Analyses of wall surface condensation risk, based on CFD model with conventional room radiator

M P Ivanov1,*, S G Mijorski2

1Technical University – Sofia, FPEPM, Department:’” Hydroaerodynamics and Hydraulic Machines”, Sofia 1000, Bulgaria
2SoftSim Consult Ltd., Consultant at TU - Sofia, FPEPM, Sofia 1000, Bulgaria

*Corresponding author’s email: m_ivanov@tu-sofia.bg

Abstract. The presented paper reveals a numerical study of the risk of condensation over exterior wall of a typical residential room. The study is based on CFD methods with implemented conjugate heat transfer. The analysis is contributed by a study of the impact of the heat released from conventional room radiator with various heat loads. This way, the resulting surface and dew point temperature distributions are derived over the simulated exterior wall. The results demonstrate that with the increase of the room relative humidity, the percentage of the external wall area, which might be affected by condensation increases significantly. This effect is observed regardless of the selected increase of the radiator heat load (in this particular study, the range is between 1000 W and 1500 W). For the range of relative humidity values from 40% to 60%, the area with increased risk of condensation increases to above 80%. This result demonstrates the crucial importance of the control of the room relative humidity within occupied premises, especially if condensation mitigation measures must be applied. The presented analyses and assessment method are rather innovative for the presented engineering area and would be in help in the buildings and indoor environment design process.

1. Introduction

The risk of condensation over the walls and windows surfaces is one of the most important factors affecting the indoor air quality in the occupied premises. It is well known that the excessive indoor humidity drives the mold growth and increases the harmful reactions between different indoor pollutants. Furthermore, the dampness in buildings is strongly related to several important health risks in the occupants, such as the development of different respiratory illnesses, allergies and asthma [1]. However, the associated mechanisms of moisture accumulation are complex and require analyses in different directions, because all the indoor environment parameters are linked to that phenomena.

Nowadays, the Computational Fluid Dynamics (CFD) methods are used for prediction of indoor climate parameters and airflow distribution, with full extend of the environmental properties and in the entire building volumes. Thus, it is recommended to use CFD for such analyses, especially in combination with the velocity distribution effect over the related surface temperatures, or simply to simulate correctly the convective processes in the studied domain. As suggested in [2], the selection of turbulence model is critical, and the most widely used k-ε model revealed difficulties in predicting natural convective heat flows, like the one from room radiators. In such case, the implementation of user-defined wall functions is good solution, but also the more precise k-ω SST turbulence model must be applied.
The room heating radiators are considered to be one of the most widely used heating devices in the residential buildings, and their performance development is still continuous process [3]. Based on that, the standard thermal output of these devices, as well as different configurations, system arrangements and testing methods are determined by the European norm EN 442-2:2014 [4]. Also, CFD based methods are used for studying the enhance effect of the position on the wall, over the temperature gradient distribution in occupied spaces, as in the work of Jahanbin and Zanchini [5]. In their work, a numerical simulation is conducted to calculate the velocity and temperature fields, the total power released by the radiator, as well as the operative temperature, which is considered as the main performance indicator of the studied radiator. Their results suggest that configuration of increasing-with-height linear temperature distribution at its surface have better performance, especially in combination with distance of 0.10 m between the radiator and the wall.

In general, all studies in the presented area aim to improve the performance of the heating radiator systems. This plays a major role in the demand for buildings energy savings and clean environment. Thus, even the flow type in the radiator configuration may be analyzed, as in the work of Embaye at al [6]. They suggest that the pulsation flow in the radiators can lead to improving the specific heat output without compromising the indoor spatial temperatures and convective velocity distribution, which means that the occupant’s thermal comfort will be unaffected.

However, there is a lack of numerical studies about the humidity related problems, with respect to the performance and configuration of indoor heating devices. For example, the properly installed and operating room radiators, may mitigate the negative effect of the moisture accumulation over existing thermal bridges in the premises [7]. The thermal bridges appear, where the building elements have higher thermal conductivity, compared with the rest of the construction envelope, and influence the energy efficiency of the buildings [8]. Modelling these complex phenomena is not an easy task and requires significant skills, experience and properly established initial and boundary conditions, in terms of conjugate heat transfer and indoor and outdoor environment parameters [9].

The aim of the presented study is to perform moisture accumulation analyses to an exterior wall of a typical residential room through numerical modelling. By CFD methods of conjugate heat transfer and thermal radiation the analysis accounts for the impact of a radiator with various heat loads over the resulting surface temperature and dew point temperature.

2. Methods of the presented study

The main methods of the study are based on CFD numerical simulations, with implemented conjugate heat transfer and radiation. For this purpose, a typical residential room is modelled, with a single wall facing the outside environment. One window and one conventional panel radiator, used as a heat source, are placed on this external wall. The conjugate heat transfer through the wall is modelled, in combination with various heat output from the radiator. Then, the resulting surface temperature over the inner side of the external wall are compared with the resulting dew point temperature in the room, in order to assess the corresponding risk of condensation.

