Analyze your car design using Autodesk Inventor with Autodesk Flow Design

Introduction

In this tutorial you learn how to quickly and easily analyze your F1 in Schools car using Autodesk® Flow Design which is an add-in for Autodesk® Inventor®. If you have not installed Autodesk Flow Design students can download a noncommercial version from: http://www.autodesk.com/education/free-software/flow-design (must be an active student or teacher with a register Autodesk Education account). A commercial version of the software can be purchased from: http://www.autodesk.com/store/flow-design and follow the instructions.
To view an introduction video for Autodesk Flow Design go to:
http://www.autodesk.com/products/autodesk-simulation-family/features/flow-design/all/gallery-view

Overview of Autodesk Flow Design

Flow Design is software that simulates air flow around vehicles, buildings, outdoor equipment, consumer products, or other objects of your choosing in a virtual wind tunnel. Flow Design technology is extremely geometry tolerant and easy to use, enabling you to begin seeing and understanding air flow behavior within seconds of starting the application. Results update almost in real-time in response to changes in wind-direction and speed that you specify. Visualization tools available within the Flow Design family of applications include 2D and 3D flow lines, shaded result planes, vector plots, and surface pressure shading. Quantified outputs include velocity, pressure, drag force, and drag coefficient.

Flow Design uses a revolutionary automatic meshing technology that can handle flow around any geometry at any stage of design. This technology is coupled with a transient, incompressible fluid flow solver and LES turbulence model in a way that delivers quick results and requires very little setup on behalf of the user.

Flow Design is available as follows:

- Flow Design standalone (installs on Windows and Mac and requires no CAD software)
- Flow Design for Inventor (add-in runs inside Autodesk Inventor 2014 or Autodesk Inventor Professional 2014)

Student notice

The design and dimensions shown in this course are for demonstration purpose only.

Check the F1 in Schools technical rules and regulations for design and dimension specifications.

Dataset files

Download the dataset file *Inventor_Course_F1_in_Schools_Dataset.zip.* Then extract the files, the default location is C:\F1 in Schools.

Free Autodesk® software

Autodesk is pleased to support your design efforts by making our industry leading 3D design software products available to students for free. Register and download software from http://students.autodesk.com.
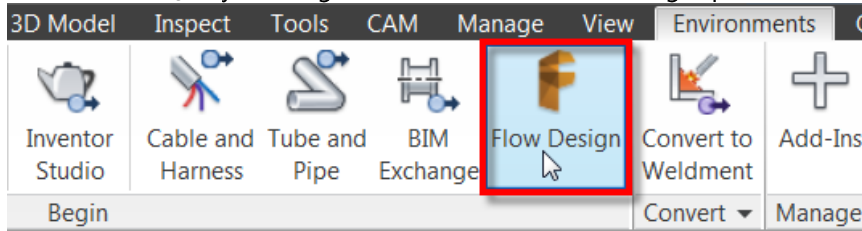
Author

Matthew Bell, Autodesk Education

Technical Editors

Paul Brincat, Autodesk Simulation | Dan Banach, Autodesk Education | Doris Fischer, Autodesk Education

Exercise

1) Start Autodesk Inventor.
2) Open the file *F1 Race Car Assembly - Appearance.iam.*
3) Start Flow Design by clicking the Environments tab > Begin panel > Flow Design.


4) The Flow Design tab with related tools will appear.
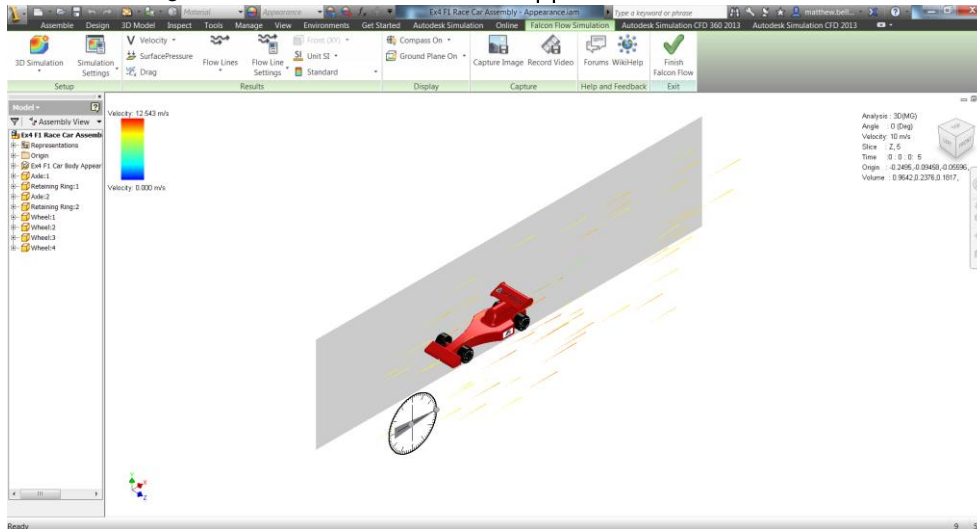
5) Adjust the speed of the air flow by clicking the Flow Design tab > Setup panel > Simulation Settings Simulation Settings_> Flow Speed.
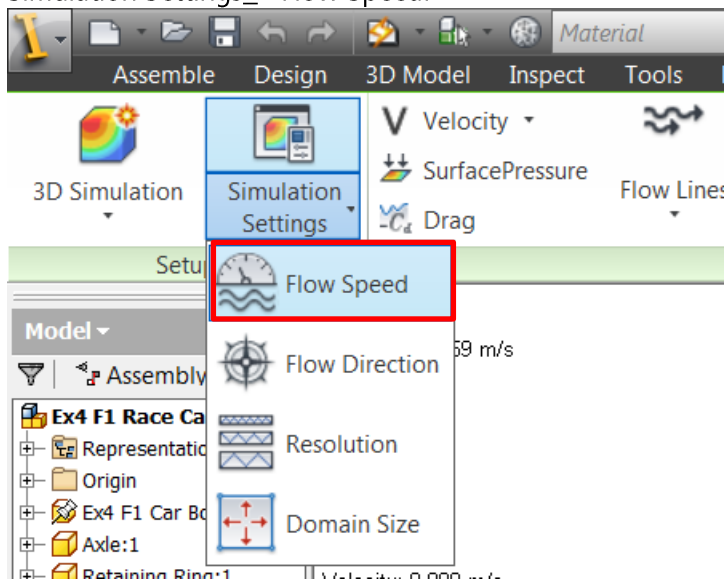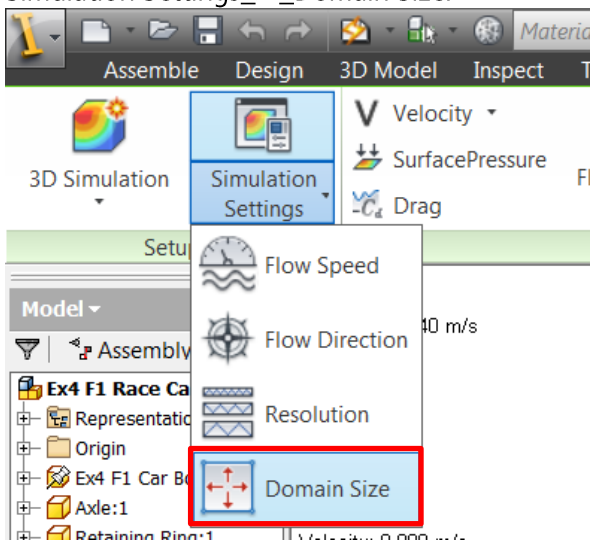

