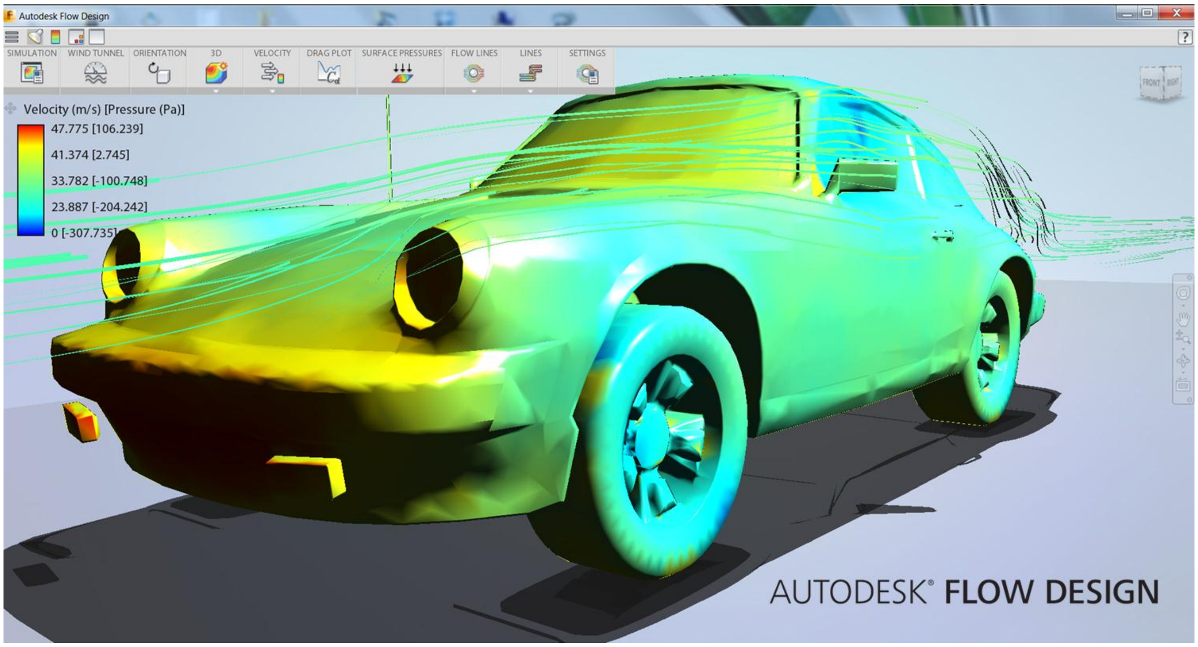
Flow Design Preliminary Validation Brief

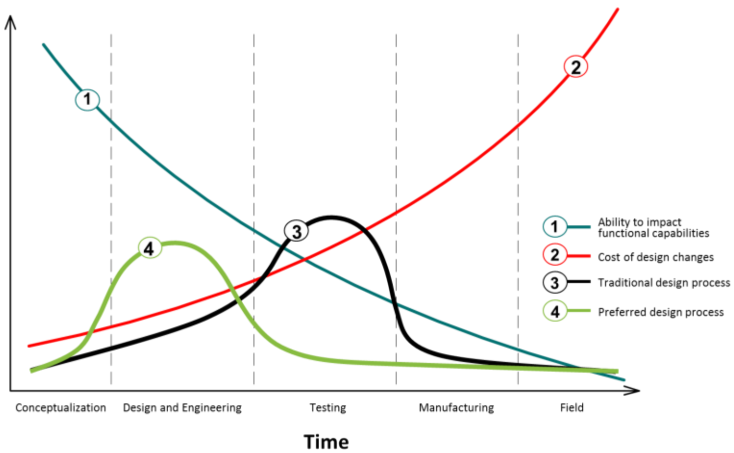


Rev 01072014

Introduction

From balancing drag and down force in automotive design, to ensuring pedestrian comfort in an architectural plan, design engineers need to understand how their design impacts or is impacted by external flow. Often, this insight is not attained until a near-final design undergoes detailed CFD analysis or wind tunnel testing. Unfortunately, by the time a design reaches that stage in the product development cycle, the ability to make a design change or improve a product's performance has declined while the cost of changes has gone up.

