

Cylinder Drag Coefficient

Flow Simulation can be used to study flow around objects and to determine the resulting lift and drag forces on the objects due to the flow. In this example we use Flow Simulation to determine the drag coefficient of a circular cylinder immersed in a uniform fluid stream. The cylinder axis is oriented perpendicular to the stream.

The computations are performed for a range of Reynolds numbers (1,1000,10⁵), where $Re = \frac{\rho U D}{\mu}$, D is the cylinder diameter, U is the velocity of the fluid stream, ρ is the density, and μ is the dynamic viscosity. The drag coefficient for the cylinder is defined as:

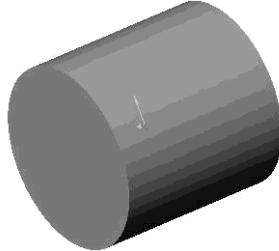
$$C_D = \frac{F_D}{\frac{1}{2} \rho U^2 D L}$$


where F_D is the total force in the flow direction (i.e. drag) acting on a cylinder of diameter D and length L .

The goal of the simulation is to obtain the drag coefficient predicted by Flow Simulation and to compare it to the experimental data presented in **Ref.1**.

Click **File, Open**. In the **Open** dialog box, browse to the **Cylinder 0.01m.SLDPRT** part located in the **Tutorial 2 - Drag Coefficient\cylinder 0.01m** folder and click **Open** (or double-click the part). Alternatively, you can drag and drop the **cylinder 0.01m.SLDPRT** file to an empty area of SolidWorks window.

The **Cylinder** analysis represents a typical Flow Simulation **External** analysis.



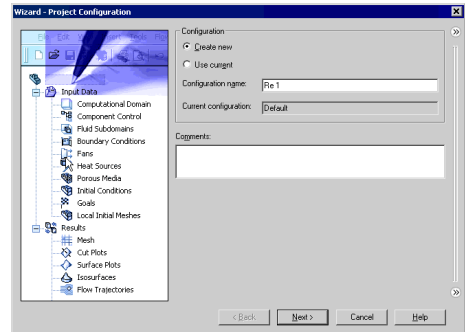
 *External flows analyses deal with flows over or around a model such as flows over aircrafts, automobiles, buildings, etc. For external flow analyses the far-field boundaries are the Computational Domain boundaries. You can also solve a combined external and internal flow problem in a Flow Simulation project (for example flow around and through a building). If the analysis includes a combination of internal and external flows, you must specify External type for the analysis.*

The first step is to create a new Flow Simulation project.

Creating a Project

- 1 Click **Flow Simulation, Project, Wizard**. The project wizard guides you through the definition of a new Flow Simulation project. In this project we will analyze flow over the cylinder at the Reynolds number of 1.
- 2 Select **Create new**. In the **Configuration name** box type **Re 1**. This is the name of the SolidWorks configuration that will be created for the associated Flow Simulation project.

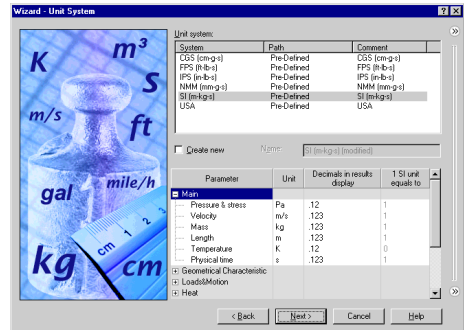
Click **Next**.



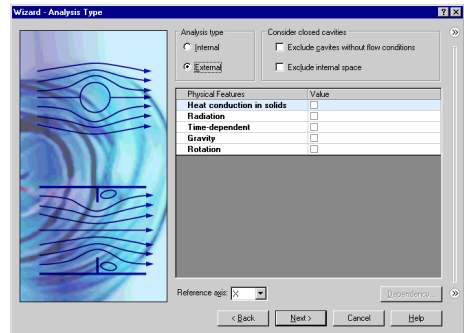
- 3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).

In this project we will specify the International System **SI** by default.

Click **Next**.



