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 $(1,1000,10^5)$, where

Re= $\frac{\rho UD}{\mu}$, *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:

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

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

CONTRACT OF A DECK	Unit system:				
m ³	Sustem	Path	Corre	ent	
V	CGS (cm-q-s)	Pre-Defin	ed CGS (cm-q-s)	
A state	FPS (It-Ib-s)	Pre-Defin	ed FPS (f	(Iba)	
C C	IPS (in-lb-s)	Pre-Defin	ed IPS (in	⊩lb-s)	
J	NMM (mm·g·s)	Pre-Defin	ed NMM	(mm-g-s)	. 1
	SI (m-kg-s)	Pre-Defin	ed SI(m-I	(g·s)	
mls	USA	Pre-Defin	ed USA		
(1/ S F+					
	E Crash and		P17 1 17 18 B		_
	1 Greake new		SI (m-Kg-s) (modined)		
					-
mila /h	Parameter	Unit	Decimats in results	T ST unit	
nal me/n	- Main		dispidy	equals to	
ya	E Prose no 1 shore	P ₂	12		
	Velocitu	n/e	123	R.	
2 No. 10	Mare	ka	123	li -	
and then y	Length	10	123	li -	
7. 18.	Temperature	ĸ	12	0	
Valor /	Physical time		123	ĭ	
	E Geometrical Characterist	ic .			
	T Loads Motion	~			
All a second sec	T Heat				-1
	- Deat	1 Ma	Count	Hab	1
	/ Dack		ALC: NOT THE REPORT OF		
ard - Analysis Type	< Deck				?
rd - Analysis Type	Analysis type C Internal	Consider o	losed cavities	v conditions	?
rd - Analysis Type	Analysis type C Internal C External	Consider o Exc Exc	losed cavities ude gavites vithout flow	v conditions	?
ıd - Analysis Type	Analysis type C Internal © External Physical Features	Consider o Excl	ilosed cavities tude gavites without flow tude internal space	v conditions	?
sd - Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in	Consider of Excl	ilosed cavities lude gavites without flow ude internal space	v conditions	?
rd - Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in R addiation	Consider of Excl Excl solids	ilosed cavities lude gavites without flow lude internal space	v conditions	?
rd - Analysis Type	Analysis type Internal External Physical Festures Heat conduction in Radiation Time dependent	Consider of Exc Exc solids	losed cavilies Lude gavites without flow Lude internal space	v conditions	?
rd - Analysis Type	Analysis lype C Internal C Sternal Physical Features Heat conduction it Radiation Time-dependent Gravity	Consider of Excl Solids	dosed cavities losed cavities lude gavites without flow lude internal space	v conditions	?
d - Analyzis Type	Analysis type C Internal C External Physical Festures Heat conduction in Time dependent Gravity Rotation	Consider o Excl Excl solids	lored covities ude gavites without flow ude internal space	v conditions	?
rd - Anslyris Type	Analysis type C Internal C External Physical Features Heat conduction in Radiation Time-dependent Gravity Rotation	Consider o Exec solids	iosed cerrites Lude gavites without flow Lude internal space	v conditions	?
ed Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in Time dependent Gravity Rotation	Consider o Excl solids	losed cevities Losed cevities Losed gavies without flow Lose internal space	v conditions	
d Aalyris Type	Analysis type C Internal Byternal Physical Features Heat conduction in Reat conduction in Time dependent Time day Rotation	Consider o Excl Excl solids	kored cevities kored cevities kude gavites without flow kude internal space	v conditions	
rd Anslysis Type	Analysis type C Internal C External Physical Features Heat conduction in Radiation Time-dependent Enrolly Rotation	Consider of Excl Excl solids	lored covilies lade gavites without flow lude internal space	v conditions	?
d Anlyris Type	C Box C Internal C Internal C Internal Physical Conductions in Relation Relation	Consider o	ilored cavilles lude gavites without flow lude gavites without flow lude internal space	v conditions	
el Andria Type	Analysis (spe C)sternal C)sternal C)sternal Physics Freatwes: Heat conduction in Rediction Time dependent Gravity Redation	Consider of Excl solids	Iosed cevilies Lobe gavies without flow Lobe gavies without flow Lobe internal space	v conditions	
el Anlyris Type	Cash Analysis loss Cipternal Ciptern	Consider of Excl solids	ilosed cavilles lude gavites without figures lude internal space	v conditions	
ed - Analysis Type	Analysis lops Analysis lops Plannal Plannal Plannal Heat conduction in Rediation Time dependent Gravity Rotation	Consider of Excl Excl solids	Iosed cevities Lobe gaves without flow Lobe gaves without flow Lobe internal space	v conditions	
ed Analytis Type	Analysis loss Analysis loss C pletrad C giannol Physical Features Heat conduction in Reduction Time to penden Time to penden Reduction	Consider o	dored cevities duce gavies without flow duce internal space	v conditions	
d Analysis Type	C period	Consider of Exel Exel	Larce Larce	v conditions	
de Anslytis Type	Analysis loss C planal C planal C planal C planal C planal Read conduction in Read conduction in R	Consider o	dored cevilies duck gavles without flow tucke internal space	v conditions	
d Anlyni Type	C peak C pland C pl	Consider o	koed caviles koed	v conditions	
de Andyris Type	C geolo C plend C p	Consider of Excl solids	loved caviles	v conditions	2
d Anlyris Type	Cash		to an internet and the second se	v conditions	
ard - Analysis Type	C period	Consider of Exel Exel solids	to carlos based carlos bude gretes without file bude internet space Value Carcel at 2 Carcel	v conditions	