Product Development Lifecycle



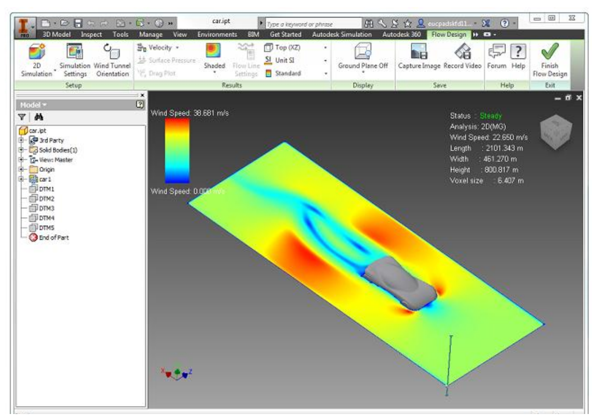
Increasingly, leaders in every industry are finding ways to move more of the product evaluation into the design phase of development, providing several advantages. Engineers are able to rapidly and inexpensively evaluate multiple design alternatives, optimizing on cost and performance. Design problems are found earlier and are thus less expensive to address and less likely to cause delays or rework later on.

Flow Design was developed to help designers understand and explore flow behavior early in the design process, providing a virtual wind tunnel on the desktop that models air flow around buildings, automobiles, sports equipment, or other consumer products. It allows designers to quickly see how air flow and wind interact with their models at various wind speeds and directions as well as provides estimates for velocity, pressure, and drag.

The following brief provides preliminary results of on-going validation studies undertaken to demonstrate its performance.

Inside Autodesk Flow Design

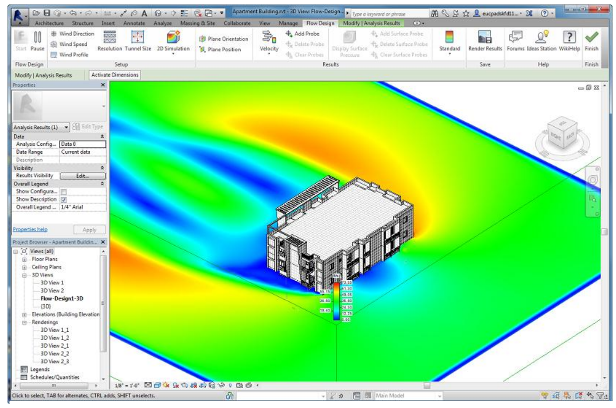
Flow Design shares many similarities with other Computational Fluid Dynamics (CFD) applications under the hood yet is tailored specifically for designers who do not require or have the time to learn and operate a full CFD application. Flow Design runs a transient, incompressible flow solver that using a Finite Volume Method approach. Turbulence is solved for using a Smagorinsky Large Eddy Simulation (LES) model similar to other CFD products. However, Flow Design was developed to be extremely geometry tolerant and not require geometry "cleanup" like other products. Its meshing technology is designed to accept geometry from the most widely used design packages. It accommodates both surface and solid 3D models and is not sensitive to small imperfections. Additionally, Flow Design was architected to start delivering results as quickly as possible and allow designers to explore the effect of changing conditions without having to set up new models, manage separate studies, or store large amounts of data. Flow Design is even available inside some CAD applications, where it can automatically build a wind tunnel around the CAD model and provide insight as part of the design process.