6) A Speed adjustment tool will appear.

7) Adjust the speed by clicking on the solid circle on the speed adjustment pop up and rotate clockwise to increase the speed, or counter-clockwise to decrease the speed.
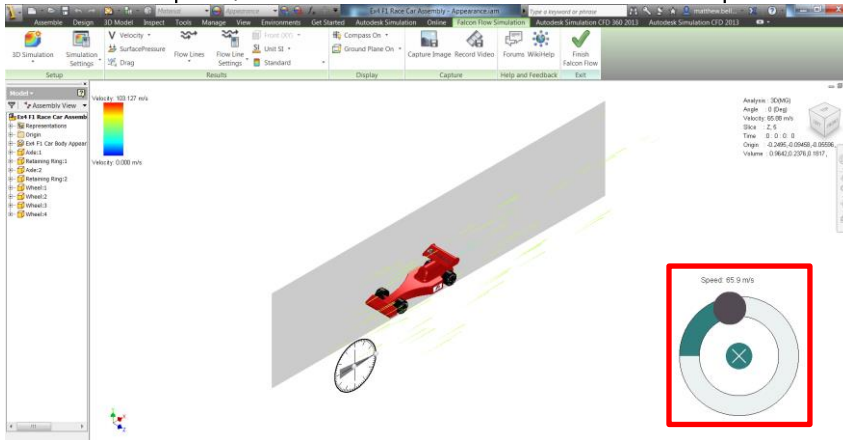
8) Next you change the simulation area, click the Flow Design tab > Setup panel > Simulation Settings Simulation Settings_> _Domain Size.

9) A Domain box will appear around the car. Adjust the domain size by clicking on an arrow on the domain box and drag to the required size.
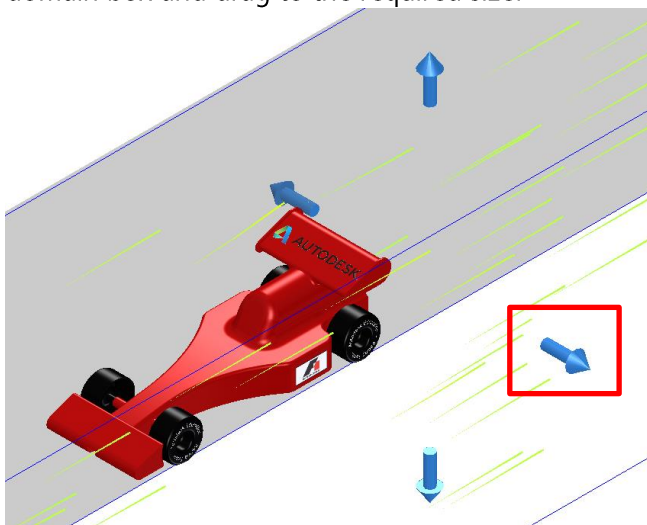
10) Click on the Left face of ViewCube to change the viewpoint so you are looking at the front of the car.
11) Replicate the track position by moving the bottom arrow up to the bottom of the wheels.

