Assessing the Potential of Ventilated Façades on Reducing a Buildings' Thermal Load using Decoupled COMSOL simulations

C. (Chris) van Dronkelaar, A.W.M. (Jos) van Schijndel*
Eindhoven University of Technology
*Corresponding author: P.O. Box 513; 5600 MB Eindhoven; Netherlands, A.W.M.v.Schijndel@tue.nl

Abstract: Solar radiation is a prominent contributor of energy in buildings (Urban, 2007), and can be transmitted directly into a building through opaque surfaces, but it can also be absorbed by building components (i.e. walls, roofs etc.). Both cause heat addition to the building interior. The application of ventilated facades can help reduce thermal loads during high temperatures and solar radiation, which in effect reduces the energy consumption due to air-conditioning systems. This is a passive cooling technique that could be developed to a greater extent in order to improve indoor climatic conditions and the microclimate around buildings (Ciampi, et al., 2003).

This study discusses the use and effect of ventilated facades, with an external façade cladding, a sub-structure anchored to the wall surface of the building under solar radiation, while designing façade elements numerically using COMSOL, to create the highest achievable velocity inside the air cavity. The mass air flow inside the cavity, due to buoyancy effects (natural convection) and wind (forced convection), can carry away heat load passively.

Results show that energy saving is increased with a ventilated façade over a conventional façade, and is more effective for higher solar radiation and higher air velocity inside the cavity. In the second part of the study it becomes clear that façade elements can be designed in such a way that they increase the air velocity inside the cavity to remove more heat efficiently. An improvement of up to 75% of the air velocity is reached in some parts of the cavity for the implemented design in comparison to the reference case.

Keywords: solar gain, air cavity, ventilation, passive cooling, ventilated façade.

1. Introduction

A façade receives on peaks up to 1000 W/m^2 solar radiation. Dependent on the geographical location, the climate, time of day, orientation to the façade and shadowing by surrounding building and trees it receives less, but still significant amount of solar radiation. This solar radiation will not be fully absorbed by the façade, a part is reflected directly (light colors reflect generally more than dark colors). Another part is carried off by wind and free convection on the outside of the façade and another part is absorbed. On a sunny day, a façade can increase to up to 40°C in comparison to the outside temperature. And while the heat flow though the façade construction is proportional to the interior of the building with the temperature difference between the cavity- and interior temperature, solar radiation leads to an increased energy use for cooling the interior of the building.

The building physical benefits of ventilated facades are generally accepted. A ventilated façade exists of an inner wall where insulation material can be applied with a damp diffusive and damp-open layer. At the front of this wall, façade elements will be placed with an air cavity in between. The air in the cavity is in open connection to the outside air. The cavity lets fresh outside air flow freely through the cavity, where moisture from condensation or rain can be transported as vapor, while the heat load due to high solar radiation can be carried off.

Objective
The objective of this study is to design façade elements that apply Bernoulli-principles to convert airflows around buildings to benefit an increase in the ventilation rate of the cavity to reduce heat transfer flows into the interior of the building, which in effect reduce the cooling load of the building.

2. Numerical simulation

To investigate the potential of wind enhancement methods for wind power utilization inside building cavities behind façade claddings, knowledge on wind aerodynamics and wind flows around buildings is crucial.

The heat transfer between the cavity and interior of the building and the wind flows in and at close distance of the cavity are simulated using COMSOL. While the wind flows around a building block have been simulated using Fluent 6.3.26 (Fluent Inc., 2012) in combination with Gambit 2.4.6 for creating the grid.

In the ideal situation, only one model is created where wind flows around buildings are simulated to show the effect they have on the air velocity inside the cavity, while at the same time calculating the effect of solar radiation on the thermal load of the building and whether the heat load inside the cavity can be carry away through forced ventilation to reduce this thermal load. Although much improvement is shown in the application of numerical simulation tools to investigate these complex subjects, the simulation of this ideal model is restrained due to limited computational resources. Therefore another method has to be followed that involves simulating the above mentioned physics on a decoupled base.

The numerical simulations for the velocity inside the cavity are based on 2 dimensional air flows, which make it apparent that an unrealistic situation is created as wind at a 3 dimensional building will flow not only upward or downward, but also sideways around the building.

Three different models have been created which are simulated independently.

1. ***Full building scale*** (Wind flows around the building using Fluent)
2. ***Building envelope scale*** (Wind flows at the ventilated façade using COMSOL)
3. ***Air cavity scale*** (Wind flow and heat transfer through the ventilated façade)

2.1 Full building scale

Wind flows around a building (30m x 30m x 30m) have been simulated, where a velocity inlet profile is made for a reference wind speed of 4 m/s to create the wind and a roughness length scale has been applied that represents an "urban environment".

A computational domain has been created according to the COST guidelines (Franke, et al., 2007). This means that prescribed values are used for the extensions from the sides of the building to the boundaries of the computational domain. The domain has a downstream length of $15H_b$ (from the leeward side of the building to the outlet of the domain), lateral width of $5H_b$, a vertical height of $5H_b$ (from the top of the building to the top of the computational domain) and a of 5Hb between the velocity inlet and front of the building, where H_b is the height of the building as shown in **Figure 1**.

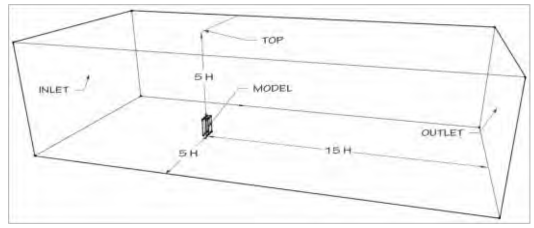


Figure 1: Computational domain.

The model is created to calculate the wind flows at certain positions parallel and perpendicular to the windward façade, which will be used in the building envelope scale model to create three types of wind flows.

2.2 Building envelope scale

In this smaller scale, the finite element method tool of COMSOL 4.2 (COMSOL, 1998-2011) has been used to simulate the oncoming wind flow for different types of façade elements to optimize the air velocity inside the cavity 2 dimensional.

To create insight in the changes in velocity with different façade designs, a model has been created with COMSOL that simulates the wind flows and their behavior at the façade. Due to the unavailability of computational resources and lack of experience with the program, it was not possible to correctly integrate the obtained velocities from the model created with Fluent. Therefore a different approach was taken that led to the inaccuracy of the actual wind flows around the building and at the inlet positions of this model. The obtained results cannot be used to show the actual velocity level inside the cavity with the corresponding velocity inlet profile from Fluent. The results are therefore only used to compare which façade design obtains the highest velocity values inside the cavity for three different types of wind flows that correspond to an upward flow, a downward flow and a wind flow coming perpendicular to the building at ~2/3 height of the building as shown in **Figure 2**.