Flow Design for Inventor

Validation of Flow Design

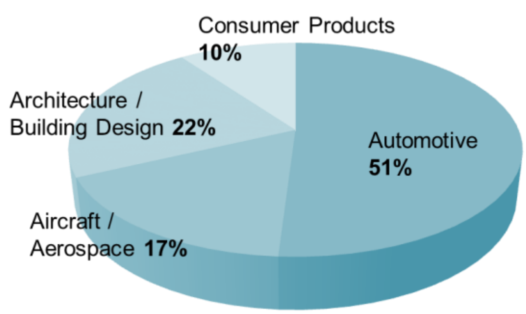
The main objective of Flow Design is to provide an understanding of how a vehicle, consumer product, or building will interact with the airflow early in the design stage. It will show where wakes will form, where there will be high and low pressure regions, and approximately where recirculation will occur. In automotive and consumer product applications, this can provide an indication of which features contribute to overall lift and drag - key factors that determine efficiency and aerodynamic downforce. In architectural applications Flow Design will show where air is stagnant in outdoor spaces, where elevated wind speeds may lead to a pedestrian comfort/safety problem, and where air will migrate from a source into the surrounding areas. These insights help inform the design of large buildings and campuses which must often provide for comfortable outdoor spaces with good air quality and natural ventilation.



Flow Design for Revit

The object of this paper is to provide initial validation of the simulation capabilities and demonstrate the level of agreement available with Flow Design as compared to established wind tunnel results or CFD analyses.

Early concepts of Flow Design were made available from fall 2011 through fall 2013 in the Autodesk Labs. Just over half of those users were running automotive applications and another 25% were running architectural applications. As such, preliminary validation efforts were focused on these two applications. Studies included:



Source: Project Falcon user survey May 2013 (n=124)

Automotive Study

- Qualitative comparison of observed behavior in wind tunnel tests and CFD analysis
- Quantitative comparison of simulated average drag coefficient against physical wind tunnel test results

Architectural Study

- Qualitative comparison of Flow Design results against published CFD analysis of low-rise building

In each of the cases, the standalone version of Flow Design was used with minimal changes made to the automated set-up in the tool in order to validate practical design workflows.

Results of these studies give an indication of Flow Design's viability as a design-level flow predictor in these applications.

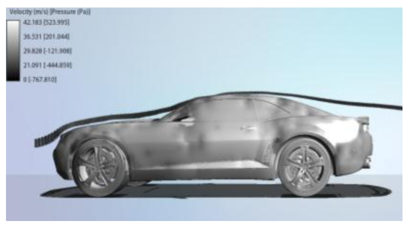
Automotive

Wind tunnel testing is commonly used in automotive design to help understand and optimize performance. But at possibly tens of thousands of dollars for each model and hundreds to thousands of dollars per hour for time in the tunnel, few concepts may actually make it that far. A virtual wind tunnel could provide valuable insight, particularly at the concept/shape-design phase. The following tests were conducted to demonstrate Flow Design's ability to approximate results of automotive wind tunnel testing and CFD analysis.

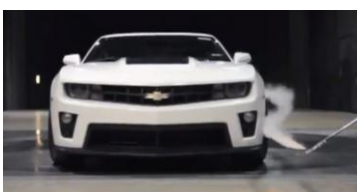
Qualitative Evaluation of Automotive Test

The following show comparisons to published wind tunnel and CFD tests. Results suggest very good correlation overall, with most major trends evident in the Flow Design results.

Chevrolet Camaro – Wind Tunnel and Flow Design Smoke Results



- Smoke is introduced ahead of grill.
- Slight separation is observed at the back end of the hood
- Flow remains attached over roof, sees mild separation on rear windscreen and exits a few inches above spoiler



- Smoke is introduced ahead of left front fender
- Slight separation is observed at the front fender we then see similar wrap and reattach behavior aft of wheel

(the torque report, 2011)

Chevrolet Volt - Wind Tunnel and Flow Design Smoke Results

- Flow introduced ahead of and slightly higher than front grill, remains attached over vehicle.
- Minimal turbulence upon passing over rear windscreen & spoiler.

