

Integration of CFD and Experimental Results at VKI in Low-Speed Aerodynamic Design

Philippe Planquart

von Karman Institute for Fluid Dynamics
1640, Rhode-St-Genèse, Belgium
philippe.planquart@vki.ac.be

Abstract

Three examples of recent investigations combining CFD analysis and wind tunnel tests at the von Karman Institute for Fluid Dynamics (VKI) are presented to illustrate the integration process between experimental tests and numerical results. For each project, the requirements and the methodology were different but the final goal was to obtain an optimized aerodynamic design. The first project presented is dealing with a fluid-structure interaction problem. The second project concerns the aerodynamic design of a new Belgian polar base station for Antarctica and the last project concerns the design of an ultra-streamlined land vehicle for the next Solar World Challenge.

Key words: fluid dynamics, vehicle aerodynamics, wind tunnel, CFD, validation, polar base, fluid-structure interaction

Introduction

Integration of CFD and experimental results in aerodynamics is required when each method alone is not reliable, fast or accurate enough to reach the final objective of the project. The advantage of each technique is then used in an integrated project. CFD analysis can be performed for example in a short period of time, but requires a validation when the design falls outside the range of standard aerodynamic design.

Based on the experience acquired at the von Karman Institute (VKI), the best integration is obtained when the scientists performing the wind tunnel tests and the CFD people are integrated in a global team knowing the final objective of the project and able to have a critical view on the results gained using wind tunnel or CFD code. To illustrate the integration process, three examples of recent investigations combining CFD and wind tunnel tests are presented. For each project, the requirements and the methodology are different but the final goal was to obtain an optimized aerodynamic design.

The first project deals with the improvement of the performance under wind load of new sun protection system to be placed alongside building. Vibrations on existing structure were observed during heavy storms. The investigation has started using wind tunnel tests. The wind tunnel tests were used to characterize the fluid-structure interaction leading to the vibrations. A new geometrical design of the blades of the protection system has been performed using CFD before a final validation in the wind tunnel. The second project concerns the aerodynamic design of the new Belgian polar station for the Antarctica. From an initial shape, the building has been modified using a combination of CFD and wind tunnel tests at reduced scale. Snow drift and snow erosion were analyzed experimentally. The results of the numerical simulations were used to understand the flow pattern around the building and to obtain the global loads on the building. New designs were tested both numerically and experimentally. Local pressure

| Report Documentation Page | | | | Form Approved OMB No. 0704-0188 | |
|--|------------------------------------|-------------------------------------|---|--|---------------------------------|
| Public reporting burden for the collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to a penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. | | | | | |
| 1. REPORT DATE JUN 2007 | | 2. REPORT TYPE N/A | | 3. DATES COVERED - | |
| 4. TITLE AND SUBTITLE Integration of CFD and Experimental Results at VKI in Low-Speed Aerodynamic Design | | | | 5a. CONTRACT NUMBER | |
| | | | | 5b. GRANT NUMBER | |
| | | | | 5c. PROGRAM ELEMENT NUMBER | |
| 6. AUTHOR(S) | | | | 5d. PROJECT NUMBER | |
| | | | | 5e. TASK NUMBER | |
| | | | | 5f. WORK UNIT NUMBER | |
| 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) von Karman Institute for Fluid Dynamics Rhode Saint-Genèse, Belgium | | | | 8. PERFORMING ORGANIZATION REPORT NUMBER | |
| 9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) | | | | 10. SPONSOR/MONITOR'S ACRONYM(S) | |
| | | | | 11. SPONSOR/MONITOR'S REPORT NUMBER(S) | |
| 12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release, distribution unlimited | | | | | |
| 13. SUPPLEMENTARY NOTES Third International Symposium on Integrating CFD and Experiments in Aerodynamics, June 2007, The original document contains color images. | | | | | |
| 14. ABSTRACT | | | | | |
| 15. SUBJECT TERMS | | | | | |
| 16. SECURITY CLASSIFICATION OF: | | | 17. LIMITATION OF ABSTRACT UU | 18. NUMBER OF PAGES 32 | 19a. NAME OF RESPONSIBLE PERSON |
| a. REPORT unclassified | b. ABSTRACT unclassified | c. THIS PAGE unclassified | | | |

distributions obtained by CFD are now used in the mechanical design of the different external panels. The third and final project deals with the aerodynamic design of an ultra-streamlined land vehicle, namely a new solar car for the next World Solar Challenge that will take place in Australia in October 2007. A change in the regulation for the new race has forced the teams to change the overall design of the cars. CFD has been used for the initial design and the definition of the car concept but the final geometrical configuration was tested and optimized with the help of the wind tunnel tests.

Description of the test facilities and CFD code

The experimental tests for the three investigations are performed in the VKI Low Speed wind tunnel L1. The L1 wind tunnel is a close-loop wind tunnel built during the fifties and continuously upgraded. The two contrarotating fans of this facility are driven by a variable speed DC motor of 590 kW, allowing a continuous variation of wind tunnel velocity. The contraction ratio of the settling chamber is 4 with a typical turbulence level of 0.3%. Two tests sections can be adapted for this wind tunnel:

- The open-jet test section L1-A (Figure 1) has a free jet test section of 3 m diameter and 4.5 m length and was designed as an aeronautical wind tunnel and is used for testing aircraft models with spans up to about 2 m and 2D wings at Reynolds number up to $4.10^6/m$. This open test section is also used for tests on ground structures and, in the current paper, has been used for the first investigation dealing with fluid-structure interaction.
- The wind engineering facility L1-B (Figure 2) consists of a specially designed test section of 2 m high, 3 m wide and 20 m long with a roughened floor to allow the growth of a turbulent boundary layer similar in nature to the lower part of the atmospheric neutral boundary layer. It is therefore an ideal tool for microclimate studies around buildings, like for the new Belgian polar base presented as second example in this paper. A special test chamber is also provided at the beginning of the section for investigation of surface vehicle or boat aerodynamics, and is equipped with a six-component floor balance and facilities for boundary layer suction. The measurements performed on the solar car were made using this balance at the entrance of the wind tunnel, meaning that the flow velocity was uniform when reaching the model.

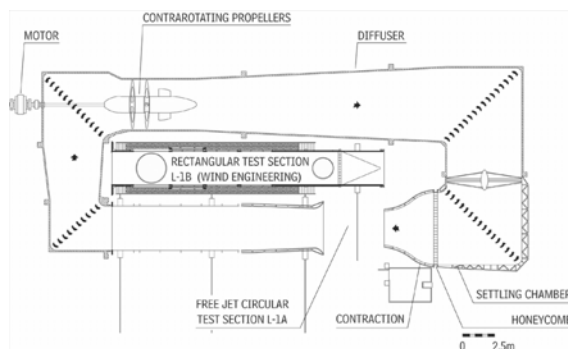


Figure 1 - wind tunnel L1-A

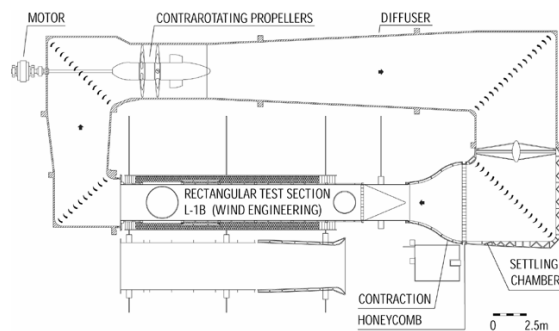


Figure 2 - wind tunnel L1-B

The CFD investigation is performed using the commercial CFD code FLUENT from the company ANSYS. This well-know commercial CFD code allows simulating fluid flow, heat transfer and chemical reaction. The code solves the conservation equations for mass, momentum, energy and chemical reaction using a finite volume method.

Flow-structure interaction

The first project deals with the improvement of the performance under wind load of new sun protection system to be placed alongside building, like shown in Figure 3. Vibrations on existing structure were observed during heavy storms, and our client has asked us to modify the aerodynamic shape of the blades to eliminate or to diminish the fluid-structure interaction.

The investigation started with wind tunnel tests to characterize the fluid-structure interaction, and more precisely to get the critical wind velocities leading to vibration, and even to damages and destruction of existing wind protection systems. High speed image recording during the tests has provided the deformation mode of the different blades as function of the wind velocity. Figure 4 shows a snapshot taken during a wind tunnel test. The deformation of a high number of blades can be seen on this figure. By analyzing in more details the video recorded during the wind tunnel tests, we conclude that the vibrations of the blades are more related to torsion than bending. Therefore, from the wind tunnel tests we came to the conclusion that the second mechanical mode is the one excited by the flow (wind).



Figure 3 - sun protection system along a building



Figure 4 - snapshot during a test in the L1-A wind tunnel showing blade deformation

The theoretical investigation has combined a mechanical analysis using a CAE package and a CFD analysis. The mechanical analysis is performed to compute the natural frequencies of the blades for different geometrical parameters and for the computation of the deformation of a blade under wind load. The CFD analysis of the flow field around existing blade is performed to get the vortex shedding frequency and the Strouhal number associated to the unsteadiness of the flow field [1]. Vibrations of the blades will occur when the frequency at which the vortices are shed from the blade will correspond to one of the natural frequency of the blade.

Figure 5 shows the frequencies of the different mechanical modes as function of the blade length (distance between the two supports). The first and the third mode correspond to a bending mode, while the second mode corresponds to torsion. As predicted by the theory for bending and torsion modes, a linear evolution of the frequencies is obtained as function of the square of the inverse of the distance between the supports of the blades (Figure 6). A first solution to eliminate the vibrations observed during the wind tunnel tests would be to shorten the distance between the supports, but this is commercially not acceptable as more supports are needed per unit length. The solution is therefore in a better aerodynamic shape of the blades.