- 4 In the **Analysis Type** dialog box select an **External** type of flow analysis. This dialog also allows you to specify advanced physical features you want to include in the analysis. In this project we will not use any of the advanced physical features

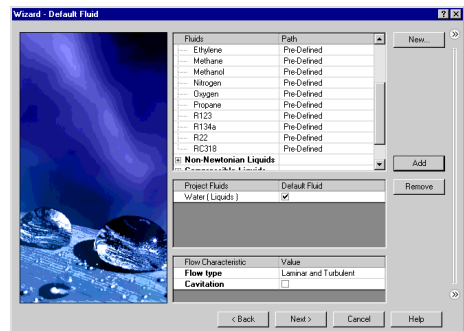


To disregard closed internal spaces within the body you can select **Exclude internal spaces**; however no internal spaces exist within the cylinder in this tutorial. The **Reference axis of the global coordinate system** (X , Y or Z) is used for specifying data in a tabular or formula form with respect to a cylindrical coordinate system based on this axis.

The flow over a cylinder is steady at a Reynolds number $Re < 40$ (see the cylinder Re definition above) and unsteady (time-dependent) at $Re > 40$. Since in this tutorial the first calculation is performed at $Re=1$, to accelerate the run, we perform a steady-state analysis.

Click **Next**.

- 5 Since we use water in this project, open the **Liquids** folder and double-click the **Water** item.



Click **Next**.

- In the **Wall Conditions** dialog box you may specify the default thermal wall conditions applied to all the model walls in contact with the fluid.

In this project we keep the default **Adiabatic wall** setting, denoting that all the model walls are heat-insulated and accept the default zero wall roughness.

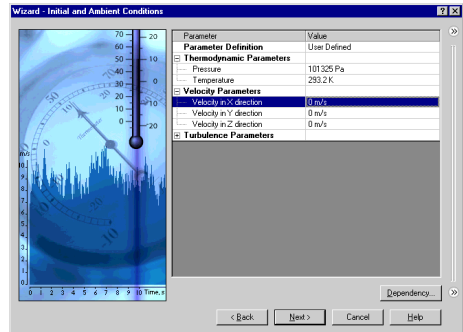
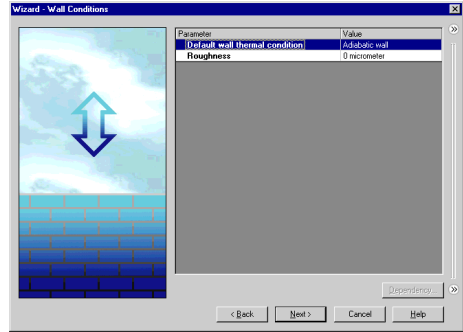
Click **Next**.


For a steady **External** problem, such as the cylinder in this tutorial, the **Initial and Ambient Conditions** dialog box asks you to specify the ambient flow conditions of the undisturbed free stream. Thus you will specify initial conditions inside the **Computational Domain** and boundary conditions at the **Computational Domain** boundaries. The ambient conditions are thermodynamic (static pressure and temperature by default), velocity, and turbulence parameters.

In this project we consider the flow under the default thermodynamic conditions (i.e., the standard atmosphere at sea level), and set the incoming stream (X-component) velocity in accordance with the desired Reynolds number.

For convenience we can use the **Dependency** box to specify the incoming flow velocity in terms of the Reynolds number.

- Click in the **Velocity in X direction** field. The **Dependency** button is enabled.
- Click **Dependency**. The **Dependency** dialog box appears.



 Using **Dependency** you can specify data in several ways: as a constant, as a tabular or formula dependency on x , y , z , r , θ , ϕ coordinates and time t (only for time-dependent analysis). The radius r is the distance from a point to the **Reference axis** selected from the reference coordinate system (the **Global Coordinate System** for all data set in the **Wizard** and **General Settings** dialog boxes), while θ and ϕ are the polar and azimuthal angles of spherical coordinate system, respectively. Therefore, by combination of r , θ , and ϕ coordinates you can specify data in cylindrical or spherical coordinate systems.

- In the **Dependency type** list select **Formula Definition**.