AUTODESK.
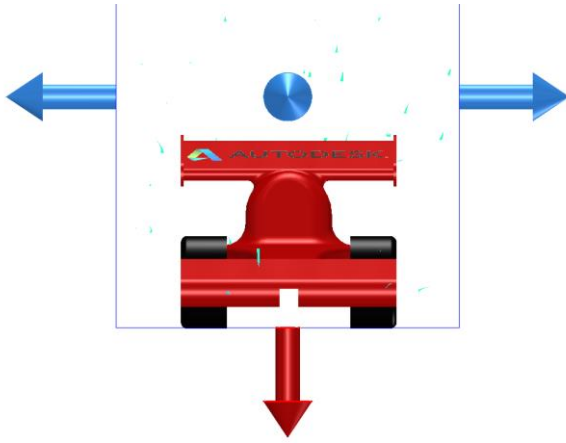
12) Next you adjust the number of flow line, click the Results panel > Flow Line Settings > Count.
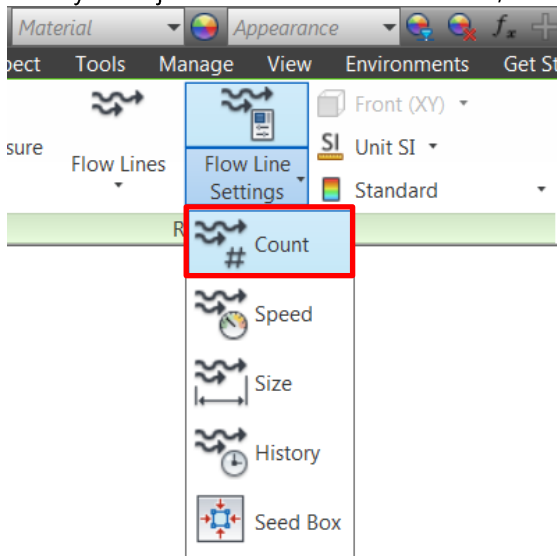
13) A Flow line count tool will appear.
14) Increase the number of flow lines by clicking and dragging the circle clockwise, or decrease the number of flow lines by rotating the circle counter-clockwise.
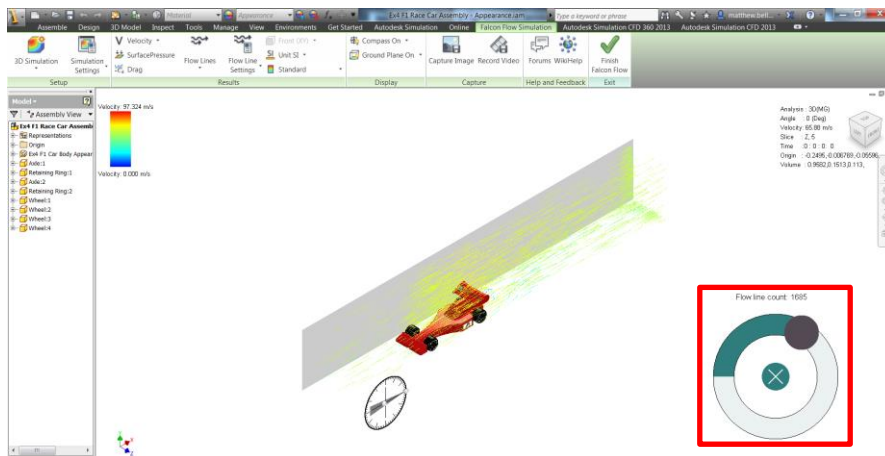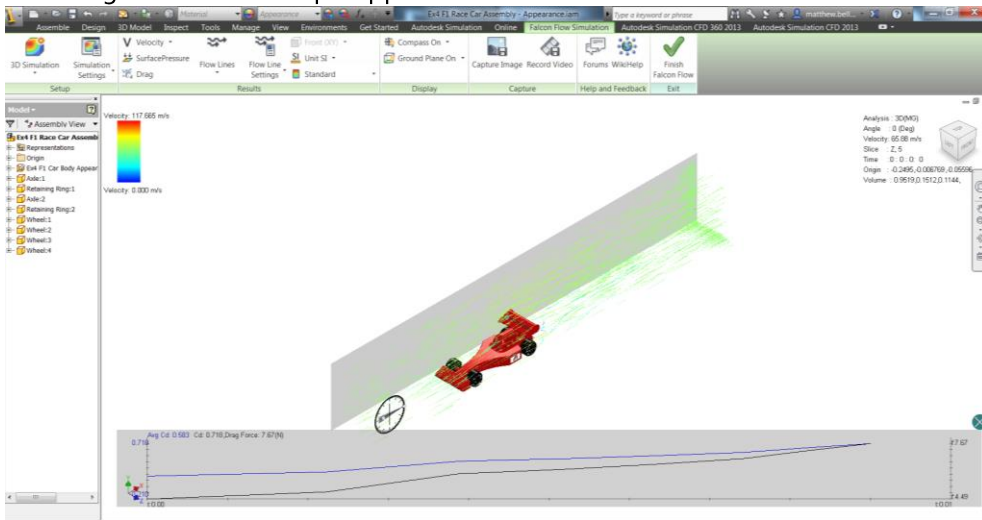

15) Change the viewpoint to the home position.
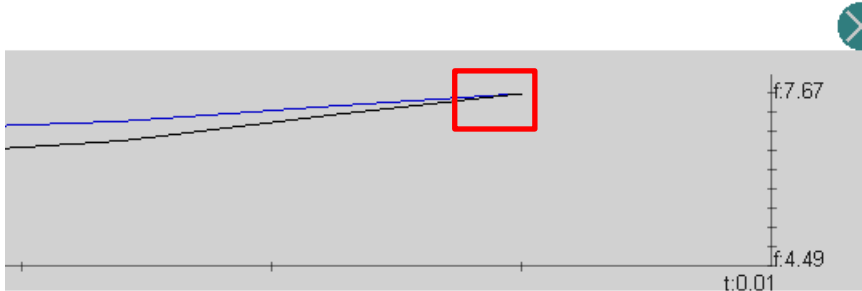16) Turn off the Flow line count tool by click the X in the center of the tool.
17) Next you adjust the drag on the car. From the Results panel > click Drag.