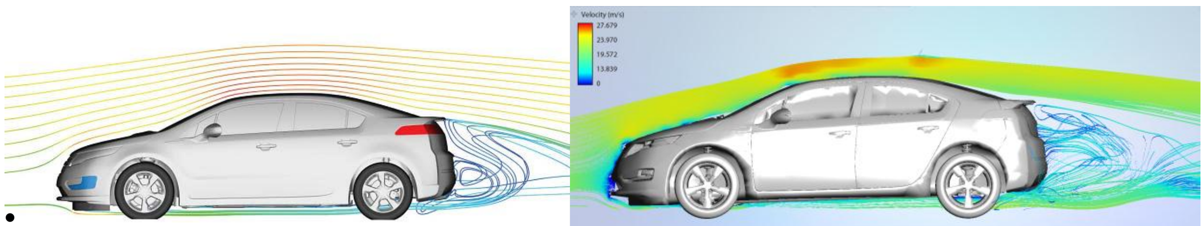
(TRANSLOGIC 67: Wind Tunnel, 2011)

- Flow introduced ahead of vehicle & remains attached over roof and rear windscreen

(Squatriglia, 2008)

Many auto makers are utilizing a combination of wind tunnel testing and CFD simulation. Such simulations can require a significant amount of model preparation, simulation set up, and computational run time. The following compares results from a detailed CFD analysis of a Chevy Volt to results from the standalone version of Flow Design, run using a simple 3D solid model of the car. Again, the results show a very good correlation in observed behavior.

Chevrolet Volt - ANSYS Fluent and Flow Design



(Shuba, 2010)

- Flow remains attached over top of vehicle nose to tail.
- Wake region draws air from underbody upward toward spoier to fill wake.

Wind Tunnel Simulation vs Experiment Study

To more thoroughly validate Flow Design, engineers at Autodesk teamed with expert aerodynamicists to test how well the tool could simulate an automotive wind tunnel test and predict drag.

The Test

Wind tunnel tests were conducted at the industrial wind tunnel facility at RMIT University, Melbourne, Australia. The team at RMIT performed tests typical to automotive bodies including measurements for drag force in a range of wind velocities. The model was then used to run a similar test in the Flow Design “virtual” wind tunnel.

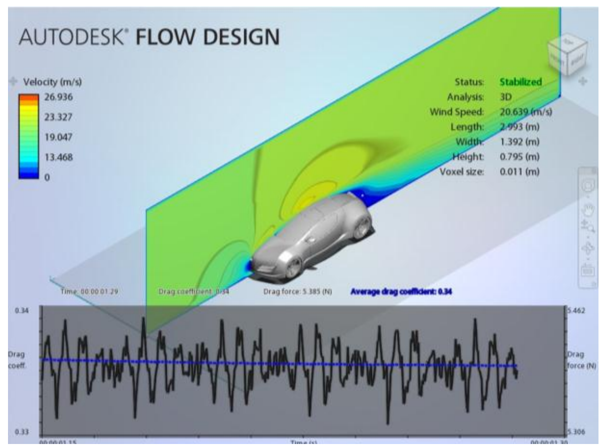
Physical Test

1/5 scale model car constructed using Autodesk Alias CAD model and 3D printing

The tunnel is a closed-jet return, constant cross-section, fixed ground wind tunnel with a test section 3m wide, 2m high and 9m long. Maximum test velocity is limited to 40 m/s (89 mph) and the longitudinal free stream turbulence intensity is 1.8%.

Velocity (m/s)	Fd (N)	Cd	Force Uncertainty (N)	Coefficient Uncertainty
6.275	0.80	0.554	0.083	0.057
8.874	1.39	0.482	0.070	0.024
11.966	2.34	0.447	0.074	0.014
14.957	3.32	0.406	0.077	0.009
17.545	4.44	0.394	0.074	0.007
20.639	5.81	0.373	0.074	0.005
23.402	7.24	0.361	0.073	0.004
26.790	9.23	0.351	0.075	0.003
29.493	11.06	0.348	0.075	0.002
32.468	13.04	0.338	0.076	0.002
35.218	15.26	0.336	0.075	0.002

Flow Design Simulation



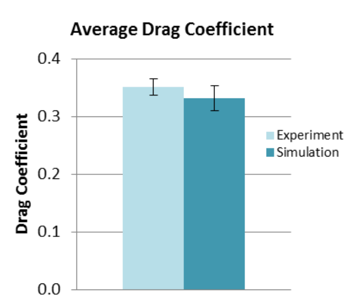
The standalone version of Flow Design was used with tunnel dimensions widened to ½ the size of the RMIT tunnel. Tests were run at 3 different resolution settings for 3 velocities.