- 10** In the **Formula** box type the formula defining the flow velocity using the Reynolds number:

$1 * (0.0010115 / 0.01 / 998.19)$. Here:

1 – the Reynolds number (Re)

0.0010115 (Pa*s) - the water dynamic viscosity (μ) at the specified temperature of 293.2 K

0.01 (m) - the cylinder diameter (D)

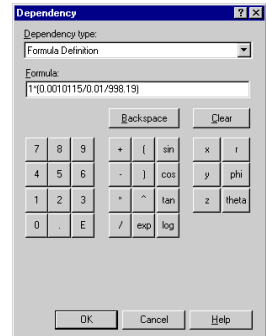
998.19 (kg/m³)- the water density (ρ) at the specified temperature of 293.2 K

- 11** Click **OK**. You will return to the **Initial and Ambient Conditions** dialog box.

For most flows it is difficult to have a good estimation of their turbulence *a priori*, so it is recommended that the default turbulence parameters be used. The default turbulence intensity values proposed by Flow Simulation are 0.1% for external analyses and 2% for internal analyses and these values are appropriate for most cases. In this project we will specify a turbulence intensity of 1%.

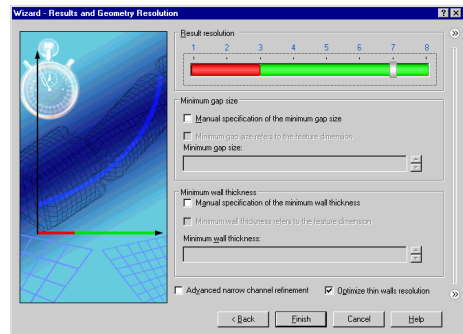
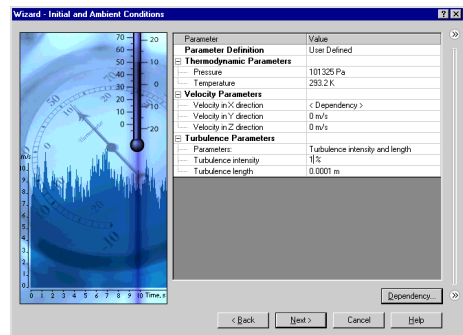
- 12** Expand the **Turbulence parameters** item and in the **Turbulence intensity** box type 1.

Click **Next**.



- 13** In the **Result and Geometry Resolution** dialog box specify the result resolution level of **7** and accept the automatically defined minimum gap size and minimum wall thickness.

Click **Finish**. The project is created and the 3D Computational Domain is automatically generated.




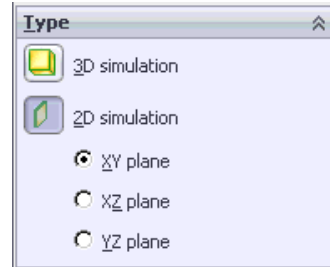
In this tutorial we are interested in determining the drag coefficient of the cylinder only, without the accompanying 3D effects. Thus, to reduce the required CPU time and computer memory, we will perform a two-dimensional (2D) analysis in this tutorial.




Specifying 2D simulation



- 1 In the Flow Simulation Analysis tree, expand the **Input Data** item.
- 2 Right-click the **Computational Domain** icon and select **Edit Definition**.



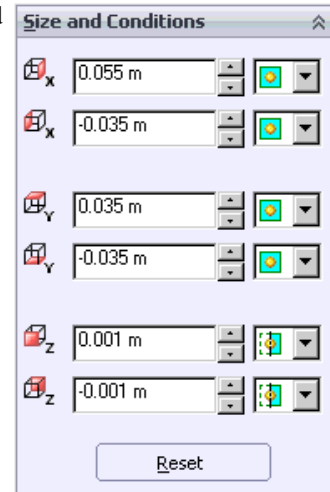
- 3 Under **Type** select **2D simulation**  and **XY plane** (since the Z-axis is the cylinder axis).




- 4 Automatically the **Symmetry**  condition is specified at the **Z min**  and **Z max**  boundaries of the Computational domain under **Size and Conditions**.

