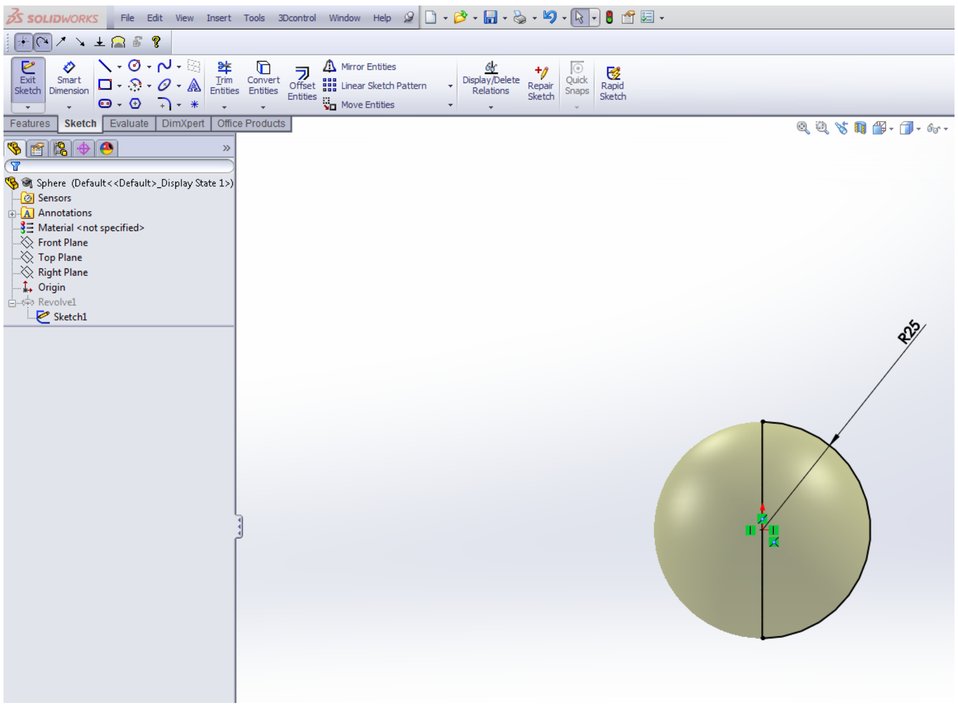
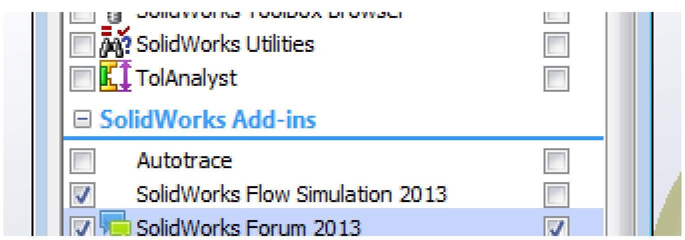
Finding Drag Coefficient using Solidworks Flow Simulation

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.

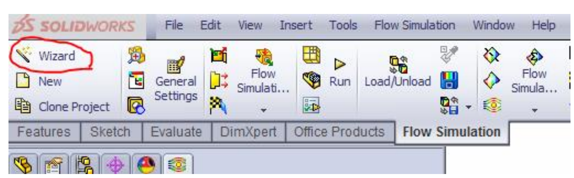
1. Model a sphere with a diameter of 50mm. This can be done with a revolved base.