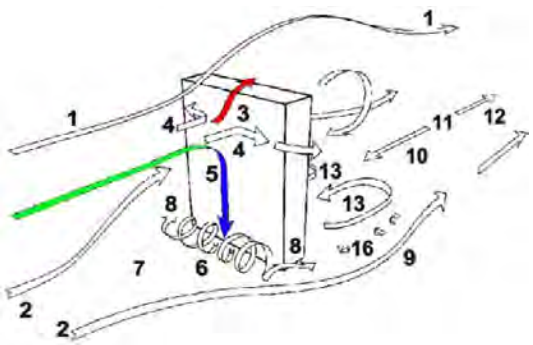


Figure 2: (adjusted from (Beranek & van Koten, 1979)**:** Wind flows around a building; 3 types of wind flows; up-wind flow (red), down-wind flow (blue) and perpendicular wind flow (green).

The flows are created by interpolating a velocity profile over the length of the inlets obtained from the Fluent model, the three flow types are shown in **Appendix I: Figure 11**, **Figure 12** and **Figure 13**. For these flows a number of different façade designs have been made that change the airflow over the façade.

There are many methods known to use the kinetic energy of the air flows efficiently. Most principles are based on the principles of Bernoulli, where a Venturi and aerofoil are examples of elements that convert flowing air in oriented airflows. These elements are particularly used to create a lower pressure, however it is also possible to create an over pressure. By shaping these elements in such a way that use can be made of the Coanda-effect which increase the effectiveness of the elements.

2.3 Air cavity scale (Heat balance)

The smallest scale is also created with COMSOL, this model consist of only the external layer of the façade, air cavity and internal wall consisting of insulation and concrete. This is the only model where heat transfer is taken into account by introducing solar radiation on the external layer and where an air flow is created at the inlet that carries away the heat load inside the cavity.

The façade is modeled with a height of 1.5m and a cavity width of 0.04m thick. A schematization of the model is shown in **Figure 3**. The material properties are given in **Table 1**.

Table 1: Material properties of the façade construction.

Material	Density [kg/m^3]	Thermal conductivity [W/mK]	Heat capacity [J/kgK]	Thickness [m]
Concrete	2300	1.8	880	0.10
Panel	23	0.3	800	0.01
Insulation	12	0.045	1300	0.05

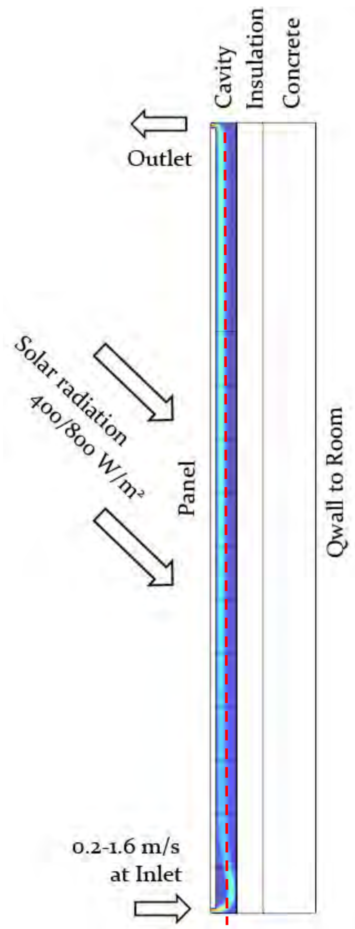


Figure 3: Schematization of the heat balance model

Steady state calculation are made to show the heat gain or heat loss for different outdoor temperatures, with a solar radiation of 400 or 800 W/m^2 and velocities fluctuating from 0.2 to 1.6 m/s at the inlet. A heat transfer coefficient of 25 W/m^{2}K is used at the outside of the external layer and 7 W/m^2K at the inside of the internal wall.

The velocity inside the cavity is measured at the dotted line shown in **Figure 3Error! Reference source not found.** at y=0.1m to 1.3m in steps of 0.1m. This information is used to show what level of velocity is to be reached inside the cavity to remove the heat inside the cavity. According to (Schwarz, 1973) a velocity of 0.2 to 0.6 m/s can be achieved by ventilating the cavity. This level is mimicked by using the fluctuating velocities at the inlet. Natural convection is simulated by closing the external layer and by applying the different outdoor temperatures and solar radiations, which results in a convective loop to be created.

2.4 Meshing

To obtain accurate results from the simulations, meshes are required to be dense enough, but not too dense as this will lengthen the computation time. For the building scale, a mesh is created with Gambit according to COST guidelines (Franke, et al., 2007). For the building envelope scale and cavity scale, a triangular mesh is created automatically by COMSOL and is set to a density "Finer" the highest density possible in most cases as higher density led to an out of memory error in the program. A few examples are shown in **Appendix I**, **Figure 14**, **Figure 15** and **Figure 16**.

A high mesh density is needed to obtain accurate results from the simulations. An example is shown in **Appendix I**, **Table 3**, where 3 different meshes are compared for the same part of design 3B, where can be seen that the difference between a mesh density of "Normal" and "Finer" has a big influence on the results. Also the appliance of an extra distribution line of 200 elements at the inner wall has a great influence. For all the models that are created, attention is given to creating the mesh in a correct way, which means that it must be dense enough, while the adjacent cells must not differ much in size and shape.

2.5 Façade designs

Façade elements can be designed in numerous forms. By starting simulations for a normal flat surface and 4 openings, a reference design is created, where other designs are compared to. The question is how this first design (design 0) can be transformed into a façade which is more effective in enhancing the air velocity inside the cavity. At first, a few normal element forms are created in symmetry along the façade to see whether these have any effect. After that, adaptations have been made to increase the velocity even more. This resulted in a number of basic designs shown in **Appendix I**, **Table 4**. The façade designs are described individually, to explain their intention and dimensions. All openings are 0.01m in height on a façade of 3m height.

- **Design 0**: The reference design, with 4 openings placed after every panel of 1m.
- **Design 1**: Blocks have been added of 0.05m thick, the steep thicker part at the outside of the façade is to locally create under pressure through the use of Bernoulli principles, as wind flows over the element and sucks air out of the cavity. The other element is made thicker to slow the flowing air and locally create overpressure. This way, outside air gets blown into the cavity to remove warmer air by other openings.
- **Design 2**: Cylindrical tubes of 3cm have been placed 2cm away from the openings in the external layer. The air flow over the façade is to increase in velocity in between the cylindrical element and the façade and part is to move into the cavity to accelerate the air velocity.
- **Design 3**: The elements in this design will lead to accelerating and slowing down the wind flows. Pressure differences are created over the surface of the façade. By making connections to the cavity at high and/or low pressures the ventilation rate can be increased. Rain is prevented from entering the cavity by creating the openings under an angle. All along the cavity there exists a difference in width, which creates a Venturi that accelerates the air flow inside the cavity. The façade element is thick to a maximum of 6cm out of the cavity, 10cm from the inner wall. This design is adapted several times to increase the performance and reduce the amount of openings, as shown in the table.
- **Design 4**: The façade elements are curved to slow the flow at the extra thickness and letting air flow in from 7 openings.
- **Design 5:** The panels have been thickened on the top to slow down the flow and letting it enter the cavity through 7 openings.

Many other elements can be thought of, and because the elements are only simulated in a 2D model, 3D designs are not analyzed.