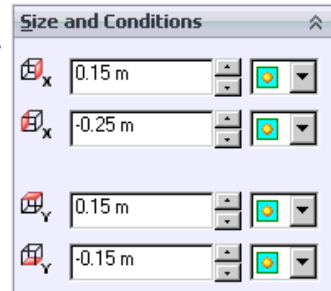
You can see that the **Z min**  and **Z max**  boundaries are set automatically, basing on the model dimensions.


Thus the reference cylinder length L in the cylinder drag (C_D) formula presented above is equal to $L = Z_{max} - Z_{min} = 0.002\text{ m}$.



 For most cases, to study the flow field around an external body and to investigate the effects of design changes it is recommended to use the default **Computational Domain** size as determined by Flow Simulation. However, in this case we will compare the Flow Simulation results to experimental results and we would like to determine the drag coefficient with a high degree of accuracy. In order to eliminate any disturbances of the incoming flow at the **Computational Domain** boundaries due to the presence of the cylinder, we will manually set the boundaries farther away from the cylinder. The accuracy will be increased at the expense of required CPU time and memory due to the larger size of **Computational Domain**.

- Under **Size and Conditions** specify the X and Y coordinates of the Computational domain boundaries as shown on the picture to the right.




- Click **OK** .

Since the incoming flow is aligned with the X-axis direction, the cylinder drag coefficient is calculated through the X-component of the force acting on the cylinder.


The X-component of force can be determined easily by specifying the appropriate Flow Simulation goal. For this case you will specify the **X - Component of Force** as a **Global Goal**. This ensures that the calculation will not be finished until **X - Component of Force** in the entire computational domain (i.e. on the cylinder surface) is fully converged.

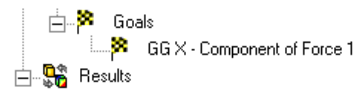
Specifying a Global Goal

- Click **Flow Simulation, Insert, Global Goals**.
- In the **Parameter** table select the first check box in the **X - Component of Force** row.
- Accept selected **Use for Conv.** check box to use this goal for convergence control.

 For the X(Y, Z) - Component of Force and X(Y, Z) - Component of Torque goals you can select the Coordinate system in which these goals are calculated. In this example the default Global Coordinate System meets the task.

| Parameter | Min | Av | Max | Bulk Av | Use |
|-------------------------|--------------------------|--------------------------|-------------------------------------|---------|-------------------------------------|
| Heat Flux | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| X - Component of Heat f | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Y - Component of Heat f | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Z - Component of Heat f | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Heat Transfer Rate | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| X - Component of Heat " | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Y - Component of Heat " | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Z - Component of Heat " | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Normal Force | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| X - Component of Norma | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Y - Component of Norma | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Z - Component of Norma | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Force | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| X - Component of Force | | | <input checked="" type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Y - Component of Force | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |
| Z - Component of Force | | | <input type="checkbox"/> | | <input checked="" type="checkbox"/> |

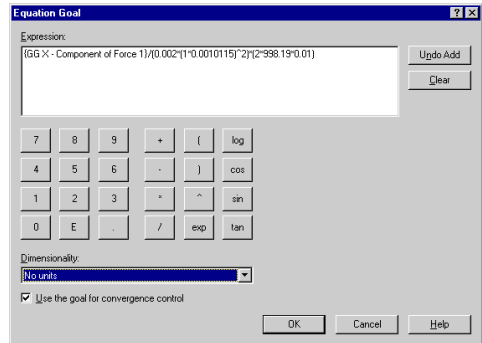
- Click **OK** . The new **GG X - Component of Force 1** item appears in the Flow Simulation Analysis tree.



Specifying an Equation Goal

When the calculation is finished, you will need to manually calculate the drag coefficient from the obtained force value. Instead, let Flow Simulation to make all the necessary calculations for you by specifying an **Equation Goal**.

- 1 Click **Flow Simulation, Insert, Equation Goal**.
- 2 In the Flow Simulation Analysis tree select the GG X - Component of Force 1 goal. It appears in the **Expression** box.