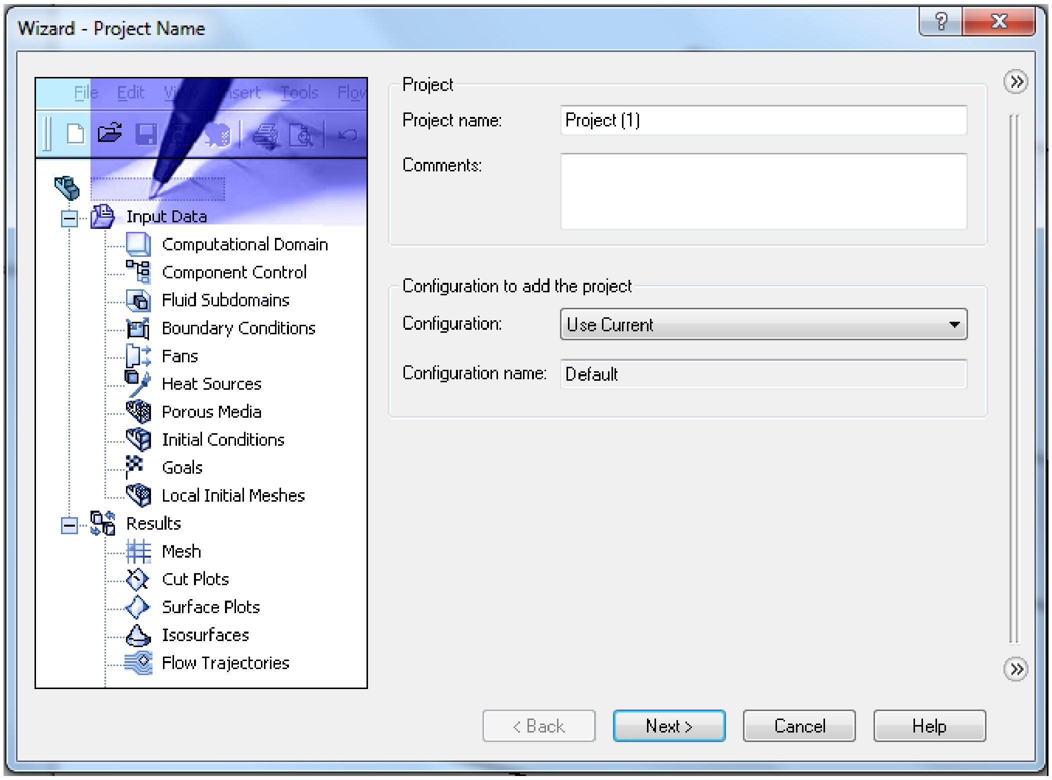
2. Go to Tools>Add-ins and load SolidWorks Flow Simulations 2013. Select OK.



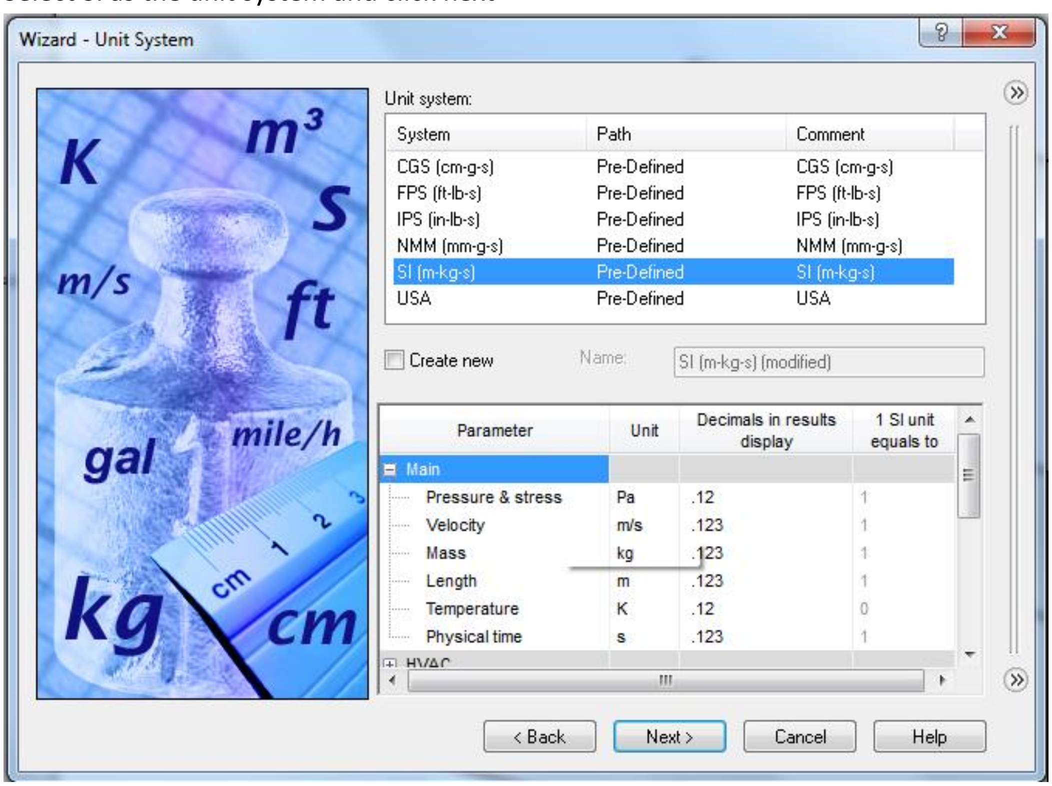
3. Save your document. In the flow simulation tab, select Wizard in the upper left hand corner to start a new simulation.



