Chapter 3 Analysis of the Flow past a Sphere and a Cylinder

Objectives

- Creating the sphere and cylinder needed for the SOLIDWORKS Flow Simulation
- Setting up Flow Simulation projects for external flow
- Running the calculations
- Using XY-Plots and Cut Plots to visualize the resulting flow fields
- Study values of surface parameters
- Cloning of the project
- Run time-dependent calculations to determine the vortex shedding frequency and the Strouhal number for the cylinder
- Compare with empirical results

Problem Description

In this chapter, we will use Flow Simulation to study the three-dimensional flow of air past a sphere with a diameter of 50 mm at different Reynolds numbers and compare with empirical results for the drag coefficient. The second part of this chapter covers the two-dimensional flow around a cylinder and we will determine the Strouhal number related to vortex shedding from the cylinder. We will start by creating the sphere needed for this simulation; see figure 3.1.

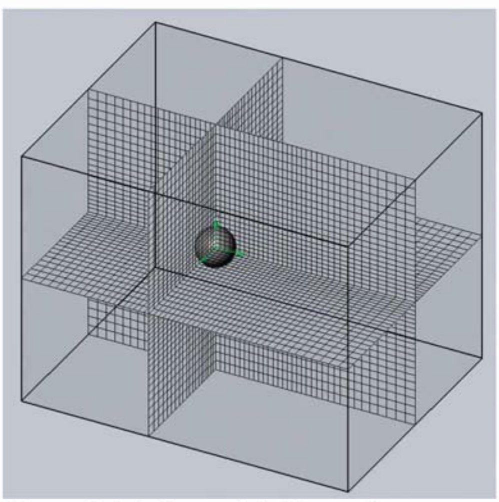

Figure 3.1 Sphere with 3D mesh

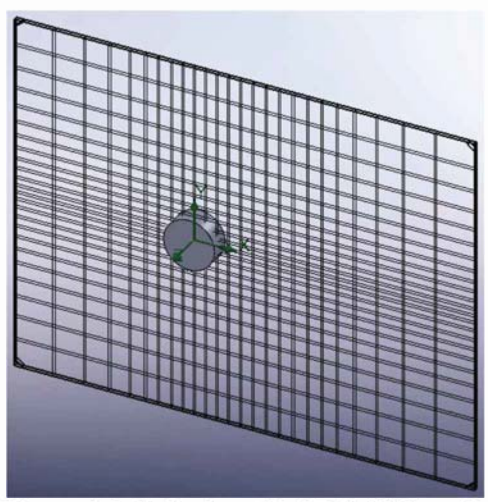

Figure 3.2 Cylinder with 2D initial mesh

Creating the SOLIDWORKS Part for the Sphere

In this exercise we will analyze the flow around a sphere. First, we have to create a model of the sphere in SOLIDWORKS and export the part to Flow Simulation. Follow these steps to create a solid model of a sphere with 50mm diameter and perform a 3D simulation of the flow field.

1. Start SOLIDWORKS and create a **New Document**.

Select **File>>New…** from the SOLIDWORKS menu.

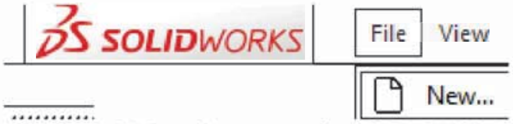

Figure 3.3 New document in SOLIDWORKS

2. Select **Part** in the **Welcome** window.

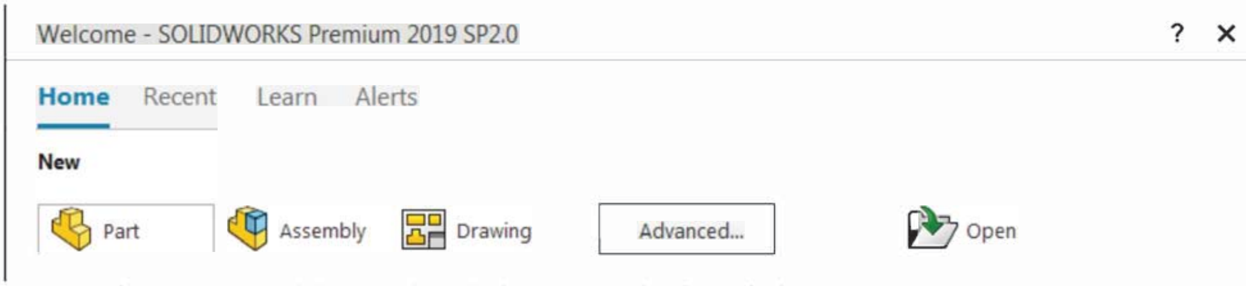

Figure 3.4 New SOLIDWORKS document selection window

In order to make the sphere, we will sketch a half circle in the Front Plane and revolve it. We start this process by making a new sketch.

3. Click on the **Front Plane** to select **Normal To**.

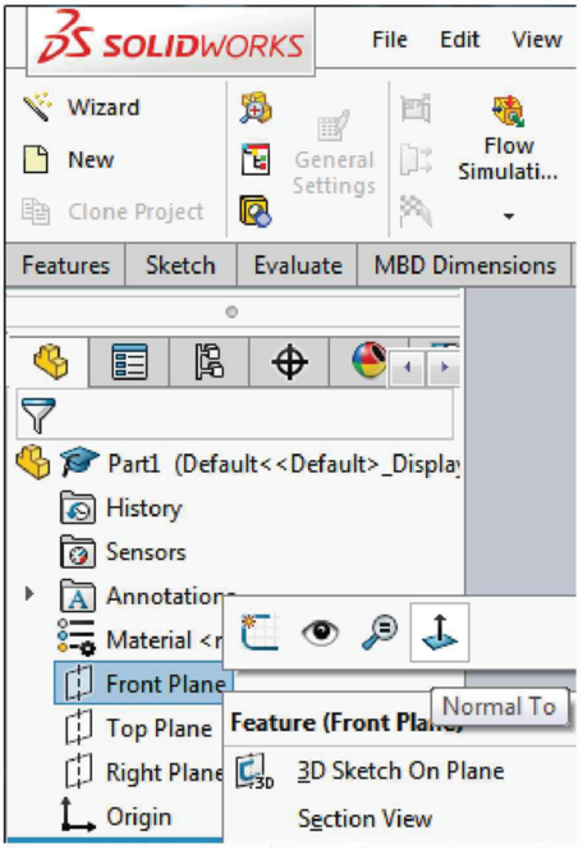

Figure 3.5 Selection of front plane and normal to

We start sketching by drawing a vertical symmetry line in the sketch plane. This centerline will be used to create the sphere as a revolved feature.