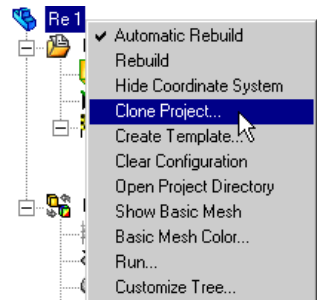
- 3 Use buttons in the calculator or keyboard to complete the expression as follows:

$$\{GG X - Component of Force 1\}/(0.002*(1*0.0010115)^2)*(2*998.19*0.01).$$
- 4 Select **No units** in the **Dimensionality** list and click **OK**. The new **Equation Goal 1** item appears in the Flow Simulation Analysis tree.
- 5 Rename the **Equation Goal 1** to Drag Coefficient.

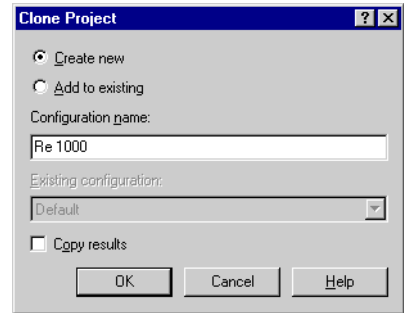
To compare the Flow Simulation results with the experimental curve taken from **Ref.1**, we will obtain the results at a Reynolds number of 1, 10^3 and 10^5 . As with $Re = 1$, the **Cylinder 0.01m.SLDPRT** is used to calculate the flow at the Reynolds number of 10^3 . The **Cylinder 1m.SLDPRT** is used to calculate the flow at the Reynolds number of 10^5 .

Cloning a Project and Creating a New Configuration


- 1 In the Flow Simulation Analysis tree, right-click the top **Re 1** icon and select **Clone Project**.
- 2 In the **Configuration name** box, type Re 1000.



- 3 Click **OK**. The new **Re 1000** configuration is created with the Flow Simulation project attached.

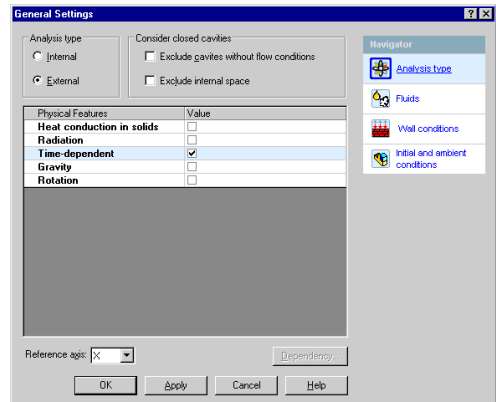


Since the new project is a copy of the **Re 1** Flow Simulation project, you only need to change the flow velocity value in accordance with the Reynolds number of 1000. Use the **General Settings** dialog box to change the data specified in the **Wizard**, except the settings for **Units** and **Result and Geometry Resolution**.

-  *The **General Settings** always presents the current state of the project parameters. You can change **General Settings** to correct the settings made in the **Wizard** or to modify the project created with the Flow Simulation **Template** in accordance with the new project requirements.*

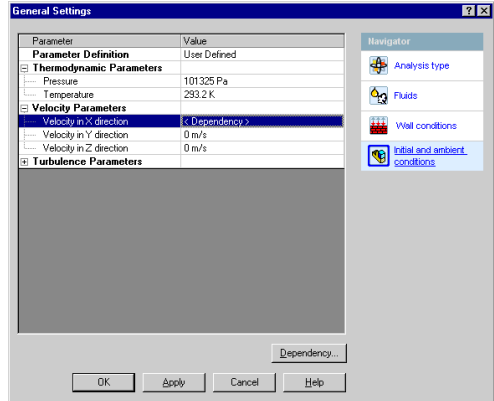
Changing Project Settings

- 1 Click **Flow Simulation, General Settings**. The **General Settings** dialog box appears.
- 2 As it has been mentioned above, since the flow over a cylinder is unsteady at $Re > 40$, select the **Time-dependent** physical feature for this project.
- 3 In the **Navigator** click **Initial and ambient conditions**.

