Bauhaus-Universität Weimar
06. / 07. März 2024
BAUPHYSIKTAGE IN WEIMAR 2024
WBA

ANALYTICAL COMPARISON AND OPTIMIZATION STUDIES OF BUILDING-PERFORMANCE SIMULATIONS ON THE BASIS OF THE FINITE-ELEMENT METHOD

Paul Schoplocher[1], Oliver Steffens[2]

[1] *OTH Regensburg, Germany, E-Mail: paul.schoplocher@oth-regensburg.de*

[2] *OTH Regensburg, Germany, E-Mail: oliver.steffens@oth-regensburg.de*

Abstract

In this paper, different building-physics related effects are examined using the finite-element method (FEM) to model convection, radiation, heat conduction, and moisture phenomena in building structures. indoor air and surface temperatures of a typical building model.

We compare these findings with the results obtained from a building-performance simulation (BPS) programme based on the lumped-parameter method (LPM). Subsequently, we study enhancements to the LPM, including those for thermal bridges and internal heat transfer coefficients, created with the help of more advanced multi-physics methods within the FEM model. These enhancements seem to yield improved results and suggest that the LPM software still has the opportunity for development.

The improved results indicate only minor differences between the FEM and LPM simulations concerning the average fluid temperature for the radiation, moisture physics, and heat transmission models. In terms of convection and thermal bridging, however, there are some notable differences and areas for development.

1. Introduction

Unlike in previous decades, nowadays the application of BPS in practice has become increasingly important. Building performance simulations have a positive effect on the environment and economy. Building behaviour can be predicted and analyzed to reduce energy use and CO_2 emissions. Consequently, a growing number of engineering companies are utilizing BPS techniques during the design stage of both residential and commercial structures (Baba et al. 2013).

The first record of building simulation dates back to 1965. Research was commissioned by the US Department of Defense (DOD) and Department of Energy (DOE) to examine how buildings function in various climates. The American Department of Energy (DoE) established the Lawrence Berkeley National Laboratory, which created the DOE-2 software tool as a result of these experiments (Crawley et al. 1997).

In 1963, the Royal Institute of Technology in Stockholm, Sweden, developed BRIS, the first computational simulation tool for buildings. The program utilized in this work, IDA ICE, was derived from this earlier version (Brown 1990).

More sophisticated tools started to appear in the 1970s as computers gained popularity and power. ESP-r, BLAST, HVACSIM+, and TRNSYS are a few examples. The development of standards like ASHRAE 90-75 came after this. The development of BPS evolved and progressed over time as a result of political and scholarly endeavours (Kusuda 1999).

Neutral Model Format (NMF) for building simulation was developed by Sahlin and Sowell (Bring et al. 1999) in the late 1980s, marking the beginning of the present state of the art. IDA ICE, which was developed in 1998, continues to use this model type. Klein (Seem et al. 1989) in that year unveiled the Engineering Equation Solver (EES), a tool used in engineering procedures to solve non-linear equations. Due to the short simulation time, the main method in building simulation remains the LPM (or zone method).

The finite-element method (FEM) was initially applied in 1982 by Cook (Cook 1974) and Rao (Rao 1982). The FEM is a numerical method for approximating differential equations in physics-based problems, which is done by breaking down the problem into smaller, easier-to-manage subproblems. The building industry uses finite element analysis (FEM) much less frequently than mechanical engineering, where it is widely used for applications. FEM is used in the building industry for a variety of purposes, including seismic and structural load calculations (Lucena et al. 2014; Besuievsky et al. 2021) in civil engineering (Mahmoud Ziada et al. 2019). FEM in building physics is sometimes used to calculate two-dimensional thermal bridges and moisture transport. Such as in the work of Berger (Berger et al. 2020).

Similar to this work, Schijndel (Jos Van Schijndel 2015) has already discussed the coupling of the FEM and LPM. Radiation and moisture, however, were not examined in this investigation. The LPM and FEM models were found to differ minimally. It was also determined that FEM is suitable to be used for building performance simulations. This work likewise comes to the same conclusion.