The dew point temperature (in °C) is one of the most convenient parameters for analysis of moisture accumulation, or as mentioned – the so-called condensation. It represents the saturation temperature of the water vapor, which corresponds to the partial pressure of water vapors in a humid air for the particular environment. The water vapor saturation pressure, as a function of its temperature is calculated by:

$$p_r = 100 \exp \left( \frac{-4030.18}{r + 235} \right) Pa,$$

where the temperature of the humid air and water vapor tr is in °C. This formula is used also in [10], and it is based on the computational code, published in ISO 7730 [11]. After mathematical conversion and considering the relative humidity calculation, the dew point temperature (tdp) can be calculated by:
\[ t_{dp} = \frac{1}{\frac{1}{235 + t_r} \ln \left( \frac{R_{H_r}}{100} \right) - 235} \]

where it is represented as a function of its room temperature \((t_r)\) and relative humidity \((R_{H_r})\).

It is considered that the air dew point temperature in the indoor environment will be the temperature of the solid surroundings at which and below which a condensation will be observed. Naturally it happens over the coldest elements in the rooms, like the surface of the external walls, or over the window frames and glazing. But also, such condensation may occur in areas with established thermal bridges as well. For this reason, one of the best practices is to place the heating radiators below windows and normally on external walls.

3. CFD model development

The modelled interior volume of the typical residential room is with dimensions 4 m by 4 m and with the addition of three solid regions at the exterior wall, including concrete panel with 0.3 m thickness, aluminium window frame with 0.05 m, and double glazing with 0.02 m (see Fig. 1). The radiator is placed under the window at 0.05 m gap between heater and exterior wall surface. This way, conjugate heat transfer is modelled only through the construction elements exposed to the ambient conditions with fixed air temperature of -10°C.

![3D model of the assessed typical residential room](image)

While for the interior walls, it is implemented 2D convective heat transfer model with following parameters: wall thickness 0.2 m, thermal conductivity 1.8 W/mK, wall exposure temperature of 20 °C, and surface emissivity \(\varepsilon = 0.91\). Also, for the purposes of the assessment any furniture or other interior details are omitted in the model.

The material properties used for the solid regions are listed in Table 1:
Table 1. Solid regions material properties

| Material type      | Density | Specific heat | Thermal conductivity | Emissivity |
|--------------------|---------|---------------|----------------------|------------|
| Concrete wall      | 2400    | 750           | 1.8                  | 0.85       |
| Aluminium profiles | 2700    | 910           | 2.8                  | 0.09       |
| Double glazing     | 2400    | 750           | 2.8                  | 0.92       |

The numerical model is developed with multipurpose CFD software Helyx® (https://engys.com/products/helyx). In total, 3 steady state simulations based on conjugate heat transfer and thermal radiation with total of 32 beams and different total heat loads of the radiator, while preserving all other model parameters. The turbulence model used in the modelling is SST k-omega. This way, the impact of the heater over the process of condensation would be outlined.

The modelled cases in the study include:

• Case 1: \( Q_{\text{total}} = 1000 \) W
• Case 2: \( Q_{\text{total}} = 1250 \) W
• Case 3: \( Q_{\text{total}} = 1500 \) W

The model discretization is completed with helyxHexMesh utility within Helyx®, creating a poly grid with predominant hexahedral elements (see Fig. 2). Total number of control volumes is 3269774 and number of cells of each type is as follows: hexahedra 3210335, prisms 12, and polyhedral 59427.

**Figure 2.** Numerical discretisation of typical residential room

All wall surfaces within the computational domain, including and solid regions had 3 prism layers projecting toward the interior of the volumes (see Fig. 2). The selected base cell size of 0.04 m is based on a grid sensitivity study completed by the authors. In this study, the base cell size of 0.05 m has been outlined as sufficient for performing grid independent solutions in the case of the conjugate heat transfer CFD modelling [9].

4. Numerical results and discussion

The numerical results for different radiator heat loads are presented in form of:

• Temperature and velocity field sections through the centre of the modelled room, shown on Fig. 3.
- Exposed wall internal surface temperature, together with risk of condensation plot, Fig. 4 to Fig. 6
- 3D graph relating the modelled radiator heat loads and different levels of air humidity to the percentage of the inner area of exposed construction elements with an increased risk of condensation, is presented on Fig. 7

The positions of the view directions and sections for presented results is illustrated on Fig. 1

**Figure 3.** Temperature and velocity fields at the centre of the room

In the temperature field figures, it is observed a separation of the radiator convective flow at the lowest heat load. This phenomenon could contribute for the process of condensation in the area of the exposed wall. With increasing the heat load, the separation reduces, and the convective flow attaches to the exposed construction element’s cold surfaces. This way, the attached warm flow would have a
beneficial impact over the possible condensation process. Same phenomenon is well observed in the velocity fields figures.

However, it should be noted that increased heat load of the radiator is connected to increase of the room air temperature. This way, zone of the exposed wall located aside of the radiator zone does not benefit from the convective flow attachment and have adverse impact resulted from increased dew point temperature of the room.