Velocity [m/s]	Resolution	Result Time [s]	Fd	Cd
6.27	100	16.66	0.457	0.308
	141	9.86	0.454	0.307
	200	3.95	0.505	0.341
20.64	100	4.93	5.535	0.346
	141	3.09	4.970	0.310
	200	1.29	5.376	0.336
35.22	100	4.57	17.141	0.368
	141	1.66	14.714	0.316
	200	0.78	16.245	0.348

The Results

The chart at right shows the average drag coefficient determined by wind tunnel experiment compared to the same determined by Flow Design simulation. Error bars indicate the standard deviation of data in each.

The results show that Flow Design was able to predict the wind tunnel results within 6%.



Architectural Applications

Comparison to CFD Analysis/Test of Low Rise Building

External flow analysis can provide several key insights for architectural applications. Results are used to supplement requirements for assessing wind loading, analyzing impact to surrounding areas, and evaluating air quality. A common use for such analysis is to verify pedestrian comfort levels.

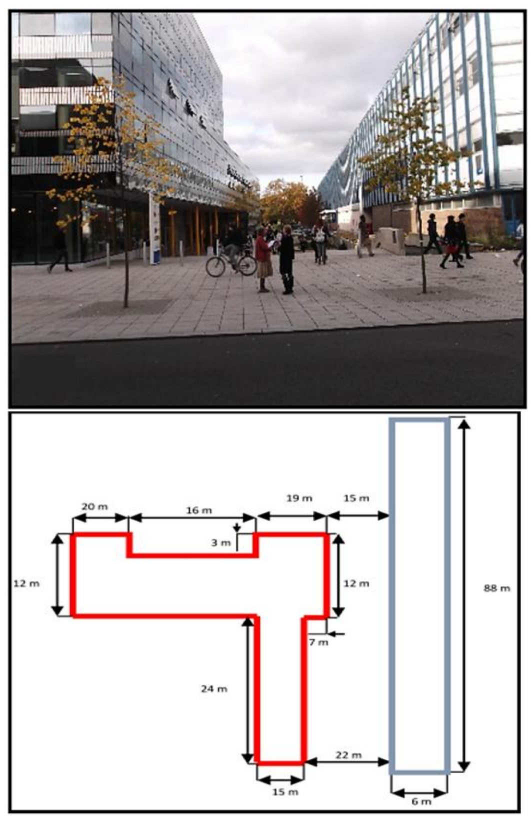
Architectural applications by their nature involve a much greater degree of uncertainty. First, wind speeds and directions are never fixed in real life; often the best the designer can do is consider statistically representative fixed wind speeds to study behavior. Next, buildings are also part of a broader landscape with other buildings and topography that are difficult or impossible to fully account for in a traditional CFD or physical wind tunnel test. Finally, there is far more potential for variation in construction techniques and materials as compared to manufactured products. Therefore for architectural applications involving pedestrian comfort, wind/wake, or contaminant studies the most important capabilities concern flow distribution and relative velocities, both of which point to potentially problematic areas. It is often more critical to match up with the overall distribution than it is to exactly match velocities on an absolute scale. Therefore, a practical goal for architectural applications is the ability to report flow distributions and show trends under different wind conditions.

The Test

Flow Design was compared to results from a published pedestrian comfort study of Coventry University Central Campus (Fadl & Karadelis). The study evaluated effects of a newly constructed building named “The Hub”. As part of the study, researchers at Coventry compared physical test measurements with simulations of common wind conditions modeled in ANSYS Fluent.

For the study, simulations were run at 4 wind directions with a wind velocity profile modeled with considerations for gradient height, type of terrain, dissipation rate, etc. Other models considerations for turbulence model, wall effects, etc. are detailed in the published report.

