## Finding Drag Coefficient using Solidworks Flow Simulation

Using solidworks to find the drag coefficient of shapes is a very useful way to cut down on the design time of a project, as it can remove tests. Running simulations also gives a visualization of how the fluid will flow around a part and give the user an idea of low and high pressure zones, fluid vectors, and ways they can streamline their parts. In this tutorial you will learn how to find the drag coefficient of a sphere and insert and view plots of velocity. You will also learn the limitations of the Solidworks Flow simulation add-in.

1. Model a sphere with a diameter of 50mm. This can be done with a revolved base.



🗞 😤 🤮 🐣 🧟

4. After selecting the wizard, name your project Flow Simulation and use default configurations. Click next.



## 5. Select SI as the unit system and click next

	System	Path	Comme	ent	
K	CGS (cm-g-s) FPS (ft-lb-s) IPS (in-lb-s) NMM (mm-g-s)	Pre-Define Pre-Define Pre-Define Pre-Define	ed CGS (d ed FPS (ft ed IPS (in ed NMM (	:m-g-s) -lb-s) -lb-s) mm-g-s)	
n/s	GI (m-kg-s) USA	Pre-Define Pre-Define	ed SI (m-k ed USA	.g-s)	
gal	nile/h Parameter	Unit	Decimals in results display	1 SI unit equals to	Ē
	Pressure & stress	s Pa	.12	1	
and a line	Mass	kg	.123	4	
10 10	Length Temperature	m K	.123 .12	1 0	
KAN		S	.123	1	

Wizard - Analysis Type	Analysis type Conside Internal Es External Es	er closed cavities xclude cavities without flow conditions xclude internal space
	Physical Features Heat conduction in solids Radiation Time-dependent Gravity Rotation	Value
	Reference axis: 🗙 💌	Dependency

6. Select external as the analysis type and leave all other boxes unchecked. Click next.

7. Expand the menu for gases and select air. Add it to the project fluids list. Click next

< Back

Next >

Cancel

Help

Wizard - Default Fluid				? <mark>x</mark>	
	Fluids Gases Pre-Defined Acetone Ammonia Argon Butane Carbon dioxide Chlorine Ethane Ethanol	Path Pre-Defined Pre-Defined Pre-Defined Pre-Defined Pre-Defined Pre-Defined Pre-Defined Pre-Defined		New	>
	Project Fluids Air ( Gases ) Flow Characteristic Flow type High Mach number flow Humidity	Default Fluid	• <u> </u>	Remove	
	< Back	Next > Cance		Help	

	Parameter	Value	C.
	Default wall thermal condition	Adiabatic wall	ſ
States Participa	Roughness	0 micrometer	
ţ			

8. Do not change the default settings in wall conditions. Click next

9. Under initial conditions, set velocity in the x direction to 0.003 m/s. This simulates a Reynolds number of approximately 10. Click next.

Vizard - Initial and Ambient Conditions		? ×
	]	
70 - 20	Parameter	Value
60	Parameter Definition	User Defined
50 - 10	Thermodynamic Parameters	
-Q40	Parameters:	Pressure, temperature
30	Pressure	101325 Pa
20 - 20	Temperature	293.2 K
10-10-	Velocity Parameters	
	Parameter:	Velocity
H dam	Velocity in X direction	.003 m/s
	Velocity in Y direction	0 m/s
	Velocity in Z direction	0 m/s
	Turbulence Parameters	
8		
7.		
6.		
5.		
4.		
3.		
2.		
1-		
0   2 3 4 5 6 7 8 9  0Time,s	5	Dependency
	K Back	ext > Cancel Help

	Result re	esolution							
	1	2	3	4	5	6	7	8	
	Minimum Man Mini Minimur	i gap size ual specific mum gap si n gap size:	ation of th	e minimun o the featu	n gap size ure dimens	ion		4	
	Minimum	wall thickr wal specific mum wall th	iess ation of th iickness re	<b>e minimur</b> efers to the	n wall thick e feature d	ness			
H		n wall thick	ness: channel re	efinement	0	otimize thir	n walls res	v	

10. Move the slider from 3 to 4 to change the mesh resolution. Click finish to set initial conditions.

- 11. Next, add a goal to find the force in the x direction. Do this by right clicking goals in the left-hand window, then select "insert global goals." Scroll down and check the box next to Force (x). Select the checkmark to insert the goal.
- 12. Next, add an equation goal to find the drag coefficient. This can be done by right clicking goals and selecting "insert equation goal." Add the equation shown to the right by clicking on the goal GG Force (X) in the left-hand window to insert it, and then typing in the rest by hand. Make sure to change dimensionality to no units. Rename this equation Drag Coefficient. This is the fluid mechanics drag equation  $F_D=.5*p*v^{2*}C_D*A$





quation Goal		8 8
Expression:		
{GG Force (X) 1}*2/(1.204*.003^2*3.14159*.025^2)	*	Undo Add
		Clear
	-	
4 5 6 · ) cos		
1 2 3 * în sin		
0 E . / exp tan		
Dimensionality:		
No units 🔹		
✓ Use the goal for convergence control		
OK Cano	el	Help

13. Run the simulation. A firewall window may appear, asking for permission. Cancel this, as it has no effect on the simulation. To view the simulation, look for the icon in the computer taskbar.