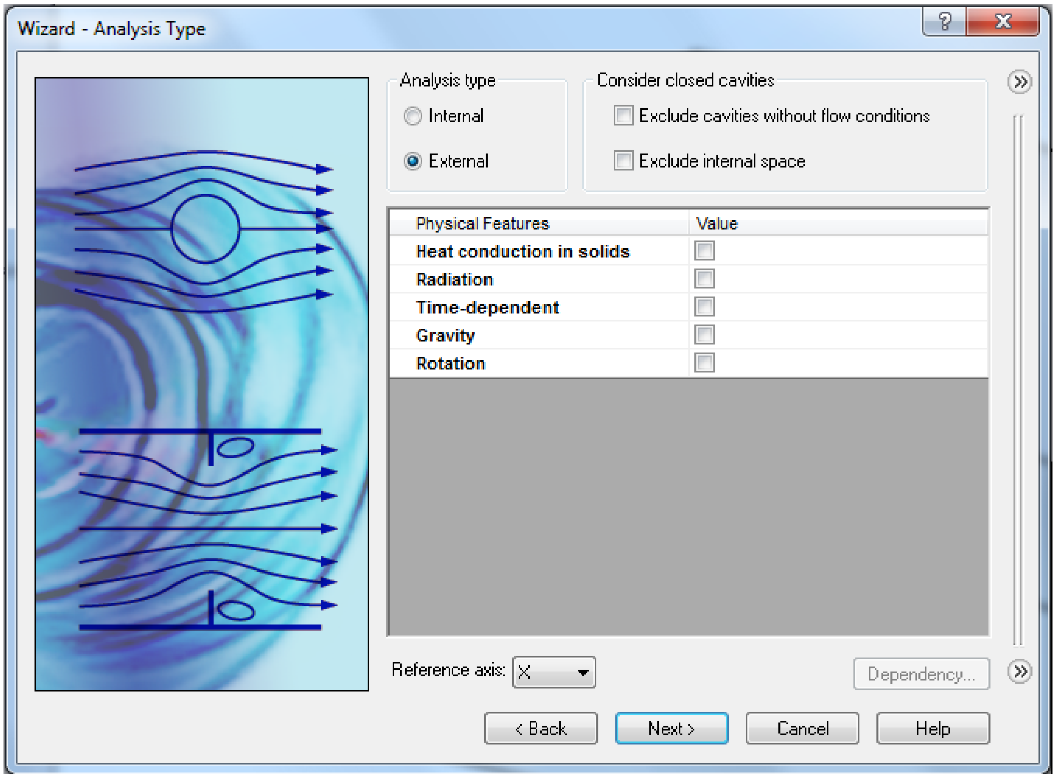
4. After selecting the wizard, name your project Flow Simulation and use default configurations. Click next.



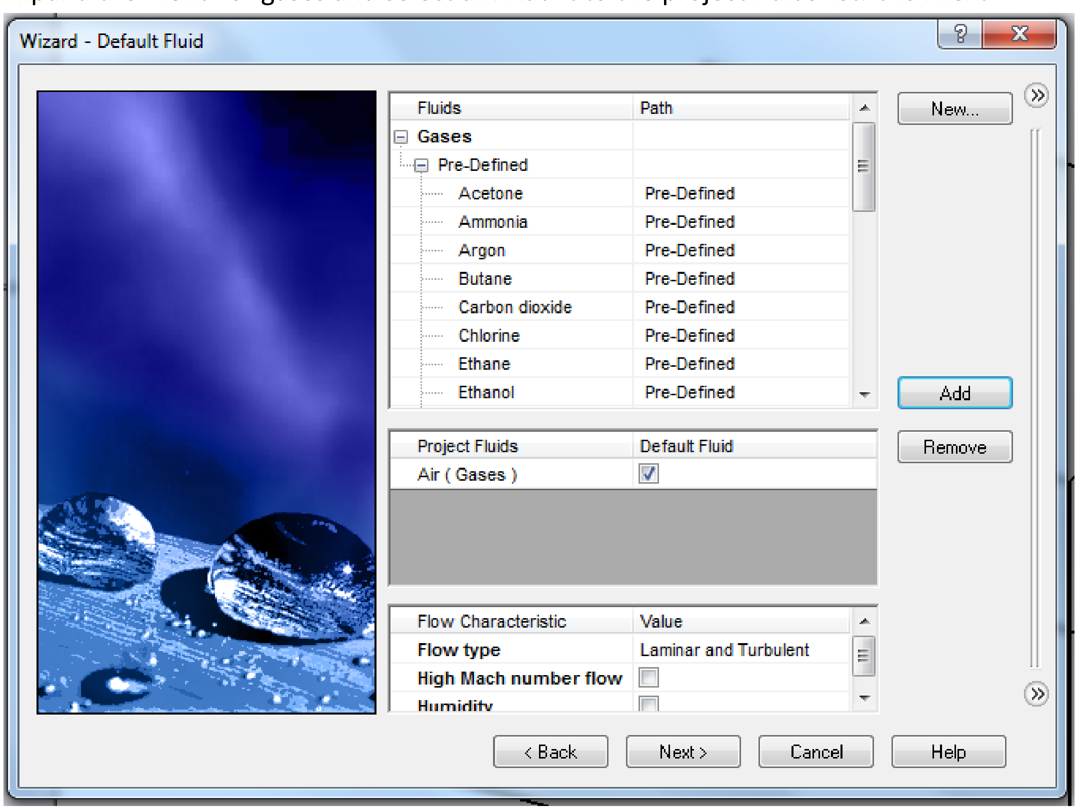
5. Select SI as the unit system and click next