Simulations in Flow Design were run at the same at 4 different orientations using simply a 10 m/s wind speed (uniform, no gradient) and tunnel size large enough minimize wall effects.



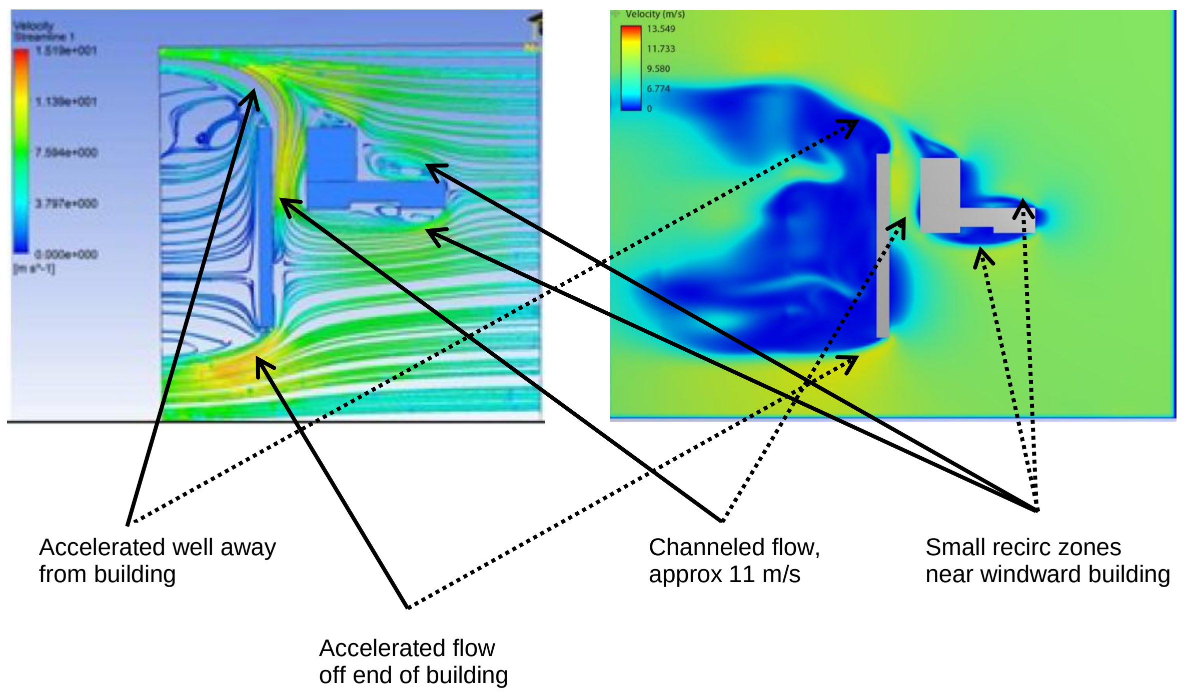
A view of the pedestrian area between the Hub and James Starley building (above) and main dimensions of the buildings (below). (Fadl & Karadelis)

The Results

The building geometry was created and input into Flow Design in less than 5 minutes. All 4 simulations were run in less than 2 hours. The following qualitative comparisons with ANSYS Fluent showed that major trends were captured, including a critical canyoning or channeled flow effect that occurs between The Hub and an adjacent building with wind coming from the East. This comparison suggests Flow Design is very well suited to predicting pedestrian comfort conditions and trends well with varying wind directions.