- □ 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.
- \square 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.

E Dylere Picoland Enders Enders	Fluids	Path		New
Hefrand Pro-Defined Hefrand Pro-Defined Notoym Pro-Defined Notoym Pro-Defined Notoym Pro-Defined Notoym Pro-Defined Notoym Pro-Defined Notoym Pro-Defined Received Received	Ethylene	Pre-Defined		
Heliard Holiard Holiard Holiard Gogen Feb Girled Gogen Feb Girled Gogen Feb Girled Gogen Feb Girled Fil3 Fil3 Feb Girled Fil3	Methane	Pre-Defined		
Nicom Pre-Drined Organ Pre-Drined Prozent Pre-Drined Prozent Pre-Drined Prozent Pre-Drined Prozent Pre-Drined Prozent Pre-Drined Prozent Pre-Drined P	Methanol	Pre-Defined		
	Nitrogen	Pre-Defined		
Processelenic Value Processelenic Value Processelenic Value	- Oxygen	Pre-Defined		
	Propane	Pre-Defined		
R134a PeoDefined R210 PeoDefined R0 PeoDefined PeoDefined PeoDefined Value (Loads) Value Value (Loads) Value Peorting Lamora and Tubulert	R123	Pre-Defined		
	R134a	Pre-Defined		
	R22	Pre-Defined		
Non-Neotonian Liquids Non-Neotonian Liquids Non-Neotonian Liquids Non-Neotonian Liquids Non-Neotonian Liquids Non-Neotonian Non	RC318	Pre-Defined		
Privac Flad Details Flad Remove View (Lloads) ∞ Prive Characteristic Valee Prive Characteristic Valee	Non-Newtonian Liquid	*	-	Add
Water (Lopids) Vides Provide Constraint of Tublect From type Forwate	Project Fluids	Default Fluid		Remove
Perr Chrastenitic Value Four Chrastenitic Usilie Four type	Water (Liquids)	×		
Flow type Laminar and Tutbulent	Flow Characteristic	Value		
Cavitation	Flow type	Laminar and Turbulent		
	Cavitation		_	

Click Next.

6 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.

- 7 Click in the Velocity in X direction field. The Dependency button is enabled.
- 8 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.
- **9** 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.

 Dependency
 ? ▼

 Dependency type:
 Formula Definition

 Formula:
 11(0.001/398.19)

 11(0.001015/0.01/398.19)
 Rackspace

 2
 Rackspace

 7
 8

 4
 5

 6
 1

 1
 2

 0
 .

