CEST 2019

16th International Conference on Environmental Science and Technology

Rhodes, Greece, 4 to 7 September 2019

Thermal bridge modelling, based on conjugated heat transfer and CFD methods

Mijorski S. [1], **Ivanov M.** [2,*]

[1] PhD, SoftSim Consult Ltd., Consultant at Technical University of Sofia, FPEPM, Department: "Hydroaerodynamics and Hydraulic Machines", Sofia 1000, Bulgaria

[2] Senior Assist. Professor, PhD, Technical University of Sofia, FPEPM, Department: "Hydroaerodynamics and Hydraulic Machines", Sofia 1000, Bulgaria

*corresponding author: Martin Ivanov: e-mail: m_ivanov @tu-sofia.bg

Abstract

In the building structure, the "thermal bridge" is defined as an isolated zone, where construction elements have higher thermal conductivity, compared with the rest of the building envelope. Thus, a significant temperature difference may exist between adjacent solid and air volumes within the developments, especially in winter conditions. The existence of thermal bridge mostly affects the energy performance of buildings, due to the increased heat losses from the occupied spaces. But also, the decreased surface temperature in these zones, could lead to moisture accumulation and substantial humidity related problems in the indoor environment.

Considering these important effects, the presented study describes the development of numerical model of a thermal bridge distribution, based on conjugated heat transfer in concrete external wall section. The thermal bridge distribution is analyzed relative to the indoor and outdoor air parameters, and the envelope thermal properties. The achieved surface temperature sensitivity results may be used for further moisture accumulation model development or enhancements.

Keywords: Thermal bridge, conjugated heat transfer, CFD, indoor and outdoor environment

1. Introduction

Nowadays, norms and standards exist for avoiding thermal bridges during the building design, which is strongly recommended and well spread practice [*Papadopoulos (2016)*]. It is also known that the linear thermal bridges may increase the heating energy need of the buildings with above 30% [*Erhorn-Kluttig (2009)*]. But still, thermal bridges exist, especially in older buildings, where no energy efficient measures have been applied. Also, they may have significant impact over the indoor air quality. The cold regions of the thermal bridges, particularly in winter conditions, may be a precondition for moisture associated problems in the buildings [*Ivanov (2019)*].

As mentioned, todays' norms and standards give the necessary parameters for the thermal bridge analyses, based on long term construction experience and scientific research in this area [*ISO 10211 (2017)*]. Nevertheless, the development of precise, multiparametric numerical models of the related phenomena, is considered to be in help of the buildings design practice.

2. Aim of the Presented Study

The aim of the presented study is to develop a numerical model of a thermal bridge distribution in concrete external wall, as well as to investigate the indoor and outdoor parameters' effect, by the methods of Computational Fluid Dynamics (CFD) and conjugated heat transfer.

3. Numerical Model and Experimental Set-Up

On Fig. 1. A) is shown the geometry used in the presented study. The modelled wall section is with dimensions 1 m by 1 m, and thickness of 0.2 m. While, entire test section is with dimensions 4 m by 4 m. The material used is a concrete with density of 2400 kg/m^3, specific heat of 750 J/(kg K), and thermal conductivity of 1.8 W/(m K).

For the purposes of the analysis a constant indoor environment conditions are modelled with air temperature of 297.15 K. This is accomplished by providing constant air flow along the interior of the test section with uniform velocity of 0.1 m/s parallel to the test section. This way the impact of the different parameters over the interior surface temperature would be highlighted. The outdoor environment parameters are model again with constant air temperature of 253.15 K and flow parallel to the exterior of the test section. The configuration is tested for different external wall roughness heights in the range of 0.3 mm to 5.0 mm and different wind speeds – from 0.5 m/s to 3 m/s.

For numerical discretisation the snappyHexMesh is utilised, creating a hexahedral and polyhedral mesh of 485608 control volume cells. All the simulations are completed with the CFD software Ansys Fluent 16.0. This way, 10 steady state simulations are performed with use of Realisable k-epsilon turbulence model and standard wall function.

The standard wall function implemented in Fluent is proposed by Launder and Spalding [*Launder-Spalding (1974)*]. The momentum at the wall boundary is calculated by law-of-the-wall for mean velocity as follows:

$$U^* = \frac{1}{\kappa}\ln(Ey^*),$$

where: $U^* = \frac{U_P C_\mu^{1/4} k_P^{1/2}}{\tau_\varpi/\rho}$ and $y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu}$; κ - von Kármán constant, 0.4187; E - empirical constant, 9.793; U_P - mean velocity of the fluid at given point; k_P - turbulence kinetic energy at given point; y_P - distance from the point to the wall surface; μ - dynamic viscosity of the fluid; ρ - density of fluid.