3. Results and Discussion

3.1 Heat balance

Because the velocities inside the cavity are not constant everywhere and the distribution is not uniform over the cavity width, a range with the highest frequent velocity has been noted for the corresponding velocities at the inlet in **Table 2**. The values are taken approximately at a distance of 0.005 to 0.025m of the cavity width, while the cavity width is 0.04m. The velocities that are measured for the corresponding velocities at the inlet at 30°C, and for a solar radiation of 400 W/m^2 and 800 W/m^2.

Table 2: Range of velocity in the cavity for different inlet velocities

Velocity at inlet (m/s)	0.2	0.4	0.6	0.8	1.0	1.2	1.4	1.6
Velocity in cavity (m/s) (for 400 W/m^2)	0.04-0.19	0.08-0.26	0.12-0.33	0.15-0.40	0.20-0.45	0.25-0.50	0.30-0.55	0.35-0.60
Velocity in cavity (m/s) (for 800 W/m^2)	0.04-0.20	0.06-0.30	0.10-0.37	0.15-0.45	0.20-0.50	0.25-0.55	0.40-0.60	0.30-0.65

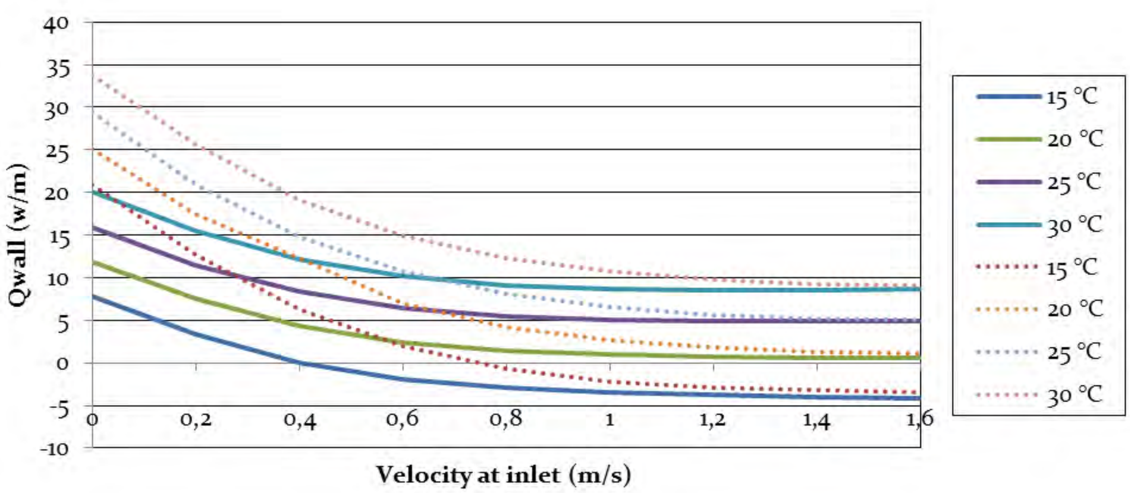


Figure 4: Graph of the heat transfer at the inside of the internal wall to the room, set off to the velocity at the inlet (dotted lines are 800 W/m^2).

The graph in **Figure 4** shows a steadily decreasing inward heat transfer (q_{wall}, measured at the inside of the concrete) for all the temperatures, but the slope at the first levels of velocity at the inlet is significantly steeper than from 0.8 m/s. The highest drop can be seen from 0 m/s (natural convection) to 0.2/0.4 m/s (ventilated façade). There is a fast reduction in the heat transfer into the room and reduction in the cooling load for velocities up to 0.8 m/s at the inlet (range of 0.15-0.45 m/s inside the cavity).

The energy saving increases remarkably as solar radiation intensity increases; the higher the solar radiation is the more efficient ventilated facades turn out to be from an energy saving point of view. The facades where the outer facing is made of reflecting materials (special steels, titanium alloys, etc.) strongly reduce the solar radiation influence and should be considered as an alternative to ventilated facades.

3.2 Façade designs

In total 5 base cases have been simulated, while some have been adjusted to increase their performance. For every design 3 simulations are made for the different types of wind flows. The first case is used as a reference to see whether applying a different shape to the façade has any benefit for accelerating the air velocity inside the cavity. In **Figure 5**, design 0 is shown for the upward-, downward- and perpendicular flow, in this reference case the 3 wind flows have different velocities inside the cavity. Therefore a comparison is made between the different types of wind flows.

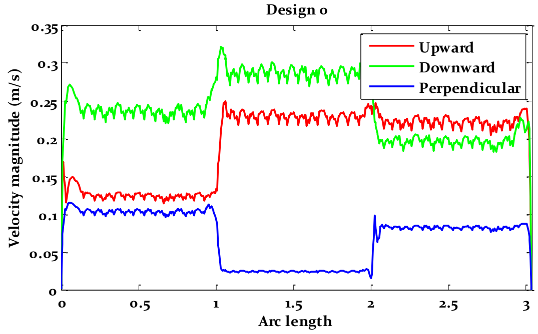


Figure 5: Velocity along the middle of the cavity for design 0.

From comparison of the different base cases, it followed that design 3 induced very high velocities inside the cavity. By keeping the same principle of the façade elements, but by adjusting the design to design 3B it led to an increased performance. A curvature on the façade is removed, while one curve is moved to the bottom as shown in **Appendix I**, **Table 4**.

An adjustment is made for the location of the opening at the bottom of the façade, which resulted in design 3C and 3D as shown in **Appendix I**, **Figure 18** and **Figure 19**. A comparison between the performance of design 3B, 3C and 3D can be seen in **Appendix I**, **Figure 20**. The graph clearly shows that the highest velocities are obtained for design 3B, where the opening in the façade construction at the bottom are a little beneath the most outer part of the curve. The wind flows that run parallel to the façade extract the air inside the cavity by creating under pressure at the specific spot, which causes the velocity inside the cavity to increase.

As design 3B shows much potential, a few other adjustments were made that led to design 3H. In this design the upper curve of 3B is removed and the cavity width in this part is thinner than was the case in design 3B. This adjustment led to a minor improvement at the upper part of the façade where the velocity increased. In **Figure 6** the results of the different adjustments are compared for design 3, 3B and 3H for the upward- and downward flows.