Nevertheless, the engineering community still uses BPS at a low rate, despite its advantages for the economy and the environment. There are still many areas where these BPS tools could be improved upon, and their development is not flawless. One of them is attempted to be addressed in this work. Clarke outlined several issues facing BPS (Clarke 2020) as well as potential future directions for this technology (Clarke 2015).

2. Simulation

For proper improvement and comparison of the two methods, it is necessary to have a reference model. This reference model is created in both LPM and FEM software to identify the significant differences and to exclude differences that are not due to physical effects. An example of this could be differences in numerical values. Each of the different models takes a specific improved thermal-physical model from the FEM software and compares it with a reference model in the FEM software. This makes it possible to analyze the specific effect, rather than filtering out which effect is causing which difference. The reference model is called the simplified FEM model (sFEM). This model will be discussed in the subchapter "2.2. Simplified Model (sFEM)".

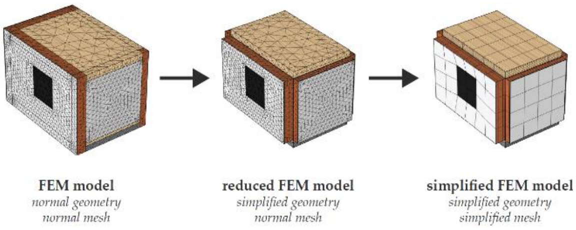


Figure 1: FEM model simplification (own illustration).

2.1. Boundary Conditions

This section describes the boundary conditions for the thermal, meteorological, and geographic conditions of the model.

The reference model is a corner room ("zone") of a residential building which is part of the co-operative housing estate Margaretenau in Regensburg. The weather data is given from the German weather agency (Deutscher Wetterdienst 2018). The weather data is for the period of 1.1.2018 at 00:00 to 31.12.2018 at 24:00. The floor plan in Figure 2 displays the dimensions. Material parameters can be provided from the authors upon request.

For the outside walls, the ambient temperature from the meteorological data serves as a boundary constraint. The wind direction and intensity are used to compute the external heat transfer coefficient. The zone-surrounding interior walls (a door has been omitted for simplicity), floors, and ceiling exhibit constant thermal boundary conditions because the room is assumed to be situated inside a heated building. The exterior surface of the interior walls and the upper story ceiling is given a constant boundary of 20°C. The surface temperature of the basement ceiling below the room is fixed at 15°C.

There is also a radiator in the zone, modelled as an ideal, virtual heating element with zero heat loss, directly connected to the air volume and producing 500 W of heat when needed. A heating system maintains the average zone fluid (air) temperature between 19 and 23°C (4 Kelvin regulation).

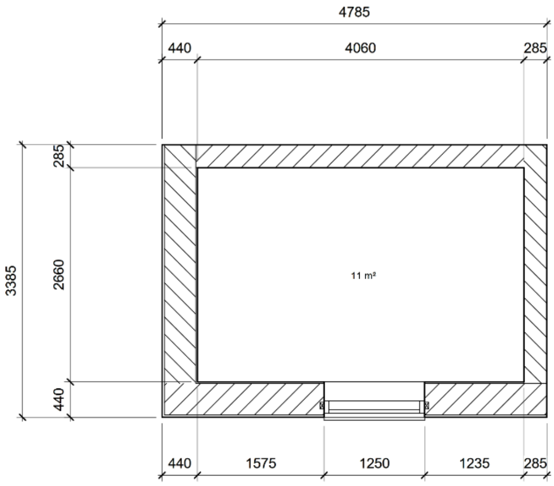


Figure 2: Floor plan and sectional view of the reference model (own illustration).