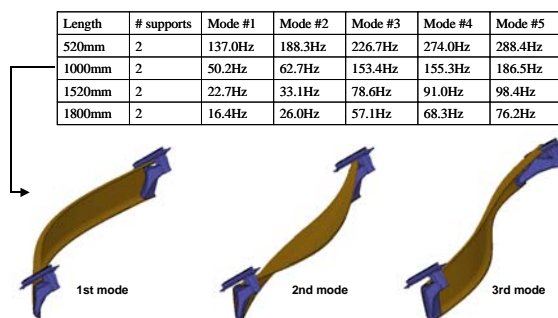


Figure 5 – mechanical analysis – blade deformation

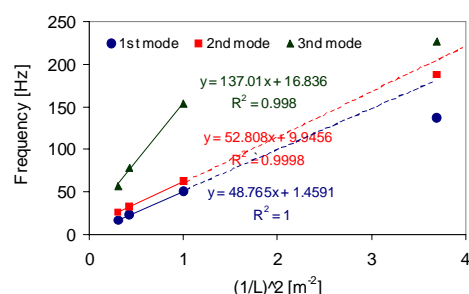


Figure 6 - Effect of the length on the vibration frequency

For the prediction of the vortex shedding frequencies, the commercial code FLUENT has been used to predict the evolution of pressure and velocity as function of time. Numerical simulations have been performed using a second-order implicit time stepping for the transient computations. The turbulence is modeled using a R.A.N.S. approach, with a k- ϵ realizable turbulence model with enhanced wall treatment. Special care has been taken in the grid generation to decrease the value of Y+ near the blade. The numerical simulations have been performed on a 2D cross-section of the blade to stay within the time constraints imposed by our client.

The simulations are performed for the same wind direction with respect to the blades as for the wind tunnel tests. Contours of velocity magnitude at two different time step are shown in Figure 7 and Figure 8, showing the velocity fluctuations as function of time. The variations of drag and lift are periodic and the vortex shedding frequency is computed for different wind velocities. The results are shown in Figure 9. A Strouhal number has been associated to the periodic fluctuations and a constant value has been found for the different numerical wind velocities.

For a wind velocity of 10 m/s, the vortex shedding frequency found numerically is equal to 33 Hz, which is close to the second mode (torsion) of the blade for a distance between the supports equal to 1520 mm. Vibrations are therefore predicted to start at a wind velocity of 10 m/s. This result has been validated by the wind tunnel tests, for which vibrations related to torsion, were observed to begin at a wind tunnel velocity of 10 m/s.

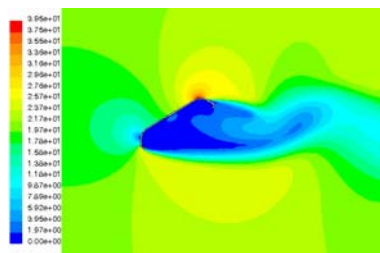


Figure 7 - contours of velocity magnitude at t=1.125 s.

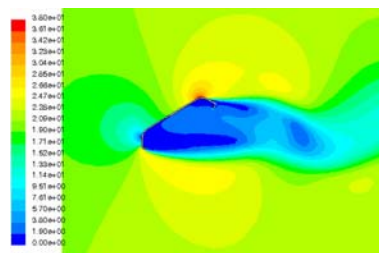


Figure 8 - contours of velocity magnitude at t=1.13 s.

| U | L | f | Strouhal |
|-----|-------|------|----------|
| m/s | m | Hz | x |
| 10 | 0,083 | 33,3 | 0,276 |
| 20 | 0,083 | 67 | 0,278 |
| 25 | 0,083 | 82 | 0,272 |
| 40 | 0,083 | 131 | 0,272 |

Figure 9 - results for one type of blade - Strouhal number

The validated theoretical approach has then been used in a next step to define a new aerodynamic design of the blades of the sun protection system. Ten new different geometries have been analyzed theoretically, both on a mechanical point of view using the CAE code and on the fluid dynamics aspects using the CFD code.

The natural vibration frequencies of the ten new blades were very close to each other for the three first modes, and therefore the best shape of blade had to be found using unsteady numerical simulation of the flow field around the blade. In order to reduce the amplitude of the fluctuations in lift obtained by CFD, the aerodynamic shape of the blade was modified. In the final design, the velocity increase on the upper part of the blade has been considerably decreased, leading to lower pressure fluctuations on the surfaces of the blade. It was believed that this would result in lower amplitude for vortex shedding and that the mechanical mode of the blade would not be excited by the flow field.

The new blade with the newly defined aerodynamic shape has been constructed and a final validation of the new design was performed in our L1-A wind tunnel. Very good results, meaning no vibrations at low speed, have been obtained with the proposed blade geometry.

The methodology for this investigation showing the integration of experimental wind tunnel tests and numerical results is summarized in Figure 10. The investigation started with a wind tunnel test reproducing a real test condition and ended with the same wind tunnel test validating the new geometry. The investigation would not have been successful without the combination of wind tunnel tests and CFD simulation.

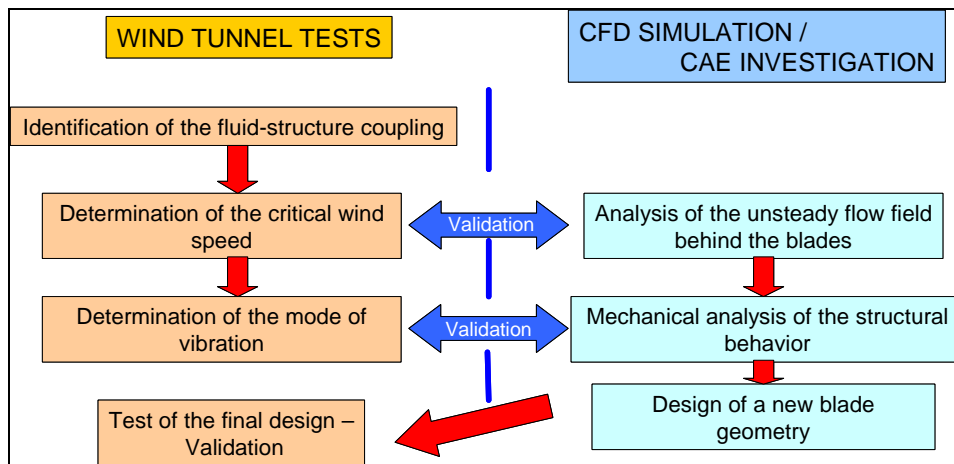


Figure 10 – integration of wind tunnel tests and CFD simulation - fluid/structure interaction study

Aerodynamic design of a new polar station

The second project concerns the aerodynamic design of the new Belgian polar station for the Antarctica. This new Belgian Research Station will be constructed in the Sor Rondane Mountains, Antarctica, during the austral summer 2007-2008. The station will have a hybrid design, with the main building above ground-level (Figure 11) and anchored onto a snow-free rock area (existing ridge). The presence of the most intense winds on Earth, so called katabatic wind leading to wind speed in excess of 65 m/s, transport large quantities of snow that can bury the building very quickly. Two important aspects are therefore to be considered in the aerodynamic design of an Antarctic building: snowdrift and wind loading.



Figure 11 - artistic impression of the future base



Figure 12 - snow erosion in wind tunnel behind the building

Today, almost all modern Antarctic stations have undergone aerodynamic studies at different stages of design. References on recent aerodynamic studies of new Antarctic stations (Neumayer III, Halley VI) can be found in the article of J. Sanz [2]. For the new Belgian polar station, an aerodynamic study has been included from the beginning of the building design process.

The aerodynamic design of the new Belgian station at VKI is divided in two phases:

- A conceptual design phase
- A detailed design phase

In the conceptual phase, we have analyzed the ability of different buildings to cope with intense drifting snow. As mentioned previously, many Antarctica stations have had serious problems with snow drifting with eventual buried structures that require costly maintenance to prevent inaccessibility and structural damage. The use of CFD for the evaluation of snow drifting and snow deposition was rejected from the beginning because it would have involved the development of new models that would require validation using wind tunnel tests. The timeframe of the conceptual phase did not allow performing this development. Therefore, the evaluation of the snow drifting and snow deposition around different building-block shapes has been made exclusively using wind tunnel tests. However, snowdrift modeling in wind tunnel is still a difficult task due to the inherent impossibility to maintain similarity of all the driving forces and the experimental study of snow drifting is also very time consuming. As a result, it was not feasible to use this technique as many building configurations had to be tested in a very limited amount of time. Instead, the sand erosion technique proved to be a very efficient tool to evaluate the wind conditions at ground level and was used to point out areas of severe wind comfort and at the same time identify snow accumulation and erosion regions. Of course, the sand erosion technique is a surface visualization and does not provide information about the development rate of the buildup height (snow accumulation). However, in the methodology developed and based on the comparison of erosion speed contour maps with and without the building, leading to an amplification factor A , taken as a measure of the action of the building on the flow, has allowed VKI to find regions of snow accumulation and snow erosion. More information on the technique can be found in the article of J. Sanz [2].

Figure 12 shows a typical result of an erosion tests with the polar station installed on the ridge topography at scale 1:100 in the L1-B atmospheric boundary layer wind tunnel. The sand erosion produced by the vortices behind the building, where the garages of the base will be placed, can clearly be seen on this figure.

The conceptual design phase has led to the evaluation of six building block concepts that were the result of a preliminary trade-off between different design parameters, with aerodynamics, energy demands and efficiency playing the leading role. At the end of this phase, the most suitable basic design was obtained that has been finalized in the detailed design phase.