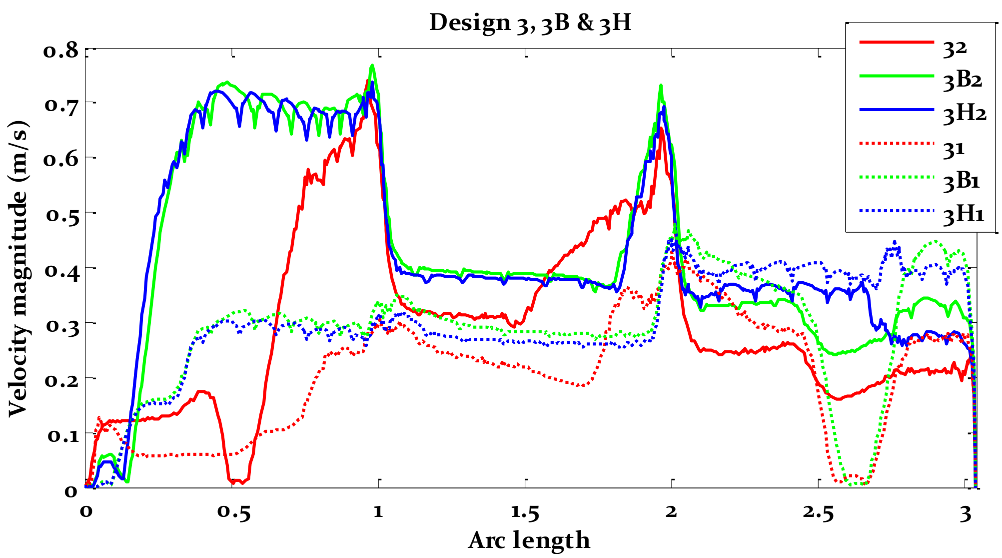
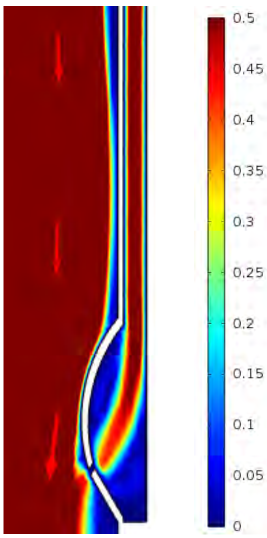


Figure 6: Graphical display of the velocity in the middle of the cavity for design 3, 3B and 3H for upward[2]- and downward[1] flow.

The graph shows that design 3B and 3H are almost similar, except that the velocities at the top of the façade (arc length 2.5-3.04m) differ by a small amount, where design 3H has higher values for upward flow and downward flow in this part. It is also clearly visible that the adjustments made to design 3 have led to the improved situation with higher air velocities inside the cavity.

The final design 3H is compared to the reference design of the ventilated façade in **Figure 7**. Where it can be seen that the velocity increases significantly in comparison to the reference design, in some parts an increase is reach of up to 75%.

The results obtained from the heat balance model simulations show that energy saving is possible with a ventilated façade and increases with higher levels of air velocity. The different façade designs show that it is possible to obtain these high velocity levels by applying specially designed façade elements that induce an increase in velocity inside the cavity.



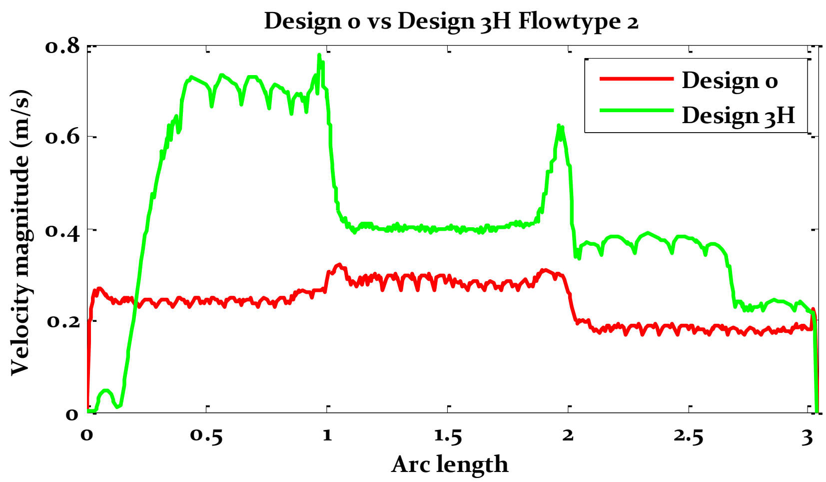


Figure 7: Velocity along the middle of the cavity for design 0 and 3H for downward flow.

4. Conclusions

Ventilated facades can play a fundamental role in the thermal performance of buildings, the outer façade construction acts as a barrier between external and internal conditions and helps to reduce the energy consumption for cooling, ventilation and air conditioning.

Steady state calculation models have been made to simulate and study the energy performance of a ventilated façade to compare different types of façade elements that increase the velocity inside the cavity. Results show that it is possible to obtain a cooling effect when using a ventilated façade which is 3m high with openings of 0.01m. The simulated heat balance shows that energy saving is increased with a ventilated façade over a conventional sealed façade, and is more effective for higher solar radiation. The higher the air velocity inside the cavity, the more heat is carried away.

A reduction of at least 50% on the inward heat transfer due to solar thermal radiation can be seen for a cavity that is ventilated with an air velocity of about 0.35 m/s in comparison to a conventional sealed façade (0 m/s).

In the second part of the study it is shown that façade elements can be designed in such a way that they increase the air velocity inside the cavity of a façade construction to remove heat more efficiently. The façade design 3H makes use of overpressure and under pressure at different parts of the façade to create higher velocities inside the cavity. When comparing design 3H with the reference case design 0, an improvement of up to 75% of the air velocity is reached in some parts of the cavity. In the different designs it can also be seen that the shape and position of the openings are very sensitive, and very quickly induce different results when changed slightly. The real air velocity results from the simulations can't be comprehended with accuracy, as the actual velocities inside the cavity induced by the wind are obtained by the application of a decoupled situation. Therefore a comparison of different designs has been made to show the potential of specially designed ventilated façades.

At this point it is difficult to give a definite criterion on the overall performance of the ventilated façade, because of absence of transient results. The façade should be evaluated over a year, because it is more beneficial when heated by solar radiation, which is higher during the summer. Nevertheless, the results in this study show that a ventilated façade could be a more energy efficient system than the conventional sealed façade, by reducing the cooling load.

5. Recommendations

Coupled or decoupled simulation

Use has been made of 2 simulation programs to simulate the air flow around the buildings and inside the cavity due to the available computational resources present, the accuracy of this method is questionable. When only one model is used to simulate the whole situation, the accuracy will be higher (Nore, et al., 2010). Therefore in further studies concerning this and related subjects the problem should be investigated using a coupled situation.

3D or 2D modeling

Another assumption in this study is the use of 2D modeling for the air flow near the façade, this means that air flow going around the sides of the building can't be assessed and façade element designs can neither be designed to relate to 3D air flows. An example of a 3D application could be the use of mushroom shaped tubes that collect air flows from outside and transport it inside at several points on the façade.

From this it also follows that the used 2D façade elements are assessed on a 2D dimensional flow, which means that only upward and downward flows are taken into account, while flows going sideways might have a large influence on the effect of the façade elements to increase the velocity inside the cavity.

Influencing factors

A more specific study on the influence of the geometry of the façade construction is needed to see, for example whether the flow rate increases with an increasing cavity width or that the roughness of the material inside the cavity affects the continuity of the heat flux and pressure losses. More investigation on the shapes and configurations of the façade elements is useful to see if the ideas created in this study can be worked out in a building element that can easily be integrated into a building. As a last it should be stated that the environmental factors also have an influence on the overall performance of the ventilated façade, for example; wind direction, wind velocity, orientation of the façade and the yearly distribution of solar radiation are to be considered and should be evaluated to conclude on the yearly energy savings.

The idea of combining the elements with closing devices, such as valves which can regulate the flow in the cavity when needed, has not been discussed in the study any further and should also be investigated as this principle makes the use of a ventilated façade more interesting for thermal load reductions during colder temperatures.