2.2. Simplified Model (sFEM)

The first challenge was to mimic the "lumpness" of the LPM model (here within the BPS software IDA ICE by Equa Solutions) in the FEM simulation software (here COMSOL). This was done by minimizing the discrete finite elements (mesh spacing) for the walls and air volume (see Figure 1 - right). However, since the FEM software cannot function on only one node for one wall the nodes were reduced to a minimal amount. Figure 1 shows the simplified FEM (sFEM) model where one rectangle represents one node.

This sFEM model is then used for all other physics simulations and models (except the geometrical model, gFEM). Here the thermal effects such as heat transfer convection and radiation were added. It is important to note that the sFEM model uses the same method of calculation as the LPM model; therefore, more advanced “multiphysics” modules of the FEM software cannot be used here.

The summary of the major physical effects for the sFEM and LPM models is presented in Figure 3.

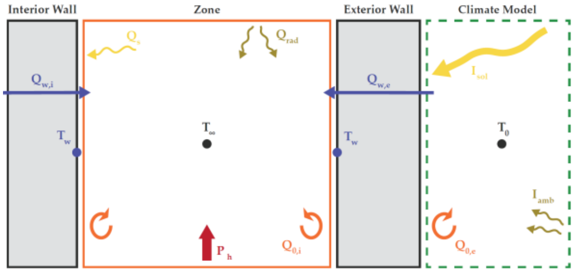


Figure 3: Summary of the physical effects used in the LPM model (own illustration).

Q_w is the interior and exterior heat flux, Q_0 is the convective heat flux for the interior and the exterior, Q_{rad} is the radiation from the objects such as wall and ceiling to the room (longwave radiation), Q_s is the direct solar radiation hitting the wall which consisting of the solar radiation I_{sol} (shortwave) and the ambient radiation I_{amb} (longwave). The temperature for the wall is T_W, the room temperature is T_∞ and the outside temperature is T_0.

The simplified FEM model uses several equations based on thermal physics to determine the equilibrium equations. One example is the simplified version of Fourier's transient law to calculate the heat transfer through solid walls:

$$\rho\ c_p \frac{\partial T}{\partial t} + \nabla(\vec{q}_c + \vec{q}_r) = \dot{Q} \tag{1}$$

Where ρ is the material density, c_p is the specific heat capacity, and $\partial T / \partial t$ is the temperature change per timestep. $\vec{q}_c$ denotes the heat flux by conduction; $\vec{q}_r$ the heat flux by radiation, and $\dot{Q}$ additional heat sources (if present).

As can be seen in Figure 4 the zone fluid temperatures of both models are very similar (0.09 K difference). Therefore, the sFEM model can be seen as a good reference for further improved FEM models. The large spikes are due to the switching of the heating system. A small shift on the time domain only indicates that the sFEM model reaches the switch point one timestep earlier. However, on the overall time scale this deviation is negligible.

The following subchapter analyzes further improved FEM models with the integrated multiphysics modules from the COMSOL software.

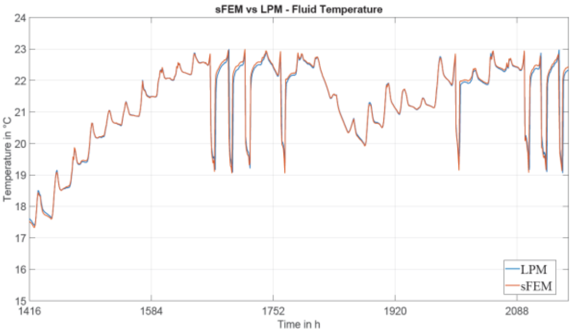


Figure 4 Fluid temperature comparison of the LPM and the simplified (sFEM) models in the month of March (own illustration).