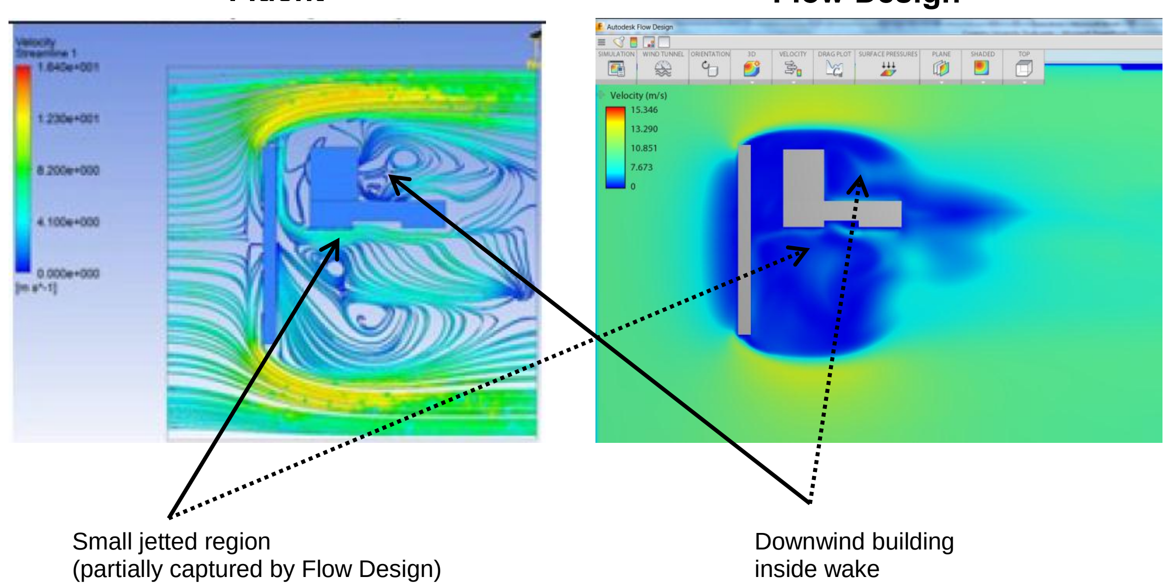
10 m/s East Wind

Fluent **Flow Design**



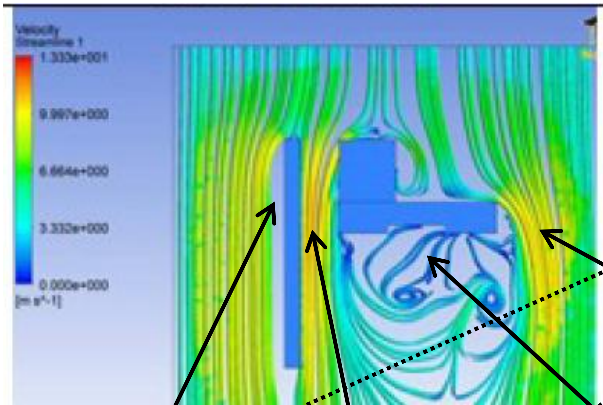
10 m/s West Wind

Fluent **Flow Design**

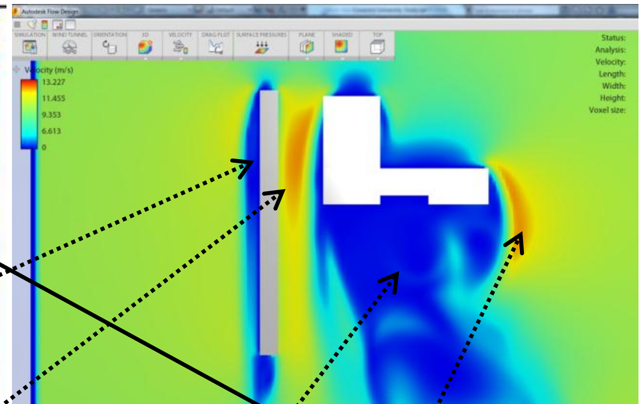


10 m/s North Wind

Fluent



Flow Design



Stagnant region
near wall

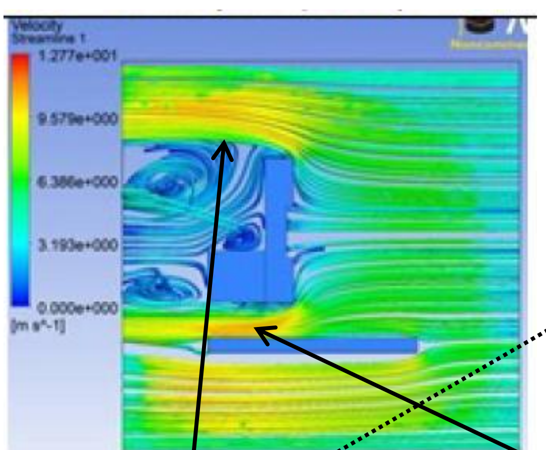
Accelerated flow,
peak velocity ~ 13 m/s