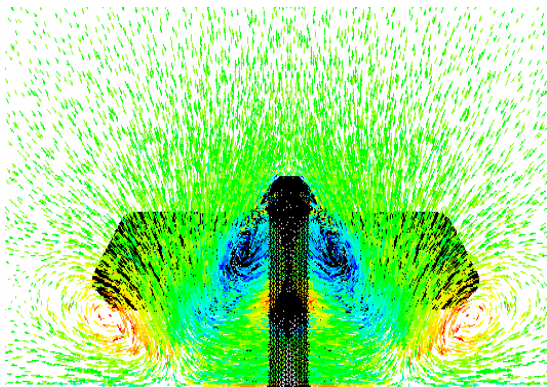


Figure 13 - numerical simulation showing the vortices behind the building

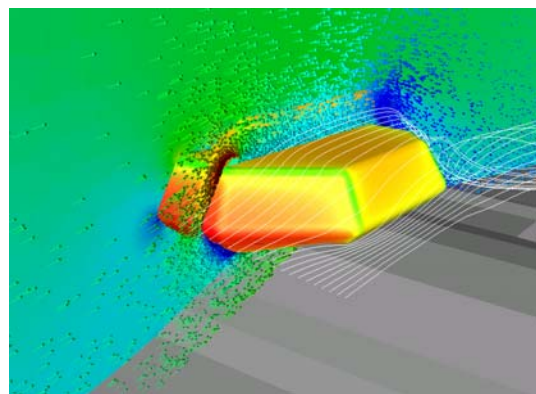


Figure 14 – “Flow visualization” using CFD

In the detailed design phase, the height and the position of the building on the ridge has been defined. Aerodynamic wind loading (drag, lift and lateral forces) on the building has been evaluated using CFD for different building-ridge configuration. A typical CFD result is shown in Figure 14, showing the pressure distribution on the building together with the velocity field and streamlines. The results of the numerical simulations have been used to understand the flow pattern around the building (Figure 13) and to obtain a first estimate of the global loads on the building. The CFD simulations have been performed for one wind direction, namely perpendicular to the building and the ridge.

In the meantime, a scaled model of the final design instrumented with pressure tabs has been constructed for a final validation. The final building is shown in Figure 15 with 159 1mm diameter pressure taps connected for four scanning valves. The pressure measured by the four pressure transducers are scaled by the freestream velocity

measured at a reference position and maps of pressure coefficient have been obtained on the different surfaces of the building for different wind directions.

Local pressure distribution obtained by CFD, like the ones shown in Figure 16 could then be validated by the wind tunnel experiments and the local data from the CFD code can then be used for the mechanical design of the external panels of the polar base.

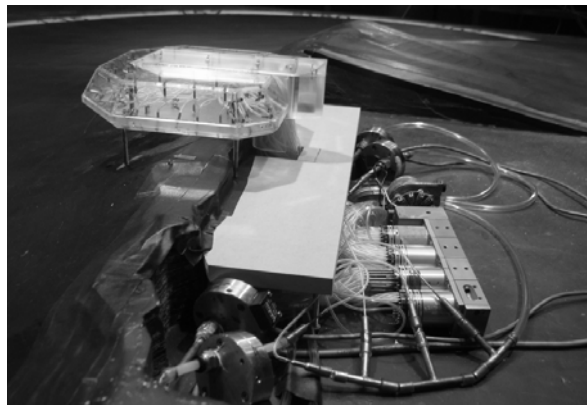


Figure 15 – final model – instrumented with pressure tabs

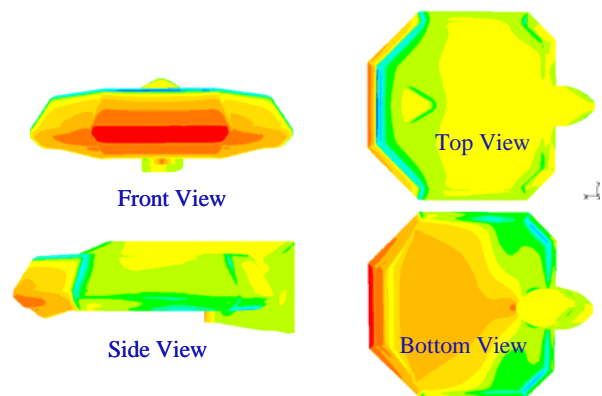


Figure 16 - CFD results showing the local pressure distribution

The methodology used in this investigation showing the integration of experimental and numerical results leading to useful results for the final design of the polar base is summarized in Figure 17. Wind tunnel tests have been used at the beginning of the study for predesign studies and at the end for the validation of the final design. In the mean time, CFD has been used to define the position of the base on the ridge and to understand the flow field behind the building.

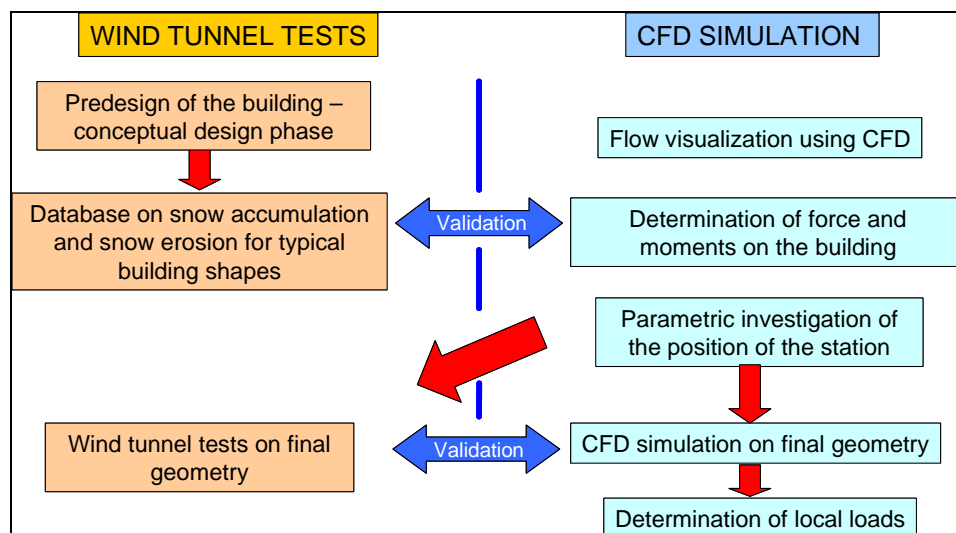


Figure 17 – methodology – design of a new polar base

Aerodynamic design of an ultra-streamlined land vehicle

The World Solar Challenge is organized every two years and is a 3000 km long race for cars powered by the sunlight. In 2005, the winner completed the journey with an average speed in excess of 100 km/h. Students from the Engineering School of Leuven (Belgium) have been the first Belgian team to participate in 2005 with a solar car named Umicar. They finished the race at the eleventh position. The initial aerodynamic design and the validation using wind tunnel tests were performed by the students with the help of VKI [3].

Faced by the possibility that speed limits may be imposed for the 2007 challenge; a new generation of solar car was created. A solar car which, with little modification, could be the basis for a practical proposition for sustainable transport. The new rules include a limit of 6 sq meters for the solar collectors, an easy driver access and an upright seating position. These rules are affecting the external aerodynamic shape of the solar car and, to participate to the 2007 World Solar Challenge, students from the Engineering School Groep T, together with the VKI, have studied and defined the aerodynamic shape of a new vehicle.

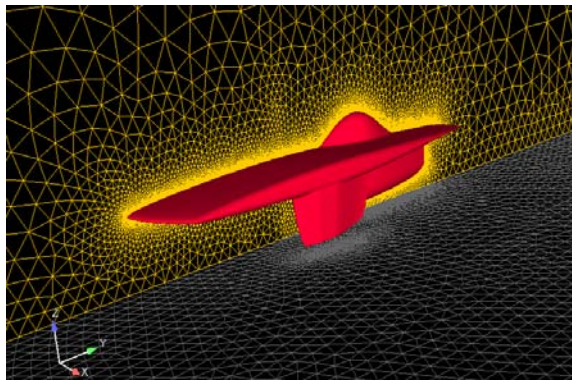


Figure 18 - computational domain + surface grid

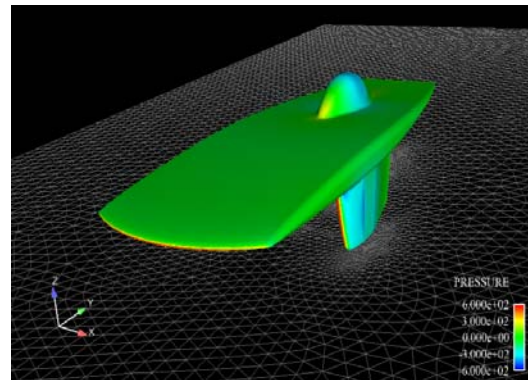


Figure 19 - contours of pressure - CFD results

The design of the new car was started using the well-known reference book for the aerodynamic design of ultra-streamlined land vehicle [4]. Numerical simulations using the commercial CFD code have been performed from the beginning of the design, first in 2D and then in 3D, for the studies of different aerodynamic shapes and the definition of an initial concept. In this first phase of aerodynamic design, the exact value of the drag was not the major concern, but we were more interested in trends to take quickly decision on for example the external shape of the car, on the number and position of the wheels, as a time period of three months was devoted for the definition of the car's concept. The concept of the car at the end of this first phase is shown in Figure 18. The car will have two front wheels and will be powered by the back wheel below the driver.