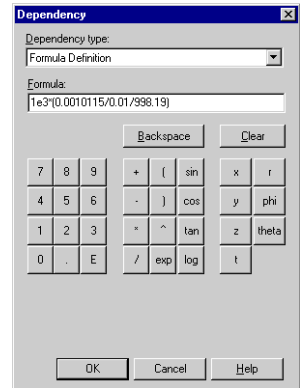


Chapter 5 Cylinder Drag Coefficient

- Click the **Velocity in X direction** field and then click **Dependency**.



- In the **Formula** box, type the formula for the new Reynolds number:
 $1e3 * (0.00101115 / 0.01 / 998.19) .$




- Click **OK** to return to the **General Settings** dialog box.
- Click **OK** to save changes and close the **General Settings** dialog box.

Changing the Equation Goal

- Right-click the **Drag Coefficient** icon under **Goals** and select **Edit Definition**.
- In the **Expression** box type the new formula for the new Reynolds number:
 $\{GG X - Component of Force 1\} / (0.002 * (0.00101115 * 10^3)^2) * (2 * 998.19 * 0.01) .$
- Select **No units** in the **Dimensionality** list.
- Click **OK** to save changes and close the **Equation Goal** dialog box.

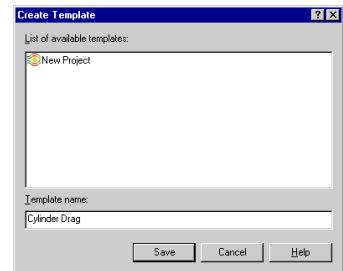
In the experiments performed with one fluid medium, the Reynolds number's large rise is usually obtained by increasing both the velocity and the model overall dimension (i.e. cylinder diameter) since it is difficult to increase only velocity by e.g. 10^5 times. Since our simulation is performed with water only, let us increase the cylinder diameter to 1 m to perform the calculation at a Reynolds number of 10^5 .


Cloning a project is convenient if you want to create similar projects for the same model. The easiest way to apply the same general project settings to another model is to use the Flow Simulation **Template**.

 **Template** contains all of the general project settings that can be used as a basis for a new project. These settings are: problem type, physical features, fluids, solids, initial and ambient flow parameters, wall heat condition, geometry and result resolution, and unit settings. Notice that **Boundary Conditions, Fans, Initial Conditions, Goals** and other features accessible from the **Flow Simulation, Insert** menu, as well as results are not stored in the template. Initially, only the New Project default template is available, but you can easily create your own templates.

Creating a Template

- 1 Click **Flow Simulation, Project, Create Template**. The **Create Template** dialog box appears.
- 2 In the **Template name** box, type *Cylinder Drag*.
- 3 Click **Save**. The new Flow Simulation template is created.



 All templates are stored as **.fwp** files in the **<install_dir>/Template** folder, so you can easily apply a template to any previously created models.

- 4 Save the model.

Next, create a new project based on the *Cylinder Drag* template.

Creating a Project from the Template

Open the **Cylinder 1m.SLDPRT** file located in the **cylinder 1m** folder.

Chapter 5 Cylinder Drag Coefficient

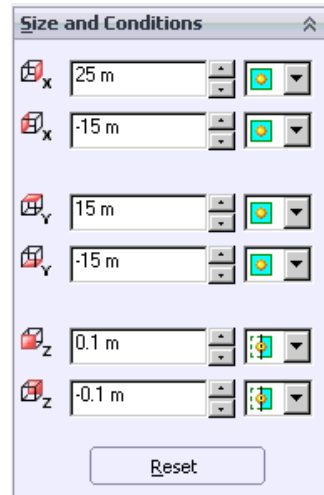
- 1 Click **Flow Simulation, Project, New**. The **New Flow Simulation Project** dialog box appears.
- 2 In the **Configuration name** box, type $Re\ 1e5$.
- 3 In the **List of templates**, select **Cylinder Drag**.
- 4 Click **OK**.