Accelerated flow,
bias towards left side

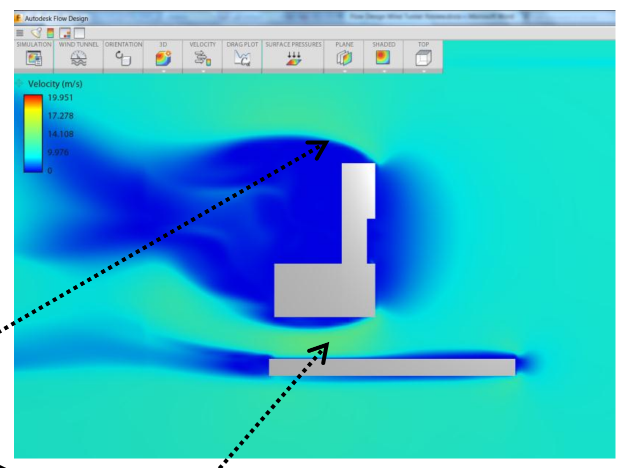
Large wake region

10 m/s South Wind

Fluent



Flow Design



Accelerated well away
from building

Channeled flow,
approx 13 m/s

Conclusions

Flow Design is developed to provide the user with early insight into aerodynamic and wind phenomena. As a design aide, it is not principally intended to provide exact measures, nor does it replace traditional CFD or physical wind tunnel testing. It does offer a limited set of quantitative measures to facilitate design comparisons and monitor the progress of a solution. These measures include drag (drag force and drag coefficient), air velocity and surface pressure.

Based on this study, users of Flow Design can expect the following:

- Flow Design will read in a variety of geometries and provide an understanding of where wakes will form, where there will be high and low pressure regions, and approximately where recirculation will occur. This understanding can provide the designer with an understanding of critical areas that should be considered when proceeding into detailed design.

- Flow Design is well suited for architectural applications. It is able to quickly model wind behavior around (not inside) closed buildings and provide an understanding of where there may be risks of elevated velocities and/or stagnant regions. This information is useful for the design of outdoor spaces and areas where outdoor air quality is of concern due to building exhausts or other contaminants.

- For automotive applications, Flow Design provides a qualitative understanding of flow characteristics around a vehicle. It can show regions where air will recirculate, provide an understanding of the size and location of the wake region, and identify high and low pressure regions on the body and approximate the drag force and coefficient.

Referenced Works

TRANSLOGIC 67: Wind Tunnel. (2011, August 15). Retrieved December 14, 2013, from Aol Autos Translogic: http://translogic.aolautos.com/2011/08/15/translogic-67-wind-tunnel/

Fadl, M., & Karadelis, J. (n.d.). *CFD Simulation for Wind Comfort and Safety in Urban Area: A Case STudy of Coventry University Central Campus.* Coventry University, Department of Civil Engineering, Architecture and Building, Coventry.

Shuba. (2010, June 1). *Leading the Way in Aerodynamic Design*. Retrieved December 14, 2013, from Desktop Engineering: http://www.deskeng.com/articles/aaaxds.htm

Squatriglia. (2008, September 16). *Video: Chevrolet Volt in the Wind Tunnel*. Retrieved December 14, 2013, from Wired: http://www.wired.com/autopia/2008/09/video-chevrolet/

the torque report. (2011). *2012 Chevy Camaro ZL1 Does Some Time in the Wind Tunnel*. Retrieved December 13, 2013, from the torque report: http://www.thetorquereport.com/2011/11/video_2012_chevy_camaro_zl1.html