4. Select the **Sketch** tab and **Line>>Centerline…**

Figure 3.6 Selection of the centerline sketch tool

5. Next, draw the vertical centerline in the sketch window. Start above the vertical coordinate axis and make sure that you get the blue dashed helpline; see figure 3.7a). Click and draw the line downward through the origin and end the line approximately the same distance below the origin as shown in figure 3.7b). Right click anywhere in the graphics window and click on **Select**. You have now finished the vertical centerline.

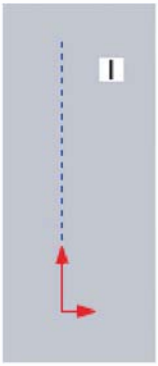

Figure 3.7a) Vertical dashed helpline

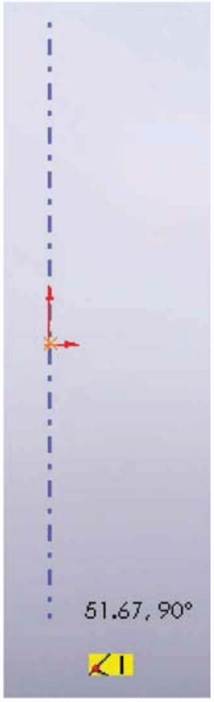

Figure 3.7b) Drawing a vertical centerline

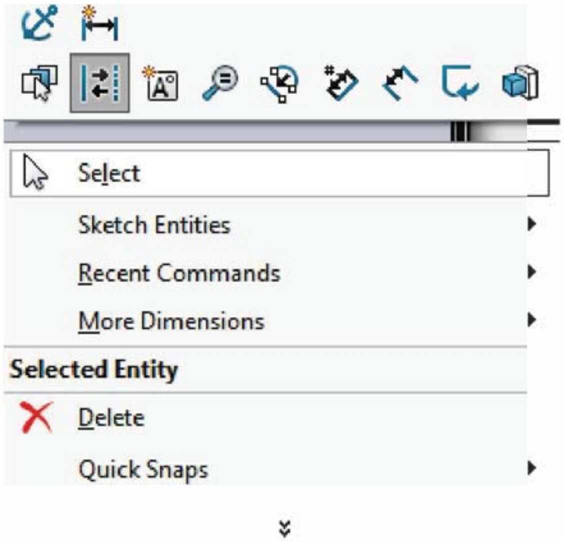

Figure 3.7c) Clicking on select

6. Select the **Centerpoint Arc** and draw the half circle. First, click on the origin. You should see an orange filled circle indicating that you are at the origin; see figure 3.8b). Next, click on the centerline above the origin of the coordinate system and draw the half circle and click on the centerline once again but this time below the origin. Right click anywhere in the graphics window and click on **Select**. Select **Tools>>Options** from the SOLIDWORKS menu and click on the **Document Properties** tab. Click on **<u>Units</u>** and select **MMGS** (millimeter, gram, second) as Unit system. Click OK to close the window.

Figure 3.8a) Selecting centerline arc

Figure 3.8b) Starting at the origin…

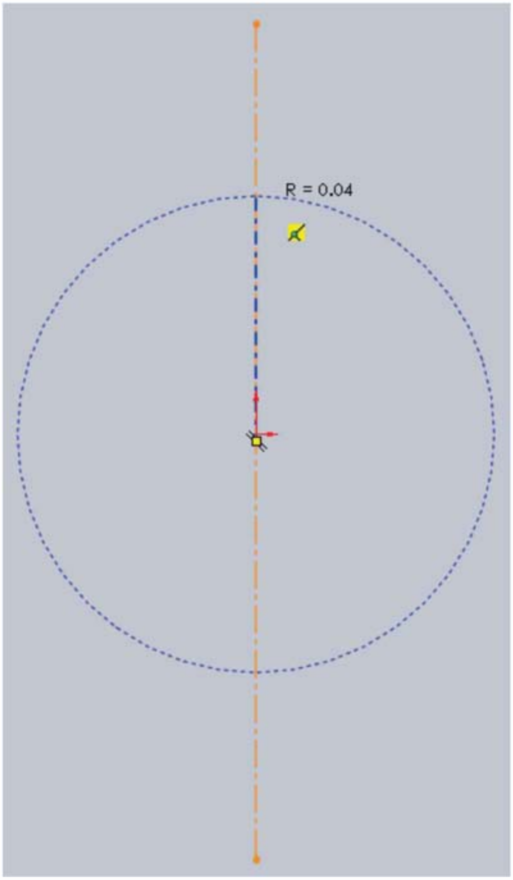

Figure 3.8c) Click above the origin…

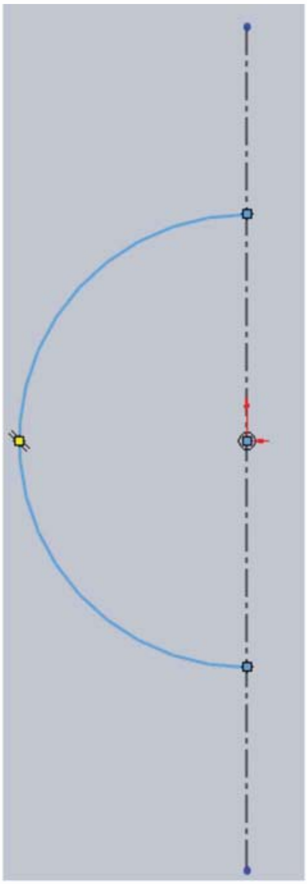

Figure 3.8d) Finished half circle

7. Next, select the **Smart Dimension** tool

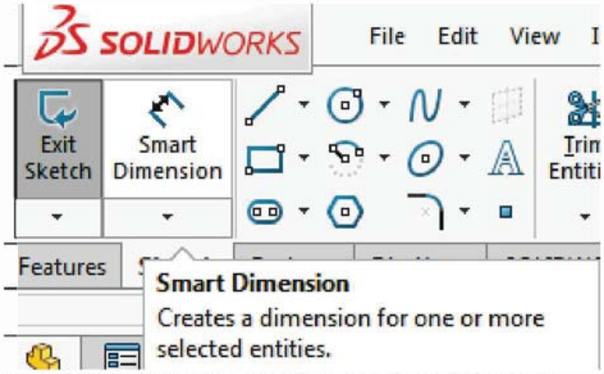

Figure 3.9 Selecting the Smart Dimension tool

8. Create a radius of 25.00 mm for the half-circle by clicking on the half-circle twice and enter the numerical value in the **Modify** window. Save the value and exit the dialog.