After this first phase, a second phase was started with more detailed analysis of the flow field around the car and the influence of local geometrical modification on the aerodynamic of the car, like for example the thickness of the wheel system, the shape of the canopy, the position of the canopy on the car, the overall shape, thickness and curvature of the main plate supporting the solar cells.

A typical result of the second phase is shown in Figure 18 (surface grid) and Figure 19 (contours of pressure on the car). A total maximum number of 5.550.000 cells are used to discretise the computational domain. A symmetry plane is used to limit the number of cells.

During this second phase, different numerical simulations have been performed by changing the grid distribution close to the body of the car to modify the cell size in the boundary layer and also by using different turbulence models. Simulations have been performed with the k- ϵ realizable turbulence model and with the Spalart-Allmaras turbulence model. A scatter in the numerical aerodynamic data has been obtained. For example, the values for the

drag area coefficient ($C_d A$ in m^2) obtained by numerical simulation were between 0.12 and 0.15, depending on the grid distribution and the turbulence model used. This scatter is quite high and this value of drag coefficient cannot be used to predict the aerodynamic performance of the car, and by this way compute the performance of the solar car. Tests in the wind tunnel were needed to get more accurate aerodynamic coefficients.

At the end of the CFDD analysis, a first overall shape was defined but the following questions were remaining and had to be answered using wind tunnel tests:

- What is the exact value of the $C_d A$? An accurate value is mandatory to predict the performance of the solar car and to develop the strategy for the race.
- How to improve slightly the geometry of the canopy to reduce the overall drag?
- What is the best geometry for the wheel protection system (thickness – length of the fairing) ?
- What is the influence of side-wind on the drag and side force?
- What would be the influence of dust or dirt on the surface of the solar cells on the aerodynamic drag?
- How are the vertical force, pitching moment and drag changing by modifying the angle of attack of the car by a few degrees? Modifications during the race will be possible by having some flexibility on the shock absorber, but the stroke of the shock absorbers has to be defined in this second phase.

Wind tunnel tests had to be performed because it was the only way to get in a short timeframe answers to these important questions. Once the different models for the wind tunnel were built, all the tests have been performed in one week.

The wind tunnel tests have been performed using two different scaled models at a scale 1/3, meaning that by performing tests at the full speed of the wind tunnel (60 m/s), the Reynolds number could be reproduced in the wind tunnel for a car driving at 20 m/s.

In ultra-streamlined vehicle, most of the drag is produced by skin friction, and therefore special care was taken to have external surfaces as smooth as possible. A front view of the solar car in the wind tunnel L1-B mounted on the six-component floor balance is shown in Figure 20. Figure 21 shows a typical result obtained with oil visualization performed on the canopy for the determination of the detachment position of the flow field. This result has been compared with the prediction of the flow detachment on the canopy using CFD, and a very good agreement was found between the experimental and numerical results.

It is worth to mention that the drag area coefficient $C_d A$ measured during the experimental tests was lower than the one predicted by the numerical simulation (reduction by more than 10%). This is one of the very good news obtained during the wind tunnel tests. It has been attributed to a low performance of the CFD code, because the numerical model was not tuned for the prediction of the transition from laminar to turbulent of the boundary layer. The repeatability of the wind tunnel tests was good, as the scatter in the $C_d A$ coefficient for five different tests was below 1%.



Figure 20 – ultra-streamlined land vehicle in wind tunnel on a six-component floor balance



Figure 21 - Oil visualization showing the detachment position on the canopy

The methodology of the design process with the integration of experimental and numerical results leading to a new ultra-streamlined land vehicle is summarized in Figure 22. The final validation of the aerodynamic design will be made during the race, for which the expectations from the current team are quite high. A position on the podium is expected.

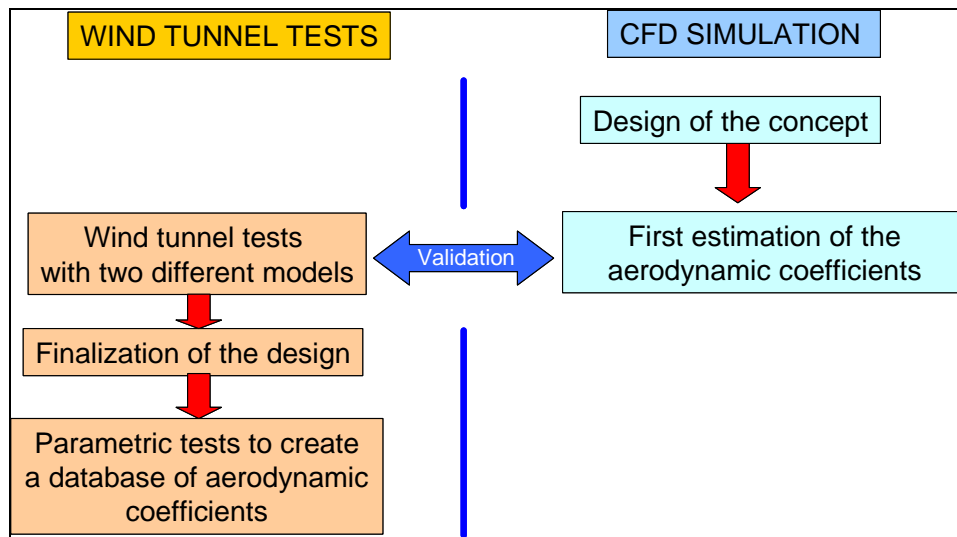


Figure 22 – methodology combining CFD and wind tunnel tests - design of ultra-streamlined vehicle

Conclusions

Using three different examples of low-speed aerodynamic design, combining wind tunnel experiments and CFD computation, we have shown how integration of both methods can lead to a better aerodynamic design. For each project, the requirements and the methodology were different, but the final goal was to obtain an optimized aerodynamic design.

We believe that the best integration is obtained when the scientists performing the wind tunnel tests and the CFD people are integrated in a global team knowing the final objective of the project and able to have a critical view on the results gained using each method. Communication between wind tunnel specialists and CFD specialists is therefore crucial throughout the study.

The three examples have also shown that wind tunnel tests are still a very useful tool necessary for aerodynamic design.

References

- [1] G.K. Batchelor: *An Introduction to Fluid Dynamics*, Cambridge University Press, 1987.
- [2] J. Sanz Rodrigo, C. Gorle, J. van Beeck, P. Planquart: *Aerodynamic Design of the Princess Elizabeth Antarctic Research Station*, 17th International Offshore and Polar Engineering Conference & Exhibition, July 2007, Lisbon, Portugal
- [3] D. Hofman, G. Janssens: *Aerodynamica: CFD studie en optimalisatie van de aerodynamische vorm van een zonnwagen*, Final year thesis, Groep T, Leuven, Belgium, 2006.
- [4] G. Tamai. *The Leading Edge; Aerodynamic Design of Ultra-streamlined Land Vehicles*, Cambridge, Robert Bentely, 1999.

**3th International Symposium on Integrating CFD and Experiments in
Aerodynamics
U.S. Air Force Academy, CO, USA
June 20-21, 2007**

Integration of CFD and Experimental Results at VKI in Low-Speed Aerodynamic Design



**Ph. Planquart , P. Rambaud
planquart@vki.ac.be**

**von Karman Institute for Fluid Dynamics
Rhode Saint-Genèse, Belgium**



Integrating CFD and Experiments in Aerodynamics
20-21 June, 2007

Outline

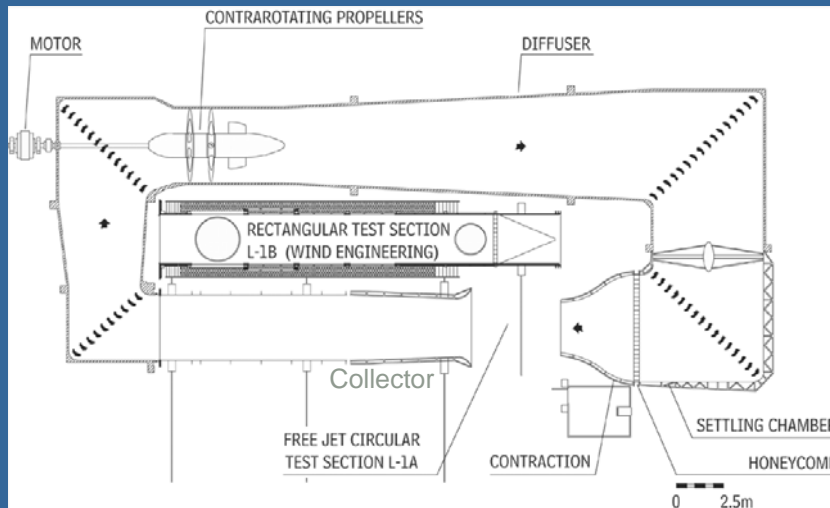
- Introduction
- Description of the test facilities and CFD code
- Example I: Flow- structure interaction problem
- Example II: Aerodynamic design of a new polar station
- Example III: Aerodynamic design of an ultra-streamlined land vehicle
- Conclusions

Introduction

- Integration of CFD and Experiments is required when each method alone is not enough to reach the final objective (with correct reliability, delay or accuracy).
 - The advantage of each method is then combined in an integrated project.
 - The best integration is obtained when Experimental and Numerical engineers are interacting each others.
-
- Illustration through three examples:
 - Example I: Improvement of sun protection system under wind load.
 - Example II: Aerodynamic design of the new Belgium Antarctica polar station.
 - Example III: Aerodynamic design of an ultra-streamlined land vehicle.
 - For each project, the requirements and the methodology are different but the final goal was to obtain an optimized aerodynamic design.