 0
 .

 0
 .

 0
 .

 0
 .

 0
 .

 0
 .

 0
 .

2 X

- 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.

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

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

Click Next.

wall thickness.

automatically generated.

70 - 20	Parameter	Value
60 -	Parameter Definition	User Defined
50 10	Thermodynamic Parameters	
	Pressure	101325 Pa
30-1-0	Temperature	293.2 K
0 00 00 00	Velocity Parameters	
3 20	Velocity in X direction	< Dependency >
10-10-	- Velocity in Y direction	0 m/s
H 0	 Velocity in Z direction 	0 m/s
	Turbulence Parameters	
SH S D	- Parameters:	Turbulence intensity and length
ma line line line line line line line line	- Turbulence intensity	1 2
	Turbulence length	0.0001 m
5 4 2 2 1 1 0 0 1 2 3 4 5 6 7 8 5 10 Time.e	< Beck N	Dependency. (>) sst > Cancel Help
Wizard - Results and Geometry Resoluti	on Result resolution	7 X
×		
	Minimum gap size	
	Manual specification of the mini	mum gap size
	Minimum gap size refers to the f Minimum gap size:	exture dimension
NO CONTRACTOR		
Nº AGADAGO	Minimum and Minimum	
	Minimum was truckness	mum wall thickness
R TEBAR	 Mg was specification of the film 	TRAIL TTOP VIEWO PDEO
AN THE REAL	Minimum wall thickness refers to	
The	Minimum wall thickness:	
KIPT TIH		
ALA PHI	1	

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.

errow channel refinement 🔽 Opimize thin wals resolution

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).



XY plane
 XZ plane
 YZ plane



You can see that the **Z** min \square_z and **Z** max \square_z 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 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**.

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

<u>S</u> ize	and Conditions	*
Ø,	0.15 m	- -
€₽ _x	-0.25 m	- ·
₫,	0.15 m	- 0 -
Ø,	-0.15 m	- 🖸 🔽

6 Click OK У .

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

- 1 Click Flow Simulation, Insert, Global Goals.
- 2 In the **Parameter** table select the first check box in the **X Component of Force** row.
- 3 Accept selected **Use for Conv.** check box to use this goal for convergence control.
- $\square For the X(Y, Z) Component of Force$ and X(Y, Z) - Component of Torque goalsyou can select the Coordinate system inwhich these goals are calculated. In thisexample the default Global CoordinateSystem meets the task.

Parameter						~
Parameter	Min	A٧	Max	Bulk Av	Use	
Heat Flux					✓	
X - Component of Heat F					✓	
Y - Component of Heat F					✓	
Z - Component of Heat F					✓	
Heat Transfer Rate					✓	
X - Component of Heat 7					✓	
Y - Component of Heat					~	
Z - Component of Heat					✓	
Normal Force					✓	
X - Component of Norma					✓	
Y - Component of Norma					✓	
Z - Component of Norma					✓	
Force					~	
X - Component of Force					✓	
Y - Component of Force					~	
Z - Component of Force					✓	

4 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.
- In the Flow Simulation Analysis tree select the GG X Component of Force 1 goal. It appears in the Expression box.

u A-	Componen	k of Folce	1)7[0:002	(1 0.0010	1101 211	2 330.131		<u>C</u> lear
7 4	8	9 6 3	+ -	[]]	log cos sin			
0	E		1	exp	tan			
mensio ounits ∐se t	naîty: he goal fo	r converge	nce contr	ol	₽			

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.