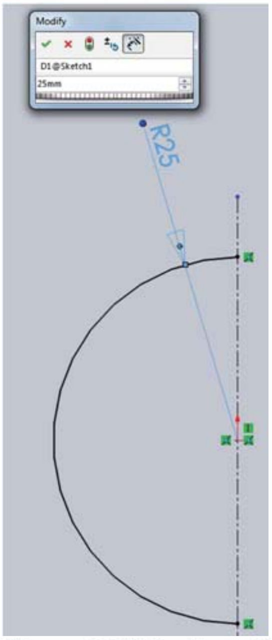

Figure 3.10 Radius of 25.00 mm for the half-circle

The next step is to make the sphere by using the revolve feature in SOLIDWORKS.

9. Click on the Features tab and Select the **Revolved Boss/Base** icon.

Figure 3.11a) Selection of the Revolved Boss-Base feature

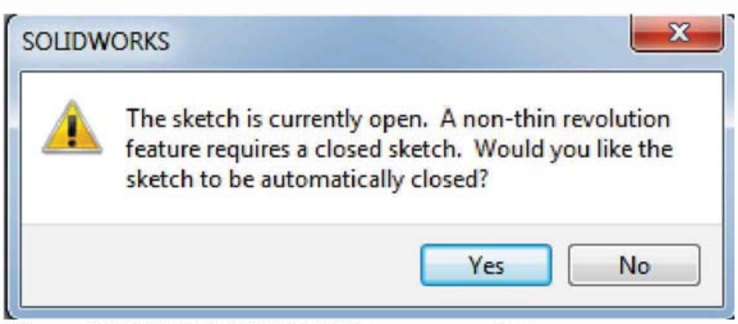

Figure 3.11b) SOLIDWORKS message window

You will get a message that the sketch is currently open and a question if you would like the sketch to be automatically closed. Choose the **Yes** button.

10. Use the default **Revolve Parameters**: Line 1, Blind Direction and 360 degrees.
Click on the OK button with the green symbol ✔ to exit the **Revolve** window.

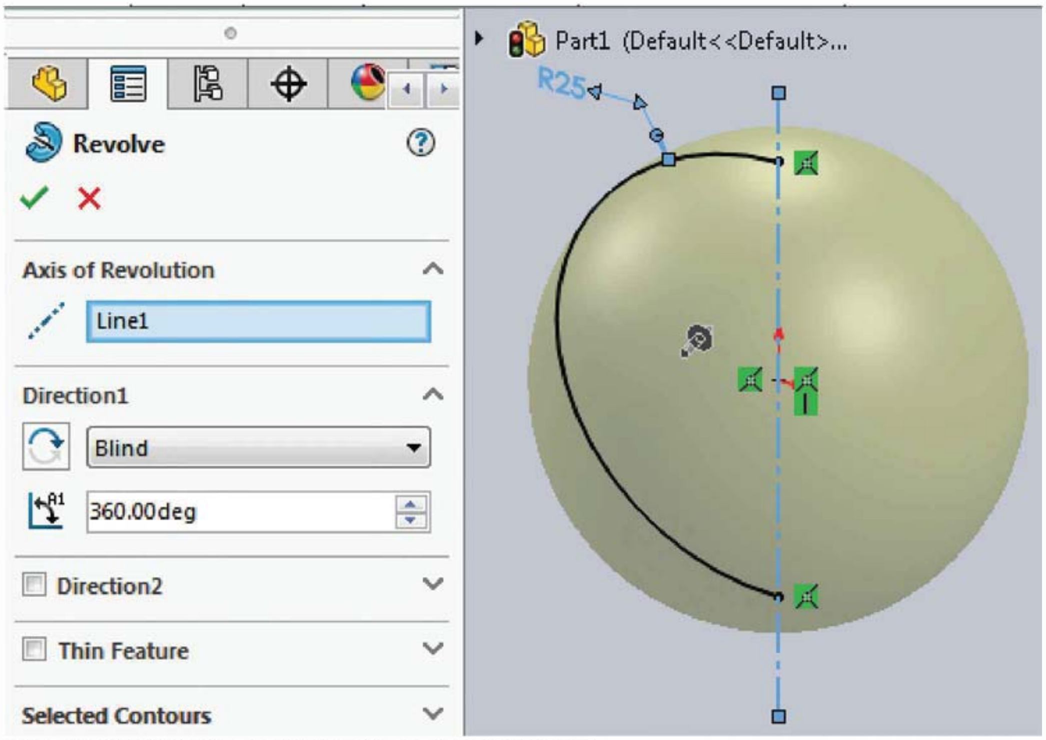

Figure 3.12 Selection of default revolve parameters

11. Move the cursor to the **File** menu and select **Save As…**
Enter **Sphere 2019** as File name and click on the **Save** button.