AUTODESK.
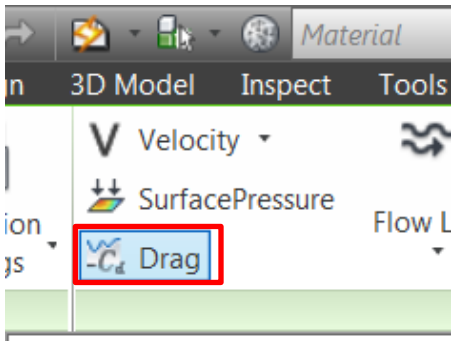
18) The Drag Calculation Graph appears at the bottom of the screen.

19) When two lines (black and blue) converge in the grid it means that the calculation is complete.

20) You can now take your readings for Drag Coefficient and Drag Force (ref appendices 1 for more explanation on this).
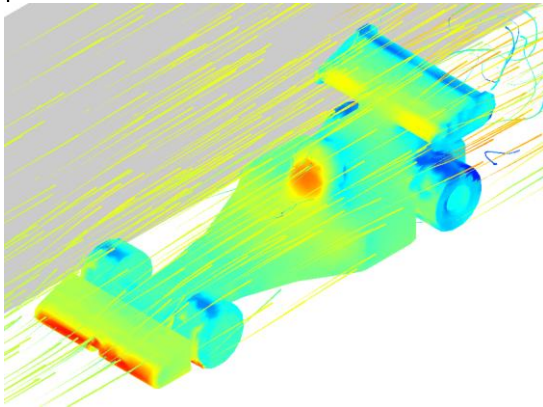Edit the features on the car body to see the affect the changes make to the airflow.

Avg Cd: 0.583 Cd: 0.718,Drag Force: 7.67(N)

AUTODESK.
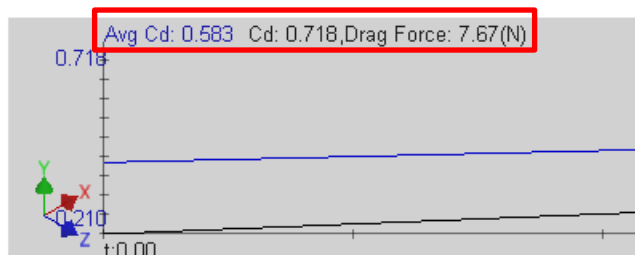
21) Next you analyze the surface pressure, From the Results panel > click SurfacePressure.
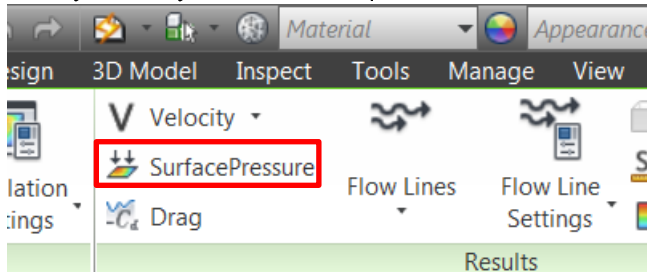
22) This will display the surface pressure on the car. Red indicates high pressure, and Blue indicates low pressure.
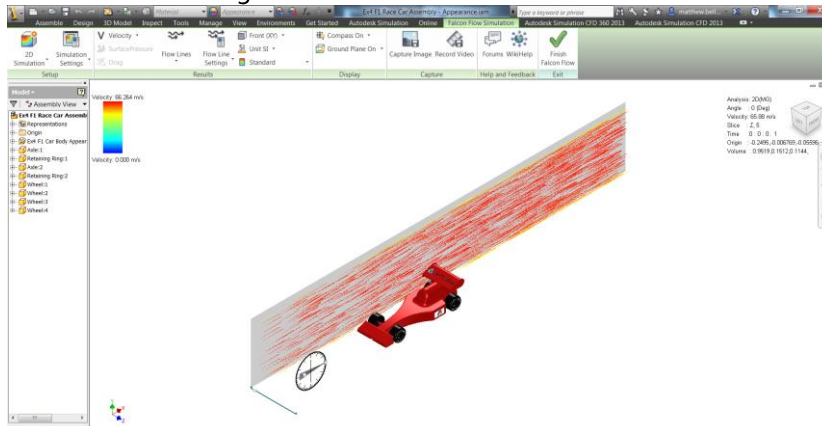

23) To complete the command click the SurfacePressure command again.
24) You can also analyze the car in a 2D plane. From the Setup panel > 3D Simulation > click 2D Simulation.
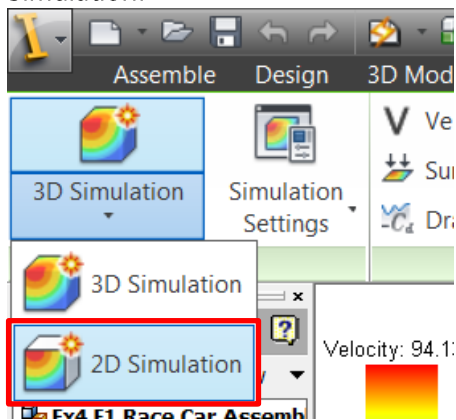
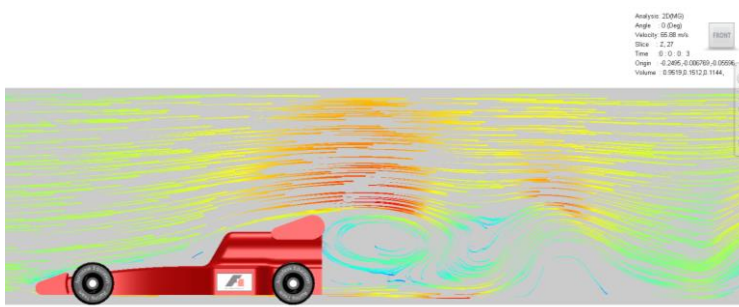25) This will now change the Domain box to a 2D slice.