As it can be observed in the Fig. 4 to 6, a slight beneficial effect in the areas above the radiator is indicated in the surface temperature and risk plots. However, no impact is observed aside of the radiator zone with increase of the heat load. It should be noted that the presented model is developed in terms of the “worst case scenario”, i.e.: non-isolated concrete wall, low external air temperature, and low powered internal heat source.

![Figure 4. Surface temperature at the exterior wall and risk of condensation, Case 1, $Q_{total} = 1000 W$](image)

![Figure 5. Surface temperature at the exterior wall and risk of condensation, Case 2, $Q_{total} = 1250 W$](image)
Thus, stronger air stratification within the studied room can be observed in the vertical temperature distribution on Fig. 3. It can be seen that with the increase of the heat load from the radiator, the vertical temperature difference increases. This leads to significantly higher temperature difference between air and floor areas, which contributes to the floor surface condensation process, especially with high levels of internal air relative humidity.

In the further analyses, Fig. 7 demonstrates again the significant importance of the room relative humidity over the external wall condensations process. The resulting numerical data is summarised in the presented 3D graph.
It can be seen that with the increase of the relative humidity, the percentage of the external wall area, which is affected by condensation, increases significantly, regardless the increase of the radiator heat load. For relative humidity values between 40% and 60%, the affected condensation area increases from 40% to above 80%. These results demonstrate the crucial importance of the control of the room relative humidity within occupied premises, especially in the process of the exposed wall condensation mitigation efforts.

5. Conclusion

Numerical study of the risk of condensation over exterior wall of a typical residential room is performed and presented. The study is based on CFD methods with implemented conjugate heat transfer and thermal radiation. The analysis is contributed by a study of the impact of the heat released from conventional room radiator with various heat loads. This way, the resulting surface and dew point temperature distributions are derived over the simulated exterior wall.

The resulting temperature and velocity fields show good correlation with the corresponding physical phenomena. However, no physical experimental tests are done at this stage for validation of the presented numerical simulations.

It is demonstrated that with the increase of the internal air relative humidity, the percentage of the exposed wall interior area, which is affected by condensation increases significantly. The effect is observed regardless of the selected increase of the radiator heat load, initially selected in the range of 1000 W to 1500 W.

For the range of relative humidity values from 40% to 60%, the area with increased risk of condensation increases to above 80%. This result demonstrates the crucial importance of the control of the room relative humidity within occupied premises, especially if moisture accumulation mitigation measures must be applied.

The presented analyses and assessment method are rather innovative for the presented engineering area and would be of help for the building and indoor environment design processes.

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 № KPI-06-M27/4, entitled: “Numerical assessment of effective measures for prevention of moisture accumulation on external walls with thermal bridge, in residential rooms without organized ventilation”.

References

[1] Bornehag CG, Sundell J, Hagerhed L, Jason S, and the DBH study group 2002 Dampness in buildings and health. Dampness at home as a risk factor for symptoms among 10 851 Swedish children (DBH-step I), Proc. to Ind. Air Conf. 2002, 431-436
[2] Risberg D, Risberg M, Westerlund L 2016 CFD modelling of radiators in buildings with user-defined wall functions, App. Therm. Eng. 94 266–273
[3] Ploskić A, Wang Q, Sadrizadeh S 2019 A holistic performance evaluation of ventilation radiators – An assessment according to EN 442-2 using numerical simulations, J. of Build. Eng. 25 100818
[4] EN 442-2:2014 Radiators and convectors. Test methods and rating
[5] Jahanbin A and Zanchini E 2016 Effects of position and temperature-gradient direction on the performance of a thin plane radiator, App. Therm. Eng. 105 467–473
[6] Embaye M, AL-Dadah R, Mahmoud S 2016 Numerical evaluation of indoor thermal comfort and energy saving by operating the heating panel radiator at different flow strategies, Energy and Build. 121 298–308
[7] Ivanov M 2019 *Dew point temperature analyses in ground floor residential room with existing thermal bridge*, Proc. “8th Int. Conf. on Thermal Equip., Ren. Energy and Rural Dev. - TE-RE-RD 2019”, Web of Conf., doi: 10.1051/e3sconf/201911201017

[8] Papadopoulos A 2016 *Forty years of regulations on the thermal performance of the building envelope in Europe: achievements, perspectives and challenges*, Energy Build. 127. https://doi.org/10.1016/j.enbuild.2016.06.051

[9] Ivanov M and Mijorski S 2020 *Development of thermal bridge numerical model, based on conjugate heat transfer and indoor and outdoor environment parameters*, E3S Web of Conf. 180, 04011 TE-RE-RD 2020, https://doi.org/10.1051/e3sconf/202018004011

[10] Ivanov M and Markov D 2010 *Analyses and assessment of the dew point temperature in Bulgarian homes, based on results of the ALLHOME project*, Proc. to “Int. Sc. Conf. in Ruse’10”, Vol.49, Series 1.2, pp. 14-18

[11] ISO 7730: 2008, *Ergonomics of the thermal environment — Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria*