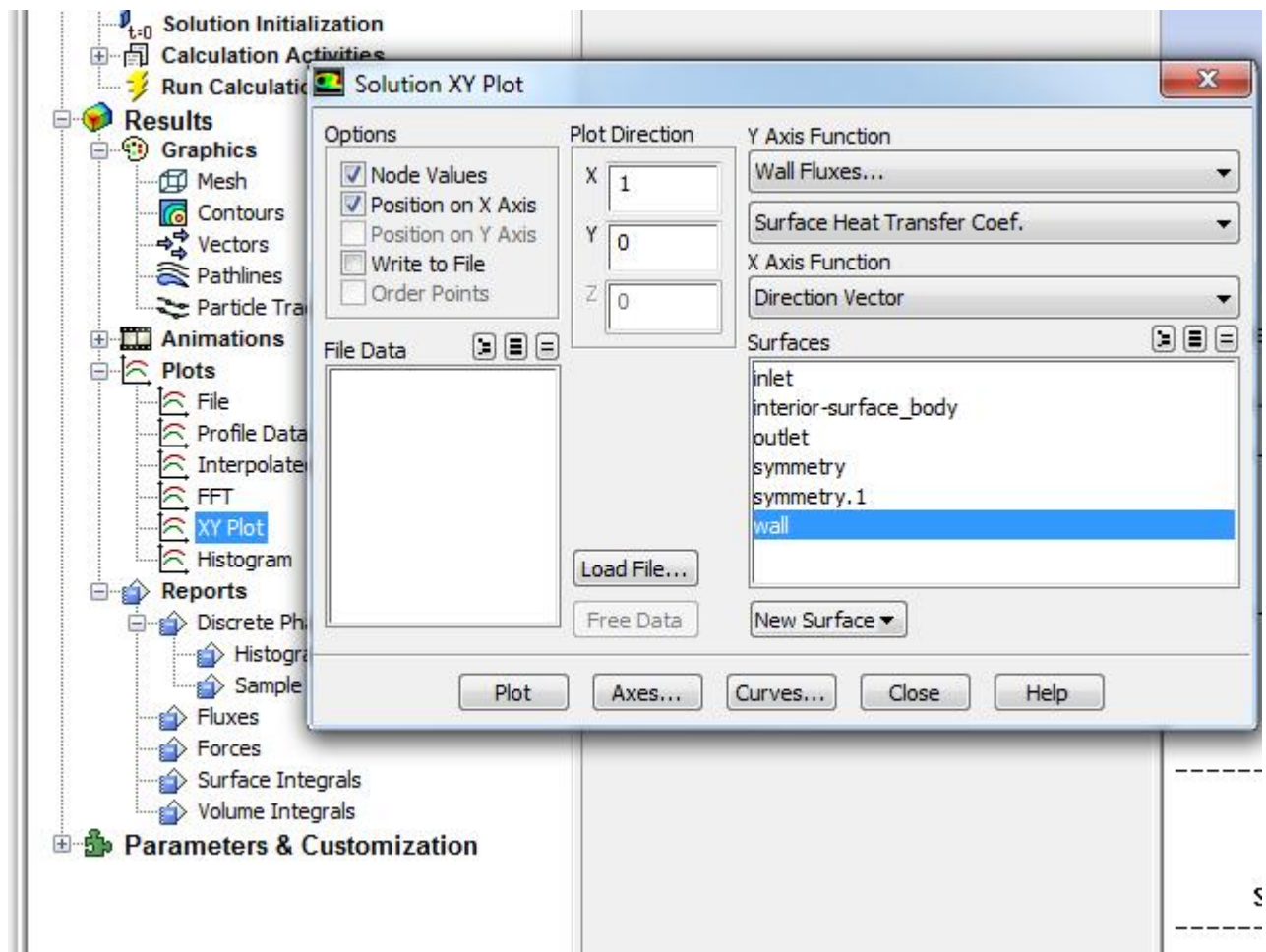
- Temperature fields
- Velocity vector fields
- Pressure fields
- Stream function fields
- Local heat transfer distribution
- Y+ distribution