26) You can move the position of the 2D slice by clicking and dragging the small ball on the sliding bar and moving it left or right to the desired position.
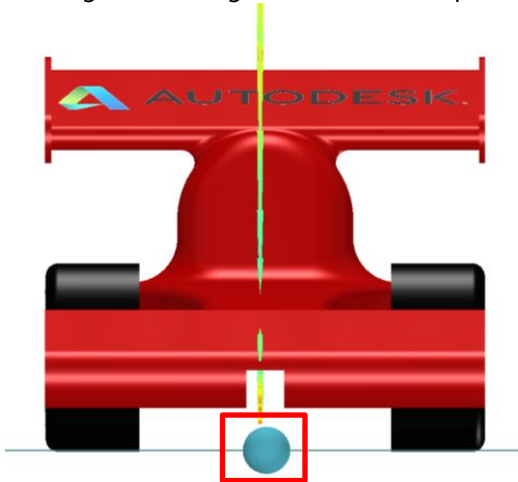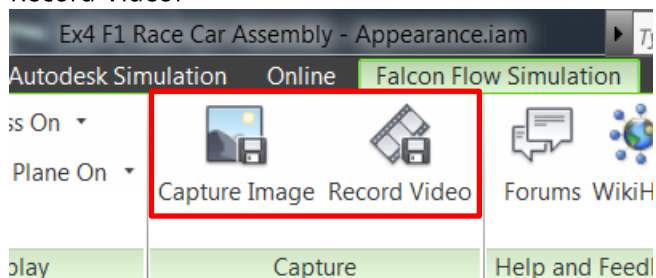

27) The results will automatically update in the graphics window.
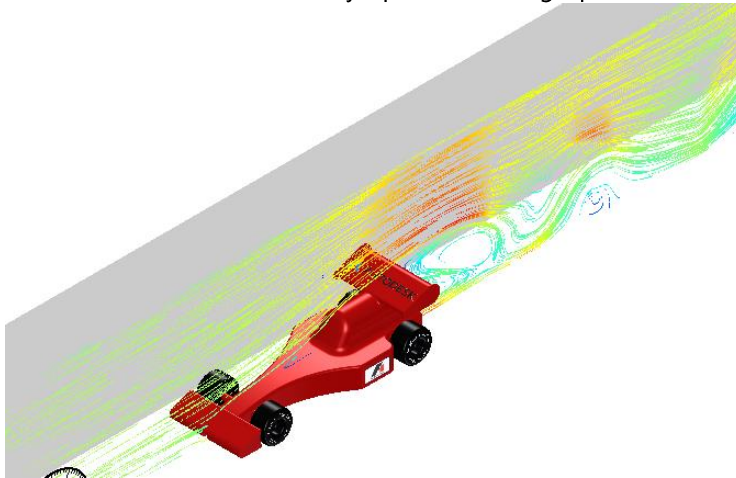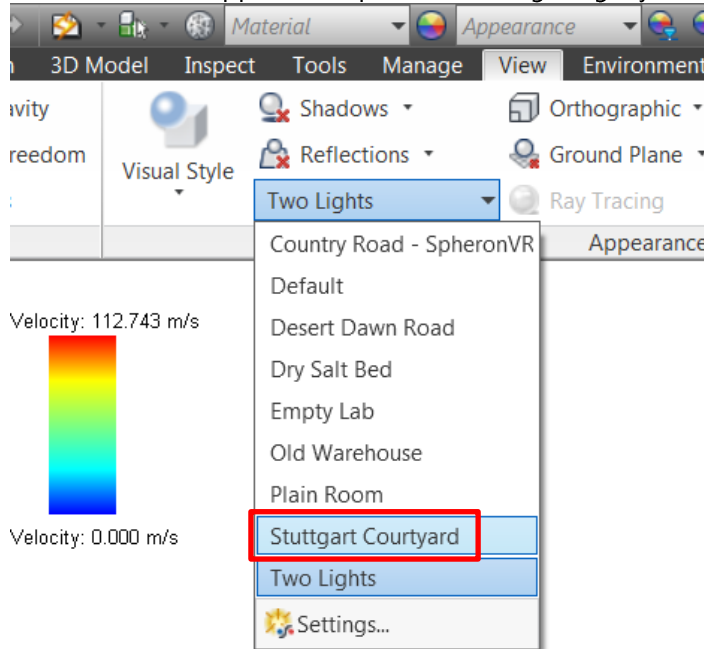

28) View the car from the side to see the effect.

29) You can save 2D snap shot or a video of the analysis by clicking the Save panel > Capture Image or Record Video.

30) You can also combine the results of Flow Design with the different lighting style in Inventor by clicking the View Tab > Appearance panel click a lighting style of your choice.
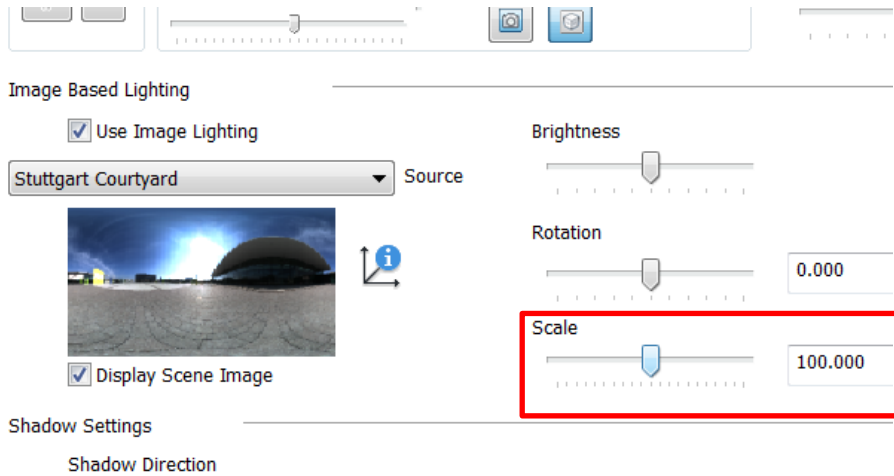