Clone Project	? ×
• Create new	
C Add to existing	
Configuration name:	
Re 1000	
Existing configuration:	
Default	7
Copy results	
OK Cancel <u>H</u> e	lp

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.
- 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.

nerar settings		
Analysis type Conside C Internal C E C External E	r closed cavities xolude gavites without flow conditions xclude internal space	Navigator Analysis type
Physical Features	Value	
Heat conduction in solids		Vial conditions
Radiation		
l me-dependent		- Conditions
Batation		
Teference agis: 🗙 💌	Dependency	
OK ,	Apply Cancel <u>H</u> elp	

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

I didilicici	Value	Navigator
Parameter Definition	User Defined	-
Thermodynamic Parameters		Analysis type
Pressure	101325 Pa	-
Temperature	293.2 K	An Fluids
Velocity Parameters		-38
Welocity in X direction	< Dependency >	
 Velocity in Y direction 	0 m/s	
Velocity in Z direction	0 m/s	Initial and ambient
Turbulence Parameters		Conditions

In the Formula box, type the formula for the new Reynolds number:
 1e3*(0.0010115/0.01/998.19).

Dep De Fo	oen rmu	lenc denc ila De	y y type efinitio	c n						×
Eor Te	mu 3"(i	la: 0.001	0115.	/0.0	1/99	3.19)	_	_	_	-
					Ba	ackspa	ace		<u>C</u> I	ear
7	•	8	9		+	ſ	sin		×	r
4	I I	5	6		•)	cos	ĺ	y	phi
1		2	3		×	^	tan		z	theta
0			Е		7	exp	log		t	
			ОК]_	Can	cel		<u>H</u> e	lp 🛛

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

Changing the Equation Goal

- 1 Right-click the Drag Coefficient icon under Goals and select Edit Definition.
- 2 In the Expression box type the new formula for the new Reynolds number: {GG X - Component of Force 1}/(0.002*(0.0010115*10^3)^2)*(2*998.19*0.01).
- **3** Select **No units** in the **Dimensionality** list.
- 4 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.

Create Template			? ×
List of available template	s:		
New Project			
Template name:			
Cylinder Drag			
,	Save	Cancel	Help

- 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.

- 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.

New Flow Simulation Project	? ×
Configuration	_
C Use current	
Configuration name:	
Re 1e5	
Basic configuration:	
Default	•
Flow Simulation template	
Cylinder Drag	•
OK Cancel He	elp

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.

<u>S</u> ize	and Con	ditions			~
₿ _x	25 m		* *	0	•
Øx	-15 m		•	0	•
æ,	15 m		* *	0	•
₫,	-15 m		•	0	•
₽z	0.1 m		*	[]	•
Øz	-0.1 m		•	[]	•
		<u>R</u> eset			

- 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*(0.0010115/1/998.19).

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.



- 5 In the General Settings dialog box expand the Turbulence parameters item. Type 0.01 m in the Turbulence length field.
- 6 Click OK.

Parameter	Value	Navigator
Parameter Definition	User Defined	
Thermodynamic Parameters		Analysis type
Pressure	101325 Pa	
Temperature	293.2 K	Fluids
Velocity Parameters		-58
Velocity in X direction	< Dependency >	Alal conditions
Velocity in Y direction	0 m/s	
Velocity in Z direction	0 m/s	Initial and ambient
Turbulence Parameters		conditions
Parameters:	Turbulence intensity and length	
Turbulence intensity	1%	
Turbulence length	0.01 m	

7 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 - Component of Force 1}/(0.2*(0.0010115*10^5)^2)*(2*998.19*1).

- 8 Select No units in the Dimensionality list.
- 9 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.

- 1 Click Flow Simulation, Solve, Batch Run.
- 2 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.

All projects cylinder 1 m.SI Re 1e5 cylinder 0.01 n				This computer (CAD session)	2		
⊟ cylinder 1 m.SI Re 1e5 Cylinder 0.01 n	LDPRT						
⊡ Re 1e5 ⊡ cylinder 0.01 n	~ ~						
🖃 cylinder U.U1n		\mathbb{M}	•	This computer (CAD session)	2		
D 1000	n.SLDPHT			TU:	0		
Re 1000	× ×		×	This computer (LAD session)	2		
ne i	• •	×.	•	This computer (LAD session)	2		
M suimum ain	ultanacus un s	t this our	onutor.		-	-	_

3 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:

1 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 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

Chapter 5 Cylinder Drag Coefficient