Data to be calculated

- Average Nusselt number
- Average heat transfer coefficient

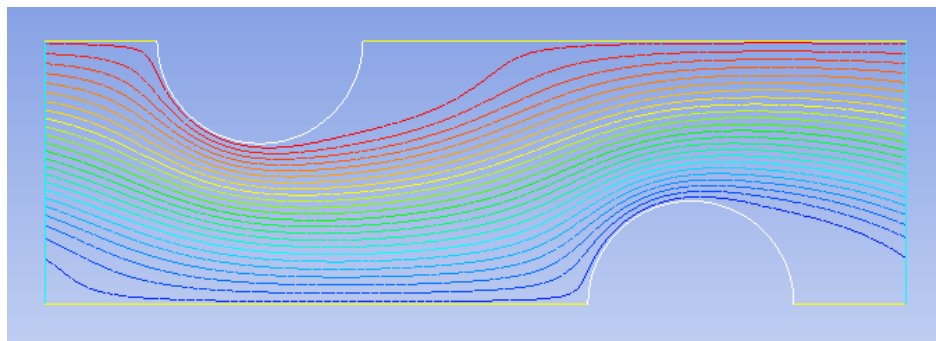
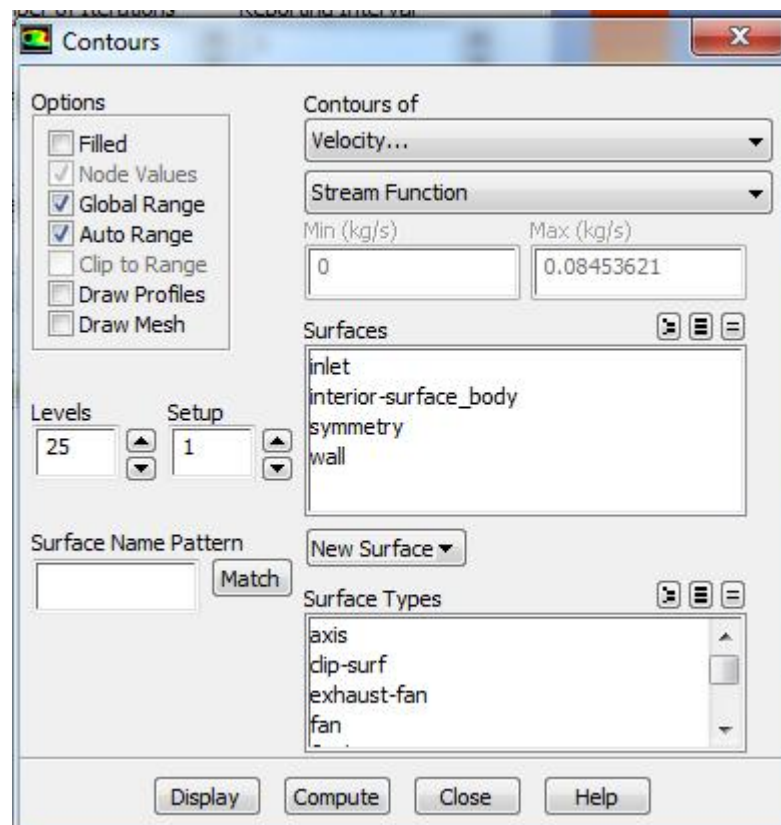
Plot a graph of local heat transfer coefficient

7.1.2. Results> Plots > XY Plot. Choose the options as suggested below.



Plot stream lines

7.1.3. Results> Graphics > Contours. Choose the options as suggested below.



7.2. Equations

7.2.1. Reynolds number and Nusselt number are defined as following:

$$Re = \frac{u \cdot d}{\nu}$$
$$Nu = \frac{\alpha d}{\lambda}$$

Here α – averaged surface heat transfer coefficient.