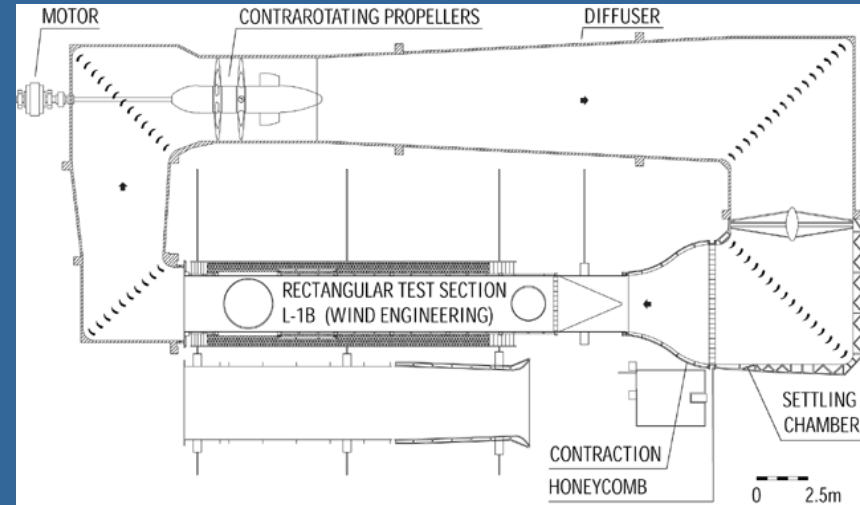
Description of the test facilities and CFD code

- Experimental tests are performed in the VKI Low Speed Wind Tunnel L1.



L1-A wind tunnel with a free jet test section

- Flow-structure interaction



L1-B wind tunnel – wind engineering facility

- Test of the Belgian polar station
- Test on the ultra-streamlined land vehicle

- The CFD investigation is performed using FLUENT from the ANSYS with RANS or URANS approaches.

Flow structure interaction

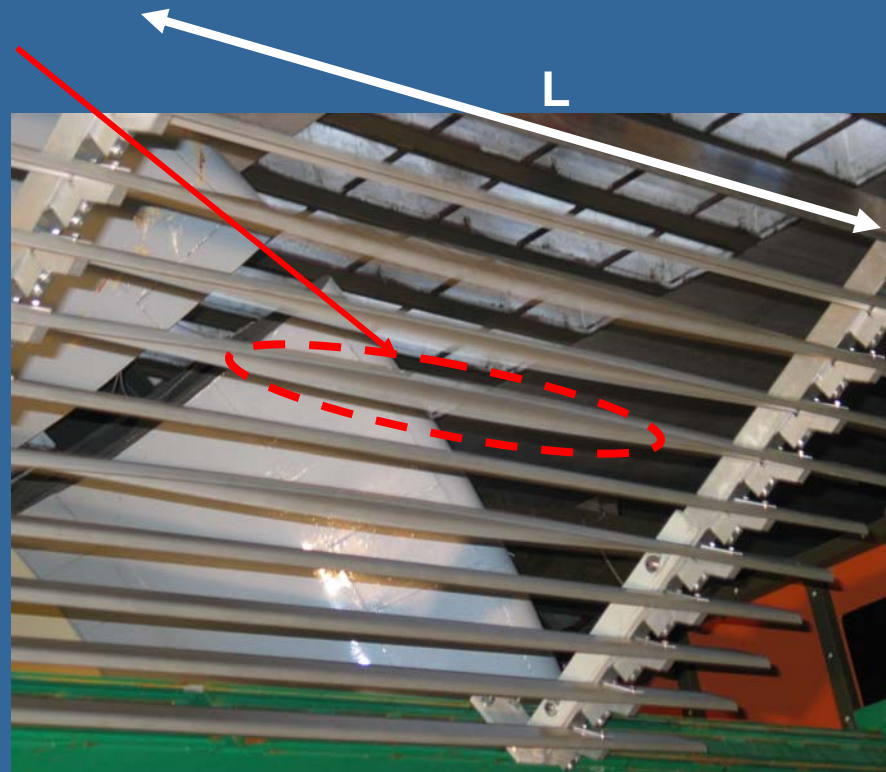
- Sun protection system to be placed alongside building.
- Vibrations were observed during heavy storms.
- Goal: Modify the blade shape to eliminate/diminish the fluid-structure interaction.



Flow structure interaction

- The investigation has **started with wind tunnel** tests.
- **High speed image** recording has provided the **deformation mode** of the different blades.
- **Snapshot** of a test shows that the vibrations of the blades are **more related to torsion** than bending.

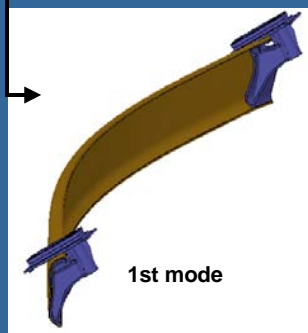
Sun protection system in
VKI - L1-A wind tunnel



Flow structure interaction

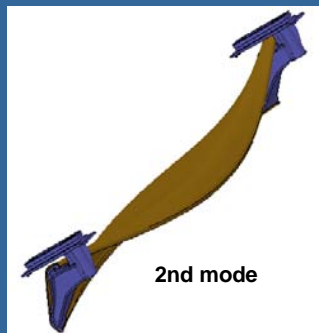
- Mechanical investigations using a Computer-Aided Engineering (CAE) package (**Solidworks**) has **provided** the natural **frequencies** of the blades, and the **deformation modes**.

| Length | # supports | Mode #1 | Mode #2 | Mode #3 | Mode #4 | Mode #5 |
|--------|------------|---------|---------|---------|---------|---------|
| 520mm | 2 | 137.0Hz | 188.3Hz | 226.7Hz | 274.0Hz | 288.4Hz |
| 1000mm | 2 | 50.2Hz | 62.7Hz | 153.4Hz | 155.3Hz | 186.5Hz |
| 1520mm | 2 | 22.7Hz | 33.1Hz | 78.6Hz | 91.0Hz | 98.4Hz |
| 1800mm | 2 | 16.4Hz | 26.0Hz | 57.1Hz | 68.3Hz | 76.2Hz |



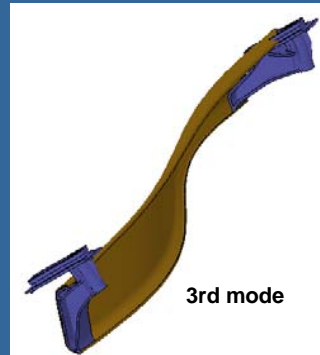
1st mode

Bending



2nd mode

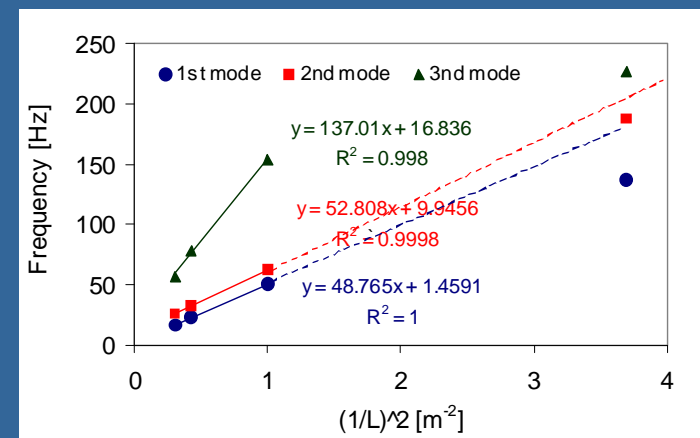
Torsion



3rd mode

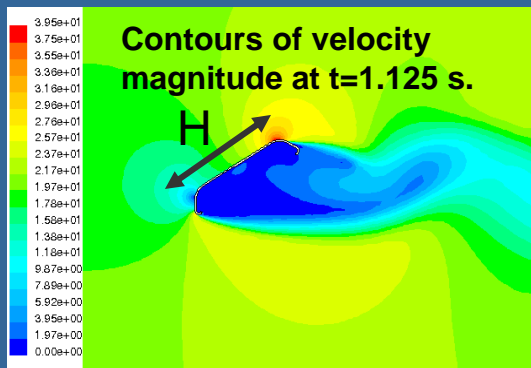
Bending

Effect of the length of the blades on the vibration frequency

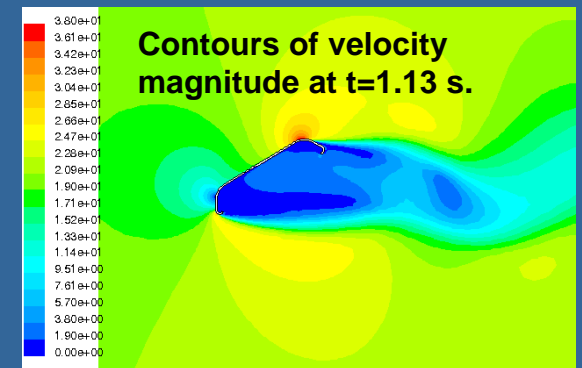


Flow structure interaction

- Vortex shedding frequencies have been obtained using a CFD approach.
- **URANS** simulation of the flow field **around a single blade** for the **same wind** direction as in the **wind tunnel** is performed.
- For a **wind velocity of 10 m/s**, the vortex shedding **frequency** found numerically is equal to **33 Hz**.