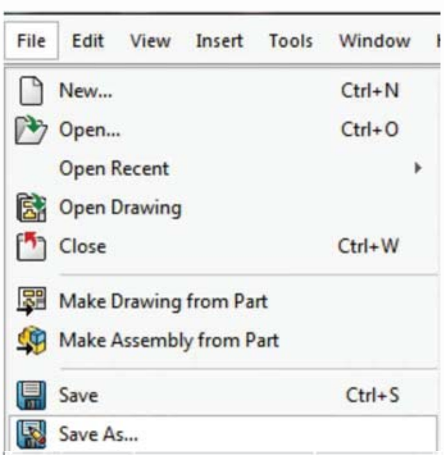

Figure 3.13a) Save the solid model

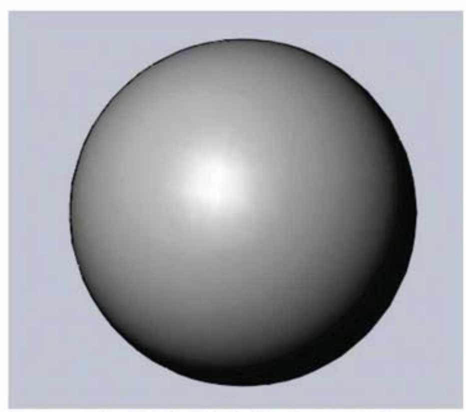

Figure 3.13b) Finished model

12. Select **Tools>>Add-Ins…**from the menu and check the **SOLIDWORKS Flow Simulation 2019** box.

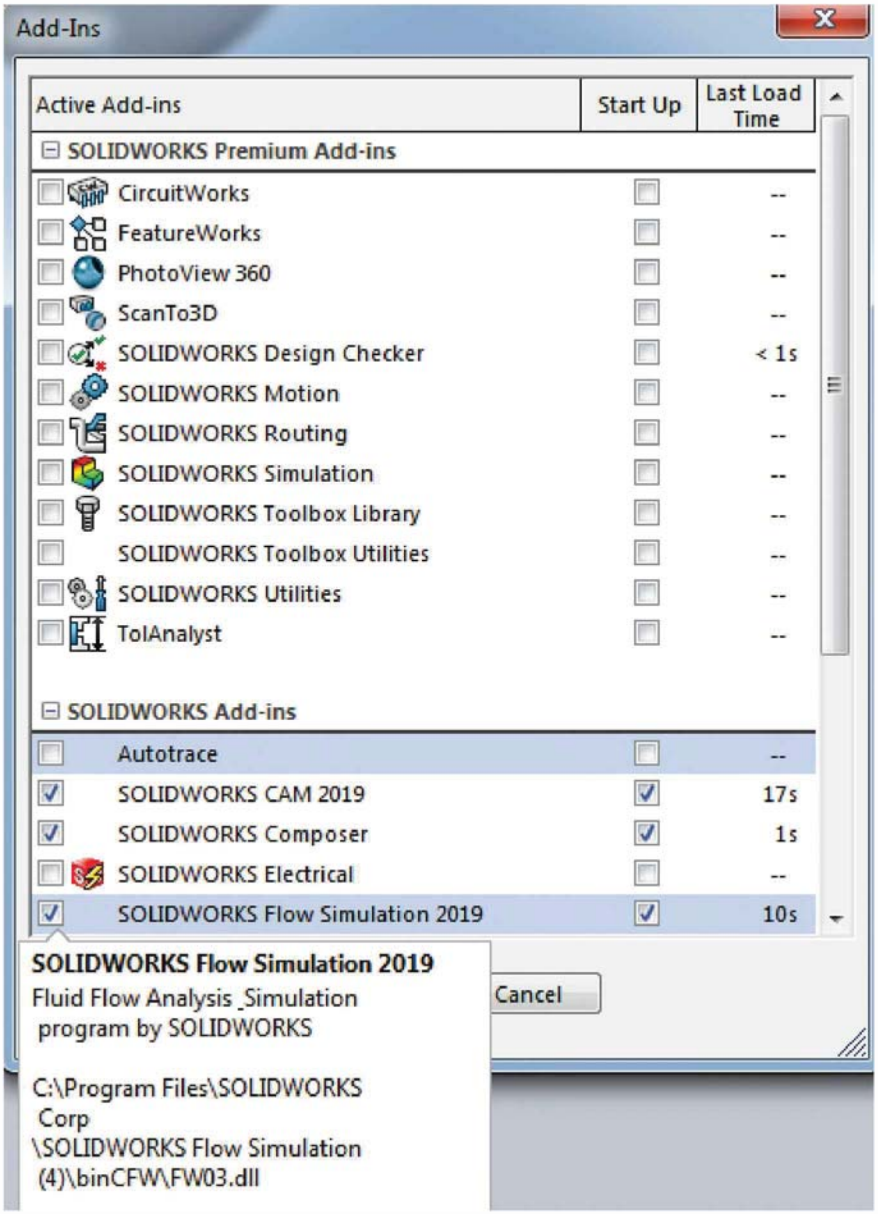

Figure 3.14 Adding Flow Simulation to the SOLIDWORKS menu

Setting up the Flow Simulation Project for the Sphere

13. We create a project by selecting **Tools>>Flow Simulation>>Project>>Wizard…** from the menu.

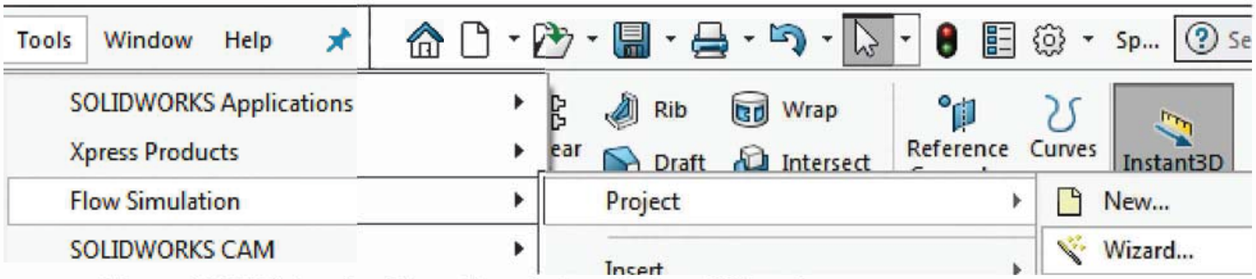

Figure 3.15 Using the Flow Simulation Project Wizard

14. Enter Project Name: "**Flow around a Sphere 2019**". Push the **Next>** button.

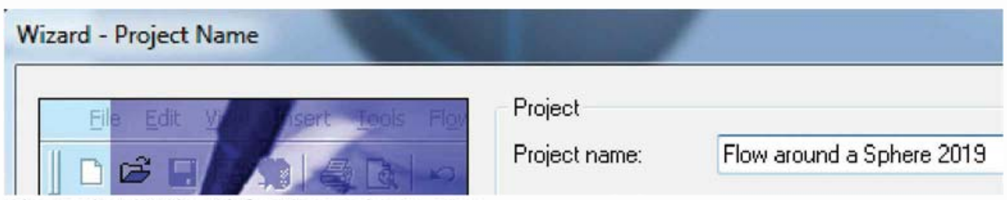

Figure 3.16 Wizard for the project name

15. We choose the **SI (m-kg-s)** unit system and click on the **Next>** button again.