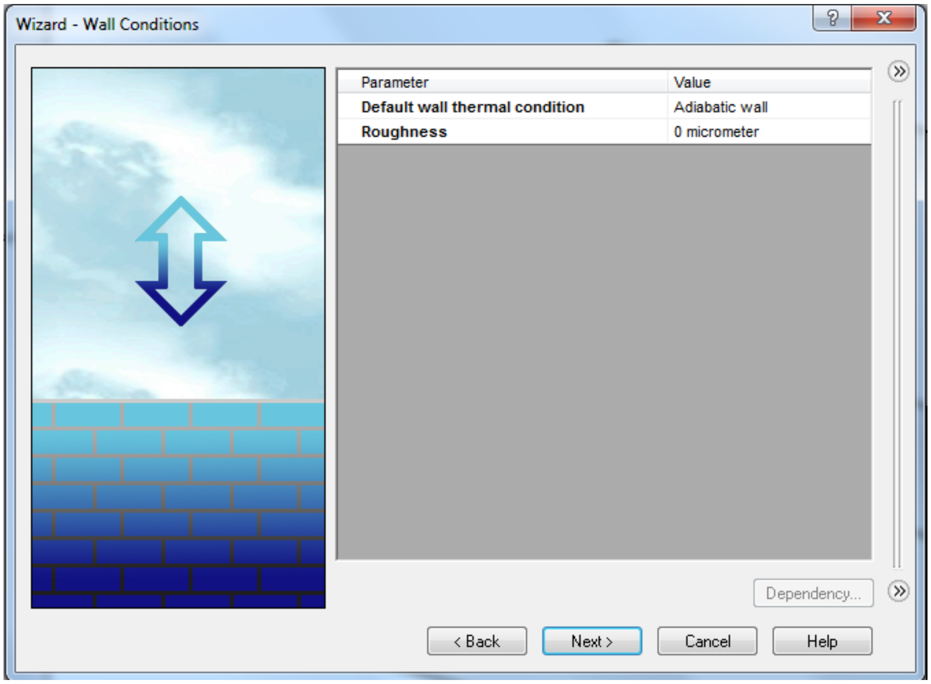
6. Select external as the analysis type and leave all other boxes unchecked. Click next.



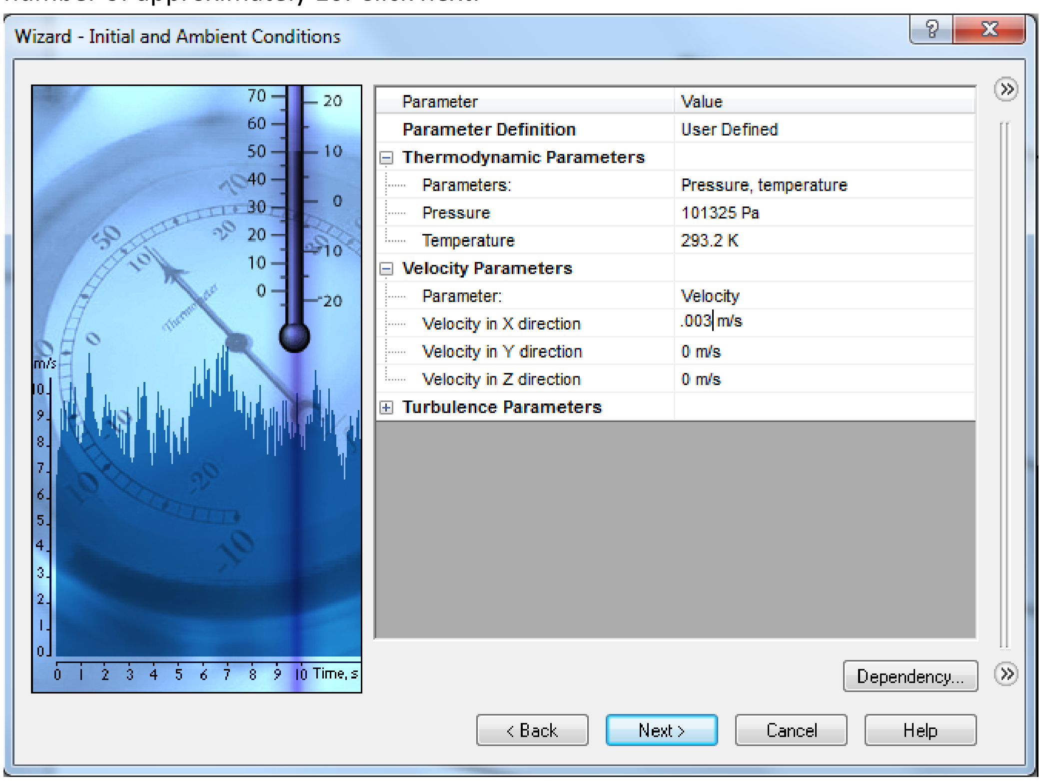
7. Expand the menu for gases and select air. Add it to the project fluids list. Click next