The newly created project has the same settings as the **Re 1000** project with the **cylinder 0.01m** model. The only exceptions are **Geometry Resolution** and **Computational Domain** size, which are calculated by Flow Simulation in accordance with the new model geometry.

Notice that the **2D simulation** setting and **Global Goal** are retained. Next, you can modify the project in accordance with the new model geometry.

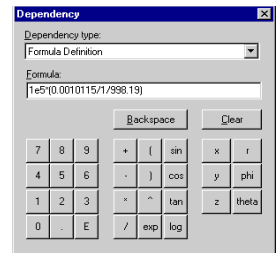
- 1 Click **Flow Simulation, Computational Domain** and adjust the computational domain size as shown at the picture to the right.



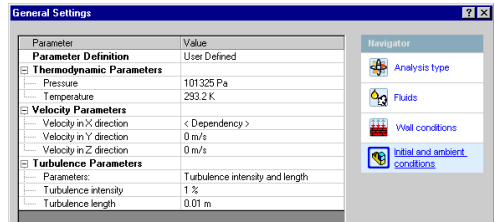
- 2 Click **OK** .
- 3 Open the **General Settings** dialog box and click **Initial and ambient conditions**, click the **Velocity in X direction** field, then click **Dependency**.
- 4 Change the velocity X component formula as follows: $1e5 * (0.00101115 / 1 / 998.19)$.

Click **OK** to return to the General Settings dialog box.

By default, Flow Simulation determines the default turbulence length basis equal to one percent of the model overall dimension (i.e. cylinder diameter). Since the **Re 1e5** project was created from the template, it inherited the turbulence length value calculated for the small cylinder ($d = 0.01m$). For the **cylinder 1m** we need to change this value.



- In the **General Settings** dialog box expand the **Turbulence parameters** item. Type 0.01 m in the **Turbulence length** field.



- Click **OK**.

- Create the **Equation Goal** for the drag coefficient of the cylinder as it was described before. In the Expression box enter the formula:

$$\{GG X - \text{Component of Force } 1\} / (0.2 * (0.0010115 * 10^5)^2) * (2 * 998.19 * 1).$$

- Select **No units** in the **Dimensionality** list.
- Click **OK**. Rename the **Equation Goal 1** to **Drag Coefficient**.

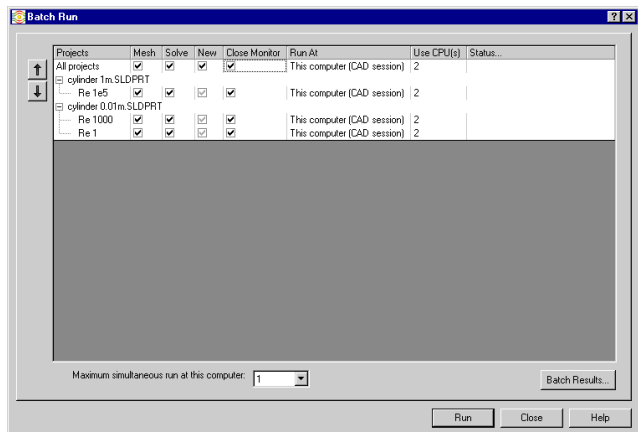
Now you can solve all of the projects created for both the cylinders.

Solving a Set of Projects

Flow Simulation allows you to automatically solve a set of projects that exist in any currently opened document.

- Click **Flow Simulation, Solve, Batch Run**.
- Select the **Solve** check

box in the **All projects** row to select **Solve** for all projects (**Re 1, Re 1000, Re 1e5**). Also select the **Close Monitor** check box in the **All projects** row. When the **Close Monitor** check box is selected, Flow Simulation automatically closes the **Solver Monitor** window when the calculation finishes.




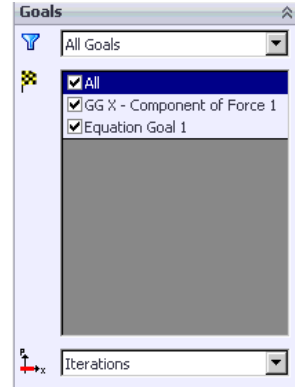
- Click **Run**.