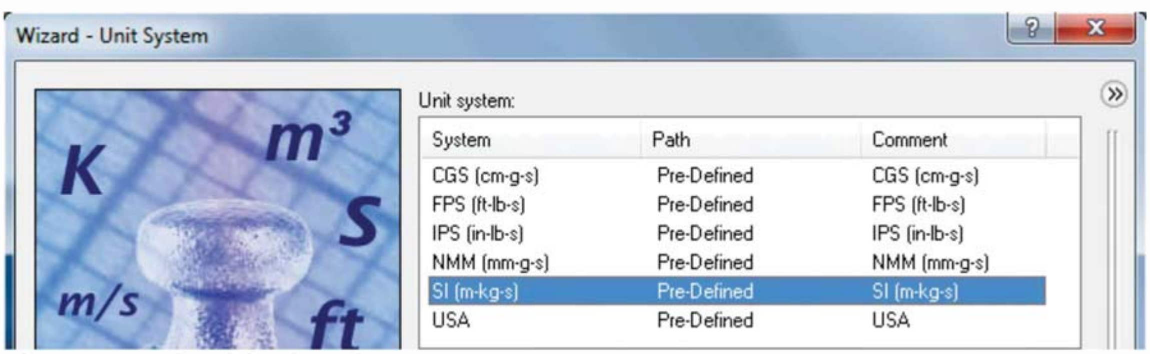

Figure 3.17 Wizard for the unit system

16. Check the **External** option for **Analysis type** and click the **Next>** button

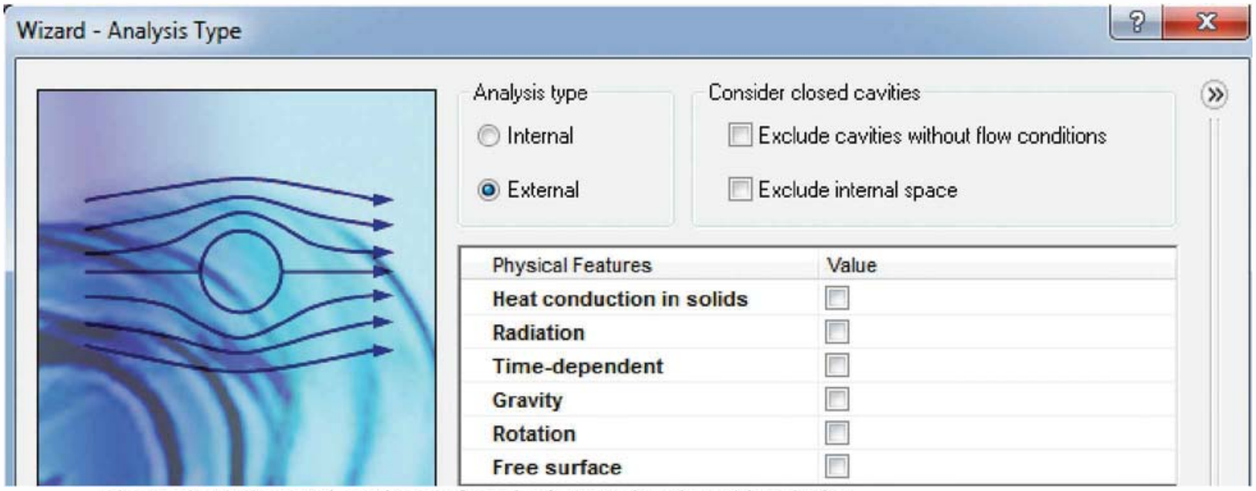

Figure 3.18 General settings of analysis type in Flow Simulation

17. Choose **Air** as the **Default Project Fluid** by clicking on the plus sign next to the **Gases** and selecting **Air**. Next, select the **Add** button. Click on the **Next>** button.

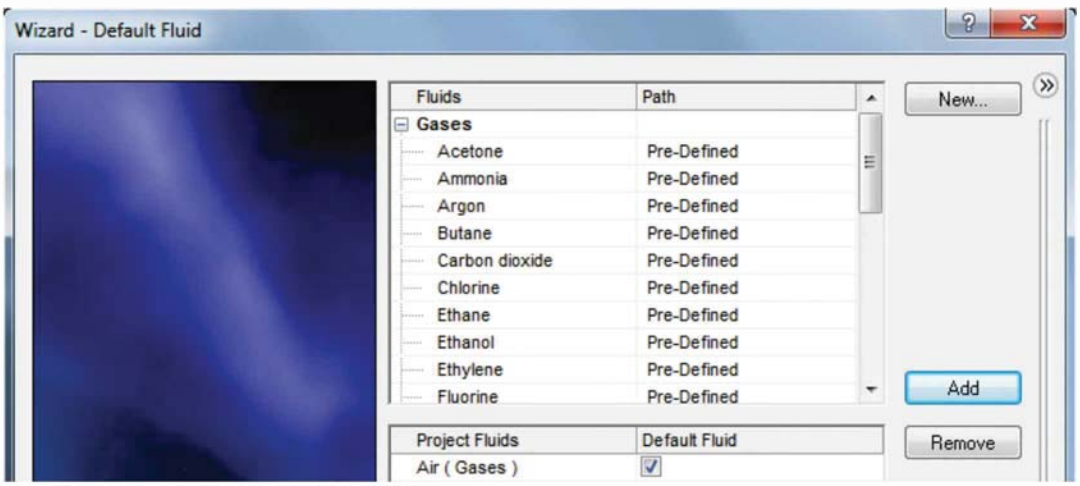

Figure 3.19 Selection of project fluid

The next part of the wizard is about **Wall Conditions**. We will use an **Adiabatic wall** for the sphere and use zero roughness on the surface of the same sphere. Next, we get the **Initial and Ambient Conditions** in the Wizard.