✓ Wizard New Clone Proj	ject <table-cell></table-cell>	General Settings	ini en	🧐	Run	Load/	Unload	₹	<ul><li>◊</li><li>◊</li><li>◊</li><li></li></ul>	Flow Simula	
Features	Sketch	Evaluate	DimXper	Offic	e Prod	lucts	Flow	Simula	ation		
Run		1					?	×	_		
Startup			🗌 Take p	revious re:	sults		Run			٥	
Solve Nev	v calculation						Help				
Con	itinue calcula	ation									
-CPU and m Run at:	This com	e puter		•							
Use	4	CPU(s)									
- Results pro	cessing after	r finishing the	calculation								
✓ Load re	sults			Batch Re	sults						

14. Selecting the flag in the flow simulation window will display progress on the goals, and when the goals are solved, the progress bars will be full. Wait until the

📀 So	olver: F	low Simulation [Def	ault] (Sphere2.SLDPF	RT)			
File	Calci	ulation View Inse	ert_Window Help				
1	н.;	> 0 🖈 🗎	0 🎮 📈 🔖	?			
					Ŷ		
ピ	Info	List of Goals					
Pa	rame	Name	Current Value	Progress	Criterion	Comment	
Sta	atus	Equation Goal 1	8.10562e-006 N	00%	0 N	No convergen	
FIL Da	ud ce	GG Force (X) 1	8.62292e-008 N	00%	0 N	No convergen	
Ite	ratio						
La	st ite						

simulation is finished solving. This simulation should solve in 89 iterations and will take approximately 5-10 minutes, depending on the computer.

15. Insert a cut plot to view the velocity around the sphere. Do this by expanding the results dropdown in the left-hand menu, right click on cut-plots and select insert. Select the front plane

as reference, and leave the default display choice of contours. Under contours, select velocity (x) and slide the slider all the way to the right for maximum steps between the minimum and maximum velocity.



🚫 Ci	ut Plot	?
<b>«</b>	× 6~ -⊨	
Sele	ction	*
	🔊 🗐 🖶	
	Front Plane	
ы	0 m	
Displ	lay	*
	Contours	
٢	Isolines	
1%	Vectors	
6	Streamlines	
	Mesh	
Cont	ours	*
	Velocity (X)	- []
↓ x	Global Coordinate System	
E <b>#</b>	255	\$
	3D profile	

Remove lighting by de-selecting the lightbulb icon in the flow simulation menu. This will brighten the image.





## Figure 1Flow cut plot on front plane

16. To export data, right click on Goal Plots, select insert and choose the goals for which data should be showed. Select show to display data in solidworks, or export to create an excel spreadsheet of data. Check the box for drag coefficient and select show to display the drag coefficient from the fluid analysis.

	Res	ults (4.fld)
-	÷#ŧ	Mesh
÷	Ø.	Cut Plots
1		🐼 Cut Plot 1
	l	🚸 Cut Plot 2
	$\diamond$	Surface Plots
	4	Isosurfaces
÷	۲	Flow Trajectories
	l	Flow Trajectories 1
	<b></b>	Particle Studies
	1.	Point Parameters
	Ŷ	Surface Parameters
	Σ	Volume Parameters
_	. <mark>12</mark>	XY Plots
1	Α.	Goal Plots
-	W	Report
L	200	Animations

	All Cools
۳.	
	GG Force (X) 1
_	
₽ <b>1</b> →x	Iterations -
₽ ↓→x	Iterations -
° ↓→x Optic	Iterations
p Dptic	Iterations
° Dptic S	Iterations

🗮 Table 💯 Chart   🍕 🗈											
Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress	Use In Convergence	Delta	Criteria		
Equation Goal 1	[]	4.1110614	4.1087576	4.1054757	4.1121136	100 %	Yes	0.0066378	0.1846560		

\*Front

17. Compare the data gathered from the case study to the graph of drag coefficient given. The case study is close to a Reynolds number of 10. Is the data exported from solidworks close enough to the actual value to be used?



24

18. Clone the study by right clicking on your project name in the projects window, then selecting clone. Rename the new project Flow Simulation: Re=10,000. Right Click on input data in the simulation window and select general settings. Under the initial and ambient conditions tab,



change the initial velocity from 0.003 m/s to 3 m/s. This simulates a Reynolds number of approximately 10,000.

General Settings		? 💌			
Parameter	Value	Navigator	S. Flaur Cinculation		
Parameter Definition	User Defined		Simulation		
Thermodynamic Parameters		Analysis type			
Parameters:	Pressure, temperature				
Pressure	101325 Pa	Fiulds			
Temperature	293.2 K				
Velocity Parameters		Wall conditions			
Parameter:	Velocity	Initial and ambient			
Velocity in X direction	3 m/s	conditions			
Velocity in Y direction	0 m/s				
Velocity in Z direction	0 m/s				
Turbulence Parameters					
OK Ap	Dependency ply Cancel Help				

19. Starting at step 11, run the new study to get a new drag coefficient. Make sure to update the velocity in the drag coefficient equation goal from .003 to 3 m/s to get an accurate result. Is the drag coefficient still accurate to the graph at Re=10,000? How large of a difference is there between the actual value and the value produced by SolidWorks?

Expression:	
{GG Force (X) 1}*2/(1.204*3^2*3.14159*.025^2)	lo Add lear
7       8       9       +       (       log       Physical Time         4       5       6       -       )       cos         1       2       3       ×       ^       sin         0       E       .       /       exp       tan         Dimensionality:       E       .       .       .       .	
Use the goal for convergence control	