2.3. Geometry and mesh-improved FEM (gFEM)

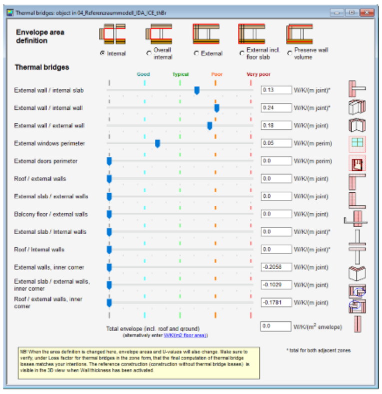
For this model (Figure 1 - left), we discard the simplified node and geometry setting of the sFEM. Thus, we are able to analyze the effect of a finer mesh and more detailed geometry. The geometrically improved model (gFEM), however, has the same simple physical models as the sFEM. Therefore, thermal bridges and inhomogeneous heat transfer can be correctly considered by the FEM software. In contrast, the LPM software considers a one-dimensional heat transfer only. Additional heat loss in the extended geometry model decreases the fluid temperature by 0.63 K on average (in March). This can also be seen in Figure 6 (gFEM).

This additional heat loss can be quantified and imported into the LPM software, therefore improving the LPM software. The resulting heat transfer coefficient is calculated as:

$$\Psi = \frac{Q_{tb}}{(T_\infty - T_0)L} \tag{2}$$

where Ψ is the heat transfer coefficient, Q_{tb} is the additional heat loss through the component acting as a thermal bridge; $T_\infty - T_0$ is the temperature difference between zone fluid and outside, and L is the length of the component.

The heat transfer coefficient Ψ is calculated for each building element. The corresponding length L of a thermal bridge represents the perimeter of the respective building element. The process of manually adding thermal bridges to the software IDA ICE is depicted in Figure 5.



Thermal bridges: object in 04_Referenzraummodell_IDA_ICE_thBr

Envelope area definition

● Internal ○ Overall internal ○ External ○ External incl. floor slab ○ Preserve wall volume

Thermal bridges

External wall / internal slab: 0.13 W/K/(m joint)*
External wall / internal wall: 0.24 W/K/(m joint)*
External wall / external wall: 0.18 W/K/(m joint)
External windows perimeter: 0.05 W/K/(m perim)
External doors perimeter: 0.0 W/K/(m perim)
Roof / external walls: 0.0 W/K/(m joint)
External slab / external walls: 0.0 W/K/(m joint)
Balcony floor / external walls: 0.0 W/K/(m joint)
External slab / internal walls: 0.0 W/K/(m joint)*
Roof / internal walls: 0.0 W/K/(m joint)*
External walls, inner corner: -0.2058 W/K/(m joint)
External slab / external walls, inner corner: -0.1029 W/K/(m joint)
Roof / external walls, inner corner: -0.1781 W/K/(m joint)
Total envelope (incl. roof and ground) (alternatively enter W/K/(m2 floor area)): 0.0 W/K/(m^2 envelope)

* total for both adjacent zones

NB! When the area definition is changed here, envelope areas and U-values will also change. Make sure to verify, under Loss factor for thermal bridges in the zone form, that the final computation of thermal bridge losses matches your intentions. The reference construction (construction without thermal bridge losses) is visible in the 3D view when Wall thickness has been activated.

Figure 5: Thermal bridge interface of IDA ICE with the imported heat transfer coefficients Ψ (own illustration)

The imported heat losses in the LPM model can be improved to an average fluid temperature difference of 0.22 K (in March), this results in an average 65% improvement for the fluid temperature (see Figure 6).

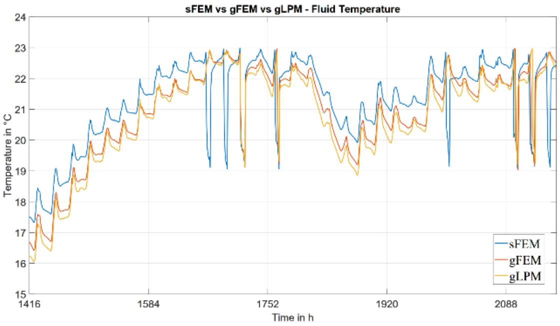


Figure 6: Fluid temperature comparison of the LPM and the simplified (sFEM), the geometrically FEM model (gFEM) and the geometrical lumped parameter model (gLPM) in the month of March (own illustration).