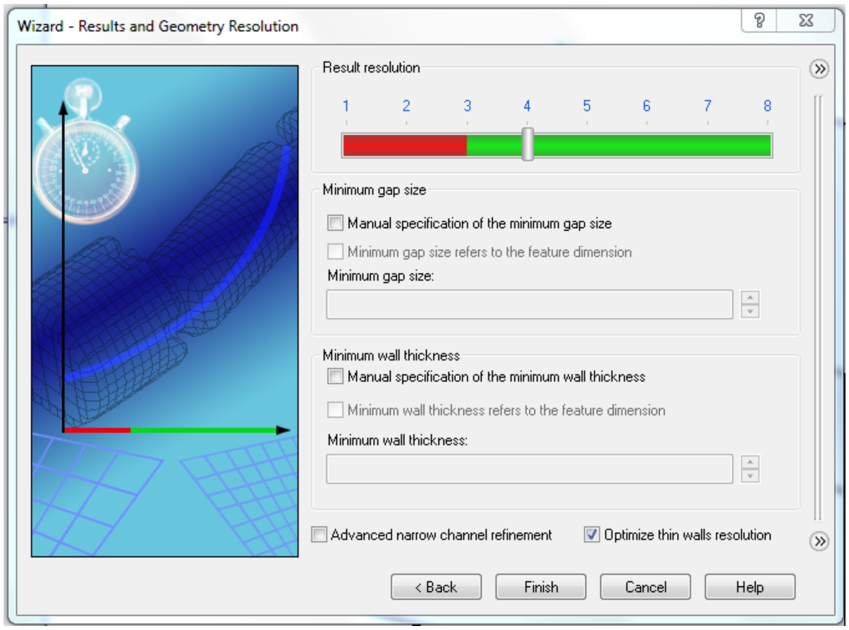
8. Do not change the default settings in wall conditions. Click next



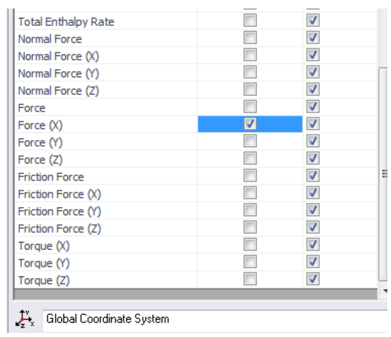
9. Under initial conditions, set velocity in the x direction to 0.003 m/s. This simulates a Reynolds number of approximately 10. Click next.