31) To make the model more realistic click Appearance panel > Perspective from the Orthographic drop down box.
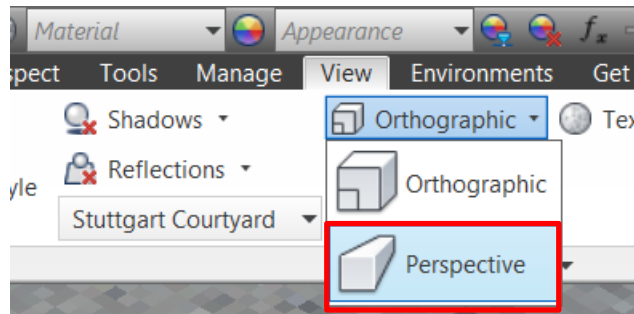
32) You can adjust the scale of the Environment via the Settings option.
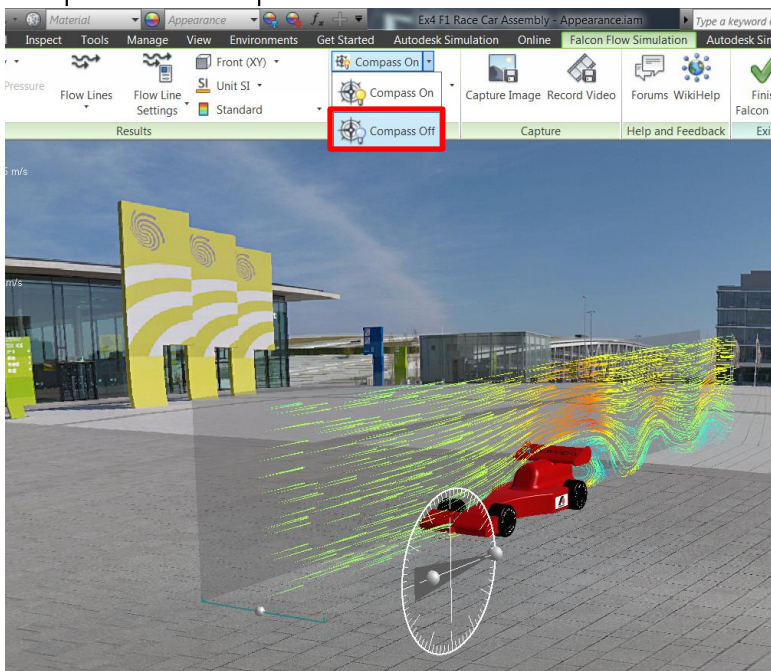
AUTODESK.
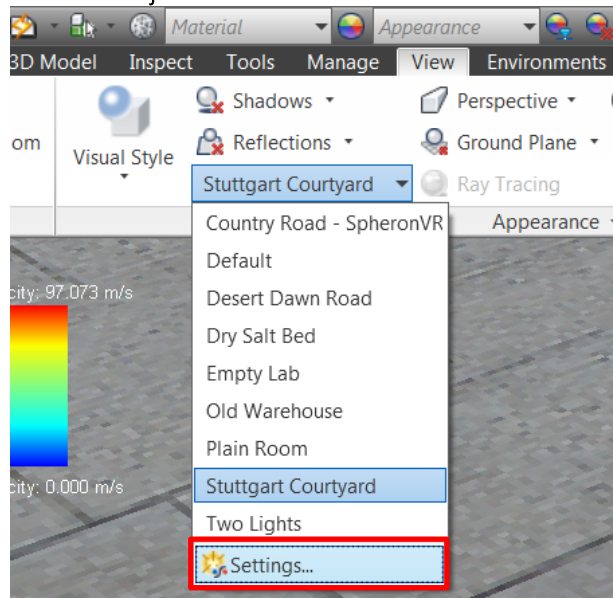
33) In the Style and Standard Editor dialog, box drag the scale slider until the image is proportioned to the car.
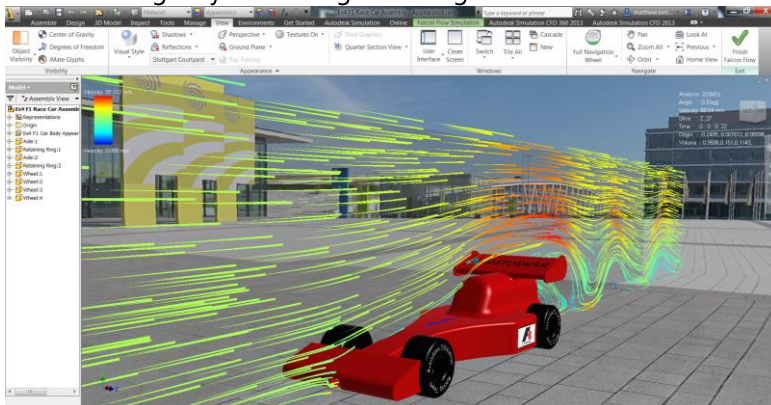