18. Click on the **Next>** button.

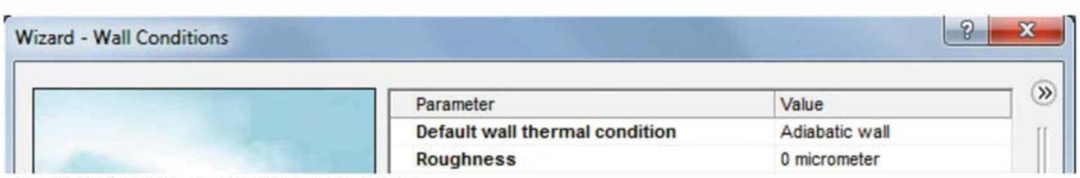

Figure 3.20 Wall conditions wizard

19. Enter **0.003 m/s** as the **Velocity in X-direction** and push the **Finish** button.

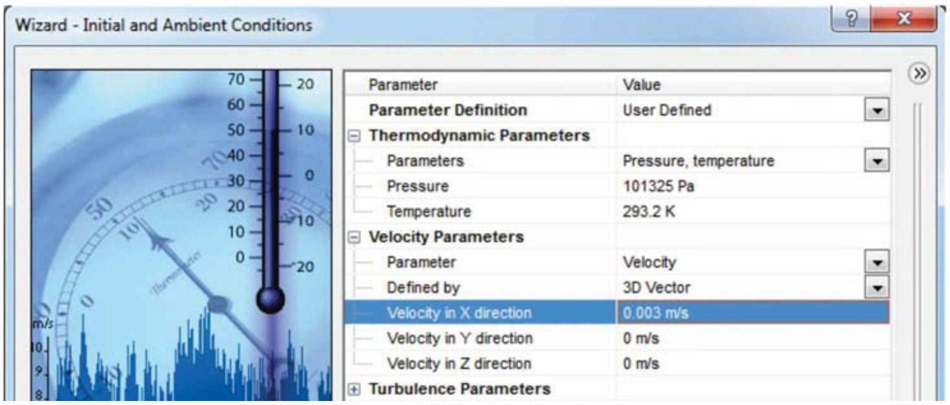

Figure 3.21 Initial and ambient conditions wizard

20. Select **Tools>>Flow Simulation>>Global Mesh** from the menu. Slide the **Level of Initial Mesh** to **4** and close the Dialog. Right click anywhere in the graphics window and select **Zoom In/Out** to see computational domain surrounding the sphere. Select **Tools>>Flow Simulation>>Project>>Show Basic Mesh** to see the mesh surrounding the sphere; see figure 3.1.

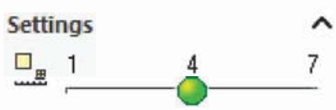

Figure 3.22a) Result resolution

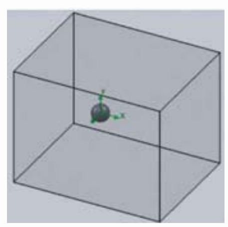
Figure 3.22b) Computational box around the sphere

Inserting Global Goal for Calculations

21. We create global goals for the project by selecting **Tools>>Flow Simulation>>Insert>>Global Goals…** from the SOLIDWORKS menu and check the box for **Force (X)**. Exit the global goals by clicking on OK ✔. Right click on **Goals** in the **Flow Simulation analysis tree** and select **Insert Equation Goal**. Select **GG Force (X) 1** from the **Flow Simulation analysis tree**. Enter the expression for the equation goal as shown in figure 3.23e). Select **Dimensionless LMA** from the **Dimensionality** drop down menu and enter the name **Drag Coefficient** for Equation Goal 1. Exit the **Equation Goal** window.

Figure 3.23a) Selection of global goals

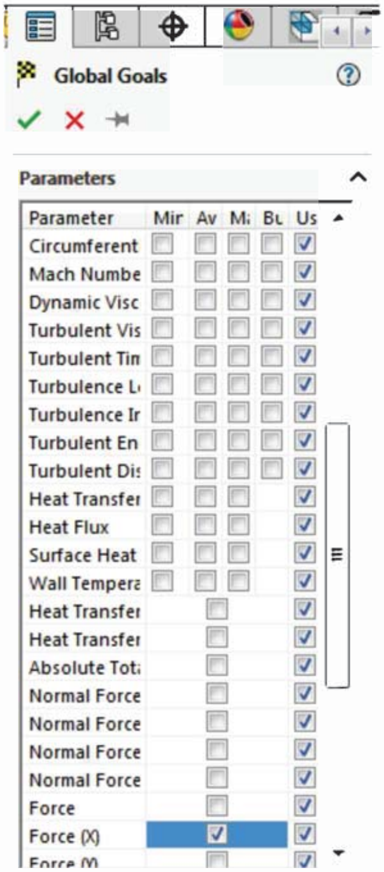

Figure 3.23b) Selection of X - Component of Force

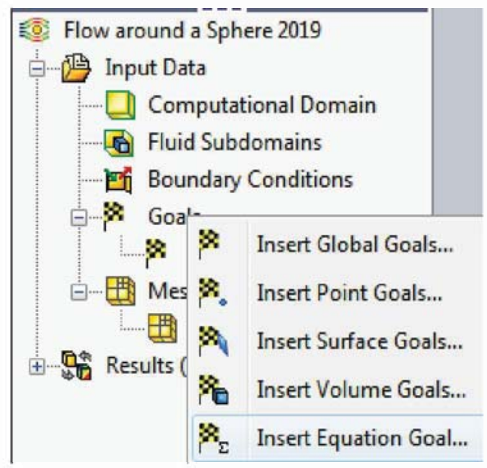

Figure 3.23c) Inserting equation goal

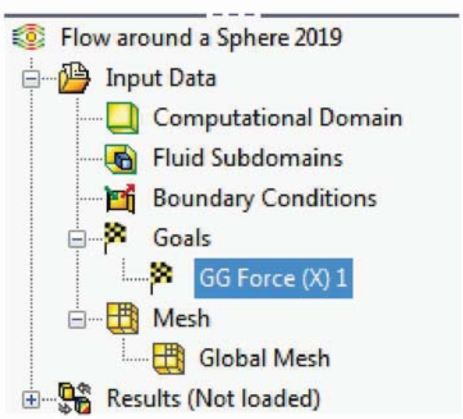

Figure 3.23d) Selection of X - Component of Force for the equation goal

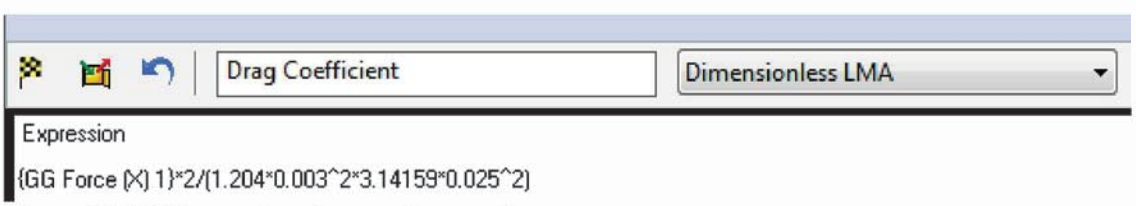

Figure 3.23e) Expression for equation goal

Running the Calculations

22. Choose **Tools>>Flow Simulation>>Solve>>Run…** Click on the **Run** button in the window that appears. Click on the goals flag to **Insert Goals Table** in the **Solver** window. You will now have the output window shown in figure 3.24a). The CPU time will depend on the speed of your computer processor and the amount of memory.