2.4. Convection

Our next model improves the heat transfer coefficient of the interior walls, ceiling and floor by including convection effects (cFEM). As the heat transfer coefficient h depends on a number of factors and parameters such as geometry, wind speed and characteristics, the calculation of h is not always easy and is therefore often an approximation.

Several methods can be used to improve the coefficient. In this work, a method known as Computational Fluid Dynamics (CFD) is used to compute the coefficient. It is one of the most accurate but also one of the most demanding methods. CFD provides a detailed simulation of fluid flow and convection within the model. However, it would not be appropriate to run a CFD simulation for each time step. Therefore, a steady-state CFD model is used to determine a correlation for the heat transfer coefficient h for the given boundary conditions and geometry. The result is the function:

$$h = C(T_\infty - T_w)^n \qquad (3)$$

Where h denotes the heat transfer coefficient, T_∞ the temperature of the zone, T_w the wall temperature; C and n are fitted constants.

This power function for h has been determined at various temperature differences between the temperature of the fluid and the (average) temperature of the wall to obtain suitable average parameters C and n. It is important to note that these parameters are only valid for given geometry and boundary conditions. The fitted constants, which vary for different building elements, are given in Table 1.

	Vertical wall	horizontal walls (upper side)	horizontal walls (upper side)
C	2.06	2.06	0.35
n	0.4	0.4	0.2

Table 1: Fitted constants from the CFD simulation (data (Schoplocher et al. 2023))

The results shown in Figure 7 indicate that less heat is transferred by convection as it has a lower heat transfer coefficient than both the sFEM and LPM models. Except for a short interval (which can be attributed to details of the heating control system), the average zone temperature in the cFEM is higher than in the sFEM. The average temperature drop is caused by a lower heat transfer coefficient for the inner surfaces. The zone temperature difference between the sFEM and the cFEM model is 1.52 K. The temperature difference of the outer walls doesn't have such a large temperature difference. This shows that the improved calculation of h can be important in any LPM software or building physics tool. The IDA ICE software used in this work, however, does not allow the direct implementation of the adjusted heat transfer coefficients. Therefore we must restrict our comparison to the different FEM models.

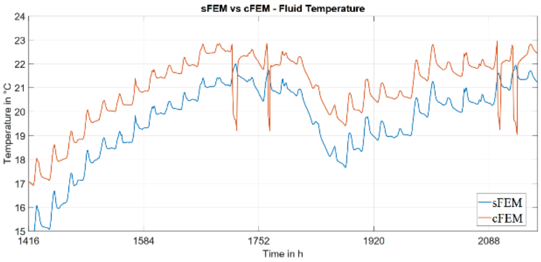


Figure 7: Fluid temperature comparison of the sFEM and the convective (cFEM) models in the month of March (own illustration).

2.5. Other models

More models were analyzed in the work. However, not all models showed a significant difference between the LPM and FEM models. Therefore, not every model is worth improving, this can only be determined for the specific case used in this work.

Models that did not show a significant difference were the following:

If only the mesh density of the model is improved without changing the geometry, the effect is rather insignificant. The reason for this is that without taking into account the additional heat losses from corners and thermal bridges, most of the heat flux into the walls and into the zone is one-dimensional, so more nodes do not improve the result.

Another interesting model is one that includes humidity (hFEM), taking into account moisture storage and transport through the solid building materials and the air volume, thus also accounting for the difference in thermal conductivity at different moisture levels. The cFEM model, in terms of moisture transfer and heat transfer coefficient as a function of moisture content, does not appear to have great advantages when only surface and fluid temperatures are considered. However, this may be different when considering more humid situations.

A detailed radiation simulation (rFEM) does not have much impact on the average temperature. The radiation model rFEM uses more advanced methods such as the hemicube or raytracing methods (COMSOL 2023). Although the calculation method is more accurate, the overall temperatures seem to remain the same. However, by using the FEM method of simulation, a finer temperature difference can be observed. One example are the local temperature peaks (because of direct radiation) on the interior walls instead of the usual average temperature increase (see Figure 8).