34) Make the Flow Design environment current by clicking the Flow Design tab.
35) You can toggle the flow direction compass on and off if desired by clicking the Display panel > Compass on or Compass Off.
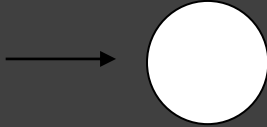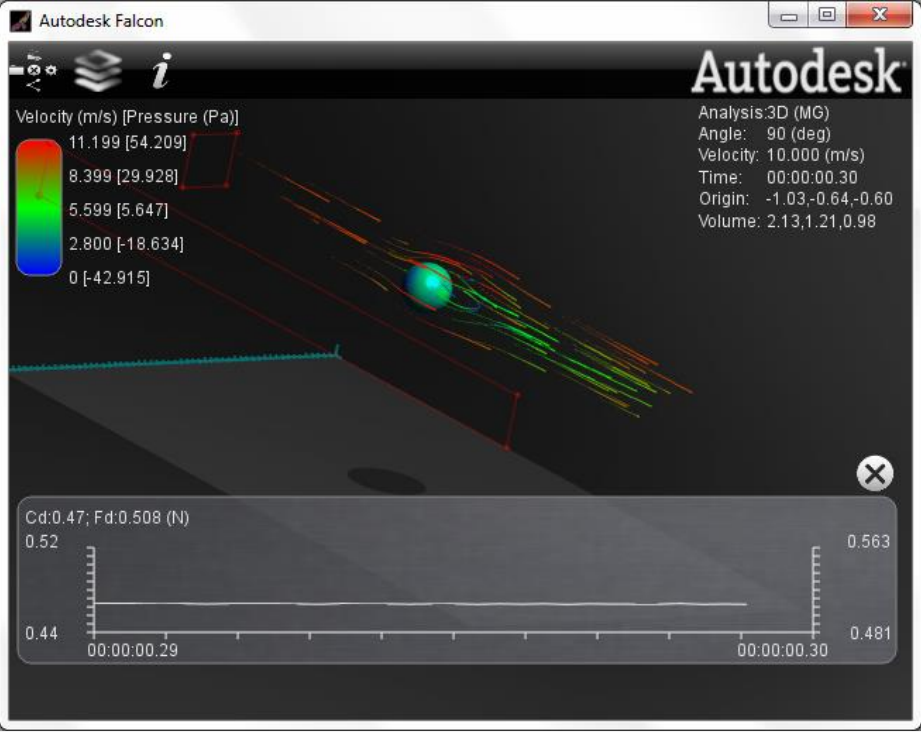

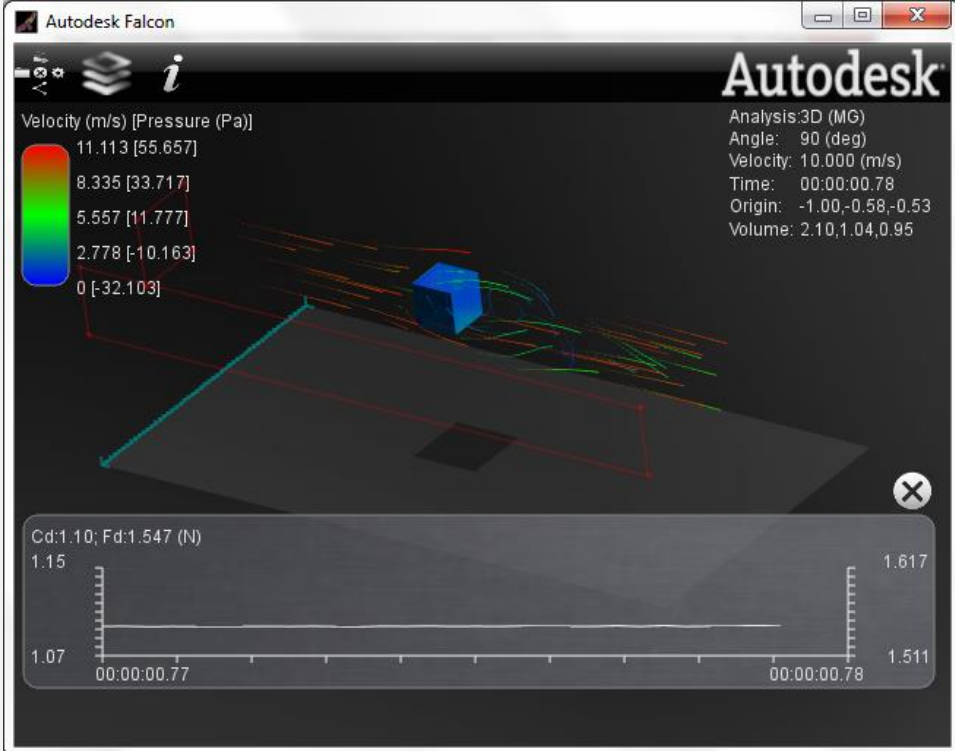36) This should give you some great images!

AUTODESK.