After completion of the calculations, select **Tools>>Flow Simulation>>Results>>Load/Unload**. ***Repeat this step once again.***

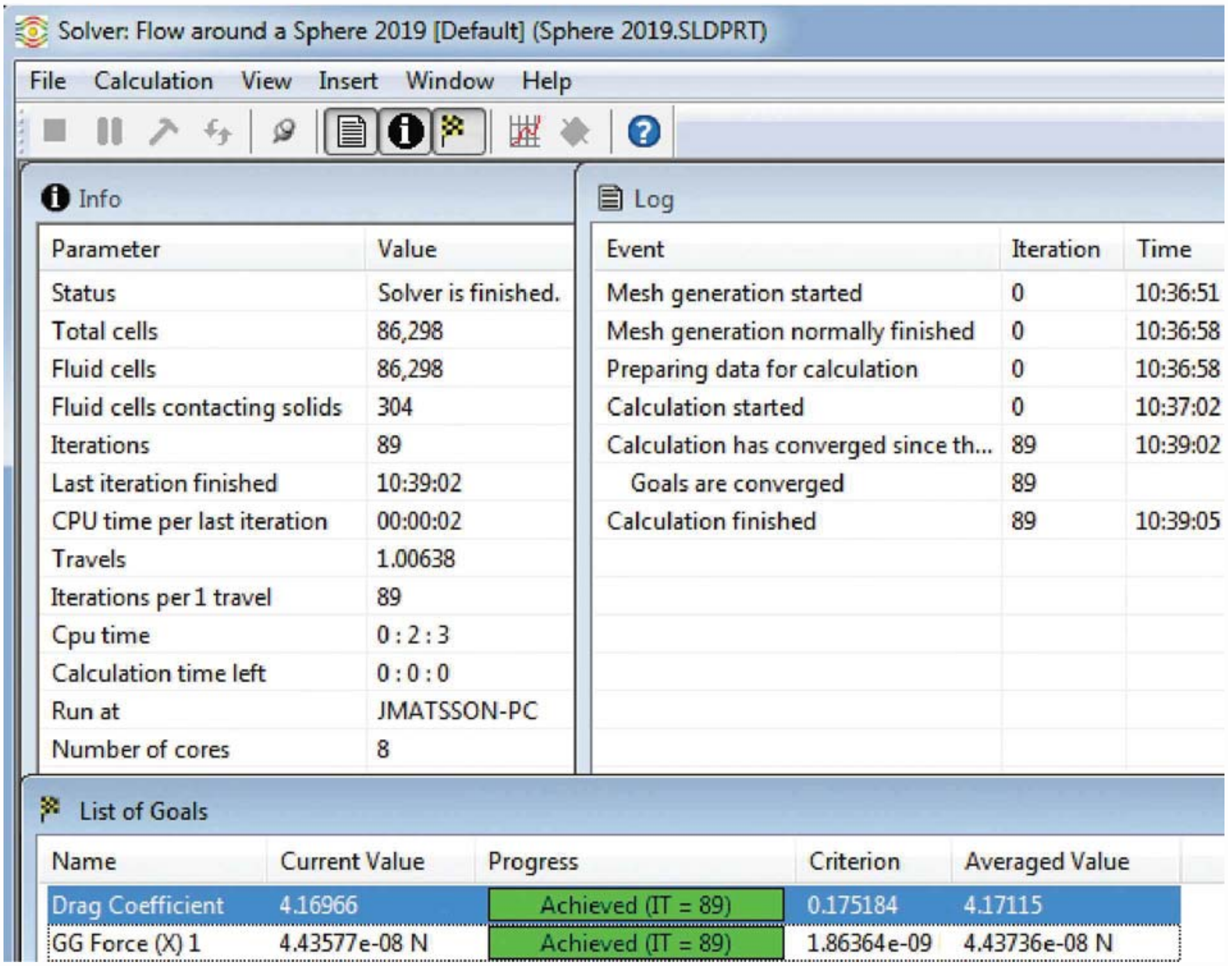

Figure 3.24a) Solver window for simulation of the flow around a sphere

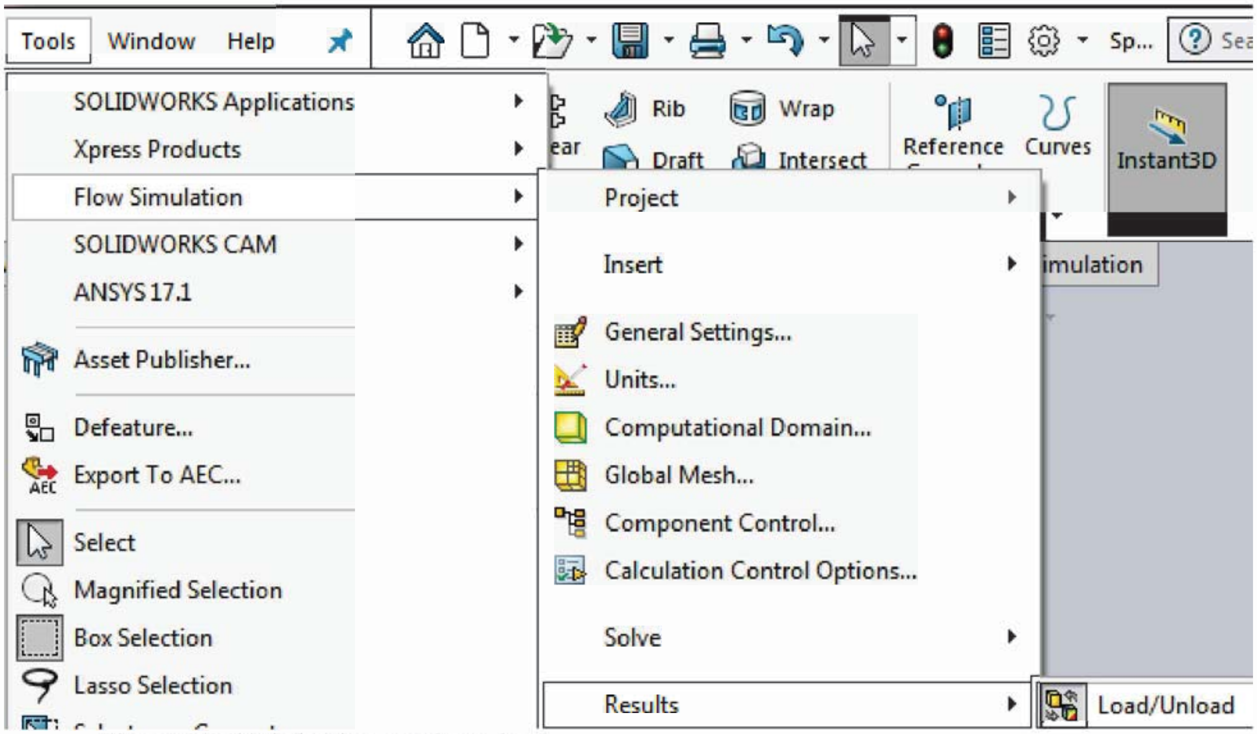

Figure 3.24b) Selecting results to load

Using Cut Plots

23. Select the **Flow Simulation analysis tree**

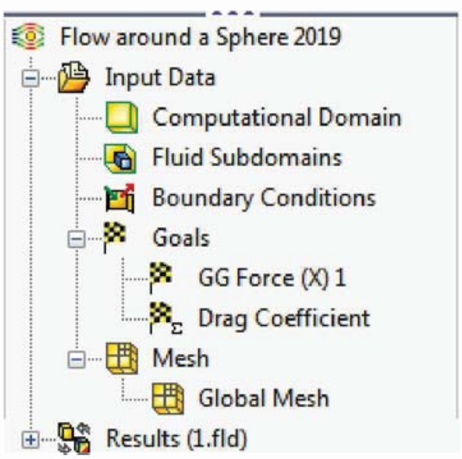

Figure 3.25 Selecting the Flow Simulation analysis tree

24. Open the Results folder, right click on the **Cut Plots** in the Results folder of the **Flow Simulation analysis tree** and select **Insert…** Click on the **Vectors** button in the **Display** section. Choose **Velocity** from the **Contours** section drop down menu. Slide **Number of Levels** to **255**. Exit the **Cut Plot** window. Rename **Cut Plot 1** to **Velocity around Sphere**. Select **Tools>>Flow**

Simulation>>Results>>Display>>Lighting from the SOLIDWORKS menu. Select the scale, right click and select **Make Horizontal**.

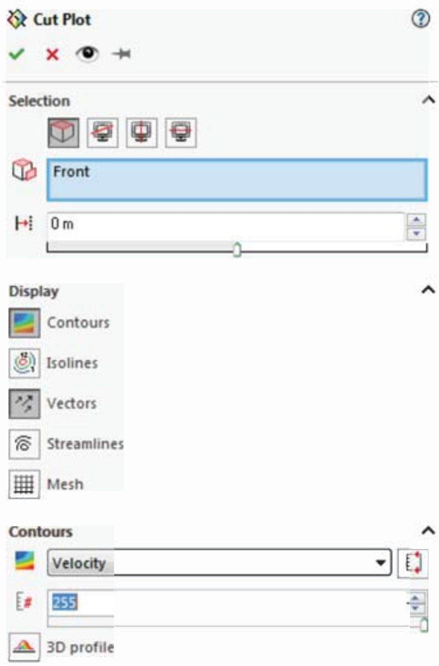

Figure 3.26a) Cut Plot window

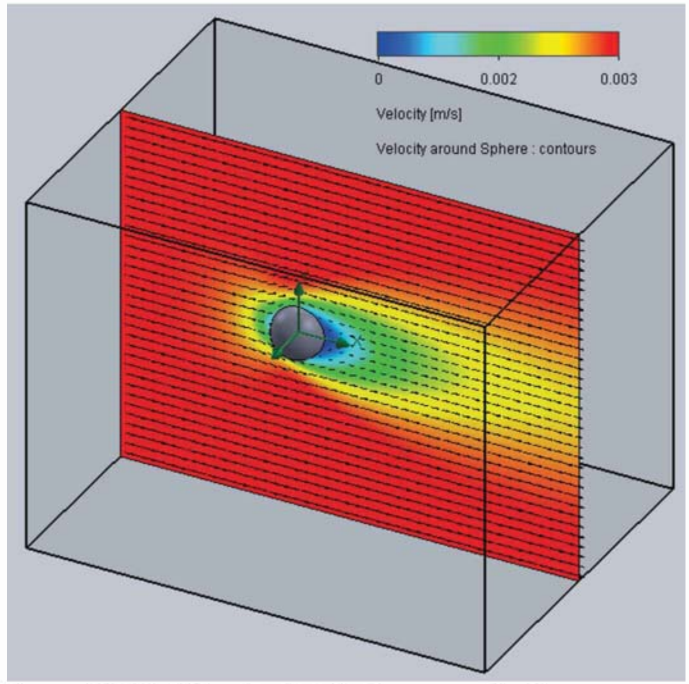

Figure 3.26b) Velocity distribution around sphere

Inserting Surface Parameters