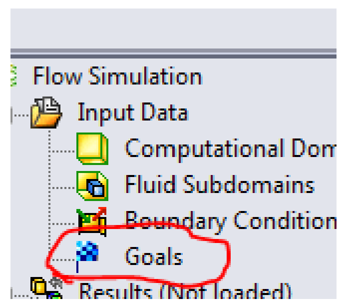


10. Move the slider from 3 to 4 to change the mesh resolution. Click finish to set initial conditions.

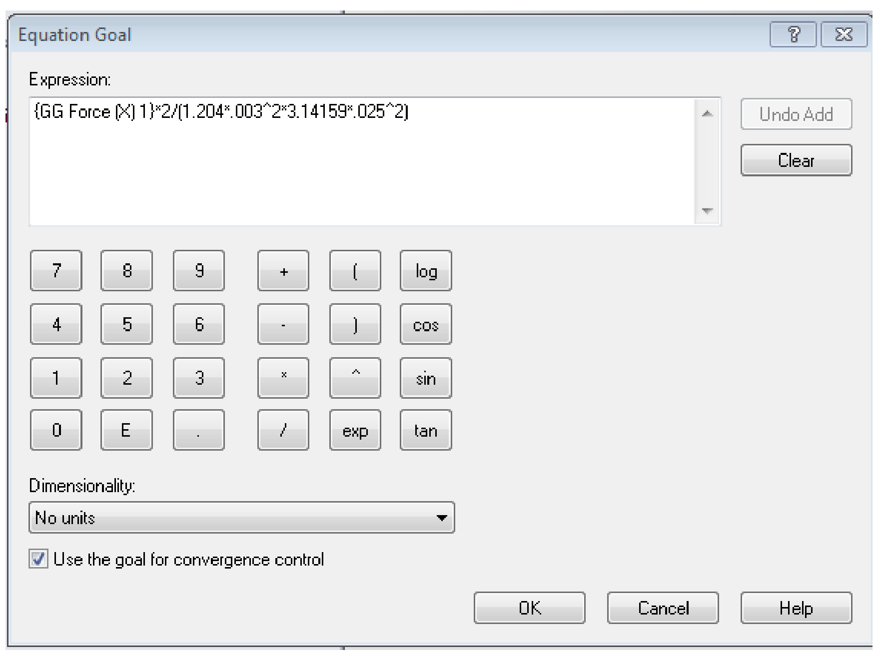


11. Next, add a goal to find the force in the x direction. Do this by right clicking goals in the left-hand window, then select "insert global goals." Scroll down and check the box next to Force (x). Select the checkmark to insert the goal.

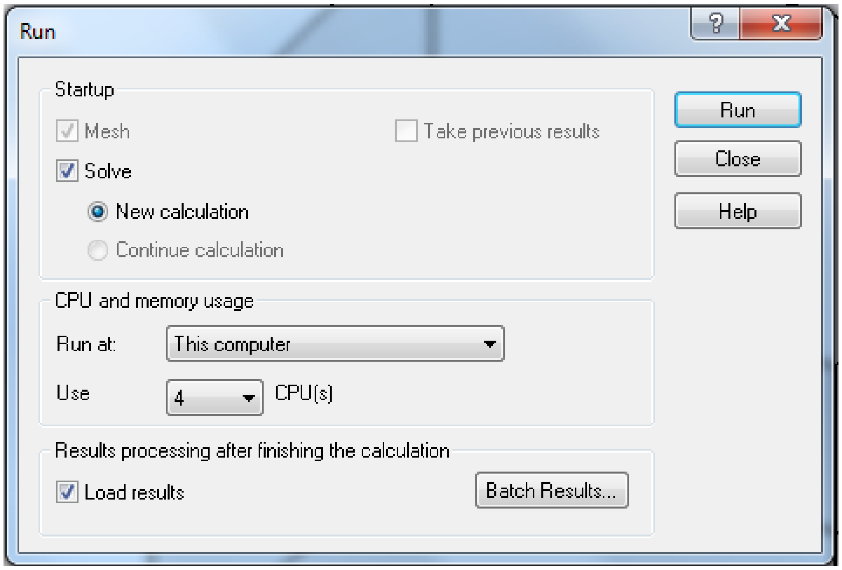


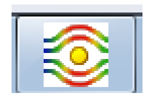


12. Next, add an equation goal to find the drag coefficient. This can be done by right clicking goals and selecting "insert equation goal." Add the equation shown to the right by clicking on the goal GG Force (X) in the left-hand window to insert it, and then typing in the rest by hand. Make sure to change dimensionality to no units. Rename this equation Drag Coefficient. This is the fluid mechanics drag equation $F_D=.5*\rho*v^2*C_D*A$

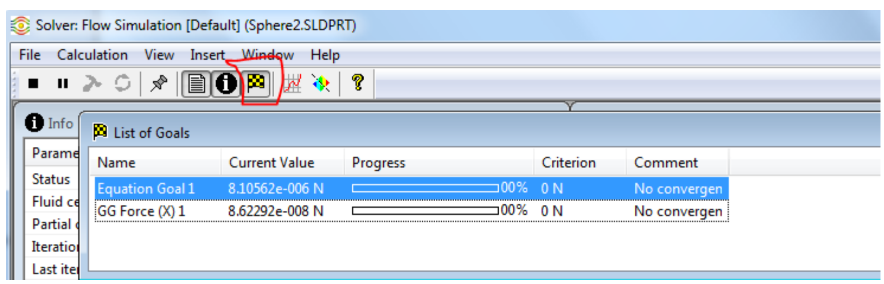


13. Run the simulation. A firewall window may appear, asking for permission. Cancel this, as it has no effect on the simulation. To view the simulation, look for the icon in the computer taskbar.