6. References

Beranek, W. & van Koten, H. (1979). Beperken van windhinder om gebouwen, deel 1. Deventer: Stichting Bouwresearch no. 65, *Kluwer Technische Boeken BV* .

Ciampi, M., F. Leccese and G. Tuoni (2003) "Ventilated facades energy performance in summer cooling for buildings." *Solar Energy* 75: 491-502.

COMSOL. *COMSOL Multiphysics 4.2.0.150*. (1998-2012).

Fluent Inc. *Fluent 6.3.26*. (2012).

Franke, Jörg, et al. (2007) "Best practice guideline for the CFD simulation of flows in the urban environment." *COST Action 732 Quality assurance and improvement of microscale meteorological models*.

Nore, K., B. Blocken and J.V. Thue. (2010) "On CFD simulation of wind-induced airflow in narrow ventilated facade cavities: coupled and decoupled simulations and modelling limitations." *Building and Environment*.

Schwarz, B. (1973) "Witterngsbeansphruchung von Hochhausfassaden." *HLH Bd.* 24.12: 376-384.

Urban, J.B. (2007) *The MIT Design Advisor: Simple and Rapid Energy Simulation of Early-Stage Building Designs*. Massachusetts.

7. Appendix I

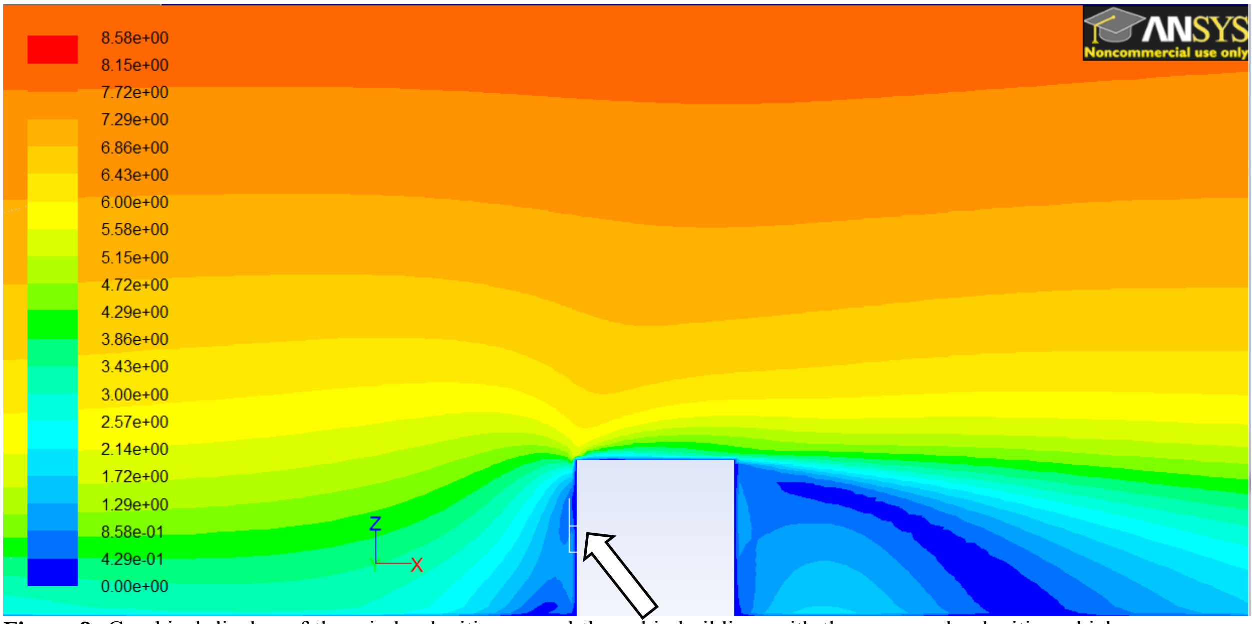


Figure 8: Graphical display of the wind velocities around the cubic building, with the measured velocities which are used for the building envelope model.

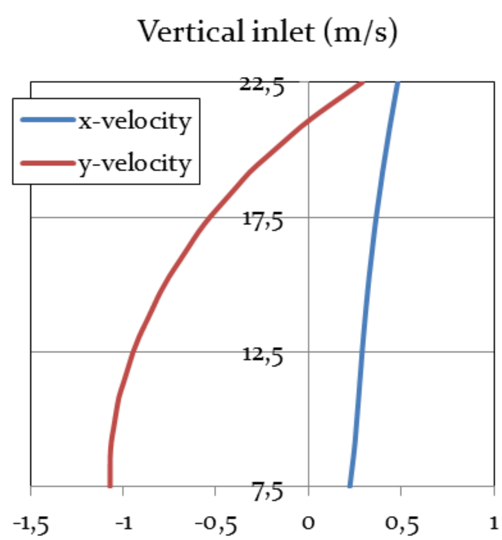


Figure 9: Velocity (m/s) measured at 1.5m from the façade at a height of 7.5m-22.5m

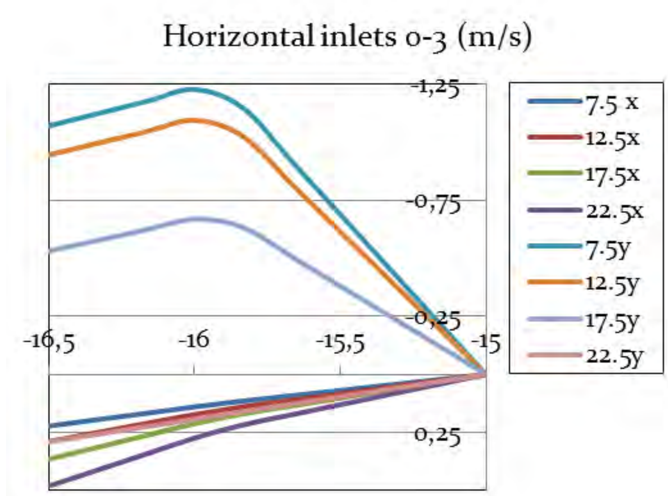


Figure 10: Velocity (m/s) measured at 7.5m, 12.5m, 17.5m and 22.5m height. (x = x-velocity, y = y-velocity).

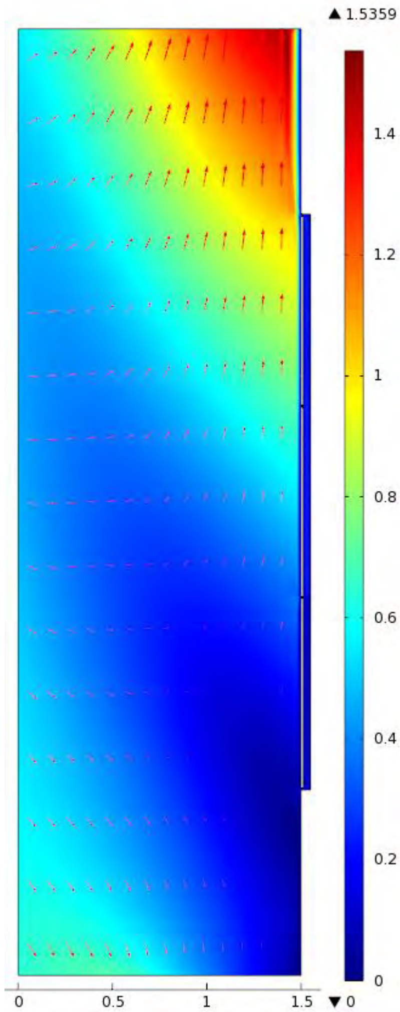


Figure 11: Type 1, (upward flow).