The logarithmic law for mean velocity is employed when $y^* > 11.225$, and when the computational grid is finer such that $y^* < 11.225$, it is applied a laminar stress-strain relationship between mean velocity and y plus: $U^* = y^*$. Thus, the law-of-the-wall for energy comprises two different laws: “linear law for the thermal conduction sublayer where conduction is important” and “logarithmic law for the turbulent region where effects of turbulence dominate conduction”, as it is implemented in the CFD software Ansys Fluent 16.0.

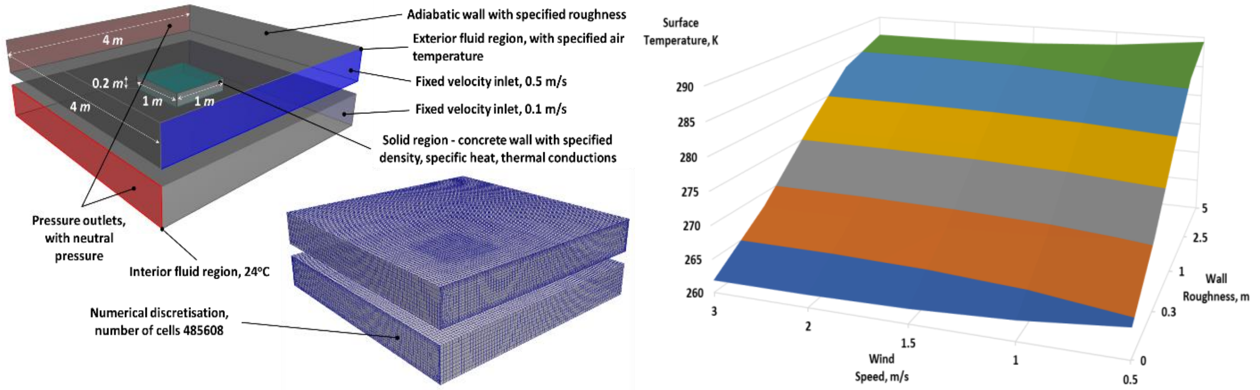

Figure 1. A) Experimental Set-up and computational grid, **B)** Results for wind speed and wall roughness sensitivity

4. Numerical Results

The results for the wind speed and wall roughness impact, over the internal wall temperature are shown on Fig. 1. B). It is seen that, the increase of the external wall wind speed from 0.5 m/s up to 3 m/s, leads to decrease of the internal wall temperature from 270.5 K to 268.0 K. This decrease is attained without changing the wind speed direction, which might have additional impact over these values. With increase of the wind speed a drop of the air temperature reached around 2 m/s. Further increase of the wind speed under model wall conditions is not observed.

More distinctive is the impact of the wall roughness, over the internal wall temperature. The results show 260.0 K at the simulated smooth wall (roughness 0 mm), which rises to almost 290.0 K, for simulated wall with roughness of 5 mm. This effect is explained with the boundary layer flow, near the solid wall’s external side, which decreases the heat energy transfer between the solid and cold fluid medias. However, this phenomenon would require additional analyses of the turbulence models and applied wall functions. Also, sufficient measurement data is not yet available, in order to validate the presented numerical model.

5. Conclusion

A numerical model of a thermal bridge in external concrete wall is developed, and the impact of the external wind speed and external wall roughness is studied, by the methods of CFD and conjugated heat transfer. The external wall roughness had more pronounced impact over the temperature drop in the inner zones of the simulated wall, however the explanation of the related phenomena requires further precise modelling in the solid-fluid boundary layer region.

Acknowledgements

The presented study is supported by the National Science Fund at the Ministry of Education and Science of Bulgaria, under the activities of: “Competition for financial support for projects of junior basic researchers and postdocs – 2018”, with Contract № КП-06-М27/4, entitled: “Numerical assessment of effective measures for prevention of moisture accumulation on external walls with thermal bridge, in residential rooms without organized ventilation”.

References

Erhorn-Kluttig H., Erhorn H. (2009), “Impact of thermal bridges on the energy performance of buildings”, Paper P148 of the *EPBD Buildings Platform*.

ISO 10211 (2017), “Thermal Bridges in Building Construction - Heat Flows and Surface Temperatures. Detailed calculations”, *International Organization for Standardization*.

Ivanov M. (2019), “Instantaneous field measurements of thermal bridge parameters in ground floor residential room”, Submitted for publishing in: Proc. of “*8th Int. Conf. on Thermal Equipment, Renewable Energy and Rural Development – TE-RE-RD’19*”.

Launder B. E., Spalding D. B. (1974), “The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering”, 3:269-289.

Papadopoulos A. (2016), “Forty years of regulations on the thermal performance of the building envelope in Europe: achievements, perspectives and challenges”, *Energy Build.* **127**.