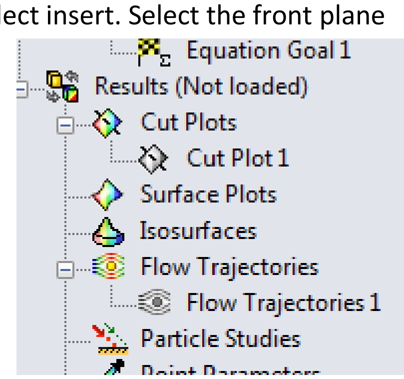


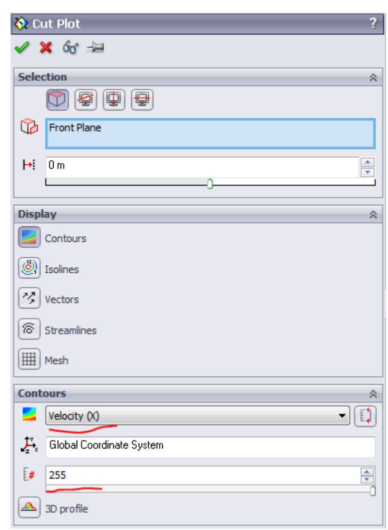


14. Selecting the flag in the flow simulation window will display progress on the goals, and when the goals are solved, the progress bars will be full. Wait until the simulation is finished solving. This simulation should solve in 89 iterations and will take approximately 5-10 minutes, depending on the 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.





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

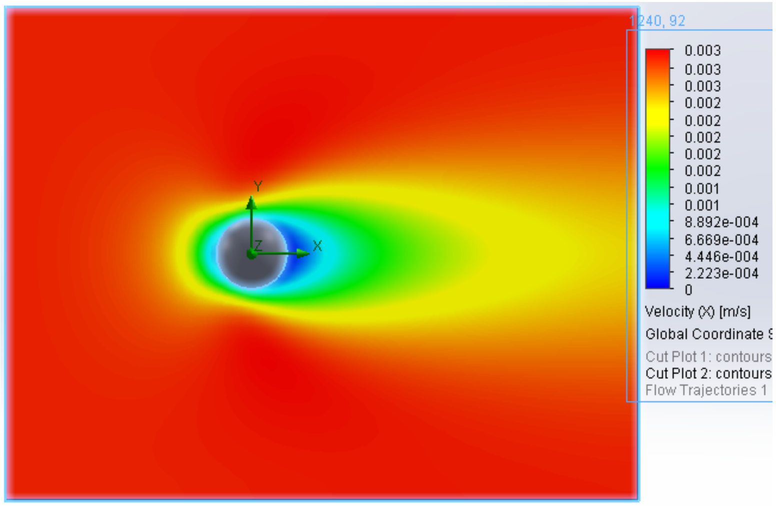
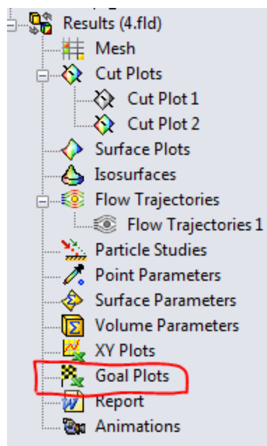
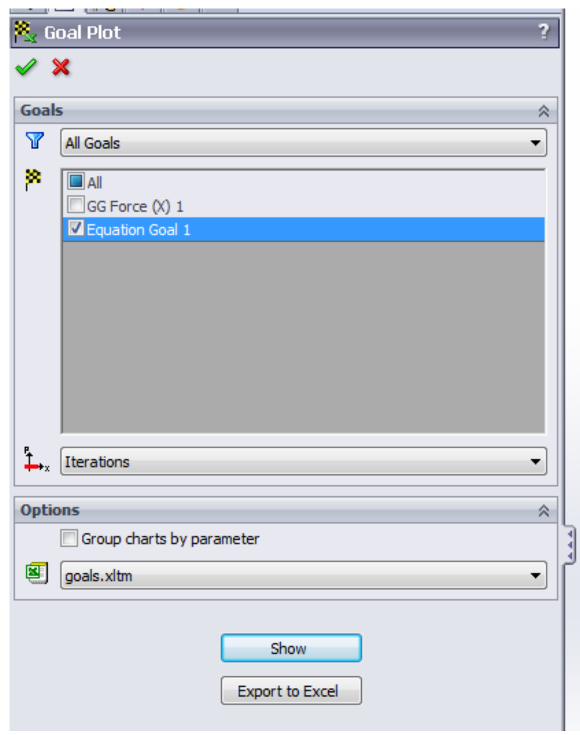


Figure 1Flow cut plot on front plane

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.