Getting Results

After all calculations are complete, go to the **cylinder 0.01m** model and activate the **Re 1000** configuration. Create **Goal Plot** to obtain the **Drag Coefficient** value:

- Click **Flow Simulation, Results, Load/Unload Results**.

- 2 In the **Load Results** dialog box, keep the default project's results file (2.fld) and click **Open**.
- 3 In the Flow Simulation Analysis tree, under **Results**, right-click the **Goal Plots** icon and select **Insert**. The **Goal Plot** dialog box appears.
- 4 Select **All**.
- 5 Click **OK** . The **Goals1** Excel workbook is created. Switch to Excel to obtain the value.



cylinder 0.01m.SLDPRT [Re 1000]

| Goal Name | Unit | Value | Averaged Value | Minimum Value | Maximum Value |
|---------------------------|------|-------------|----------------|---------------|---------------|
| GG X - Component of Force | [N] | 0.000118661 | 0.000112731 | 0.000102502 | 0.000122439 |
| Drag Coefficient | [] | 1.157683117 | 1.099830977 | 1.00002738 | 1.194542459 |

- 6 Activate the **Re 1** configuration and load results. Create the goal plot for both the goals.

cylinder 0.01m.SLDPRT [Re 1]

| Goal Name | Unit | Value | Averaged Value | Minimum Value | Maximum Value |
|---------------------------|------|-------------|----------------|---------------|---------------|
| GG X - Component of Force | [N] | 1.14446E-09 | 1.16833E-09 | 1.12811E-09 | 1.8234E-09 |
| Drag Coefficient | [] | 11.1656052 | 11.3984466 | 11.00608012 | 17.78943946 |

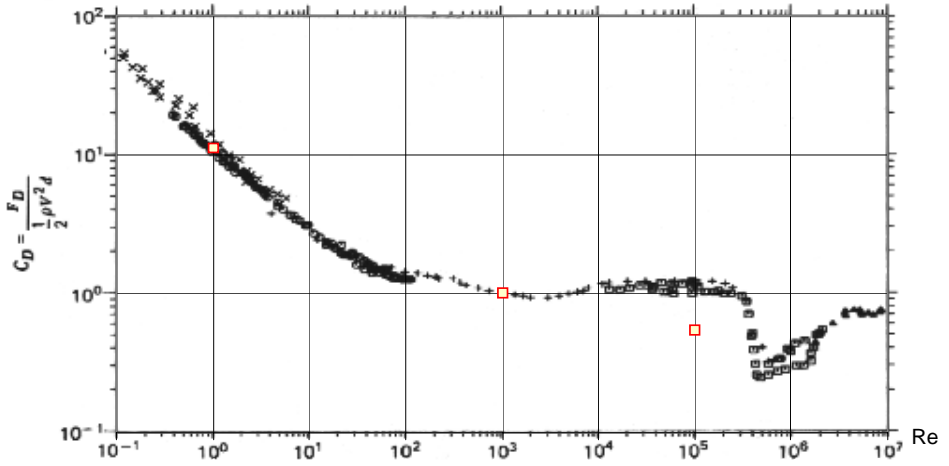
- 7 Switch to the **cylinder 1m** part, activate the **Re 1e5** configuration, load results and create the goal plot for both the goals.

cylinder 1m.SLDPRT [Re 1e5]

| Goal Name | Unit | Value | Averaged Value | Minimum Value | Maximum Value |
|---------------------------|------|-------------|----------------|---------------|---------------|
| GG X - Component of Force | [N] | 0.44242974 | 0.442499134 | 0.429568256 | 0.451472343 |
| Drag Coefficient | [] | 0.431644044 | 0.431711746 | 0.419096102 | 0.440466204 |

Even if the calculation is steady, the averaged value is more preferred, since in this case the oscillation effect is of less perceptibility. We will use the averaged goal value for the other two cases as well.

You can now compare Flow Simulation results with the experimental curve.



Ref. 1 Roland L. Panton, "Incompressible flow" Second edition. John Wiley & sons Inc., 1995