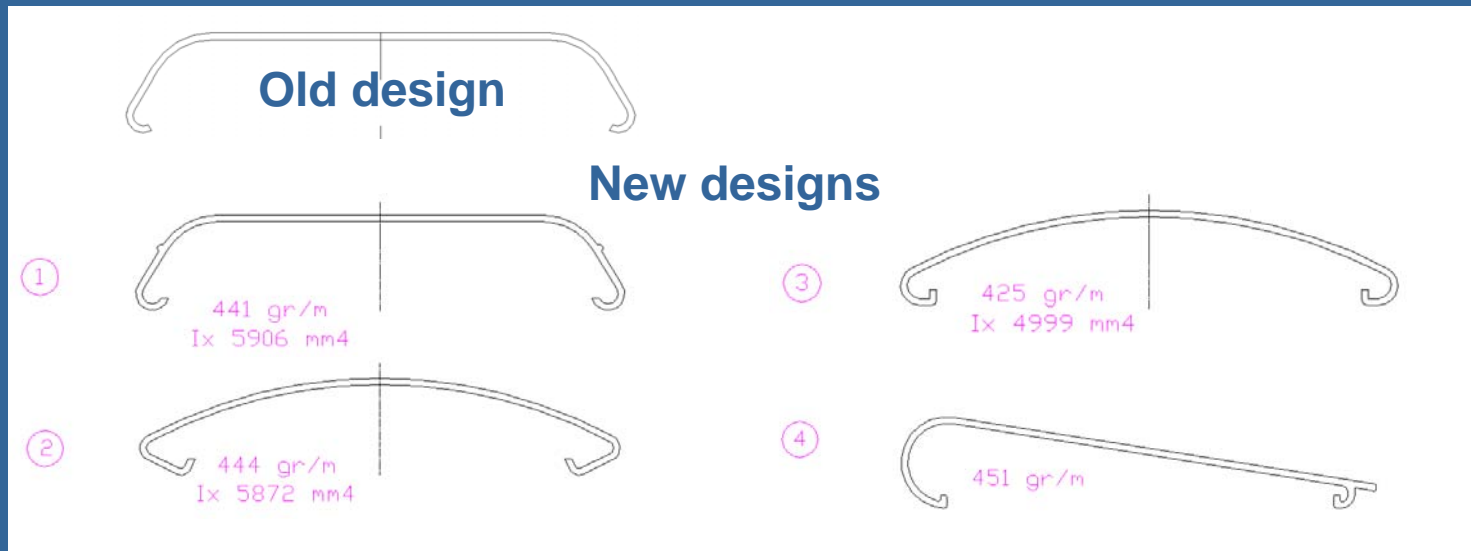
| U | H | f | Strouhal |
|-----|-------|------|----------|
| m/s | m | Hz | [-] |
| 10 | 0,083 | 33,3 | 0,276 |
| 20 | 0,083 | 67 | 0,278 |
| 25 | 0,083 | 82 | 0,272 |
| 40 | 0,083 | 131 | 0,272 |



- This **result** has been **validated** by the **wind tunnel tests**, for which vibrations related to torsion were observed to begin at a wind tunnel velocity of 10 m/s.

Flow structure interaction

- Ten different geometries have been proposed and analyzed (CAE).



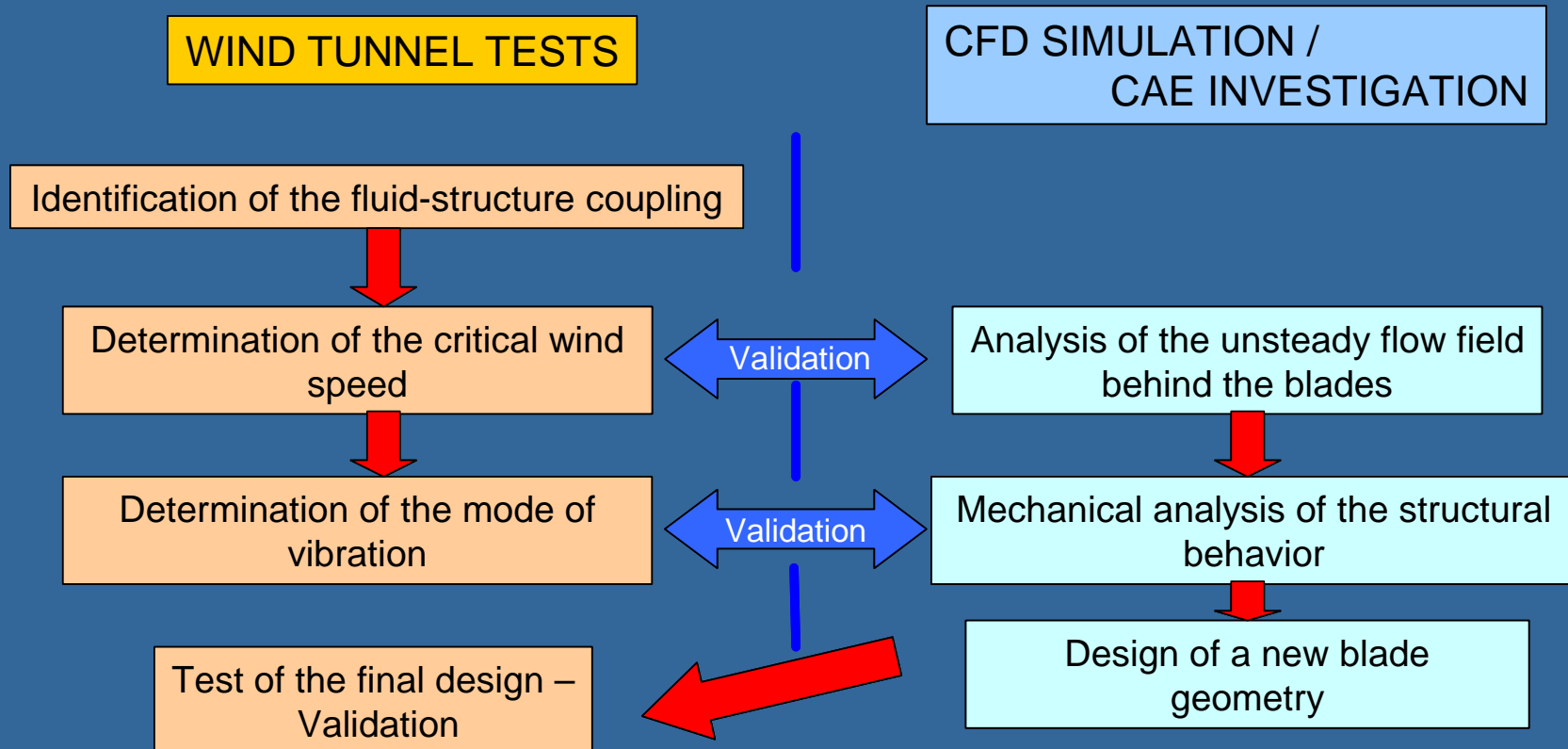
Flow structure interaction

- Natural **frequencies** of the ten new blades were **close to each other**.
- The **best shape** had to be found **based on CFD** (flow field around the blade).
- The **final blade** presents **reduced** velocity **acceleration** leading to lower pressure fluctuations and lower amplitude of vortex shedding.



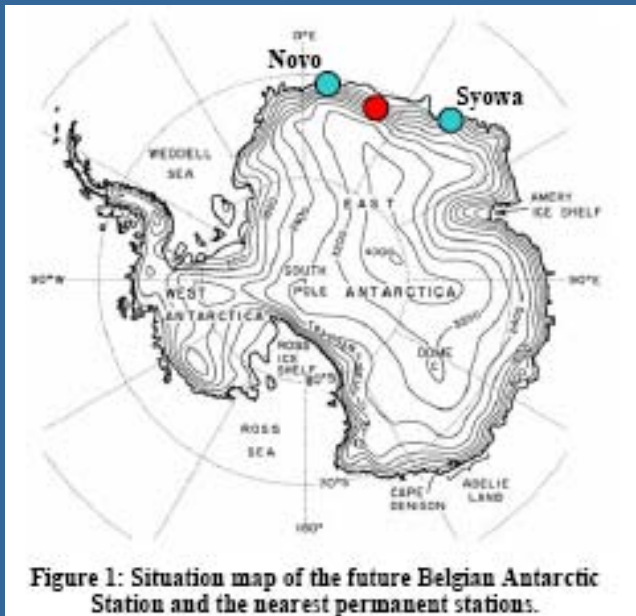
- The **final blade** was built and then **tested** in the **wind tunnel**.
- Very good results, meaning **no fluid-structure interactions**, have been observed at low speed with the proposed blade geometry.

Flow structure interaction



Aerodynamic design of a new polar station

- Aerodynamic **design** of the new **Belgian Polar station** for the **Antarctica**.
- The presence of the **most intense winds** on Earth, so called the katabatic wind leading to wind speed in excess of **65 m/s**.
- Transport of large quantities of **snow** that can bury the building very quickly.



Most important aspects have to be considered:

- **Snowdrift**
- **Wind loading.**

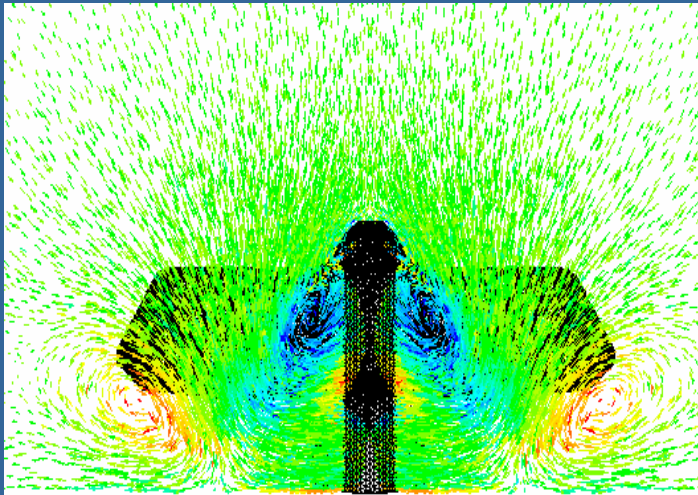
Aerodynamic design of a new polar station

- The aerodynamic design is divided in two parts:
 - A conceptual phase
 - A detailed analysis phase
- In the **conceptual phase**, the ability of different buildings to cope with intense drifting snow has been tested.
- This **investigation could only** be made with **wind tunnel tests**.
- The methodology developed by VKI has allowed to find experimentally regions of snow accumulation and snow erosion.

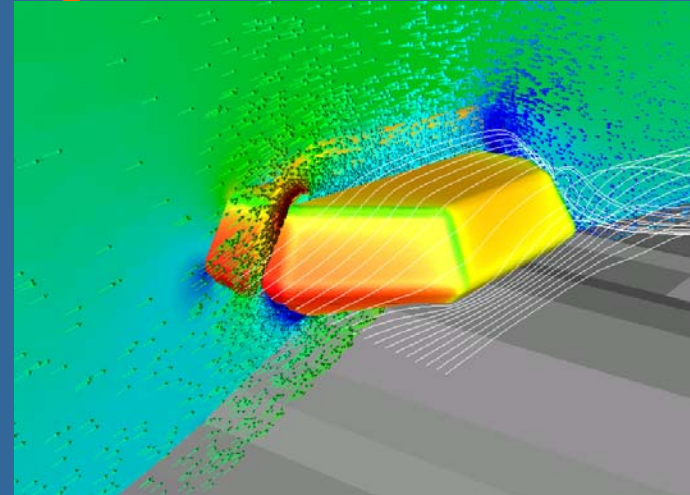


Aerodynamic design of a new polar station

- In the **detailed analysis phase**, the height and position of the building on the ridge has been defined.
- **Aerodynamic wind loading** on the building has been evaluated **using CFD** for **different positions** of the building.
- **CFD results** were also used **to understand** the **flow** pattern around the building. The vortices behind the building are responsible of the snow erosion and snow accumulation.
- **Wind tunnel results** have been **revisited using CFD** “flow visualization”.



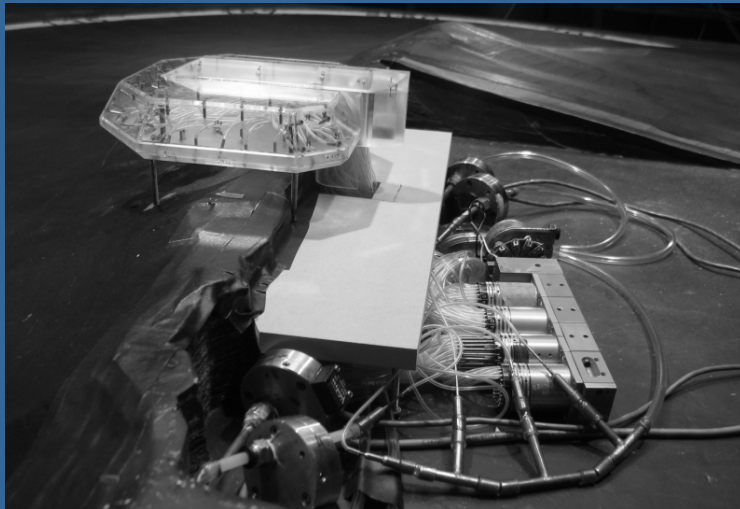
Numerical simulation showing the vortices behind the building



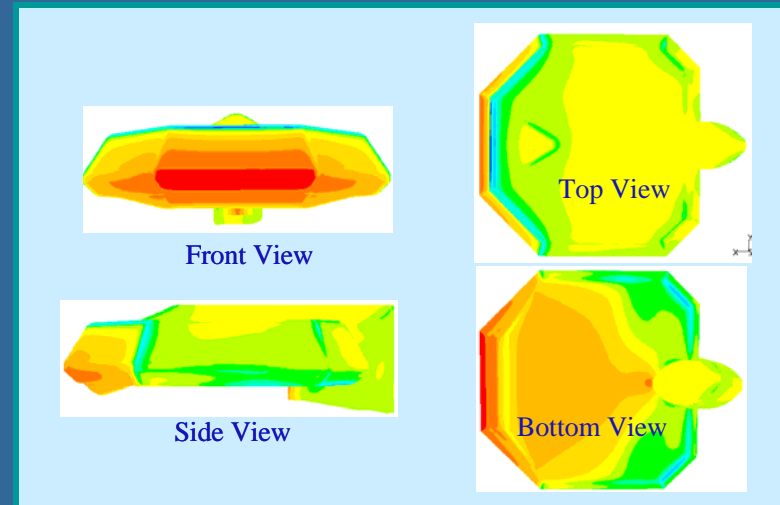
Flow visualization using CFD

Aerodynamic design of a new polar station

- Final building configuration was built and tested in the wind tunnel.
- Local pressure measured in the wind tunnel have been compared to CFD results.
- Local CFD data are used for the mechanical design of the external panels.

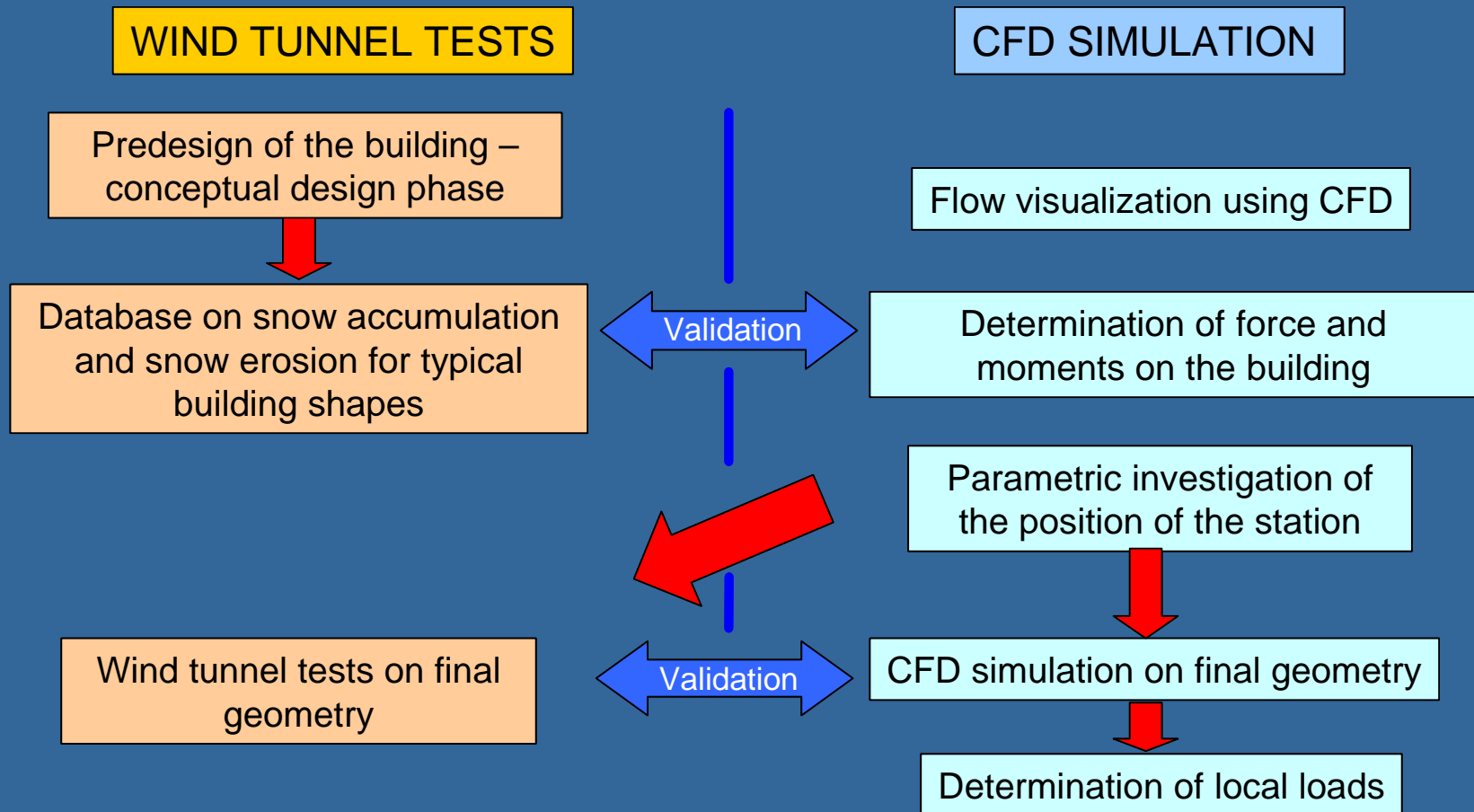


Final model – instrumented with pressure tabs



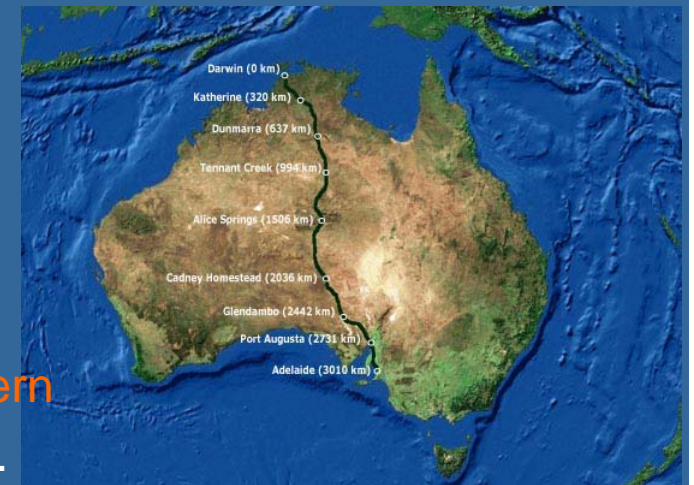
CFD results showing the pressure distribution on the different parts