Appendix

Flow Design 2013 for Autodesk® software is a wind tunnel simulator that allows you to interactively investigate the aerodynamic performance of your design at any stage using a robust meshing technique.
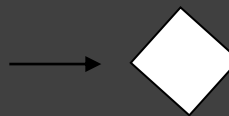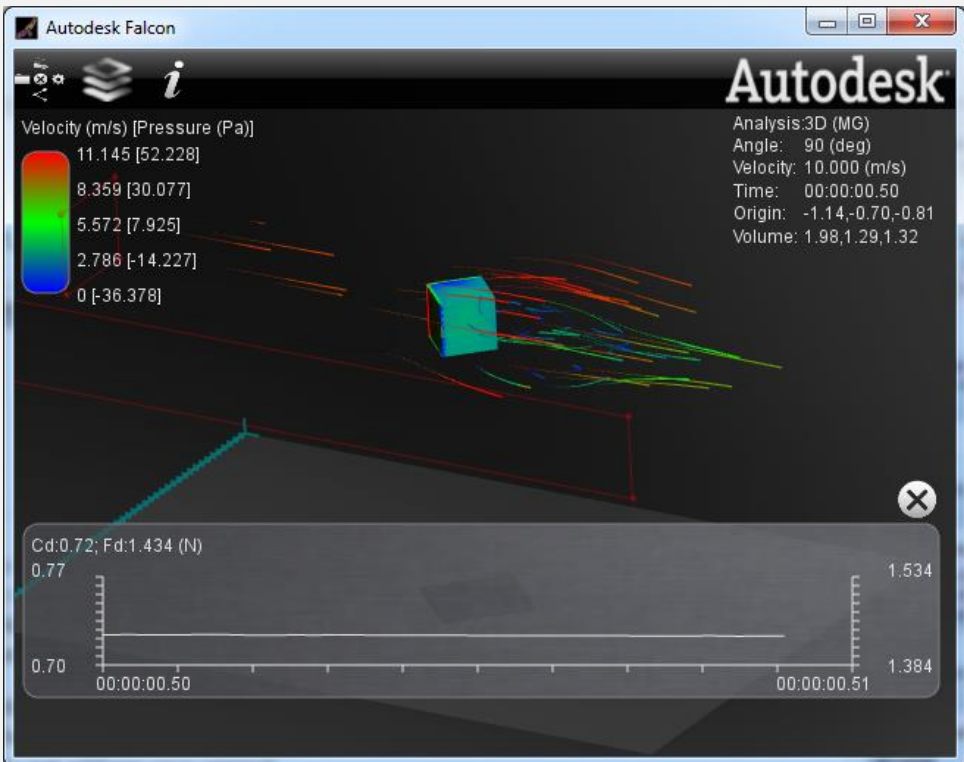
The solver technology that drives **Flow Design** utilizes the following CFD techniques to simulation the air flow over the body:
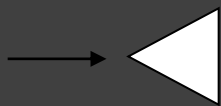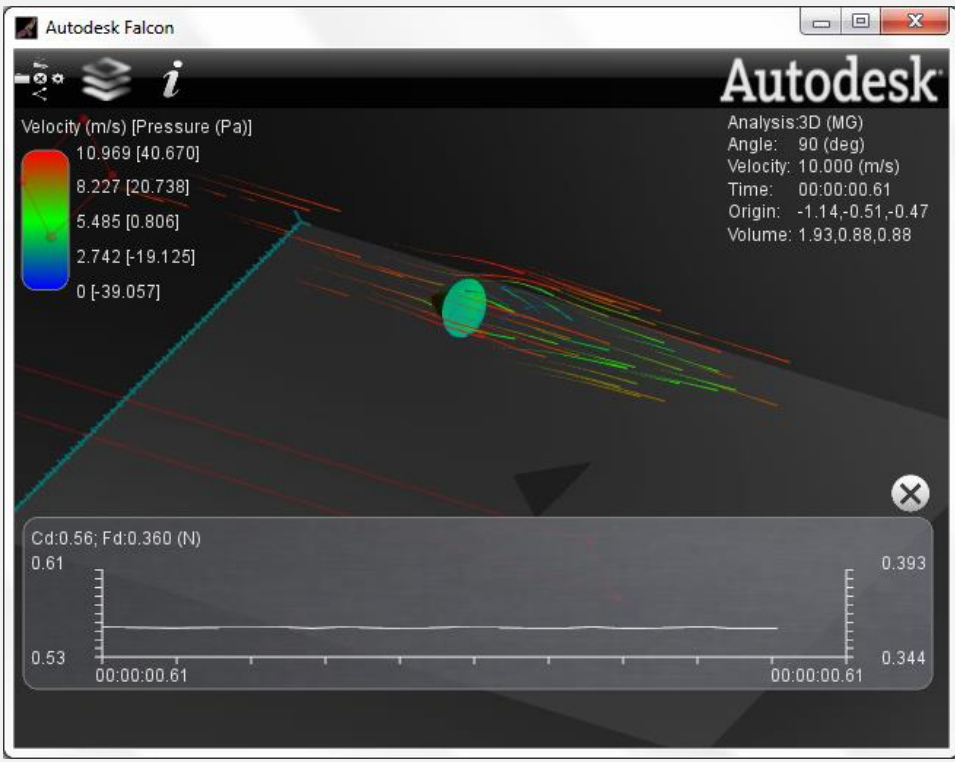
- Transient, Incompressible fluid flow solver
- Finite Volume Method
- Full 2D and 3D Navier-Stokes fluid solution
- LES turbulence model

To highlight the accuracy of this solver technology, **Flow Design** full 3D analysis was applied to a number of aerodynamic cases. As shown by the plot of the coefficient of drag, a measure of the aerodynamic drag of the object, in each of these cases **Flow Design** was able to settle on a value that closely matched the published values.

AUTODESK®

Case	Flow (10m/s) past a sphere (Ø150mm)
Observed	Drag Coefficient: 0.47 Source: http://en.wikipedia.org/wiki/Drag_coefficient
Flow Design	Drag Coefficient: 0.47 Analysis Resolution: 150%

AUTODESK.

Case	Flow (10m/s) past a cube (150mm)
Observed	Drag Coefficient: 1.05 Source: http://en.wikipedia.org/wiki/Drag_coefficient
Flow Design	Drag Coefficient: 1.10 Analysis Resolution: 150%

Case	Flow (10m/s) past an angled cube (150mm)
Observed	Drag Coefficient: 0.8 Source: http://en.wikipedia.org/wiki/Drag_coefficient
Flow Design	Drag Coefficient: 0.72 Analysis Resolution: 150%

Case	Flow (10m/s) past a 60° cone
Observed	Drag Coefficient: 0.50 Source: http://en.wikipedia.org/wiki/Drag_coefficient
Flow Design	Drag Coefficient: 0.56 Analysis Resolution: 150%

AUTODESK.