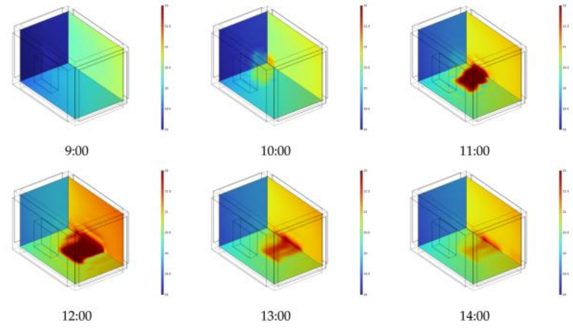


Figure 8: North and west wall surface temperature on 7 *November* inside the rFEM simulation (own illustration)

3. Summary

A summary table of all the models and their deviations from the LPM model is presented in Table 2. To exclude the initial sFEM model deviations from the LPM model, the modified (improved) models are only compared with the sFEM model.

Model	Average temperature difference (in relation to sFEM model)		Simulation time
	March (zone)	June (Zone)	Time
Reference LPM model (LPM)	0.08	0.13	13s
Simplified FEM model (sFEM)	0	0	1min 22s
Improved geometry (gFEM)	0.63	0.21	35min 12s
Improved lumped geometry (gLPM)	0.82	0.16	14s
Improved radiation (rFEM)	0.05	0.17	5min 19s
Improved humidity (hFEM)	0.07	0.12	3min 22s
Improved convection (cFEM)	1.52	0.29	4min 15s

Table 2: Comparison of all the physical models in relation to the sFEM model (own illustration)

4. Conclusion and Outlook

There are several notable differences between the FEM simulations (in COMSOL) and the LPM model in IDA ICE. We have derived suitable improvements for the LPM model concerning geometry, mesh, and internal convection. In contrast to this, improved radiation, heat transfer and moisture models in COMSOL do not show any significant potential for improvement. IDA ICE shows good accuracy in these areas, despite the limitations imposed by the physical simplifications. Compared to the FEM programme, the simulation time is significantly shorter by about 13 seconds.

In the geometric model (gFEM), improved mesh and additional heat loss through thermal bridges have been taken into account. These differences can be imported into IDA ICE using table values. As a result, the average temperature difference for the fluid improved from 0.63 K to 0.2 K. This allows the LPM model to produce more accurate results (gLPM).

The convective model (cFEM) also has room for development. With an average zone temperature difference of 0.98 K, the internal heat transfer coefficients h and their dependence on zone and surface temperatures can be modified. Although our set of parameters is adjusted to a particular combination of boundary conditions and geometry, this approach can be pursued for other situations in subsequent work.

The LPM software still needs to be improved in a few areas. These improvements will lead to more accurate and effective results. As a result, more accurate design can lead to a potential reduction in emissions and CO_2 emissions.

There are many factors to be taken into account that cannot be addressed in this work. So far, all research has used the same geometric model. Future research can use FEM to analyze individual rooms, an external wall or even entire buildings. There will be a new understanding of the complex thermal exchanges inside and outside the building. The study of HVAC (Heating, Ventilation and Air Conditioning) systems, which include underfloor heating, radiators and ventilation, is also an important objective for the future. This can be used in conjunction with specific, geometrically modelled heating devices and CFD simulation. It may also be possible to analyze convection in larger structures, taking into account phenomena such as the Marangoni effect.

Another emerging issue will be how to integrate these isolated aspects in an effective and reasonably straightforward manner. By improving already powerful and fast BPS software using FEM simulations techniques as an add-on tool, it might be possible to study an entire building structure in terms of radiation, convection, local heating units and possibly even fluid flow, along with even longer time intervals. This additional, more detailed information has the potential to advance our understanding of building behaviour, stimulate innovation and perhaps identify new technological areas that have not been targeted for development.