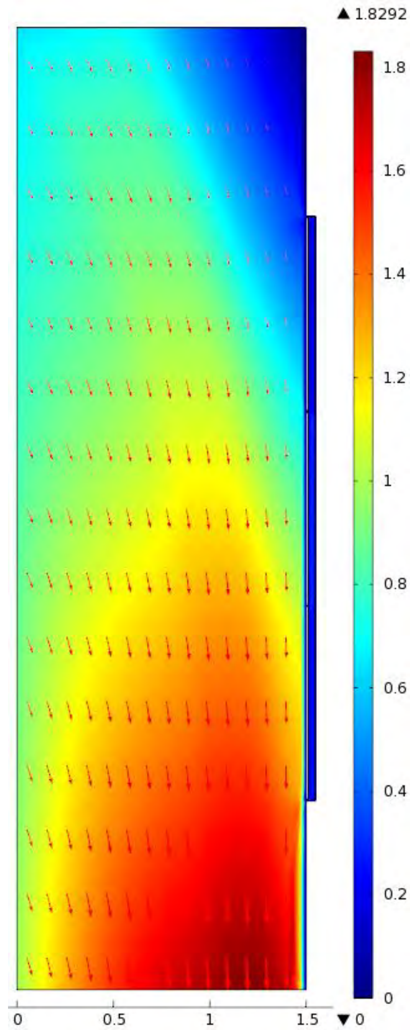


Figure 12: Type 2,(downward flow).

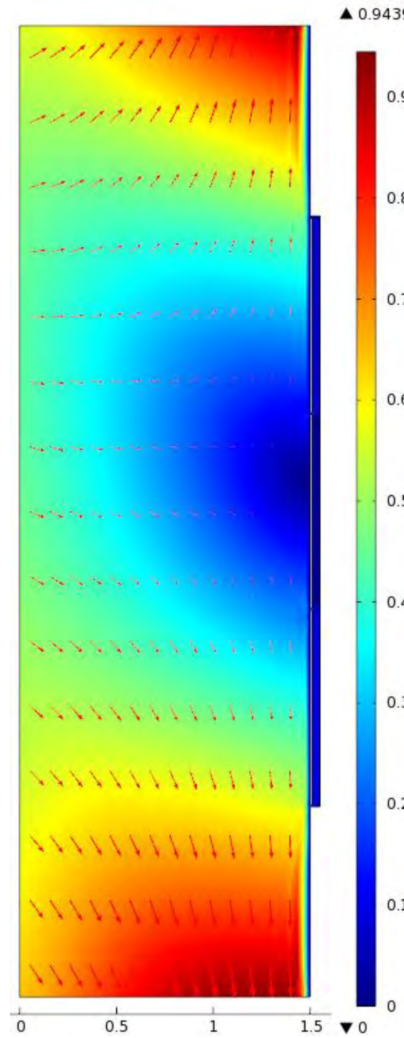


Figure 13: Type 3, (perpendicular flow).

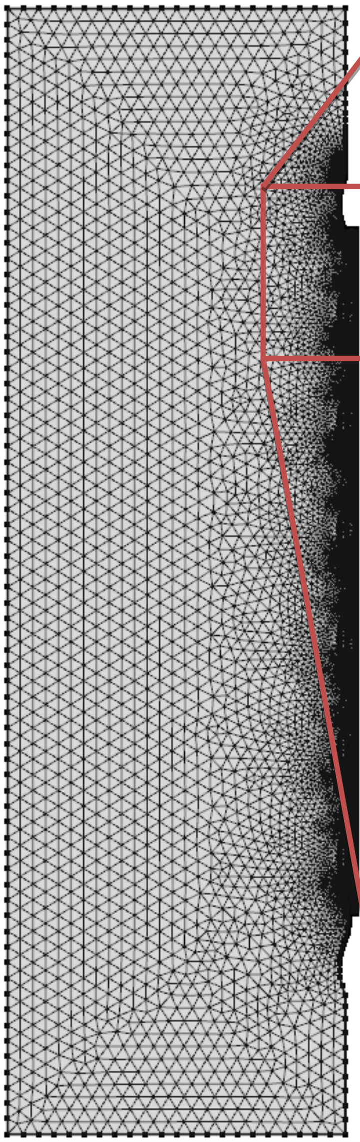


Figure 14: Mesh of design 4A with curved facade elements.

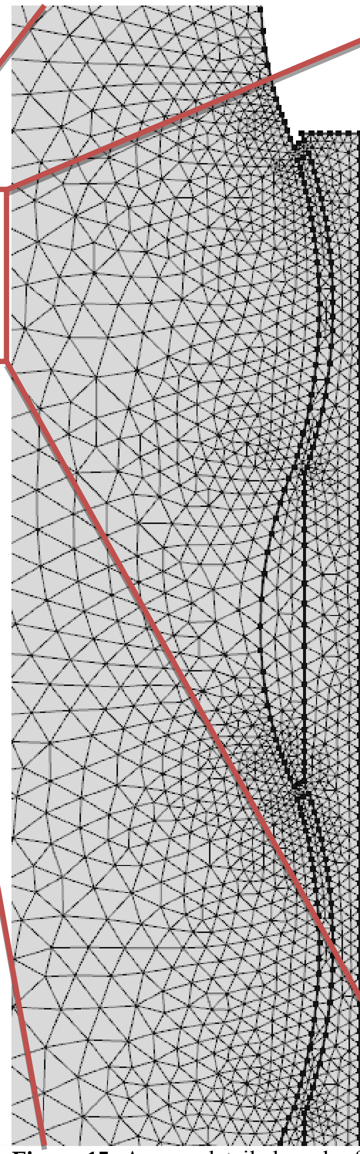


Figure 15: A more detailed mesh of design 4A with curved facade elements.

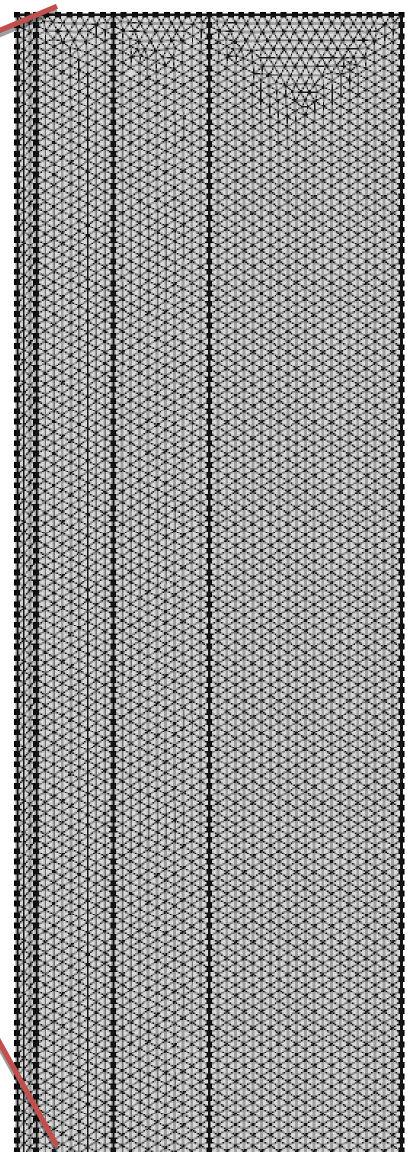
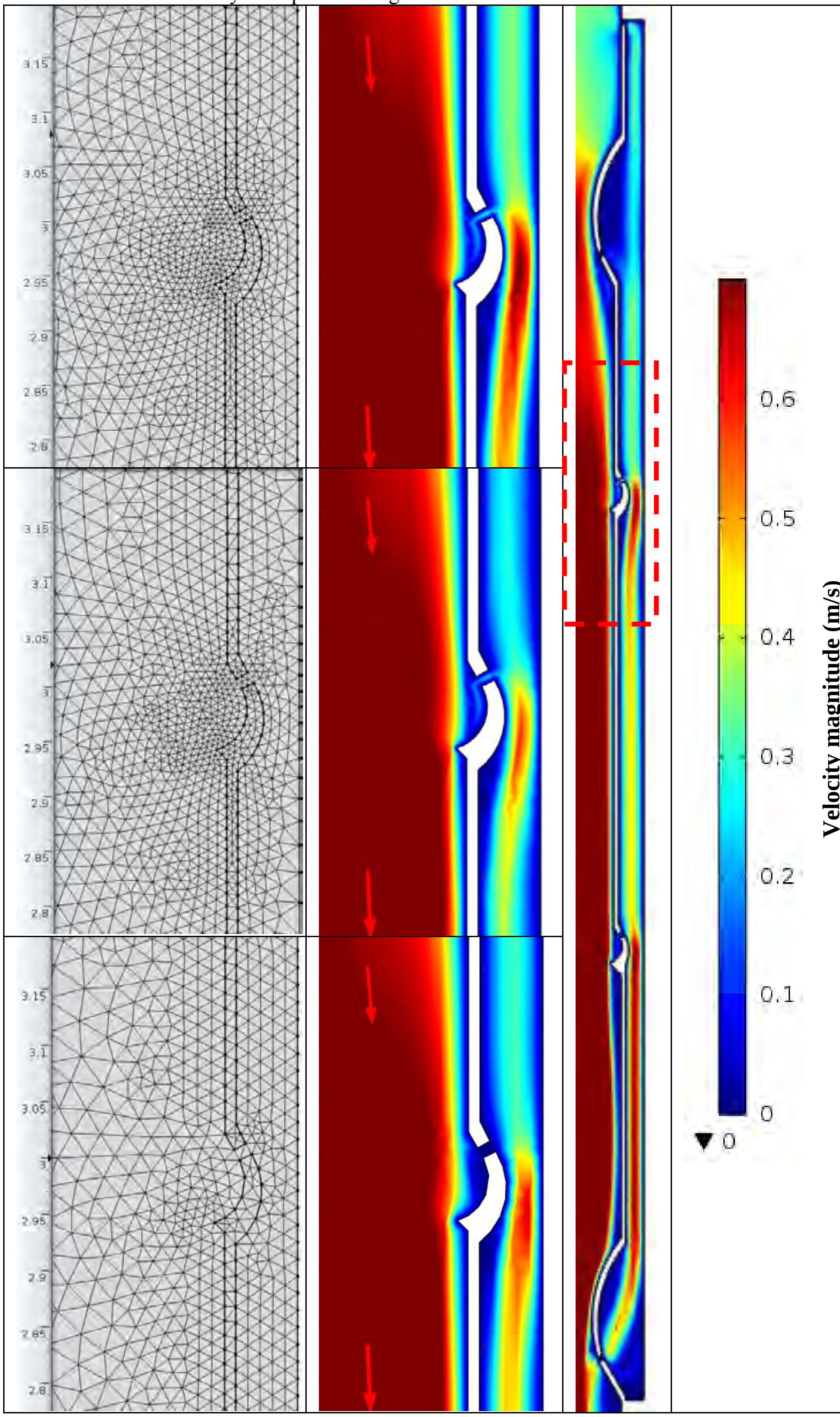


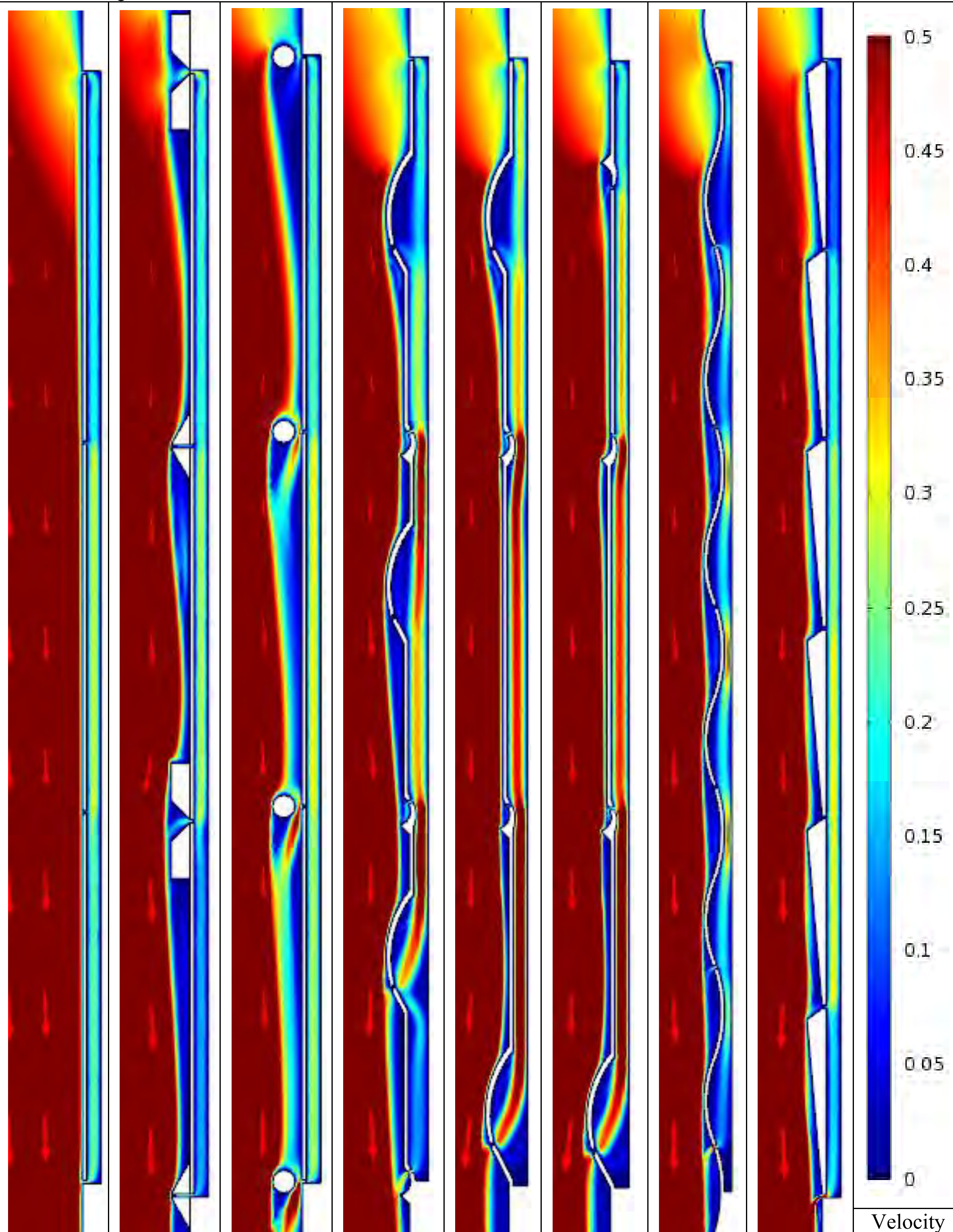
Figure 16: Mesh of the Heat balance model.

Table 3: Mesh sensitivity for a part of design 3B for downward flow.

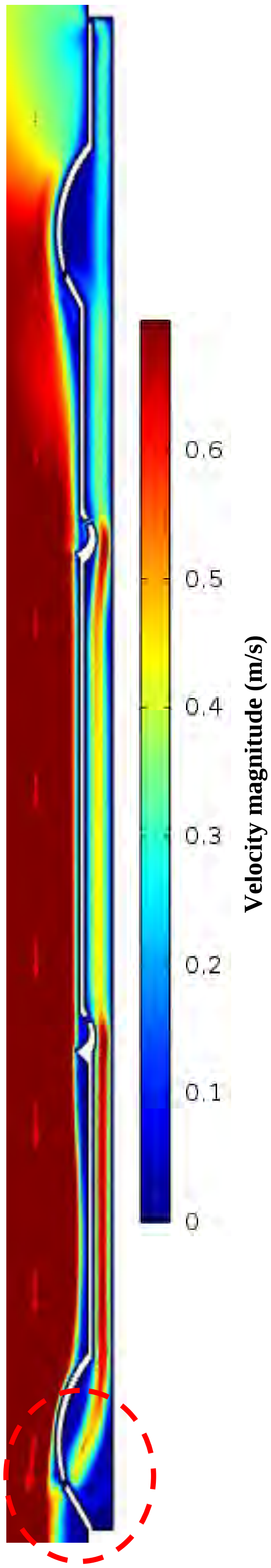


Mesh size: Normal Distribution inner wall: 200	
Mesh size: Finer	
Mesh size: Finer Distribution inner wall: 200	

Table 4: Designs simulated with COMSOL for the 3 different wind flows.



0	**1**	**2**	**3**	**3B**	**3H**	**4**	**5**	Velocity range scaled



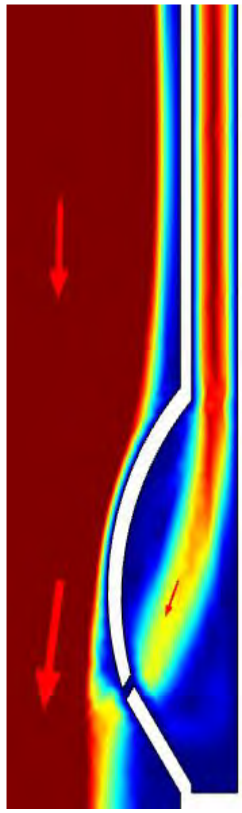


Figure 17: Bottom facade, design 3B.

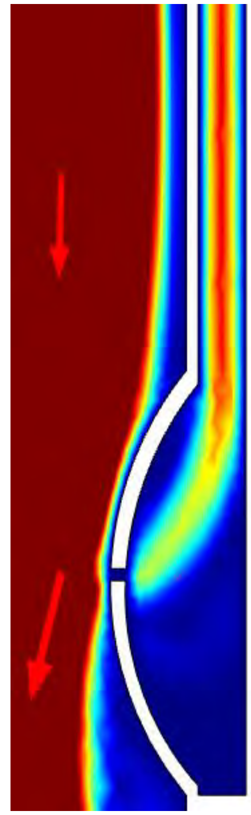


Figure 18: Bottom facade, design 3C.

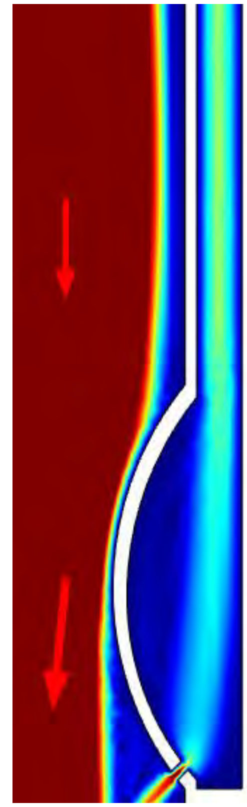


Figure 19: Bottom facade, design 3D.

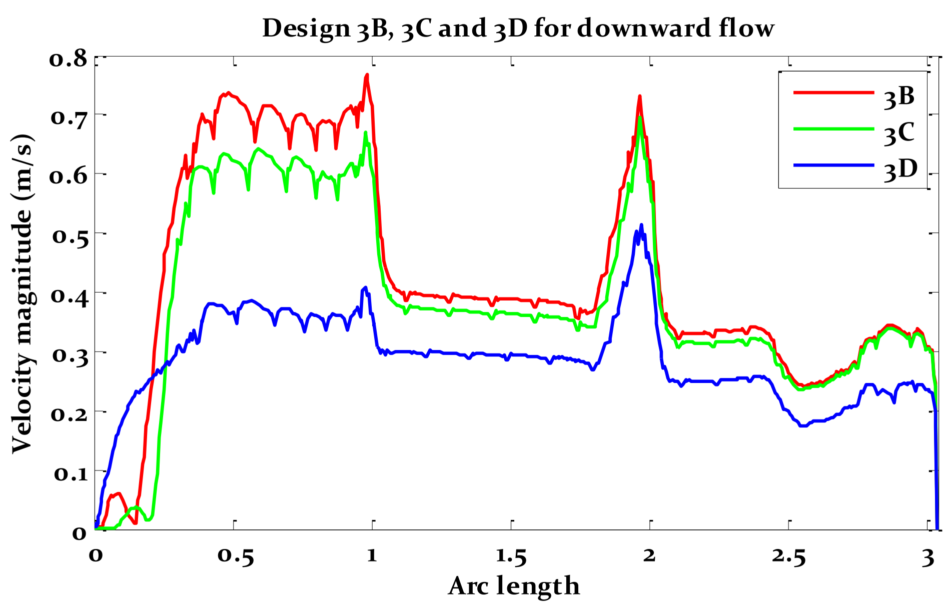


Figure 20: Graphical display of the velocity in the middle of the cavity for design 3B, 3C and 3D for downward flow.