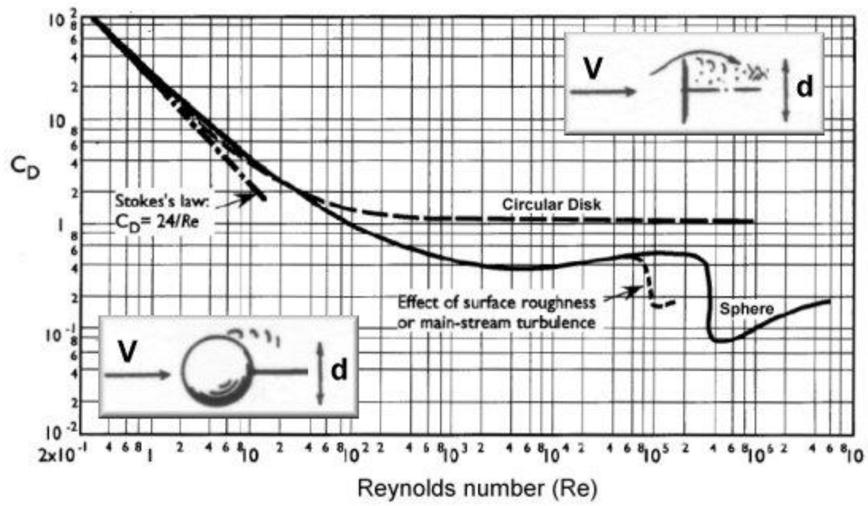


*Front

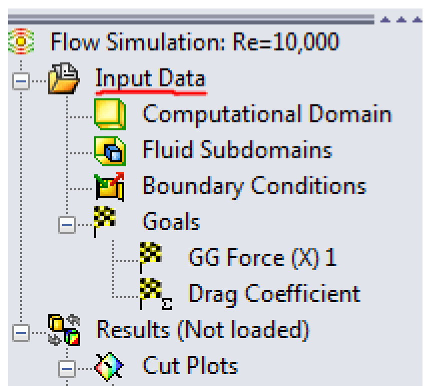
Table Chart

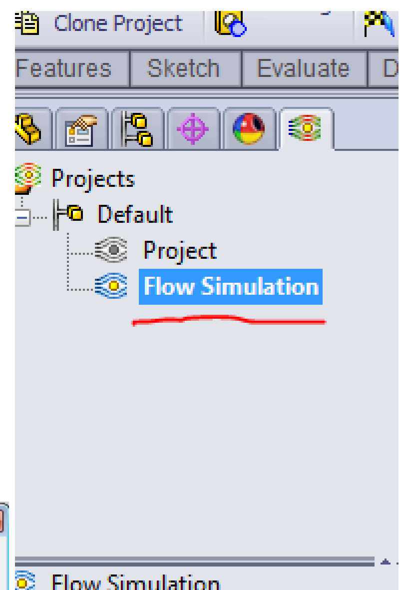
Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress	Use In Convergence	Delta	Criteria
Equation Goal 1	[]	4.1110614	4.1087576	4.1054757	4.1121136	100 %	Yes	0.0066378	0.1846560

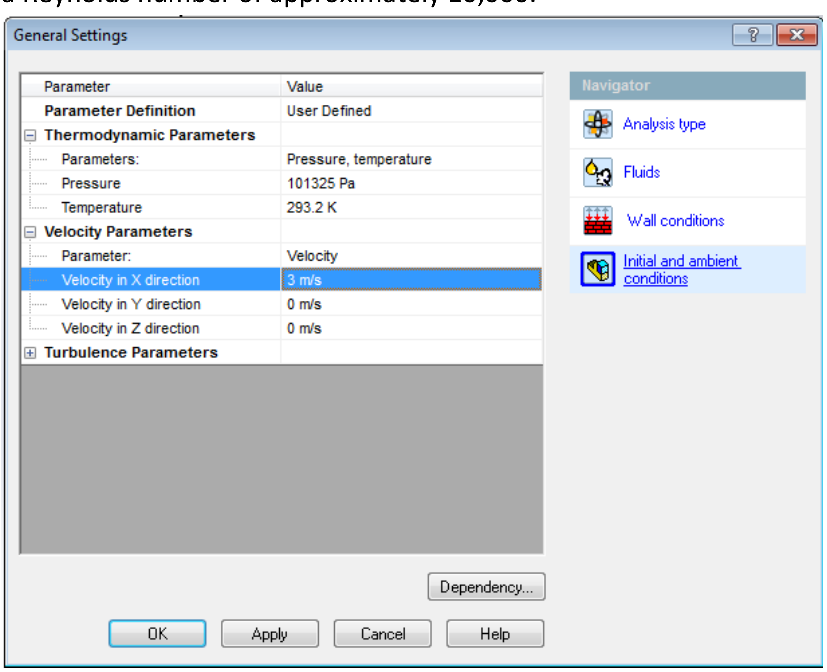
17. Compare the data gathered from the case study to the graph of drag coefficient given. The case study is close to a Reynolds number of 10. Is the data exported from solidworks close enough to the actual value to be used?



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.







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?