25. Right-click on **Surface Parameters** in the **Flow Simulation analysis tree** and select **Insert….** Open the **FeatureManager design tree** in the graphics window and select the **Revolve1** feature. Check the **All** box in the **Parameters** window. Push the **Export to Excel** button in the **Surface Parameters** window. An Excel file is generated with local and integral parameters.

Integral Parameters

Integral Parameter	Value	X-componen	Y-componen	Z-componen	Surface Area [m^2]
Heat Transfer Rate [W]	0				0.00773814
Normal Force [N]	1.82419E-08	1.82418E-08	4.72604E-11	4.68917E-11	0.00773814
Friction Force [N]	2.6116E-08	2.6116E-08	3.08364E-11	2.96744E-11	0.00773814
Force [N]	4.43579E-08	4.43577E-08	7.80968E-11	7.65661E-11	0.00773814
Torque [N*m]	5.33237E-12	1.65908E-13	3.78598E-12	-3.7514E-12	0.00773814
Surface Area [m^2]	0.00773814	2.03288E-20	1.55854E-19	6.77626E-21	0.00773814
Torque of Normal Force [N*m]	2.10165E-14	-1.6016E-14	1.19285E-14	-6.5499E-15	0.00773814
Torque of Friction Force [N*m]	5.31982E-12	1.81923E-13	3.77406E-12	-3.7448E-12	0.00773814
Heat Transfer Rate (Convective) [W]	0				0.00773814
Uniformity Index [ ]	1				0.00773814
Area (Fluid) [m^2]	0.007853982				0.007853982

Figure 3.27a) Integral surface parameters