7.2.2. Velocity **corresponding to Fluent simulation** can be calculated using a following formula:

$$\dot{m} = u \cdot \rho \cdot 0,5 \cdot S_1$$

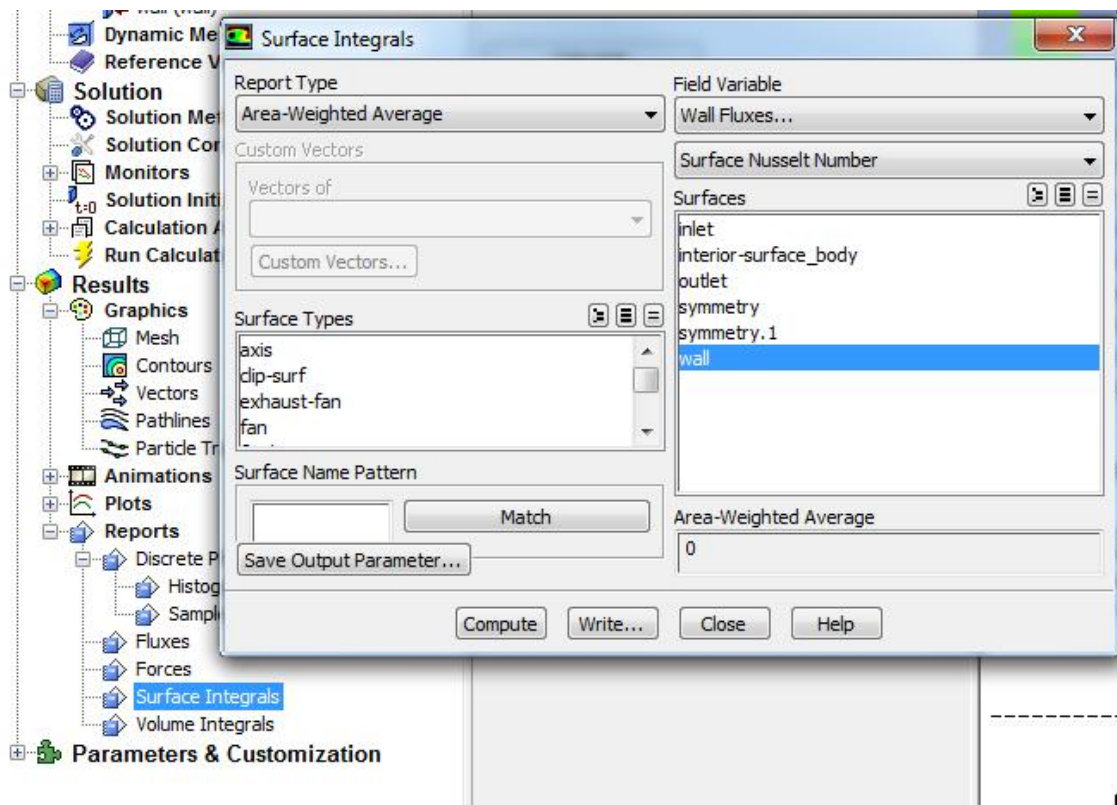
7.2.3. **Results>Reports>Surface Integrals>Wall Fluxes> Surface Nusselt number.** Click **Compute**. The Nusselt number calculated by Fluent will appear in **Console**.

Check that in **Reference Values** the following values are chosen:

Length = tube diameter

Temperature = 300 K

These will provide a correct Nusselt calculations.



7.2.4. Alternatively, the Nusselt number can be calculated by using the following **empirical equations** for heat exchangers:

$$Nu = 0.36 \cdot Re^{0.6} \cdot Pr^{0.33} \cdot C_s,$$

$$\text{where } C_s = \left(\frac{\sigma_1 - 1}{\sigma_2 - 1} \right)^{0.1}, \quad \sigma_1 = \frac{s_1}{d}, \quad \sigma_2 = \frac{s_2}{d}, \quad \sigma_2 = \sqrt{\frac{\sigma_1^2}{4} + \sigma_2^2}$$

C_s is used to take into account the influence of tube bank geometry.

Pr is a Prandtl number. It characterizes the fluid properties.

Fill the following tables after calculations

\dot{m} , kg/s	U, m/s	Re	Pr	Nu_{calc}	Nu_{fluent}	ΔNu , %