Aerodynamic design of a new polar station



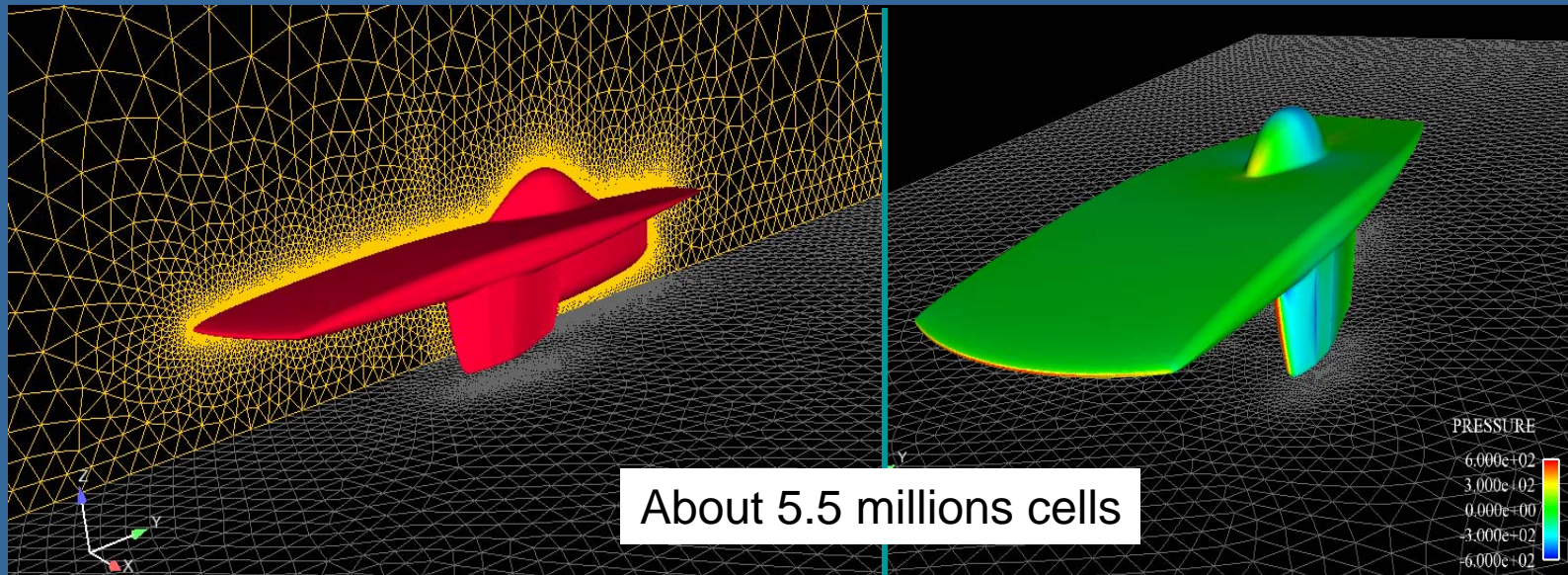
Aerodynamic design of an ultra-streamlined land vehicle

- The **World Solar Challenge** is organized every two years and is a 3000 km long race for **cars powered by sunlight** (www.wsc.org.au/2007/).
- New **rules** for the **2007 Challenge** **are affecting** the external aerodynamic **shape** of the solar car.
- Two phases for the design
 - **Definition of the concept**
 - **Detailed analysis of the aerodynamic shape**
- Design started by using CAD package. **CAD geometries** were then inserted in the **CFD package** to get aerodynamic data.
- **First phase:**
 - **Exact value** of the **drag** was **not the major concern**
 - **Quick information** to define the **shape** of the car.
 - **Final car concept** includes **two front wheels** and **a back wheel** below the driver that will be used to power the car.



Aerodynamic design of an ultra-streamlined land vehicle

- **Second phase:** detailed investigation of the flow field around the car (CFD).
- **Parameters** that were tested:
 - Thickness and **shape of the main plate** supporting the solar cells
 - **Shape of the canopy**
 - **Thickness of the wheel system**
 - **Position of the canopy** on the car
- **Scatter** is obtained in the **drag area coefficient** depending on **turbulence models!!**



Aerodynamic design of an ultra-streamlined land vehicle

- The following questions were remaining at the end of the CFD analysis:
 - What is the exact value of $C_d.A$?
 - How to improve the geometry of the canopy to reduce the overall drag?
 - What is the best geometry for the wheel protection system ?
 - What is the influence of side-wind on the drag and side force ?
 - What would be the influence of dust or dirt on the surface of the solar cells on the aerodynamic drag ?
 - What would be the sensitivity to a small change of the angle of attack of the car on the lift, the pitching moment and the drag?
- Wind tunnel tests had to be performed because it was the quickest way to answers to these questions.

Aerodynamic design of an ultra-streamlined land vehicle

- Wind tunnel tests were performed on a model at scale 1/3.
- All the tests have been performed in one week.
- Drag area coefficient ($C_d A$) was lower than the one predicted by CFD (reduction by more than 10%).



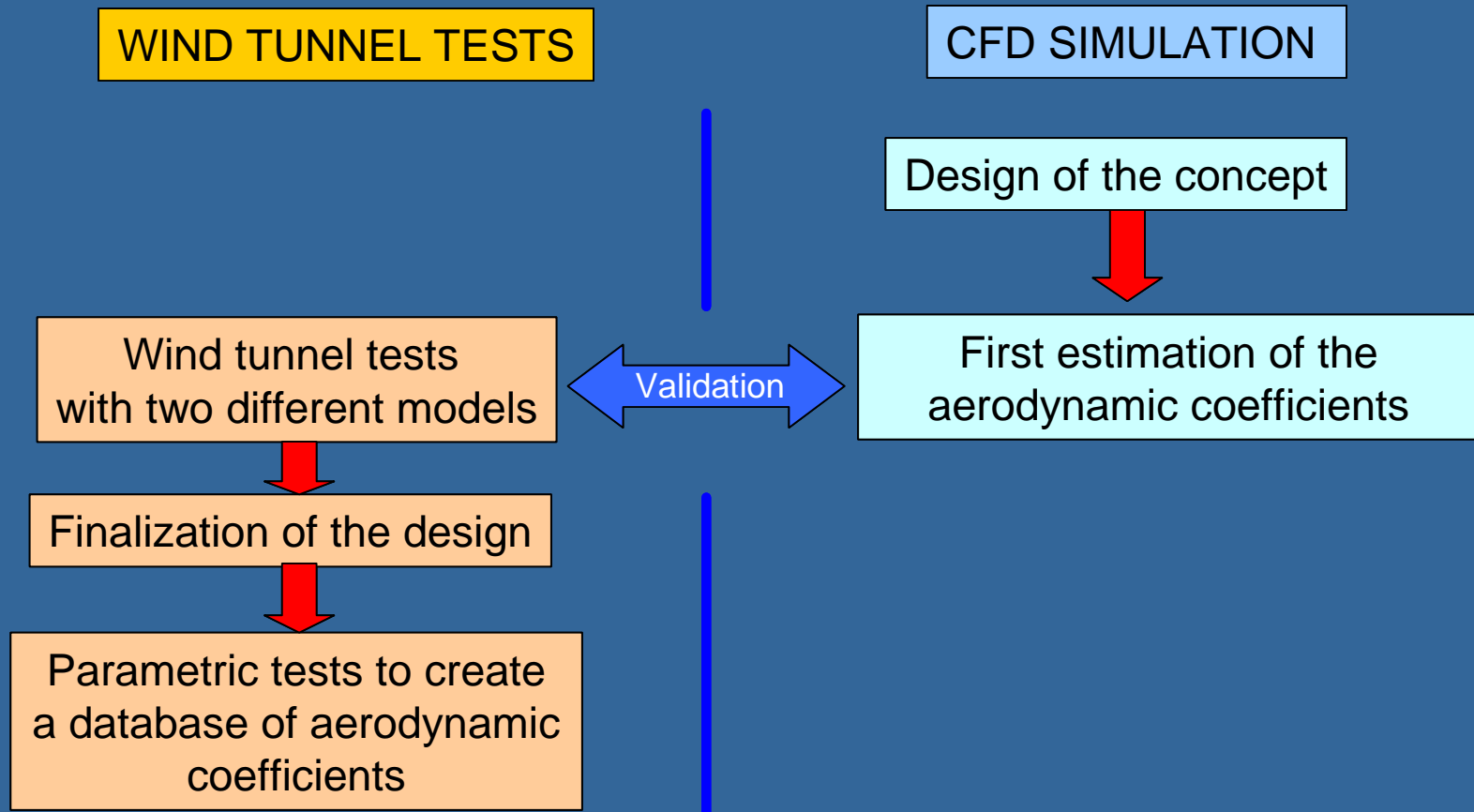
Ultra-streamlined land vehicle in wind tunnel on six-component floor balance



Oil visualization showing the detachment position on the canopy

- Final validation will be the race in October 2007.

Aerodynamic design of an ultra-streamlined land vehicle



Conclusions

- Integration of CFD and Experiments can lead to a better aerodynamic design.
- Communication between wind tunnel specialists and CFD specialists is crucial throughout the studies.
- The three examples have also shown that a single approach is often not enough to obtain the optimized design.