5. Acknowledgement

Part of this work was funded by the German Federal Ministry of Economic Affairs and Climate Action (BMWK) under ZIM Grant No. 16KN077922. The authors would like to acknowledge the fruitful cooperation with Consolinno Energy GmbH and Luxgreen Climadesign, as well as the support by Regensburg Center of Energy and Resources (RCER) at OTH Regensburg.

Literaturverzeichnis

Baba A, Mahdjoubi L, Olomolaiye P, Booth C, 2013. State-of-the-art on buildings performance energy simulations tools for architects to deliver low impact building (LIB) in the UK, 2013, 1867–1884.

Berger J, Gasparin S, Dutykh D, Mendes N, 2020. On the comparison of three numerical methods applied to building simulation: Finite-differences, RC circuit approximation and a spectral method. Building Simulation, 13 (1), 1–18.

Besuievsky G, García-Nevado E, Patow G, Beckers B, 2021. Procedural modeling buildings for finite element method simulation. Journal of Physics: Conference Series, 2042 (1), 12074.

Bring A, Sahlin P, Vuolle M, 1999. Models for Building Indoor Climate and Energy Simulation. Building Energy Analysis Tools Model Documentation, Stockholm, 110 S.

Brown G, 1990. The BRIS simulation program for thermal design of buildings and their services. Energy and Buildings, 14 (4), 385–400.

Clarke J, 2015. A vision for building performance simulation: a position paper prepared on behalf of the IBPSA Board. Building Performance Simulation, 1–7.

Clarke JA, 2020. Why tools for buildings and cities performance simulation need to evolve. Building and Cities.

COMSOL, 2023. Introduction to Computing Radiative Heat Exchange, 09.03.2023. Abgerufen am 09.03.2023, https://www.comsol.de/blogs/introduction-to-computing-radiative-heat-exchange

Cook RD, 1974. Concepts and applications of finite element analysis. A treatment of the finite element method as used for the analysis of displacement strain and stress. Wiley, New York u.a.

Crawley DB, Lawrie LK, Winkelmann FC, Buhl WF, Edem AE, 1997. What Next for Building Energy Simulation— A Glimpse of the Future. Building Simulation '97.

Deutscher Wetterdienst, 2018. Wetter und Klima - Deutscher Wetterdienst - Regensburg, 12.07.2022. Abgerufen am 12.07.2022, https://www.dwd.de/DE/wetter/wetterundklima_vorort/bayern/regensburg/_node.html

Jos Van Schijndel, 2015. BES with FEM: Building Energy Simulation using Finite Element Methods. COMSOL Conference.

Kusuda T, 1999. Early history and future prospects of building system simulation. Proceedings of Building Simulation '99, 1999.

Lucena CAT, Queiroz PCO, Debs ALHCE, Mendonça AV, 2014. DYNAMIC ANALYSIS OF BUILDINGS USING THE FINITE ELEMENT METHOD. In: Proceedings of 10th World Congress on Computational Mechanics, 08/07/2012. Editora Edgard BlücherSão Paulo, S. 4712–4726.

Mahmoud Ziada, Sertaç Tuhta, Eren Hayati Gençbay, Furkan Günday, Yosra Tammam, 2019. Analysis of Tunnel Form Building Retrofitted with CFRP using Finite Element Method. International Journal of Trend in Scientific Research and Development (IJTSRD).

Rao SS, 1982. The finite element method in engineering. Pergamon Press, Oxford, New York, Toronto, Sydney, Paris, Frankfurt Main i.e. Kronberg-Taunus, XXVI, 625 S.

Schoplocher DP, Ettengruber S, Steffens O, 2023. Improvements for building-performance simulations by a comparative finite-element method analysis. Energy and Buildings, 278, 112563.

Seem JE, Klein SA, Beckman WA, Mitchell JW, 1989. Transfer Functions for Efficient Calculation of Multidimensional Transient Heat Transfer. Journal of Heat Transfer, 111 (1), 5–12.